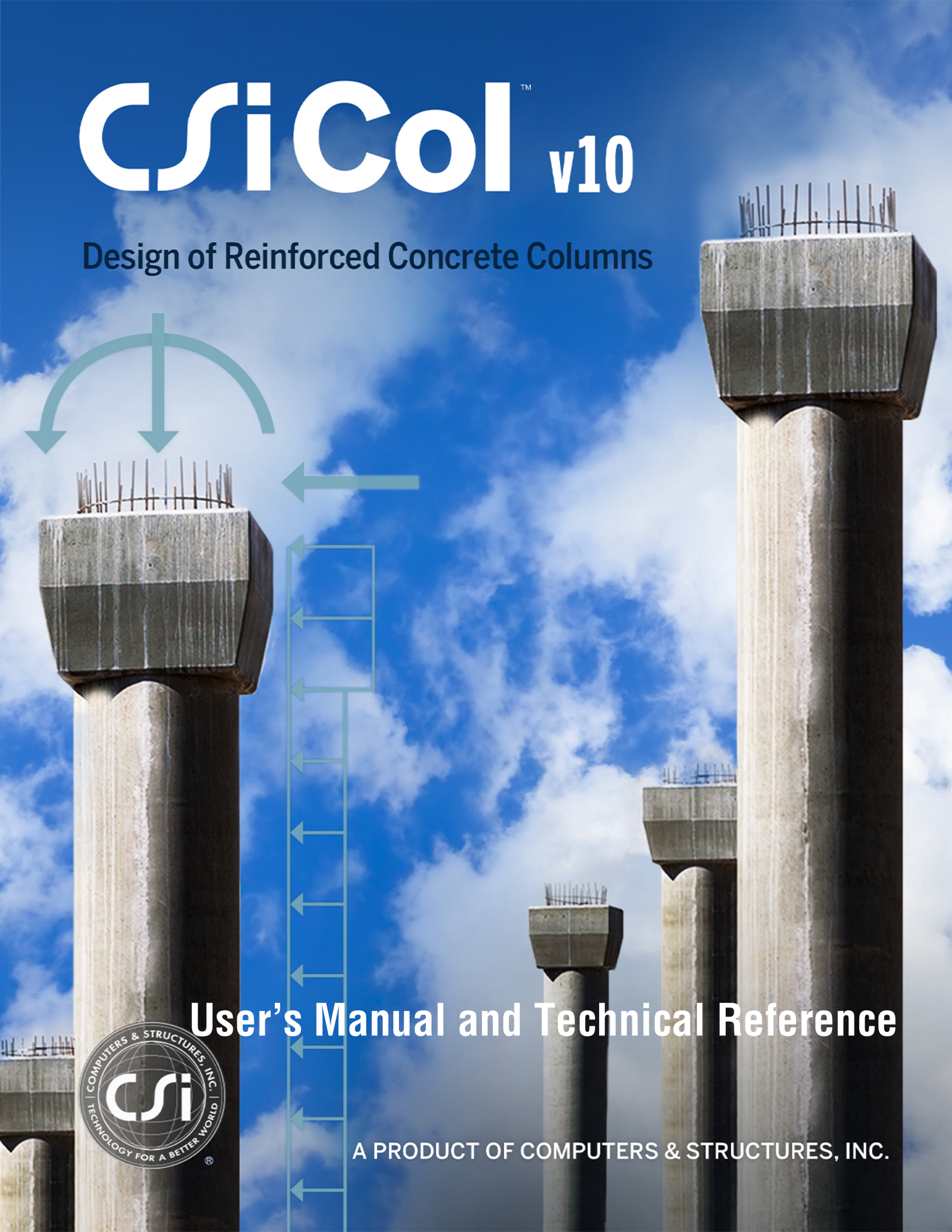
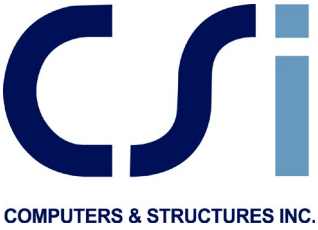
CSiCol™ v10

Design of Reinforced Concrete Columns



User's Manual and Technical Reference

A PRODUCT OF COMPUTERS & STRUCTURES, INC.



CSiCol™

User's Manual and Technical Reference

ISO COL050120M1 Rev. 0
Proudly developed in the United States of America **May 2020**

Copyright

Computers & Structures, Inc.
1464 N. California Blvd.
Walnut Creek, California 94596 USA
Phone: (510)649-2200
FAX: (510)649-2299

e-mail: info@csiamerica.com (for general information)
e-mail: support@csiamerica.com (for technical support)
web: www.csiamerica.com

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

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 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 design tool helps in automatically selecting the optimum reinforcement for specified actions using user-defined rules.

The program can generate a wide variety of results. Output includes the capacity interaction surface, PM curves, MM curves, moment-curvature curves and flexural stress contours 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 in any patterns such as point, line, rectangle and circle as required using several drawing and placement tools. Several standard (ASTM, Metric, and Imperial) and user-defined rebar sets may be used.

1.1. Compatibility with CSiCol v9

CSiCol files saved in version 9 (.cdb files) can be imported to CSiCol v10. Multiple columns can be imported, and the columns are imported along with their framing conditions, loadings, material assignments and slenderness properties. By default, the working units will be the same as that used in the older version of CSiCol. As of now, only US (inch) and SI (mm) unit systems are supported

in this version of CSiCol. . If the units in the CDB file are in any other system, it will be converted to either US (inch) or SI (mm) unit system. If the CDB file uses an older version of the design code available in CSiCol v10, it will open with the new version of the design code.

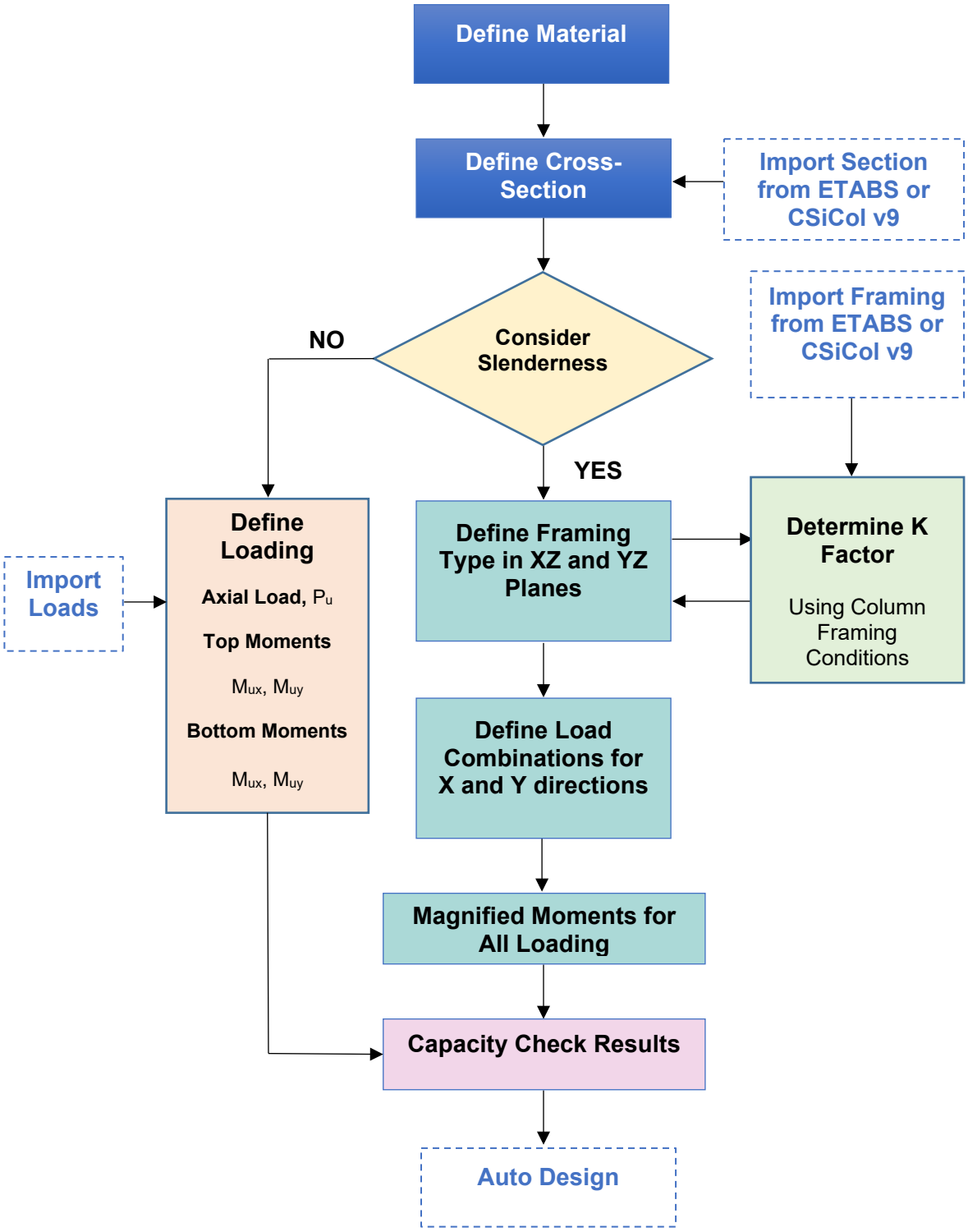


Figure 1.1 Overview of CSiCol Design and Analysis Process

1.2. Key Features

1.2.1. Design and Analysis Capabilities

- Use the Quick Design Wizar0d capabilities to guide you through the entire column design process.
- Import columns from ETABS along with the materials, sections, loads and framings
- 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 slender and short columns respectively.
- Perform analysis and design considering slenderness effects.
- Use Auto Design feature to determine the optimum reinforcement to satisfy user-defined parameters.

1.2.2. Slenderness Considerations

- Define the framing or import it from either ETABS or CSiCol v9.
- 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.

1.2.3. 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 predefined parametric shapes, including a variety of solid, hollow, and flanged shapes, in addition to a large collection of Standard Steel shapes.
- Import shapes from text files and .DXF Files
- Import sections from ETABS or CSiCol v9 files.
- 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 using several rebar patterns.
- Apply ASTM, Metric, Imperial or user-defined rebar sets.

1.2.4. Material Properties

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

1.2.5. Results Generated

- Compute and report basic geometric properties, such as area, moments of inertia, shear area and rebar area.

- Plot the flexural stress resulting from actions (P, Mx, M_y) on a column's cross-section.
- Generate capacity interaction surfaces and diagrams.
- View capacity details of the P-M diagram such as axial load, major axis moment, force and moment on concrete and rebar, strain, etc at various neutral axis depth.
- Plot multiple P-M or M-M curves in the same graph with varying values for either Compressive Strength of Concrete (fc'), Steel Tensile Strength (fy) or rebar sizes.
- Generate moment-curvature curves for any arbitrary column shape to determine performance and ductility.
- Display the location and orientation of the neutral axis corresponding to a specific loading.
- Compute capacity ratios based on different methods and determine adequacy of the section for the specified loads.
- Design the column in accordance with the specifications of the selected design code.

1.2.6. Miscellaneous

- Model composite cross-sections using several materials.
- Set the working units to US or SI.
- Use the comprehensive Help to apply the program efficiently.
- Create detailed, customized reports by adding information and graphics.
- Display a summary of some critical information regarding all the columns such as the governing load, rebar configuration, rebar ratio and the corresponding D/C ratio.
- Generate a one-page report for all the columns in the *Project File.*

1.3. Terminology

Some important terms that require clear understanding to help make the program more useful are defined below:

- **Project File** is where newly defined or pre-created columns can be saved for later use. Several columns can be added to an individual *Project File*.
- **Column** is a complete member with a cross-section, framing condition and load combinations. Several tools are available to create columns from within the *File Menu* including *Quick Design Wizard*, *Rectangular*, *Circular*, *Blank*, *Import ETABS* and *Import CDB*.
- **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 (unless a section has only one shape). Figure 1.2 shows a section made up of 2 shapes.
- **Shape** is the basic component that is used to create sections. Rebar can be added to individual shapes or anywhere in the section. Predefined shapes from standard libraries can be used and modified to create sections. Several tools are available to create shapes, including the *File Menu>Rectangular*, the *File Menu>Circular*, the *Draw Menu>Concrete Shapes*, the *Draw Menu>Steel Shapes*, the *Draw Menu>Add Shapes>Shape Library*, the *Draw Menu>Add Shapes>Steel Shapes* and the *Draw Menu>Add Shapes>Coordinates* buttons. Shapes can also be imported from a .txt file or a .DXF file using the tools available under *Draw Menu>Import Shapes.*
- **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 several libraries.

- **Modes:** You can switch between three different modes in CSiCol which are as follows:
  1. **Selection Mode** is activated when the *Edit Menu>Select* button is clicked. While in selection mode, you can select various shapes in a column section but cannot edit the shapes using the mouse. However, shapes can be edited using the *Property* grid that gets activated in the *Selection Mode*. Shapes can be aligned using the various buttons available in *Edit Menu>Align* and moved using the *Edit Menu>Edit>Move Sections* button while in the *Selection Mode*.
  2. **Drawing Mode** is activated when the drawing options for rebars or basic concrete and steel shapes are selected from *Home Menu>Draw>Rebars*, *Home Menu>Draw>Shapes*, *Draw Menu>Rebars*, *Draw Menu>Concrete Shapes* or *Draw Menu>Steel Shapes.* While in the *Drawing Mode*, the cursor in the working area changes to +. Multiple objects can be drawn in the *Drawing Mode.* The *Drawing Mode* can be exited by pressing the Esc key.
  3. **Results Mode** is activated when the results for the column section are viewed from the *Results Menu* or from the *Project Explorer>Columns>Results.* When this mode is active, the user is not allowed to make any changes to the section. Therefore, the tools under the *Edit Menu* and the *Draw Menu* are inactive in this mode. To exit the *Results Mode* and make changes to the section, select the desired section from the *Project Explorer>Columns.*
- **Axis:** Various editing operations, computed properties, and shape locations are referenced to the global X and Y axes. These axes help 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 CSI's SAP2000 and ETABS programs. In those programs, the 1-1 axis is used as the axis that passes through the members' longitudinal axis. A section showing the local axes is presented in Figure 1.3.

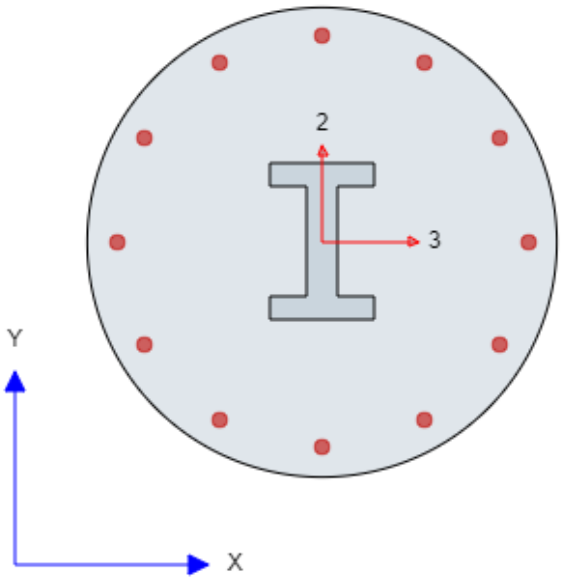


Figure 1.2 Section made of 2 Shapes

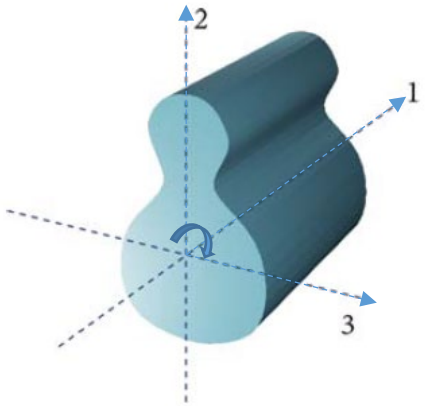


Figure 1.3: Section in 3D showing local axes

1.4. Results and Output

CSiCol generates the following types of results:

- Section Properties
- Stress Distribution
- Magnified Moments
- Section Capacity
- Moment Curvature
- Multiple Curves

1.4.1. Section Properties

The software reports the following properties for the section:

- Basic Properties: Includes the area, shear areas, moment of inertia and torsional constant of the selected section.
- Section Bounds: Includes the overall dimensions of the selected section and the centroid location with respect to both the global origin as well as the local origin.
- Additional Properties: Includes the radii of gyration and section moduli of the selected section.
- Rebar Properties: Displays the rebar area and rebar ratio in the selected section.

Note: For composite sections, concrete is considered as the base material and the transformed section properties are reported.

1.4.2. Stress Distribution

The program can display the Flexural Stress on the section for the selected load combination and section location. The stress is displayed independently as 2D or 3D color-coded contours on the working area.

1.4.3. 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. The final design loads can be viewed by using the *Results Menu>Design Loads* button or the *Project Explorer>Columns>Results>Design Loads* which gives access to the *Design Loads* form. Select the loading direction (Along X or Along Y) by clicking on the respective tab to view the loading in the desired direction. The moments under Along X and Along Y tab correspond to the moment about the X-axis and the Y- axis respectively.

The magnified moments are calculated based on the selected design code. Currently, the following design codes are supported for magnified moments calculation:

- ACI 318-08
- ACI 318-11
- ACI 318-14
- IS 456:2000
- CSA A23-3-04
- BS 8110-97
- Euro Code 2: 2004

1.4.4. Section Capacity

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

- P-M Interaction Curves

The capacity interaction curves are displayed as follows:

- Interaction Surface: The variation of all three parameters governing the section capacity (the axial load P, moment M_2 and moment M_3) of a column section is plotted in a color-coded 3D graph.
- P-M Curve: Load-moment interaction curves showing the relationship of axial load capacity and resultant moment capacity for a specified angle of neutral axis.
- M-M Curve: Moment-moment interaction curves showing a plot between moment capacity about the local 2-axis and moment capacity about the local 3-axis at a specified axial load level.

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.

- Capacity Ratio Check
  The program is capable of calculating and reporting the capacity ratios for column sections using three different methods (Moment sum at P, Moment Vector at P and Axial Capacity Ratio). In addition, the program can display the resultant moment and resultant moment angle. The program also reports whether the section is or is not adequate based on the maximum value of the capacity ratio calculated from different methods. For capacity ratios greater than one, the program reports the section as inadequate.

1.4.5. Moment Curvature

The moment-curvature curves may be plotted for any orientation of the neutral axis and for a specified level of axial load.

1.4.6. Multiple Curves

The program can generate multiple P-M or M-M diagrams in the same graph with varying values for either Compressive Strength of Concrete (fc'), Steel tensile strength (fy) or rebar sizes.

1.5. Auto Design

The Column Auto Design feature is a very effective, efficient and powerful tool for designing reinforcement for 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 program optimizes the design of the columns for a user-specified maximum capacity ratio limit. Limits may be imposed on the maximum and minimum rebar size and maximum rebar ratio used in the section. This feature is explained in further detail in Chapter 3.

1.6. Other User Support Documents

In addition to this manual, support documentation includes a help file. 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.

2. CSiCol's User Interface

CSiCol's user interface is highly user friendly and intuitive. The interface consists of a working window with various menus. Figure 2.1 shows CSiCol's main screen. The components of the interface are described in this chapter.

2.1. Working Window

The CSiCol graphical user interface is shown in Figure 2.1. It includes the main window, title bar, ribbon menu, project explorer and context-sensitive area. Each of these items is described in the bulleted list that follows.

- **Main Window**
  This window may be moved, resized, maximized, minimized, or closed using standard Windows operations. Refer to Windows help available on the Start menu for additional information about these items.
- **Main Title bar**
  The main title bar includes the program and model names. The main title bar is highlighted when the program is in use. Move the main window by left clicking in the main title bar and holding down the mouse button as you drag the window around the computer screen.
- **Ribbon Menu Bar**
  This menu bar contains the program's menus from which various sub menus can be selected to perform specific actions. Such as *File, Edit, Define, Draw, Results, Options* and *Help*.
- Working Area
  All creation and modification of shapes and sections take place in the working area. CSiCol automatically updates the working area as shapes are added, moved, resized, rotated and combined to create new sections of arbitrary geometry and size. The working area is filled with a "graph paper" type grid that is useful for placing, aligning, and resizing shapes and sections.
- **Project Explorer**
  The project explorer has 4 main views namely *Materials, Stress-Strain, Columns* and *Report*. The first two views of this explorer allow easy access to view and modify the properties of the materials and stress strain curves defined in the project file. These can be used for any of the columns in the project. Under the *Columns* view, the shapes,

rebar patterns, framings and loadings of all the columns in the project file can be accessed and modified. The results for the columns can also be accessed under the *Columns* view for each specific column. The *Report* view can be used to generate a one-page and the detailed report for the selected column.

- **Context-sensitive Area**
  This area contains the context-sensitive grids that get activated in response to the user action. Different grids get activated based on what button the user clicks or which shape the user selects. For example, the *Property* grid gets activated in this area when a shape or rebar is selected.

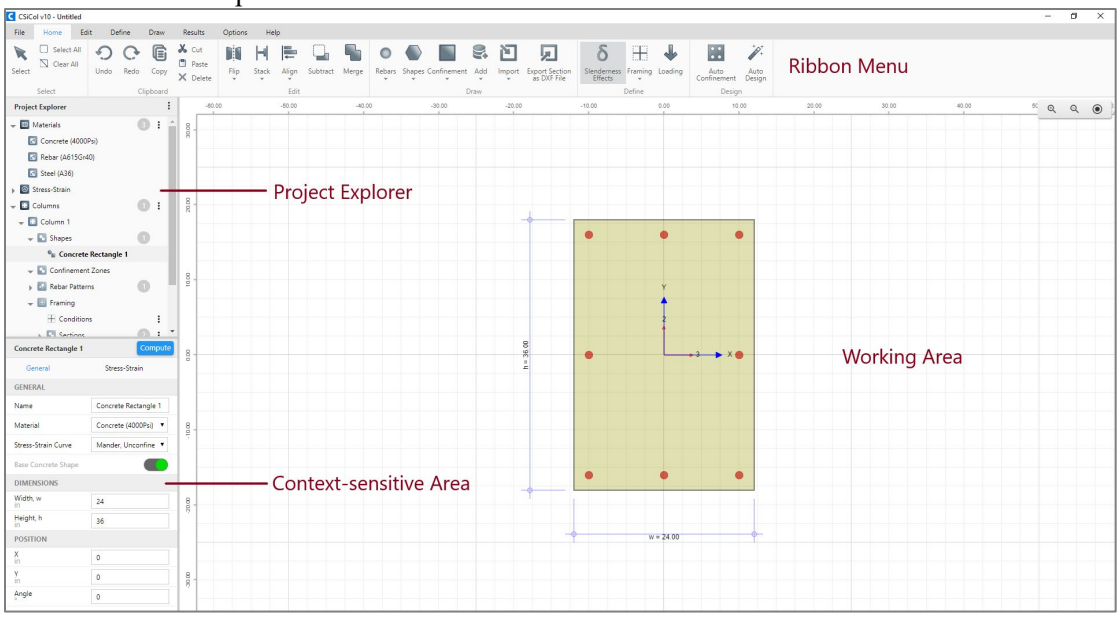


Figure 2.1: CSiCol Working Window

2.2. Customize the Program

This section briefly describes the different settings available under the *Options Menu*. CSiCol provides different settings to modify the different parameters used globally in the current *Project File*. Those parameters include the working units, the design code, view options, number formats and other display settings. These options have been placed under the *Options Menu* on the *Ribbon Menu Toolbar*.

2.3. General Options

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

- *Units Drop-down list*: The *Units* drop-down list identifies the various units available for use in the model. The selected units apply globally to all the sections in the Project file. Currently, US (kip-in) and SI (N-mm) units are available.
- *Code Drop-down list*: The *Code* drop-down list identifies the design codes available in CSiCol. Select the desired code from the drop-down list by clicking on it. The specified code will be used to calculate 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. Updating the design code will erase all previous results. So, the user will be asked to confirm before changing the design code.

- *Rebar Sets* button: The *Rebar Sets* button gives access to the *Add Rebar Sets* form where new rebars can be added and existing ones removed. New rebars can be added either by clicking on the  button and manually inputting the name and diameter of the bar or added from standard libraries such as ASTM by clicking on the Add button.

2.4. Number Formatting Options

The settings related to the number formats used to display all the numerical values in the *Project file* are grouped under the *Options Menu>Number Format*.

- *Decimals Drop-down list*: Use this drop-down list to change the number of decimal places to be displayed for all the numeric values.
- *Separator Drop-down list*: Use this drop-down list to select different separators for thousands and decimals.
- *Show Exponent* button: Turn on the *Show Exponent* toggle to display the numeric values in exponential notation.

2.5. View Options

The settings to show or hide various elements of the working area such as grid, ruler and global axis are grouped under the *Options Menu>View*.

- *Grid* button: Use this toggle to display or hide the grid in the working area.
- *Ruler* button: Use this toggle to display or hide the ruler in the working area.
- *Global Axis* button: Use this toggle to display or hide the global x and y-axis in the working area.
- *Dimensions* button: Use this toggle to display or hide the dimensions of the shapes and rebars in the working area.

2.6. Diagram Options

The settings to display the points and labels of diagram data are grouped under *Options Menu>Diagrams.* See section 5.6 of this manual for more details.

- *Points* button: Use this toggle to display or hide the data points in the diagrams.
- *Labels* button: Use this toggle to display or hide the data point labels in the diagrams.

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.

3.1. 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, Rebar Patterns and the Auto Design feature, which are described in the subsequent sections of this chapter. Tools for editing shapes in a section are described in Chapter 4.

3.2. Methods for Creating Columns

Columns can be created using the following methods:

- ***File Menu>Quick Design Wizard*** button: This accesses the *Quick Design Wizard*. *The Quick Design Wizard* form provides access to all 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 > Blank Model*** button: This 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 button under the *Draw Menu* and modified to obtain the desired section using the buttons available under the *Edit Menu*. Defining the column framing and loading and generating results must be completed independently using the various buttons available under the *Define Menu* and the *Results Menu*. Reports for the column can be viewed using the buttons available under the *Home Menu>Report* or from the *Project Explorer>Report*.
- ***File Menu > Rectangular Model*** button: This can be used to create a rectangular column. Defining the column framing and loading and generating results must be

completed independently using the various buttons available under the *Define Menu* and the *Results Menu*. Reports for the column can be viewed using the buttons available under the *Home Menu>Report* or from the *Project Explorer>Report*.

- ***File menu > Circular Model*** button: This can be used to create a circular column. Defining the column framing and loading and generating results must be completed independently using the various buttons available under the *Define Menu* and the *Results Menu*. Reports for the column can be viewed using the buttons available under the *Home Menu>Report* or from the *Project Explorer>Report*.

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.

3.2.1. Quick Design Wizard

The *Quick Design Wizard*, shown in Figure 3.1, appears when you start CSiCol.

The *Quick Design Wizard* makes creating a model quick and easy because this form provides access to all the other forms needed to create and analyze the model as well as to generate results. The general process involved in using the *Quick Design Wizard* is as follows:

1. Click on the *File Menu > Quick Design Wizard* button to access the *Quick Design Wizard* form. The *Wizard* provides access to all the other forms needed to create and analyze the model as well as to generate results.
2. **Project Details**: Click on the *Define* button next to *Project Details* to access the *Project Information* form where the details of the project such as Project Name, Project Code, Engineer and Company can be filled in.

   The *Project Information* form can be accessed later for viewing and modification from the *Define Menu>Project>Project Details* button. The information specified on the *Project Information* form displays at the start of the detailed reports generated for the Project File.
3. **Units and Code:** Click on the *Define* button next to *Units and Code* to access the *Options* form. Review and accept the defaults on the various parameters on the form such as working units, design codes and number format or use the drop-down lists and edit boxes to specify the desired values.

   These settings can also be modified later from the drop-down lists available under the *Options Menu*.
4. **Material Properties**: Click on the *Define* button next to *Material Properties* to access the *Material Properties* form. The *Material Properties* form has separate tabs for concrete and rebar materials. Review and accept the defaults or use the edit boxes to specify the desired values.

   The *Material Properties* form can also be accessed later from the *Define Menu>Project>Materials* or by clicking on the ⋮ button next to the *Project Explorer>Materials.*

5. **Column Name:** Accept the default name or modify it be typing in the edit box next to *Column Name.*

6. **Section Shape:** Select the shape type (Rectangular or Circular) to access the *Rectangular Column* or the *Circular Column* form and define the shape name, cross-section dimensions, clear cover, rebar size and rebar distribution.

   The shapes and rebars can be modified later by selecting them and editing the dimensions and other parameters in the *Property* grid.

7. **Consider Slenderness Toggle:** When this toggle is turned on, CSiCol will consider slenderness effects and column framing will need to be defined using the *Define Framing* button. When this toggle is turned off, CSiCol will ignore slenderness effects, the *Define Framing* button will be unavailable, and you can skip to Step 9.

8. **Define Framing:** This button is active only if slenderness effects are being considered. Clicking this button accesses the *Column Framing Conditions* form.

   This form can also be accessed later from the *Define Menu>Slenderness> Framing Conditions* or by clicking on the button next to the *Project Explorer>Columns>Framing>Conditions.*

9. **Define Loading:** Detailed or Simple Loading may be defined. If the *Consider Slenderness* toggle is turned on, 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* toggle is turned off, CSiCol will automatically display the form for defining the load case in the simple mode.

   The *Define Loading* form can also be accessed later using *Define Menu> Loading>Loading* or by clicking on the button next to the *Project Explorer>Columns>Loading.*

10. **Results:** After the column section and its loading and framing conditions have been defined, the results can be viewed using the various options available under the Results section. Click on the *Capacity Ratio* button to determine if the column section is adequate for the loading defined. To view the detailed results, click the *Detailed Results* button. The *Interaction Diagrams* button can be used to access the PMM interaction diagrams and the *Generate Report* button generates a one-page report for the defined column. All these results along with some additional results can be viewed from the *Results Menu* or from the *Project Explorer>Results* after closing the wizard. The report can be viewed from the *Project Explorer>Report.*

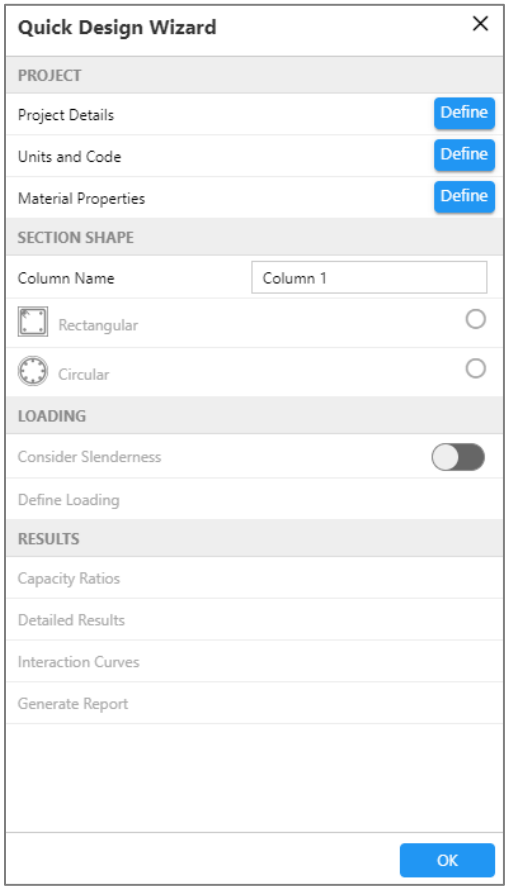


Figure 3.1 *Quick Design Wizard* Form

3.2.2. Blank Model

When the *Blank Model* button is used, CSiCol adds a blank working area to the current *Project File*.

The basic procedure for creating a section using the *File Menu>Blank Model* is as follows:

1. Select the *File Menu>Blank Model* button to add a blank working area to the Project file.
2. The program will open with the default working units and design code. Use the drop-down lists in the Options Menu>General to change these settings, if necessary.
3. Use the Define Menu>Stress Strain>Concrete button or click on the ⋮ button next to the Project Explorer>Stress-Strain>Concrete to access the Define Concrete Confinements form and define new confinement models or to modify the default one.

4. Use the Define Menu>Stress Strain>Steel button or click on the button next to the Project Explorer>Stress-Strain>Steel to access the Define Steel *Stress-Strain* form and define new steel stress-strain models or to modify the default one.

5. Use the *Define Menu>Project>Materials* button or the button next to the *Project Explorer>Materials* to access the *Material Properties* form and define the concrete and rebar properties.

6. Use one of the following tools to add a shape to the current column section:

    a. Select any of basic concrete shapes available in *Draw Menu>Concrete Shapes* to draw in the working area.

    b. Select any of basic steel shapes available in *Draw Menu>Steel Shapes* to draw in the working area.

    c. Click on the *Draw Menu>Add Shapes>Shape Library* button to access a selection list of different shapes.

    d. Click on the *Draw Menu>Add Shapes>Steel Shapes* button to access a library of standard steel shapes.

    e. Click on the *Draw Menu>Add Shapes>Coordinates* button to access the *Add Points* form where the coordinate values of the desired shape can either be entered directly or imported from external sources using the *Import* button.

    f. Click on the *Draw Menu>Import Shapes>Text File* button to import a shape from coordinates saved in a text file.

    g. Click on the *Draw Menu>Import Shapes>DXF File* button to import a shape from a DXF file.

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

8. Use the buttons available on the *Edit Menu* such as *Flip, Merge, Stack, Align* and *Edit* to modify the shapes to obtain the desired section.

9. Use the various options available in *Draw Menu>Rebars* to add rebars into the section. The rebars can be added as points, line or in rectangular or circular pattern as required.

10. If necessary, click on a shape or rebar to activate the *Property* grid and edit their dimensions and other properties numerically. The *Property Grid* for concrete and steel shapes have a tab for the stress strain model where the properties of the stress strain model assigned to the shape can be viewed and modified, if necessary. The *Property* grid can also be accessed by selecting the shape or rebar from the *Project Explorer* under *Columns* view.

    Note: The dimensions of shapes created using the *Draw Menu>Concrete Shapes>Polygon* button can only be edited by modifying the nodal coordinates either from the *Points* tab of *Property* grid or from *Edit Menu>Edit>Edit Points.*

The parametric shapes can be converted to polygon from *Edit Menu>Edit>Edit Points.*

11. If the column is being designed as slender, click the *Define Menu>Slenderness>Slenderness Effects* button to enable this feature. If the column is not slender, skip this step and Step 12.
12. If the column is slender, use the *Define Menu>Slenderness>Framing Condition* button or click on the ⋮ button next to the *Project Explorer>Columns>Framing>Conditions* to define the framing scenario for the column cross-section.

    Note: This button is activated only when slenderness effects are being considered.

13. Use the *Define Menu>Loading>Loading* button or click on the ⋮ button next to the *Project Explorer>Columns>Loading* to define loads for the column cross-section.
14. Use the *Home Menu>Design>Auto-Design* button to apply an iterative process to determine the minimum reinforcement and the reinforcement layout for the governing load combination that satisfies the user-defined limits for smallest and largest bar sizes, clear cover, clear spacing, maximum rebar ratio and maximum D/C ratio.

    Note: This option is only available for rectangular and circular reinforced sections.

15. Use the various options available under the *Results Menu* or the *Project Explorer>Columns>Results* to review outputs for the column cross-section.
16. Click on the options available under the *Report* view of the *Project Explorer* to view the column summary, generate a one-page report or a detailed report for the selected column.

3.2.3. Rectangular Model

Use the *Rectangular Model* button to add a new rectangular column section to the current Project file. The following provides a basic approach for adding the rectangular column to the active *Project file*.

1. Select the *File Menu>Rectangular Model* button to add a rectangular column to the Project file.
2. The program will open with the default working units and design code. Use the drop-down lists in the Options Menu>General to change these settings, if necessary.
3. Use the Define Menu>Stress Strain>Concrete button or click on the ⋮ button next to the Project Explorer>Stress-Strain>Concrete to access the Define Concrete Confinements form and define new confinement models or to modify the default one.

4. Use the Define Menu>Stress Strain>Steel button or click on the button next to the Project Explorer>Stress-Strain>Steel to access the Define Steel Stress-Strain form and define new steel stress-strain models or to modify the default one.

5. Use the Define Menu>Project>Materials button or the button next to the Project Explorer>Materials to access the Material Properties form and define the concrete and rebar properties.

6. If necessary, click on the shape or rebar to activate the Property grid and edit their dimensions and other properties numerically. The Property Grid for concrete and steel shapes have a tab for the stress strain model where the properties of the stress strain model assigned to the shape can be viewed and modified, if necessary. The Property grid can also be accessed by selecting the shape or rebar from the Project Explorer under the Columns view.

7. If the column is being designed as slender, click the Define Menu>Slenderness>Slenderness Effects button to enable this feature. If the column is not slender, skip this step and Step 8.

8. If the column is slender, use the Define Menu>Slenderness>Framing Condition button or click on the button next to the Project Explorer>Columns>Framing>Conditions to define the framing scenario for the column cross-section.

   Note: This button is activated only when slenderness effects are being considered.

9. Use the Define Menu>Loading>Loading button or click on the button next to the Project Explorer>Columns>Loading to define loads for the column cross-section.

10. Use the Home Menu>Design>Auto-Design button to apply an iterative process to determine the minimum reinforcement and the reinforcement layout for the governing load combination that satisfies the user-defined limits for smallest and largest bar sizes, clear cover, clear spacing, maximum rebar ratio and maximum D/C ratio.

    Note: This option is only available for rectangular and circular reinforced sections.

11. Use the various options available under the Results Menu or the Project Explorer>Columns>Results to review outputs for the column cross-section.

12. Click on the options available under the Report view of the Project Explorer to view the column summary, generate a one-page report or a detailed report for the selected column.

3.2.4. Circular Model

Use the *Circular Model* button to add a new circular column section to the current *Project file*. The following provides a basic approach for adding the circular column to the active *Project file*.

1. Select the File Menu>Circular Model button to add a circular column to the Project file.
2. The program will open with the default working units and design code. Use the drop-down lists in the Options Menu>General to change these settings, if necessary.
3. Use the Define Menu>Stress Strain>Concrete button or click on the ⋮ button next to the Project Explorer>Stress-Strain>Concrete to access the Define Concrete Confinements form and define new confinement models or to modify the default one.
4. Use the Define Menu>Stress Strain>Steel button or click on the ⋮ button next to the Project Explorer>Stress-Strain>Steel to access the Define Steel Stress-Strain form and define new steel stress-strain models or to modify the default one.
5. Use the Define Menu>Project>Materials button or the ⋮ button next to the Project Explorer>Materials to access the Material Properties form and define the concrete and rebar properties.
6. If necessary, click on the shape or rebar to activate the Property grid and edit their dimensions and other properties numerically. The Property Grid for concrete and steel shapes have a tab for the stress strain model where the properties of the stress strain model assigned to the shape can be viewed and modified, if necessary. The Property grid can also be accessed by selecting the shape or rebar from the Project Explorer under the Columns view.
7. If the column is being designed as slender, click the Define Menu>Slenderness>Slenderness Effects button to enable this feature. If the column is not slender, skip this step and Step 8.
8. If the column is slender, use the Define Menu>Slenderness>Framing Condition button or click on the ⋮ button next to the Project Explorer>Columns>Framing>Conditions to define the framing scenario for the column cross-section.

   Note: This button is activated only when slenderness effects are being considered.

9. Use the Define Menu>Loading>Loading button or click on the ⋮ button next to the Project Explorer>Columns>Loading to define loads for the column cross-section.

10. Use the Home Menu>Design>Auto-Design button to apply an iterative process to determine the minimum reinforcement and the reinforcement layout for the governing load combination that satisfies the user-defined limits for smallest and largest bar sizes, clear cover, clear spacing, maximum rebar ratio and maximum D/C ratio.

    Note: This option is only available for rectangular and circular reinforced sections.

11. Use the various options available under the *Results Menu* or the *Project Explorer*>Columns>Results to review outputs for the column cross-section.
12. Click on the options available under the Report view of the Project Explorer to view the column summary, generate a one-page report or a detailed report for the selected column.

3.3. Methods for Importing Columns

Columns can be imported from the following two sources:

- ETABS
- Older version of CSiCol

3.3.1. Import ETABS Model

The *Import ETABS Model* button allows the user to select an existing ETABS model and import the concrete columns defined in the model into CSiCol. The basic procedure for importing an ETABS model into CSiCol is as follows:

1. Click on the *Import ETABS Model* button to access the *Open ETABS Model* form. Find and select the ETABS model file to be selected and click on the *Open* button.
2. Select the load cases or load combinations that you wish to import into CSiCol.

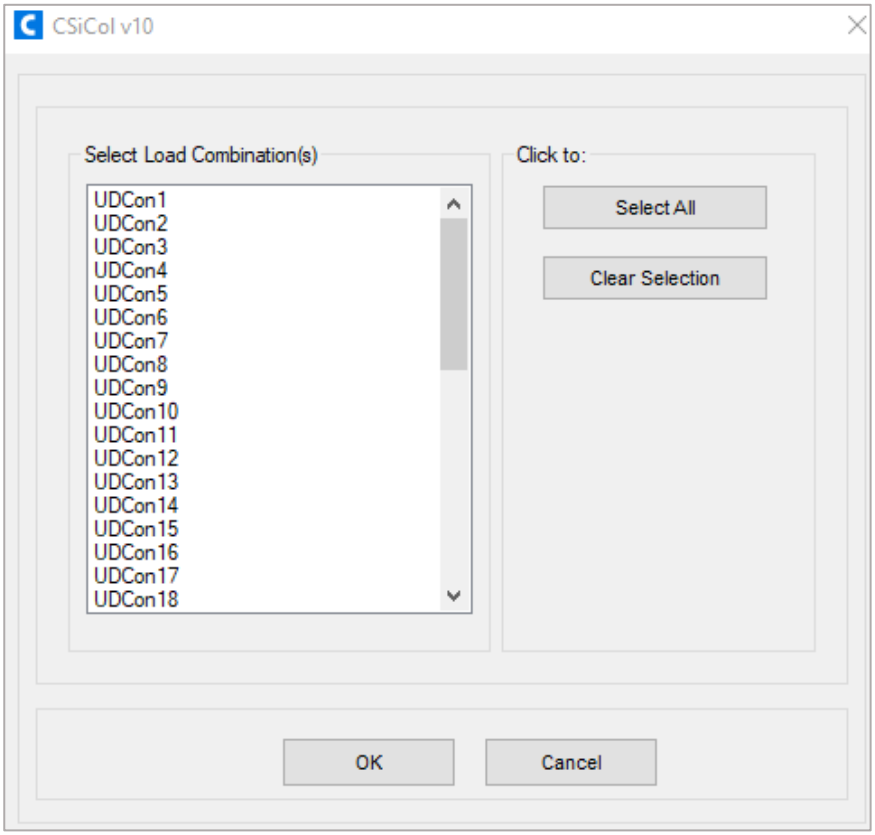


Figure 3.2: Select Load Cases/Combinations to Import

3. Once you select the load combinations, the *ETABS Model* form will be accessed where you can view the 3D model and visually select the columns you wish to import. Multiple columns can be selected, and the properties of the selected columns can be viewed under the *Properties* grid. Columns can also be selected based on their section and story location by clicking on the  button in the context sensitive toolbar.

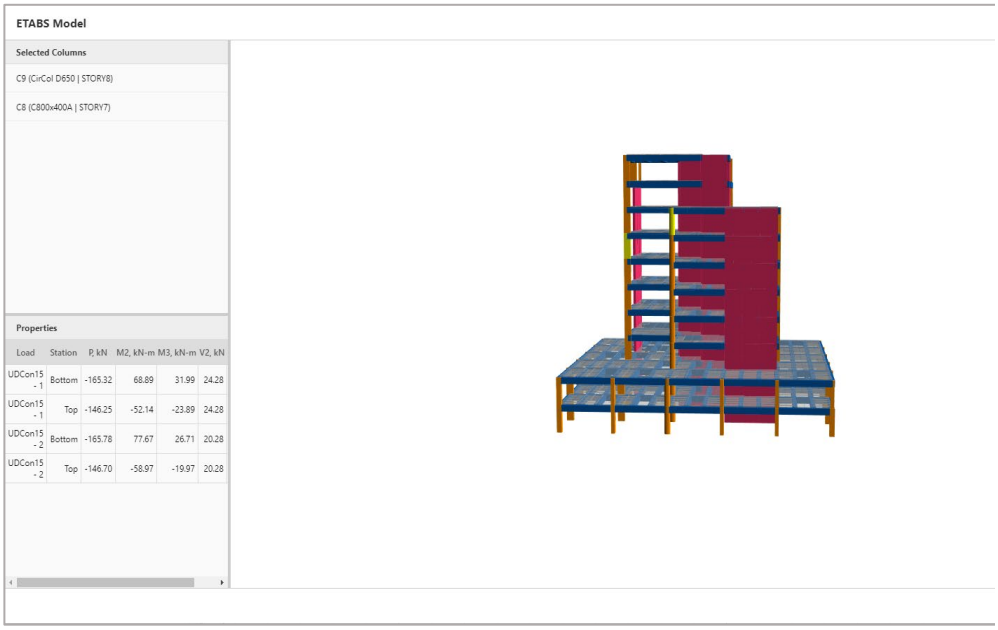


Figure 3.3 Importing Columns from ETABS Model

Note: This feature will only work if ETABS is installed on the computer.

If the file path for ETABS executable has not been set before, you will first be asked to set it. The column sections are imported along with their loading conditions, material assignments and framing conditions. The Effective Length factors (k) are not imported. By default, the working units will be the same as the database unit in ETABS. As of now, only US (inch) and SI (mm) unit systems are supported in CSiCol.

Note: If the units setting in the ETABS file is in MKS system, it will be converted to SI (mm) unit system.

If the building model has not been analyzed and designed in ETABS, only the model geometry will be imported. If the column is failing in ETABS, it is still possible to import the column, but the reinforcements will not be available.

Important Note: The clear cover in CSiCol refers to the cover to the MAIN reinforcing bars. Since ETABS defines the clear cover based on the ties / confinement bars, the tie diameter will be added to the clear cover when importing from ETABS into CSiCol.

3.3.2. Import CDB File

This button can be used to import a CDB file i.e a CSiCol file of an older version.

Note: Multiple columns can be imported and the columns are imported along with their framing conditions, loadings, material assignments and slenderness properties. By default, the working units will be the same as that used in the older version of CSiCol. As of now, only US (inch) and SI (mm) unit systems are supported in this version of CSiCol. If the units in the CDB file are in any

other system, it will be converted to either US (inch) or SI (mm) unit system. If the CDB file uses an older version of the design code available in CSiCol v10, it will open with the new version of the design code.

3.4. 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:

- Basic Concrete Shapes
- Basic Steel Shapes
- Shape Library
- Steel Shapes

3.4.1. Basic Concrete Shapes

Select any of basic concrete shapes available under *Draw Menu>Concrete Shapes* to draw in the working area. There are several basic shapes available such as rectangle, box, circle and tee. Shapes of any arbitrary geometry can also be drawn graphically in the working area by selecting the *Polygon* shape

3.4.2. Basic Steel Shapes

Select any of basic steel shapes available under *Draw Menu>Steel Shapes* to draw in the working area. There are several basic shapes available such as tube, I shape, pipe, angle and tee.

3.4.3. Shape Library

Click on the *Draw Menu>Add Shapes>Shape Library* button to access a selection list of different shapes. The shapes available in the *Shape Library* include AISC Steel Shapes, BS Steel Shapes, CISC Steel Shapes, Basic Concrete and Steel Shapes, Box Girder Shapes, Bridge Pier Shapes, Cold Formed Shapes, Fillet and Chamfer, Plate Library, Rounded Steel Shapes and Shear Walls.

3.4.4. Steel Shapes

Select any of the standard steel shapes by clicking on the *Draw Menu>Add Shapes>Steel Shapes* button.

3.5. Add Shape by 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 comma separated, tab separated or spaced .txt file.

- Click on the *Draw Menu>Add Shapes>Coordinates* button to access the *Add Points* form where the coordinate values of the desired shape can either be entered directly or imported from external sources using the *Import* button.
- Click on the *Draw Menu>Import Shapes>Text File* button to import a shape from coordinates saved in a text file.

3.6. Add Shape by Importing .DXF File

A shape can be imported from a .DXF file created using AUTOCAD.

- Click on the *Draw Menu>Import Shapes>DXF File* button to import a shape from a DXF file.

Note: This feature can only be used for importing shapes. Rebar layouts cannot be imported using .DXF File.

3.7. Export Shapes

The *Export Section as DXF File* button under *Draw Menu>Export* allows the user to save the selected shape as a DXF file which can be opened using AutoCAD.

3.8. Add Rebars

Rebars can be added to the section in different patterns as required using the buttons available under *Draw Menu>Rebars.*

- *Point*: Use this button to draw a single rebar in the working area with a mouse click at the desired location.
- *Line*: Use this button to draw a single line of rebar in the working area.
- *Rectangle*: Click on this button to draw rebar in a rectangular pattern.
- *Circle*: Click on this button to draw rebar in a circular pattern.

Important Note:

- The number of rebar in the pattern, clear cover, and minimum clear spacing can be modified using the *Property* grid (Figure 3.4) which is activated in the context sensitive area by selecting the rebar either by clicking on it in the working area or selecting it from the *Project Explorer>Columns>Rebar Patterns.*
- The clear cover in CSiCol is defined based on the clear cover to the MAIN reinforcement bars, not the ties / confinement bars.
- *Base Rebar Pattern* – This assignment is required for the calculation of Magnified Moments.
  - By default, the first rebar/ rebar layout drawn will be set as the "*Base Rebar Pattern*" under **General** Tab in property grid.
  - This field is allowed to toggle only when there are two or more rebar/ rebar layouts drawn in the canvas.
  - Assigning a rebar/ rebar layout as the *Base Rebar Pattern* will automatically remove the *Base Rebar Pattern* assignment from the previous rebar/ rebar layout.
  - When the rebar/rebar layout that is currently set as the *Base Rebar Pattern* is deleted, the first rebar/ rebar layout from the order drawn is set as the *Base Rebar Pattern.*

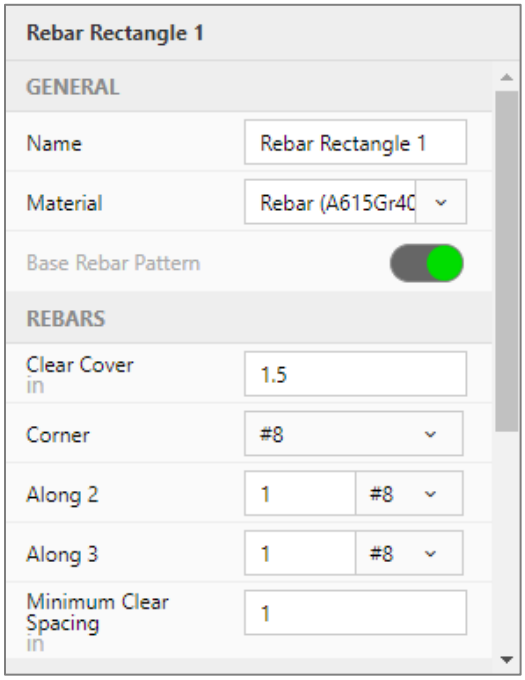


Figure 3.4 *Property* grid for rebars

3.9. Add Columns to Project File

Use the *Define Menu>Project>Columns* button or the ⋮ button besides *Project Explorer>Columns* to access the *Define Column* form where new columns can be added to the Project file and existing ones can be renamed or removed. The columns in the *Project File* can be viewed and modified by selecting the desired column from the *Project Explorer>Columns.*

3.10. Create Composite Columns

Composite columns can be created by simply placing the shapes of different materials one over the other in the desired way. It is not necessary to create holes before placing shapes over one another since CSiCol already accounts for the overlapped area. The procedure to move shapes to a desired location has been explained in Chapter 4.

Note: For composite sections, concrete is considered as the base material and the transformed section properties are reported in the *Results*.

3.11. Materials, Stress Strain Models and Confinement Models

3.11.1. Define Material Properties

Use the *Define Menu>Project>Materials* button to access the *Define Materials* form (Figure 3.5) where existing materials can be viewed or modified, and new materials can be added from the *Material Library.* A list of existing materials is displayed under the *Materials* section and the

properties of the selected material is displayed under the *Material Properties* section. The properties can be modified by directly typing in the desired values into the edit boxes provided. New materials can be added using the *Material Library* form which can be accessed by clicking on the  button. The *Material Library* form provides a selection of concrete, rebar and steel materials from several material libraries such as USA, China and Europe.

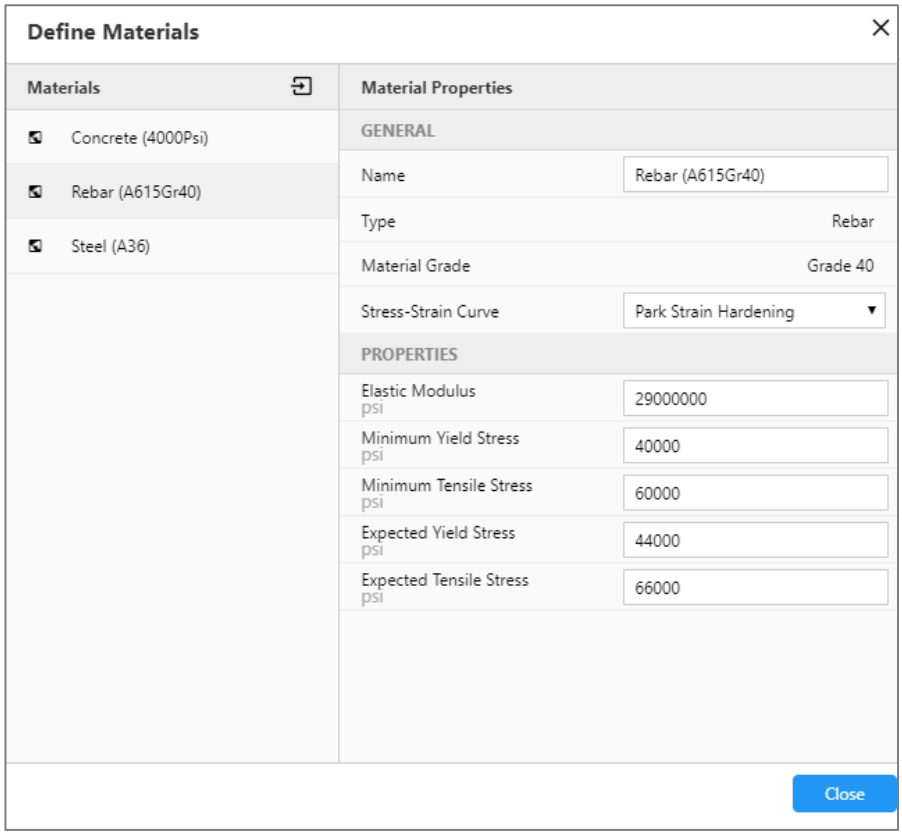


Figure 3.5 *Define Materials* Form

Note: The properties of existing materials can also be modified by selecting them from the *Project Explorer* under the *Materials* view. The *Define Materials* form can also be accessed by clicking on the button besides *Project Explorer>Materials.*

3.11.2. Define Confinement Models and Stress Strain Models

The buttons available under *Define Menu>Stress Strain* allow the user to define the stress strain models for concrete and steel.

- *Concrete* button: Click on this button to access the *Define Concrete Confinements* form where existing confinement models can be viewed or modified, and new confinement models can be added. A list of existing confinement models is displayed under the *Confinement Models* section and the properties of the selected confinement model is displayed under the *Properties* section. The properties can be modified by directly

typing in the desired values into the edit boxes provided.
New confinement models can be added using the *Add Confinement Model* form which can be accessed by clicking on the + button. The *Add Confinement Model* form provides a selection of confinement models. The *Define Concrete Confinement* form can also be accessed by clicking on the ⋮ button next to the *Project Explorer>Stress-Strain>Concrete.*

- *Steel* button: Click on this button to access the *Define Steel Stress-Strains* form where existing stress-strain models can be viewed or modified, and new stress-strain models can be added. A list of existing stress-strain models is displayed under the *Stress-Strain Models* section and the properties of the selected stress-strain model is displayed under the *Properties* section. The properties can be modified by directly typing in the desired values into the edit boxes provided.
  New stress-strain models can be added using the *Add Stress-Strain Model* form which can be accessed by clicking on the + button. The *Add Stress-Strain Model* form provides a selection of stress-strain models. The *Define Steel Stress-Strain* form can also be accessed by clicking on the ⋮ button next to the *Project Explorer>Stress-Strain>Steel.*

Note: The properties of existing concrete confinement models or steel stress-strain models can also be modified by selecting them from the *Project Explorer* under the *Stress Strain* view.

The default concrete model is the Mander's Unconfined Model. The default model for rebar is the Park Strain Hardening Model.

The stress-strain model for concrete and steel are assigned to the shape. The stress-strain model used for a concrete or steel shape can be changed by choosing the desired stress-strain model from the Stress Strain Curve drop down list in the General tab of the Property grid. The properties of the stress-strain model assigned to the shape can be modified under the Stress-Strain tab of the Property grid. The Property grid can be activated by selecting the desired shape either by clicking on it in the working area or selecting it from the Project Explorer under the Columns view.

The stress-strain model for rebar is assigned to the rebar material. The stress-strain model used for the rebar can be changed from the Stress-Strain Curve drop down list of the Material Properties grid. The properties of the stress strain model assigned to the rebar material can be modified under the Stress-Strain tab of the Material Properties grid. The Material Properties grid can be activated by selecting the desired material from the Project Explorer under Materials view.

Important Note: The properties assigned to the materials will be used for capacity calculation while the properties assigned to the stress strain models or confinement models will be used for moment-curvature calculation. By default, these two values will be the same i.e. every concrete/steel shape will have a corresponding stress-strain model and every rebar material will have a corresponding stress strain model. However, the user may manually change the material properties assigned to the stress strain model to be different from that assigned to the shape or material without affecting the capacity results. This is allowed because the user may wish to calculate the moment-curvature based on strength higher than the design strength of the material.

3.11.3. Define the Base Shape/Rebar Pattern

This assignment is required for the calculation of Magnified Moments. The equation for calculating the effective stiffness in ACI-318 assumes a section with a single type of concrete material, a single type of rebar material and a single type of steel material as seen in Equations (3.1) and (3.2). But, for a column using different types of concrete, rebar and steel materials in the same section we need to obtain equivalent I_g, equivalent I_{se} and equivalent I_{sx} for concrete, rebar and steel respectively. In order to get these separate transformed sections, we need to define 3 separate base materials for concrete, rebar and steel.

$$(EI)_{eff} = \frac{(0.2E_cI_g + E_sI_{se})}{1 + \beta_{dns}} \quad (3.1)$$

$$(EI)_{eff} = \frac{(0.2E_cI_g)}{1 + \beta_{dns}} + E_sI_{sx} \quad (3.2)$$

- Base Concrete Shape:
    - By default, the first shape drawn will be set as the "*Base Concrete Shape*" under **General** Tab in *Property* grid.
    - This field may be toggled only when there are two or more concrete shapes drawn in the canvas.
    - Assigning a concrete shape as the *Base Concrete Shape* will automatically remove the *Base Concrete Shape* assignment from the previous concrete shape.
    - When the concrete shape that is currently set as the *Base Concrete Shape* is deleted, the first concrete shape from the order drawn is set as the *Base Concrete Shape.*
- Base Rebar Pattern –
    - By default, the first rebar/ rebar layout drawn will be set as the "Base Rebar Pattern" under General Tab in Property grid.
    - This field may be toggled only when there are two or more rebar/ rebar layouts drawn in the canvas.
    - Assigning a rebar/ rebar layout as the Base Rebar Pattern will automatically remove the Base Rebar Pattern assignment from the previous rebar/ rebar layout.
    - When the rebar/rebar layout that is currently set as the Base Rebar Pattern is deleted, the first rebar/ rebar layout from the order drawn is set as the Base Rebar Pattern.

- Base Steel Shape
    - By default, the first steel shape drawn will be set as the "Base Steel Shape" under General Tab in Property grid.
    - This field may be toggled only when there are two or more steel shapes drawn in the canvas.
    - Assigning a steel shape as the Base Steel Pattern will automatically remove the Base Steel Shape assignment from the previous steel shape.
    - When the steel shape that is currently set as the Base Steel Shape is deleted, the first steel shape from the order drawn is set as the Base Steel Shape.

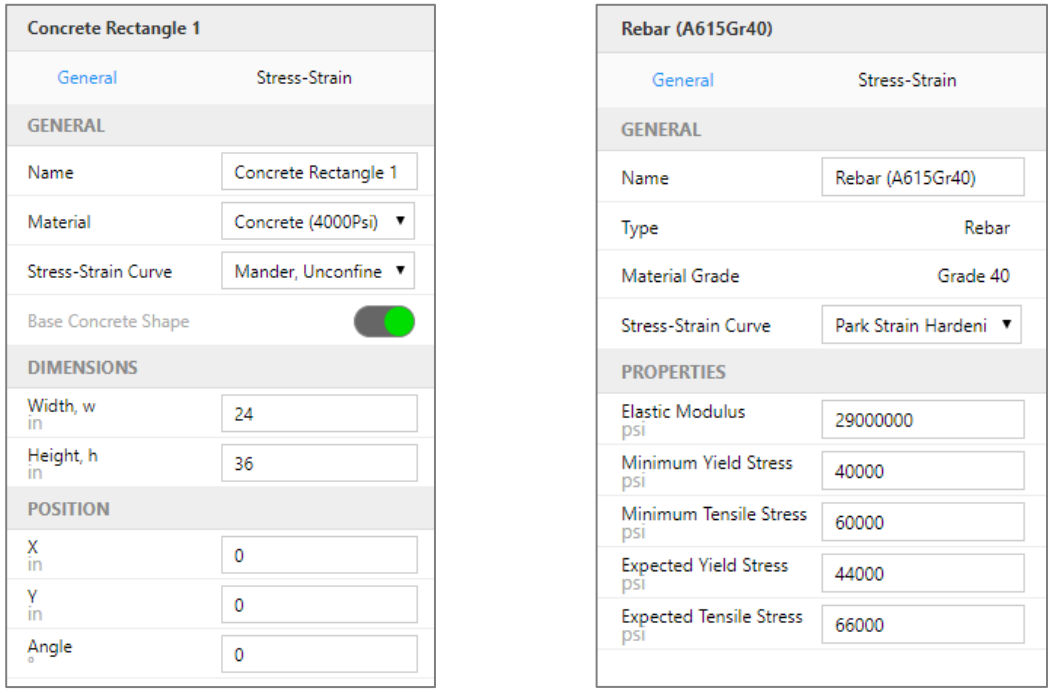
3.11.4. Assign Materials

Materials can be assigned to the shapes and rebars by selecting the desired material from the *Materials* drop-down list in the *Property* grid (Figure 3.4). The *Property* grid gets activated by selecting the shapes or rebar either by clicking on it in the working area or by selecting it from the *Project Explorer* under *Columns* view.

3.11.5. Assign Confinement Models and Stress Strain Model

For concrete and steel shapes, the stress-strain models are assigned to the shape. The stress strain model can be assigned by selecting the desired stress strain model from the *Stress-Strain Curve* drop-down list in the *General* tab of the *Property* grid (Figure 3.6a). The properties of the stress-strain model assigned to the shape can be modified under the *Stress-Stain* tab of the *Property* grid. The *Property* grid gets activated by selecting the shapes or rebar either by clicking on it in the working area or by selecting it from the *Project Explorer* under *Columns* view.

For rebars, the stress strain model can only be assigned to the material. Click on the *Define Menu>Project>Materials* button to access the *Define Materials* form (Figure 3.5) or alternatively, select the desired rebar material from the *Project Explorer* under *Materials* view to activate the *Material Properties* grid (Figure 3.6b) in the context-sensitive area. Change the stress-strain model assigned to the rebar material by selecting the desired stress strain model from the *Stress-Strain Curve* drop-down list. The properties of the stress-strain model assigned to the rebar material can be modified under the *Stress-Strain* tab of the *Material Properties* grid.



a) b)

Figure 3.6 a)*Property* Grid for Concrete Shape b) *Material Properties* Grid for Rebar Material

Important Note: The properties assigned to the materials will be used for capacity calculation while the properties assigned to the stress strain models or confinement models will be used for moment-curvature calculation. By default, these two values will be the same i.e. every concrete/steel shape will have a corresponding stress-strain model and every rebar material will have a corresponding stress strain model. However, the user may manually change the material properties assigned to the stress strain model to be different from that assigned to the shape or material without affecting the capacity results. This is allowed because the user may wish to calculate the moment-curvature based on strength higher than the design strength of the material.

3.12. Create Confinement Zones

Confinement zones can be created in two ways:

- Auto Create Confinement
- Draw Confinement

3.12.1. Auto Create Confinement

In this method, a confinement zone is created by the software automatically. The basic procedure for auto creating confinement is as follows:

1. Select the desired column section from the *Project Explorer*.

Note: This feature is available for sections with only one rebar pattern (rectangular or circular patterns only).

2. Click on the *Home Menu>Design>Auto Confinement* button to create a confinement zone in the section.
3. By default, the confinement zone will enclose the outer perimeter of rebars. Accept the default or change the dimensions of the confinement zone by typing in the edit boxes available under the *General Tab* of the *Property Grid.* The *Property Grid* is activated in the context-sensitive area by selecting the confinement zone from the working area or from the *Project Explorer* under *Confinement Zones*.
4. By default, the Mander's Confined model is assigned to the confinement zone. This can be changed from the Stress-Strain Curve drop down list in the *General Tab* of the *Property Grid.* New confinement models must first be defined from the *Define Concrete Confinements* form which can be accessed from the *Define Menu>Stress Strain>Concrete* button or by clicking on the ⋮ button next to the *Project Explorer>Stress-Strain>Concrete.*
5. Once the appropriate confinement model has been assigned to the confinement zone, select the *Stress Strain Tab* of the *Property Grid*. Review the values of the parameters of the confinement model such as tie diameter and tie spacing under the *Ties* heading and modify them by typing in the edit boxes available as required. The total main steel area will be computed from the number of rebars provided. But the value may be changed manually by the user, if required.
6. Review the material properties under the *Materials* heading in the *Stress-Strain Tab* of the *Property Grid.* By default, the material properties will be the same as that of the material assigned to the main section. Modify the values if necessary, by typing in the edit boxes available.

Note: The properties assigned to the materials will be used for capacity calculation while the properties assigned to the stress strain models or confinement models will be used for moment-curvature calculation. By default, these two values will be the same i.e. every concrete/steel shape will have a corresponding stress-strain model and every rebar material will have a corresponding stress strain model.

However, the user may manually change the material properties assigned to the stress strain model to be different from that assigned to the shape or material without affecting the capacity results. This is allowed because the user may wish to calculate the moment-curvature based on strength higher than the design strength of the material.

For auto confinement zones, the material properties will be connected to the material properties assigned to the main section. The total main steel area will also be computed automatically by the software. But both these parameters may be changed by the user, if required.

Also, when the material of the section is changed after creating the auto confinement, the user will be prompted to decide whether to update the material properties assigned to the stress strain model or not.

3.12.2. Draw Confinement

In this method, CSiCol allows the user to draw their own confinement zones. The basic procedure for drawing confinement zones is as follows:

1. Select the desired column section from the *Project Explorer*.
2. Click on the *Draw Menu>Confinement>Rectangle* or *Circle* button and draw a confinement zone on the section with the mouse.

   Note: Only rectangular and circular confinement zones are available.

3. The dimensions of the confinement zone can be changed by typing in the edit boxes available under the *General Tab* of the *Property Grid* of the confinement zone. The *Property Grid* is activated in the context-sensitive area by selecting the confinement zone from the working area or from the *Project Explorer* under *Confinement Zones*.
4. The confinement zone can be moved to a desired location in the same way as any other shape.
5. By default, the Mander's Confined model is assigned to the confinement zone. This can be changed from the Stress-Strain Curve drop down list in the *General Tab* of the *Property Grid.* New confinement models must first be defined from the *Define Concrete Confinements* form which can be accessed from the *Define Menu>Stress Strain>Concrete* button or by clicking on the button next to the *Project Explorer>Stress-Strain>Concrete.*
6. Once the appropriate confinement model has been assigned to the confinement zone, select the *Stress Strain Tab* of the *Property Grid*. Review the values of the parameters of the confinement model such as tie diameter, tie spacing and total main steel under the *Ties* heading and modify them by typing in the edit boxes available as required.
7. Review the material properties under the *Materials* heading in the *Stress-Strain Tab* of the *Property Grid.*

   Note: For manually drawn confinement zones, the material properties will not be connected to the material properties assigned to the main section. The total main steel area will also need to be computed manually by the user.

Operations such as merging, subtracting, aligning, and stacking can be performed on confinement zones in the same way as for other shapes to obtain complex confinement zones. Merging is only allowed for confinement zones with the same material properties and same confinement models

(rectangular confinement model or circular confinement model). After merging, the user must verify the dimensions in the *Stress Strain* tab of the confinement zone and modify them as required.

3.13. Consider Slenderness Effects

Toggle the *Define Menu>Slenderness>Slenderness Effects* button to specify whether the slenderness effects for the current column are to be considered or not. If slenderness effects are to be considered, the *Framing Condition* also needs to be specified. The framing condition is used to compute the effective length factor as well as other factors related to the moment magnification depending on the design codes.

The following design codes are supported for slender columns:

- ACI 318-08
- ACI 318-11
- ACI 318-14
- IS 456:2000
- CSA A23-3-04
- BS 8110-97
- Euro Code 2: 2004

3.13.1. Specify Column Framing Conditions

Use the *Define Menu>Slenderness>Framing Conditions* button when the *Slenderness Effects* toggle is turned on to access the *Column Framing Conditions* form and specify the information needed to compute the magnified moments while considering slenderness effects. In this form, click on the framing condition that best represents the column end/support condition (e.g. Fixed on both, Fix-Pin, Fix-Roller, Fix-Free, Pin-Roller and lowest story framed columns with pinned base, fixed base or intermediate story columns connected to frame elements on both ends). Figure 3.7 illustrates the Column Framing Conditions relative to the XZ and YZ planes for a general case of Framed Columns.

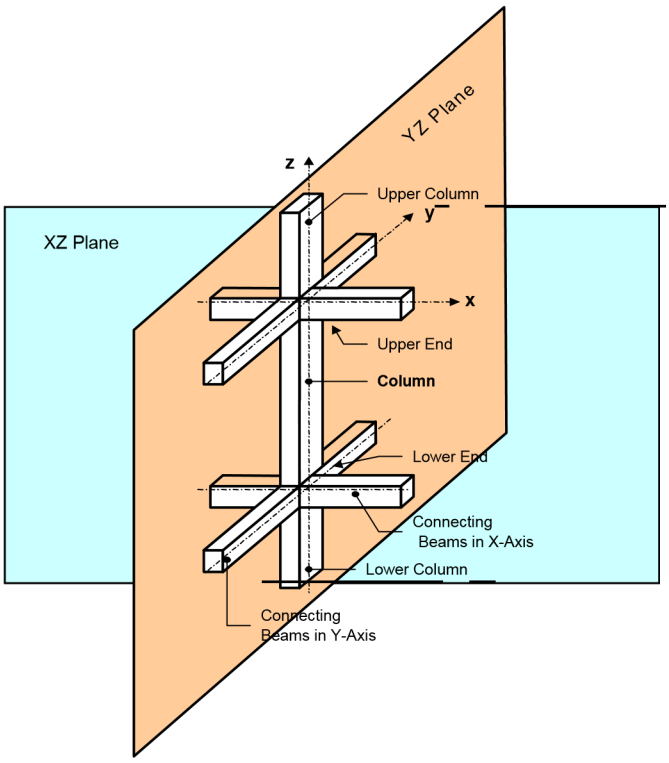


Figure 3.7 Column Framing Conditions

After selecting the framing condition, the framing conditions in XZ plane is displayed in the working area as seen in Figure 3.8. The framing condition can be viewed either in 2D (either XZ or YZ plane) or 3D by selecting the desired option in the context-sensitive toolbar.

- XZ/YZ or 3D buttons: Click on the XZ or YZ button to view and change the framing parameters in the XZ or YZ planes respectively. Click on the 3D button to view the framing in 3D.
- Zoom In button: Click the Zoom In button to zoom in (make the image larger). Zooming in can also be done by moving the scroll button on the mouse upwards.
- Zoom Out button: Click the Zoom Out button to zoom out (make the image smaller). Zooming out can also be done by moving the scroll button on the mouse downwards.
- Rest and Refresh View button: Use this button to restore the original default display of the interaction surface.

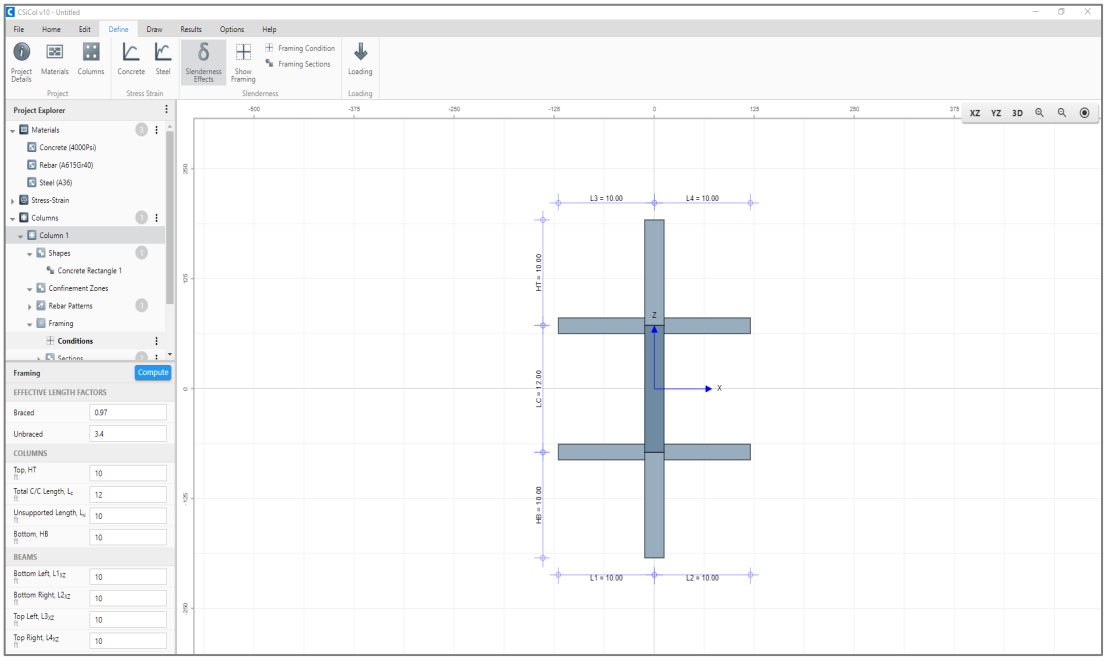


Figure 3.8 Framing Conditions View, 2D

3.13.2. Effective Length Factor

Use the following procedure to calculate the effective length factors:

1. Click on the *Define Menu>Slenderness>Show Framing* button to view the framing conditions.
2. For each plane (XZ or YZ), the *Framing* grid (Figure 3.9) gets activated in the context-sensitive area where the framing parameters such as unsupported length of column, c/c length of columns and length of beams need to be entered.

   Note: By default, Eurocode 2:2004 does not consider the contribution of the stiffness of the top and bottom columns to the joint stiffness. So, for Eurocode, an additional "Consider Non-failing Columns" toggle has been introduced in the *Framing* grid to override this. This toggle allows the user to consider the contribution of 'non-failing' columns to the joint stiffness, if desired.

   For BS 8110-97 code, there is an additional option to consider the beams framing into the column to be simply supported. Turn on the "Simply Supported, Top" or "Simply Supported, Bottom" toggles to consider simply supported beams framing the column at the top and bottom nodes, respectively.

3. Select each of the connected beams and columns to activate the *Frame Member* grid where you can view the properties of the section assigned to them and change them, if necessary. By default, the connected beams and columns are assigned a program defined beam and column framing section respectively. If you want to

define your own sections or change the dimensions and other properties of the existing ones go to the next step. If not, **skip to step 7**.

4. Click on the *Define Menu>Slenderness>Framing Sections* button or the button next to the *Project Explorer>Columns>Framing>Sections* to access the *Define Framing Sections* form where a list of existing framing sections is displayed. Using this form, new framing sections can be added, and existing ones renamed or removed.
5. Select the desired framing section from the *Project Explorer>Columns>Framing>Sections.* The selected section will be displayed in the working area and the *Property* grid will get activated in the context-sensitive area. Change the dimensions and other properties of the selected framing section as required using the *Property* grid.
6. After defining new sections and modifying the properties of the sections as required, click on the *Define Menu>Slenderness>Show Framing* button to view the framing condition.
7. Select the connecting beam or column whose section you would like to change. The *Frame Member* grid gets activated. Select the desired section under the *Section* drop down list.
8. Once the parameters have been correctly entered and sections have been correctly assigned, click on the *Compute* button in the *Framing* grid to calculate the Effective Length (k) factors.
9. Perform the above steps for both XZ and YZ planes. Alternatively, the effective length values can be specified through direct input.

Note: The *Framing Condition, Framing Sections* and *Show Framing* buttons are activated only when the *Slenderness Effects* toggle is turned on.

The framing view and *Framing* grid can also be accessed from the *Project Explorer* under *Columns>Framing>Conditions.*

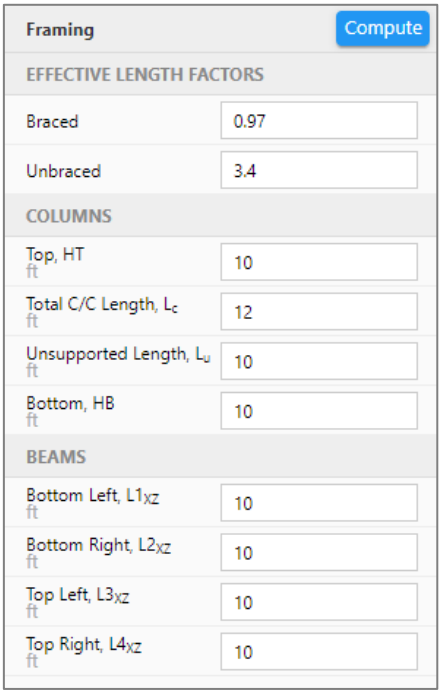


Figure 3.9 *Framing* Grid

3.14. 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. CSiCol automatically opens the Loading form in either the detailed mode or the simple mode depending on whether the Slenderness Effects toggle is turned on or off.

Figure 3.10 illustrates loading on the column and cross-section. Unlimited load combinations may be defined for a column cross-section by clicking on the Define Menu>Loading button.

Note: Sign Convention used by the software:

Axial Load:
For Axial Load, compression is considered to be positive and tension is considered to be negative.

Bending Moment:
Similar to ETABS, for columns bending in double curvature, the top and bottom end bending moments are of opposite signs and for columns bending in single curvature, the top and bottom end bending moments are of the same sign

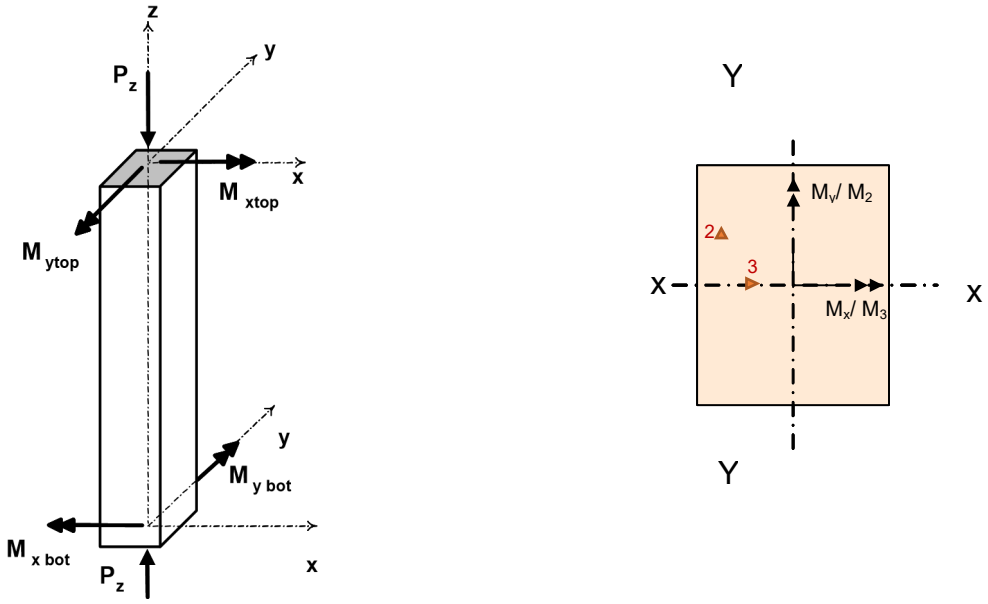


Figure 3.10 Column Loading on Column and Cross-Section

3.14.1. Simple Loading Mode

Simple loads apply when (a) the column is to be designed as a short column or when slenderness effects can be ignored; (b) the loads and bending moments have already been magnified by separate analysis, such as the P-Delta analysis option in ETABS or by other procedures; and (c) detailed loading or column framing information is not available.

For the simple load combination case, the combination name, axial load value (P_u), top and bottom bending moments about X-axis (M_{ux}) and about Y-axis (M_{uy}) must be specified. Define the simple loading on the column as follows:

1. Specify the load combination name, axial load and the bending moments.
2. Specify as many load combinations as required for analysis and design.
3. Import load combinations from previously saved text files using the *Import* button.

3.14.2. Detailed Loading Mode for Slender Columns

Detailed load combinations can be defined along X and Y, separately. The moments under Along X and Along Y tabs correspond to the moments about the X-axis and about the Y- axis respectively. Detailed loads are needed only if slenderness effects need to be considered. Define a detailed load combination as follows:

1. Select the loading direction (Along X or Along Y) by clicking on the respective tab, and then specify the combination name.
2. Specify the following code-dependent parameters:

For the ACI-318 Design Code:

- Specify the axial load, non-sway top and bottom bending moment in the spreadsheet cells.
- Click on the *Check* button under the *Check Sway* column to access the *Sway Conditions* form. There are 3 methods available for sway check: Stability Index, Second order Analysis and Relative Stiffness of Bracing to Columns. Select the desired method for sway check and fill in the required parameters. Click on the *Check Sway* button to check for sway condition. Click on *OK* once done.
- If the column is found to be of sway type, the *Consider Sway* toggle in the *Define Loading Slenderness* form will be automatically turned on. If the column is found to be of non-sway type, the software can still be forced to consider it as a sway frame by manually turning on the *Consider Sway* toggle.
- When the *Consider Sway* toggle is turned on, the input fields for the values of the sway parts of axial load, top and bottom bending moments, Story Load, Story Shear, Relative Sway, Critical Load and Stability Index will get activated. Edit the values as and if required. These values will be used to calculate the magnified moments.

For the IS 456-2000 Design Code:

- Specify the axial load, bottom bending moment, and top bending moment values.
- Click on the *Check* button under the *Check Sway* column to access the *Sway Conditions* form. There are 3 methods available for sway check: Stability Index, Second order Analysis and Relative Stiffness of Bracing to Columns. Click on the *Check Sway* button to check for sway condition. Click on *OK* once done.
- If the column is found to be of sway type, the *Consider Sway* toggle in the *Define Loading* Slenderness form will be automatically turned on. If the column is found to be of non-sway type, the software can still be forced to consider it as a sway frame by manually turning on the *Consider Sway* toggle.

For the CSA A23-3-04 Design Code:

The process for defining detailed loads for CSA Design Code is the same as that for ACI Design Code.

For the Euro Code 2:2004 Design Code:

- Specify the axial load, top moment and bottom moment values. By default, the column is considered to be braced in the selected direction. Turn on the *Consider Unbraced* toggle to change this default assumption.

Note: For Eurocode 2, moment magnification is not considered for composite sections (i.e. the steel area coming from the steel shape in the composite section will not be considered. Only the rebar area will be considered)

The parameters required for the calculation of magnified moments in Eurocode2 such as concrete partial safety factor (γ_c), reinforcing partial safety factor (γ_s), long term compressive strength factor (α_{cc}) can be specified in the *Options Menu>Partial Safety Factors* tab and ambient relative humidity (RH%) and ratio SLS to ULS moments (rm) can be specified in the *Options Menu>Design Parameters* tab. Long term compressive strength factor (α_{cc}) , long term tensile strength factor (α_{ct}), ambient relative humidity

(RH%) and ratio SLS to ULS moments (rm) can be specified in the *Options Menu>Slenderness Parameters* tab.

For the BS8110-97 Design Code:

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

3. Click the + button to add a new load combination.
4. Edit the existing load combinations by directly typing in the edit boxes.

Note: The final design loads after the moment magnification can be viewed from *Results Menu>Capacity>Design Loads* or from the *Project Explorer* under *Results>Capacity> Design Loads.*

3.14.3. Sway Check (Based on ACI 318-14)

Click on the *Check* button under the "Check Sway" Column of the *Define Loading Slenderness* form to access the *Sway Conditions* form. The *Sway Conditions* form is shown in Figure 3.11. The sway check can be performed based on the following three methods:

- **Stability Index**: This 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 is assumed to be non-sway. The data required to perform this check includes the Story Load, Story Shear, Relative Lateral Deflection and c/c length of column. If the column is found to be of sway type, the parameters entered in this form gets transferred to the sway part of the Loading form and can be edited, if necessary.
- **Second Order Analysis**: 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 Columns**: A column is 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 as compared to the lateral stiffness of the column (greater than or equal to 12), 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.

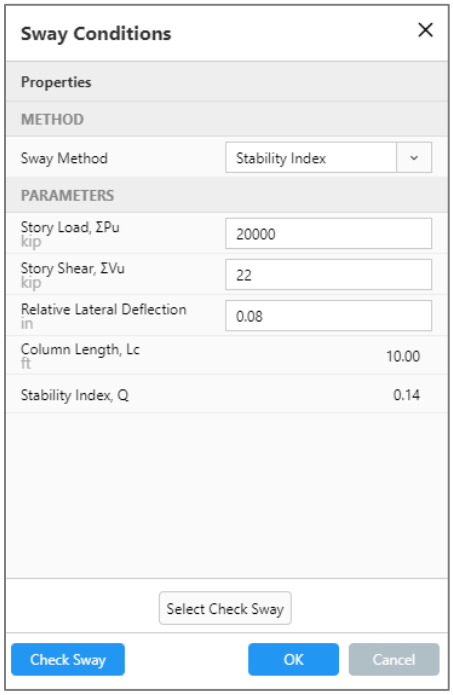


Figure 3.11 *Sway Conditions* form

3.15. Column Auto Design

The *Home Menu>Design>Auto Design* feature is a very effective, efficient and powerful tool for the design of column reinforcement with or without considering slenderness effects. Use the Column Auto Design feature to design the column reinforcement in accordance with the various design parameters defined/specified by the user.

The *Auto Design* tool in CSiCol uses an iterative process to identify the minimum reinforcement and the reinforcement layout that can be provided to the selected column to satisfy the user-specified maximum capacity ratio limit for the governing load combination (load combination which gives the highest D/C ratio). The basic procedure for using this tool is as follows:

1. Select the desired column from the *Project Explorer*.

Note:

- This feature is only available for rectangular and circular reinforced concrete columns with only one rectangular or circular rebar pattern.
- This option is available for rectangular and circular reinforced concrete columns with only one rectangular or circular rebar pattern imported from ETABS.
- When columns are imported from CDB files, they are imported as polygons. So, this feature is not available for columns imported from CDB files.

2. Click on the *Auto Design* button to access the *Design* form (Figure 3.12).
3. Accept the default settings for the rebar selection or change them by using the drop-down menus, edit boxes and toggles provided, if necessary. The settings that are available are as follows:
    a. Smallest Bar: Use this edit box to specify the smallest rebar that can be used for the design.
    b. Largest Bar: Use this edit box to specify the largest rebar that can be used for the design.
    c. Clear Cover: Use this edit box to specify the clear cover that must be provided in the section.
    d. Clear Spacing: Use this edit box to specify the minimum clear spacing between the rebars.
    e. Maximum Rebar Ratio: Use this edit box to specify the maximum rebar ratio.
    f. Maximum D/C Ratio: Use this edit box to specify the desired maximum D/C ratio that must be satisfied by the design.
    g. Follow Provided Layout: When this toggle is turned on, the rebar layout remains fixed and Auto Design only increases the bar size to satisfy the maximum D/C ratio limit. When this toggle is turned off, Auto Design increases the number of bars while satisfying the clear spacing.
    h. Allow Bundled Bars: When it is turned on, Auto Design allows bundled bars if the maximum D/C ratio limit cannot be satisfied using single bars. When this toggle is turned off, bundled bars are not allowed.
    i. Status: This shows the status of the design process. If the design has not been run yet, it displays "***Run Design"***. If the design process has been completed successfully, it displays "***Success*** "and if the specified D/C ratio limit cannot be satisfied with the current settings of rebar selection, it displays "***Failed***".
4. Click on the *Run* button to start the auto design process.
5. CSiCol displays the design information for each iteration, including the current rebar layout, load combination, rebar ratio and current capacity ratio. The information may change depending on the selections made on the Rebar Selection settings. CSiCol displays a "***Success"*** status when the design process has been completed successfully and a "***Failed***" status when the maximum D/C ratio limit cannot be satisfied with the current settings of rebar selection for the given column section and load combination. Update the settings for the rebar selection and repeat the design process until the "***Success***" status is displayed.

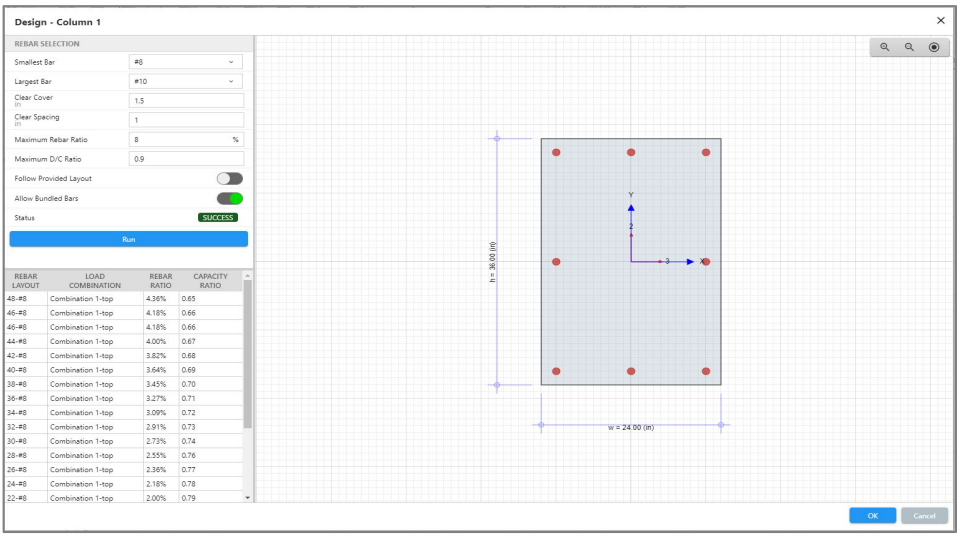


Figure 3.12 CSiCol *Auto Design*

4. Editing Column 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.

4.1. Types of Shapes

After adding a shape, its dimensions and properties can be modified. However, the process is different for different types of shapes. There are basically two types of shapes based on how they are created:

- **Parametric Shapes**: This type includes the rebar shape patterns, basic steel/concrete shapes defined parametrically, and the standard steel shapes obtained from the database. In this type of shapes, the user adds the shape using the *Quick Design Wizard*, the ***File Menu>Rectangular Model*** or ***File Menu>Circular Model*** button or selects the shapes from one of the available shape libraries, adding it to the current section (except the *Polygon* shape).

Note: The parametric shapes can also be converted to polygon (non-parametric shape) using *Edit Menu>Edit>Edit Points* button which allows you to add, remove or modify nodal point coordinates of the selected shape.

- **Non Parametric Shapes**: This type of shapes are created in several ways: by drawing on the screen using the ***Draw Menu>Concrete Shapes>Polygon*** button, by modifying the library based shapes by either flipping the shapes, merging more than one shape, creating holes or performing other operations to achieve the desired shape and by inputting the coordinates of the shape.

4.2. Editing Parametric Shapes

The dimensions and position of parametric shapes can be modified from the *Property* grid. The *Property* grid gets activated in the context-sensitive area when you select a shape either by clicking on it in the working area or selecting it from the *Project Explorer* under the *Columns* view. Figure 4.1 shows the *Property* grid for parametric shapes.

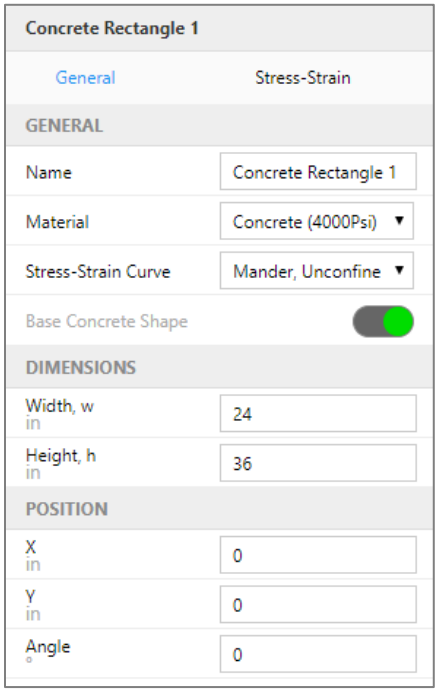


Figure 4.1 *Property* grid for Parametric Shapes

The parametric shapes can also be edited using *Edit Menu>Edit>Edit Points* button which allows you to add, remove or modify nodal point coordinates of the selected shape. However, editing a parametric shape in this way will require you to convert the parametric shape to a polygon (non-parametric).

4.3. Editing Non-Parametric Shapes

The dimensions of non-parametric shapes can be edited by using the *Edit Menu>Edit>Edit Points* button which allows you to add, remove or modify nodal point coordinates of the selected shape. The dimensions of the shape and the holes can also be edited from the *Property* grid under the *Points* and *Holes* tab respectively as seen in Figure 4.2. The *Property* grid gets activated in the context-sensitive area when you select a shape either by clicking on it in the working area or selecting it from the *Project Explorer* under the *Columns* view.

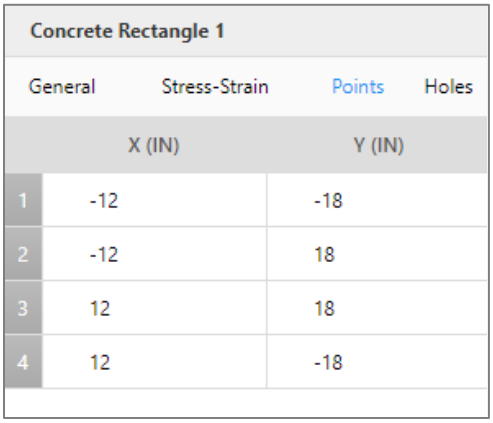


Figure 4.2 *Property* Grid for Non-Parametric Shapes

4.4. 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. Choose the appropriate alignment button from the following options

- *Align Left* button: Use this button to align the selected shapes along their left edge. The alignment is done with respect to the lowest value of x-coordinate of all the selected shapes.
- *Align Right* button: Use this button to align the selected shapes along their right edge. The alignment is done with respect to the highest value of the x-coordinate of all the selected shapes.
- *Align Center* button: Use this button to align the selected shapes along the vertical centerline. The alignment is done with respect to the x-centroid of all the selected shapes.
- *Align Top* button: Use this button to align the selected shapes along their top edge. The alignment is done with respect to the highest value of the y-coordinate of all the selected shapes.
- *Align Bottom* button: Use this button to align the selected shapes along their bottom edge. The alignment is done with respect to the lowest value of the y-coordinate of all the selected shapes.
- *Align Middle* button: Use this button to align the selected shapes along the horizontal centerline. The alignment is done with respect to the y-centroid of all the selected shapes.

4.5. Flip Shapes

CSiCol has options for flipping the shapes. Select the shape to be flipped and click on the appropriate button on *Edit Menu>Flip*:

- Flip Horizontal button: This button flips the selected shape about the vertical axis (mirror).
- Flip Vertical button: This button flips the selected shape about the horizontal axis (mirror).

Note: The shapes are converted into an editable polygon after flipping.

4.6. Stack Shapes

Stacking is performed based on the relative location of the shapes. Select the shapes to be stacked and choose the appropriate stack button:

- Stack Horizontal button: Use this button to stack the selected shapes horizontally, side by side. The y-coordinates of the shapes do not change.
- Stack Vertical button: Use this button to stack the selected shapes vertically, one on

top of the other. The x-coordinates of the shapes do not change.

4.7. Merge Shapes

Two shapes with the same material properties and have a common edge or that overlap can be merged. Merge shapes as follows:

1. Select the shapes to be merged.
2. Click on the Edit Menu>Merge button.
3. Verify the final shape after merging.

Note: The shapes are converted into an editable polygon after merging.

4.8. Create Holes in Shapes

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

1. Create the shape or the section to which you want to add a hole (first shape or section).
2. Add a shape whose shape and size are the same as that of the desired hole (second shape).
3. Move the second shape to the place where the hole is to be created.
4. Select both the shapes and click on the Edit Menu>Merge>Subtract button.

Note: The shapes are converted into an editable polygon after creating holes. The dimensions of the hole can be modified from the *Property* grid under the Holes tab.

4.9. Move Shapes

Selected shapes in the working area can be moved either using the mouse or by displacement values from the *Edit Menu>Edit>Move Sections* button.

Move the shapes using the mouse as follows:

1. Click on the *Edit Menu>Select* button to activate the *Selection Mode.*
2. *Select a single shape to be moved or multiple shapes by holding the Ctrl or Shift key.*
3. *Hold down the left mouse button and drag the mouse to move the shapes to the desired location.*

Move the shapes by displacement value as follows:

1. Click on the Edit Menu>Select button to activate the Selection Mode.
2. Select the shape(s) to be moved.
3. Click on the Edit Menu>Edit>Move Sections button to access the Move Points form.
4. Type in the displacement values in X(dx) and Y direction (dy) on the form.

5. Obtain and Interpret Results

This chapter briefly describes the different results generated by CSiCol.

5.1. 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:

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

5.2. Section Properties

To view a summary of the overall dimensions and geometric properties of the current column section in the working area, use the *Results Menu> Section> Section Properties* button or the *Project Explorer>Columns>Results> Section> Section Properties* to access the *Section Properties* form. This form is divided into 3 sections:

- Basic Properties: Includes the area, shear areas, moment of inertia and torsional constant of the selected section.
- Section Bounds: Includes the overall dimensions of the selected section and the centroid location with respect to both the global origin as well as the local origin.
- Additional Properties: Includes the radii of gyration and section moduli of the selected section.
- Rebar Properties: Displays the rebar area and rebar ratio in the selected section.

The following context-sensitive toolbar is available for Section Properties:

- Zoom In button  : Click the Zoom In to zoom in (make the image larger. Zooming in can also be done by moving the scroll button on the mouse upwards.

- Zoom Out button : Click the Zoom Out button to zoom out (make the image smaller). Zooming out can also be done by moving the scroll button on the mouse downwards.
- Rest and Refresh View button : Use this button to restore the original default display of the section.
- Add to Report button : Click on this button to add the section properties to the detailed report.

Note: For composite sections, concrete is considered as the base material and the transformed section properties are reported.

5.3. Section Stresses

After the column section has been created and load combinations have been assigned, CSiCol can display Flexural Stress as 2D or 3D color-coded contours on the working area.

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.
2. Define the load combinations for the section.
3. Click on the Results Menu> Section> Section> Section Stresses button or the Project Explorer> Columns> Results> Section> Section Stresses to view the Section Stresses.
4. Select the loading and section location from the *Section Elastic Stresses* grid (Figure 5.1) that gets activated in the context-sensitive area.
5. The Stresses can be viewed in 2D or 3D as seen in Figure 5.2 by selecting the desired option in the associated context-sensitive toolbar.
6. The value of stress at any point in the section can also be read by hovering the mouse/cursor over the point. A color-coded key is displayed at the extreme right of the working area to help interpret the stresses generated on the section.

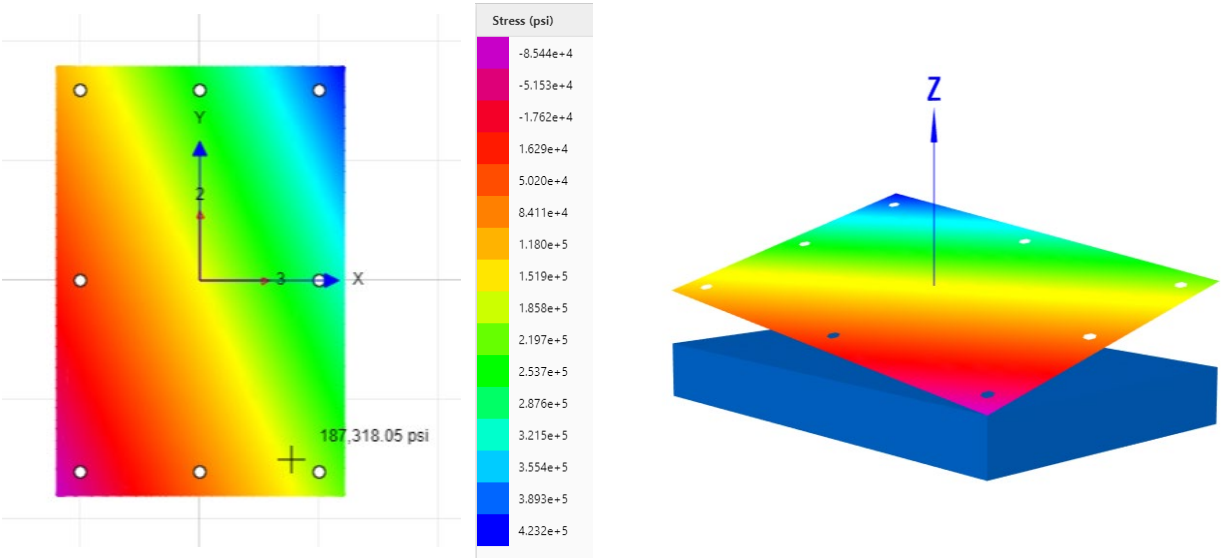
The following context-sensitive toolbar is available for Section Stresses:

- 2D button : Use this button to view the section stresses in 2D view.
- 3D button : Use this button to view the section stresses in 3D view.
- Zoom In button : Click the Zoom In to zoom in (make the image larger. Zooming in can also be done by moving the scroll button on the mouse upwards.
- Zoom Out button : Click the Zoom Out button to zoom out (make the image smaller). Zooming out can also be done by moving the scroll button on the mouse downwards.
- Rest and Refresh View button : Use this button to restore the original default display of the section stresses.

- Add to Report button  : Click on this button to add the section stresses to the detailed report.

Section Stresses	
STRESS TYPE	
Stress Type	Flexural Stress
LOADING	
Loading	Combination 1
Section Location	Top
Axial Load, P_f kip	1,000.00
Moment, M_{fx} kip-ft	300.00
Moment, M_{fy} kip-ft	300.00
STRESS RANGE	
Minimum Stress, σ_{min} psi	-578.70
Maximum Stress, σ_{max} psi	2,893.52

Figure 5.1 *Section Elastic Stresses* Grid



a) b)

Figure 5.2 Section Flexural Stresses in a) 2D b)3D

5.4. Interaction Surface and Curves

Three stress resultants (axial load, moment about 3-axis and moment about 2-axis) 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_3 and M_2 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 Interaction Curve (PM 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 3 and 2 axes.
- **Moment-Moment Interaction Curve (MM Curve)**: The capacity surface can be sliced on the 2-3 plane to obtain the plot between moments M_3 and M_2. This plot between the moment capacity about the 3 and 2 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.

5.5. 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_3 and M_2 along two orthogonal horizontal axes.

Generate the Interaction Surface for any given column cross-section and display the interaction surface and curves of a column section as follows:

1. Define the column section (geometry and material properties) for which you want to generate the surface (see Chapter 3 for more information).
2. Use the Results Menu> Capacity> Interaction Diagram button or the Project Explorer> Columns> Results> Capacity> Interaction Diagram to view the interaction diagrams.
3. Clicking on this button activates the *Interaction Diagram Settings* grid in the context-sensitive area where various settings for the interaction diagram such as the option to include or exclude the strength reduction factor, ϕ (or partial safety factors), number of curve points and number of curves can be adjusted.

4. After making the necessary changes in the *Interaction Diagram Settings* form, click on the *Refresh* button to update the diagrams.

Note: The strength reduction factors specific to the design codes can be modified from the *Options Menu>Strength Reduction Factors (*or *Partial Safety Factors* depending on the selected design code). The type of stirrup (tied or spiral) can be selected from the *Define Menu>Stirrup Type.*

The working area is divided into three sections with associated toolbars that can be used to modify the display on each section as seen in Figure 5.3. Each section has been described below with their associated toolbars.

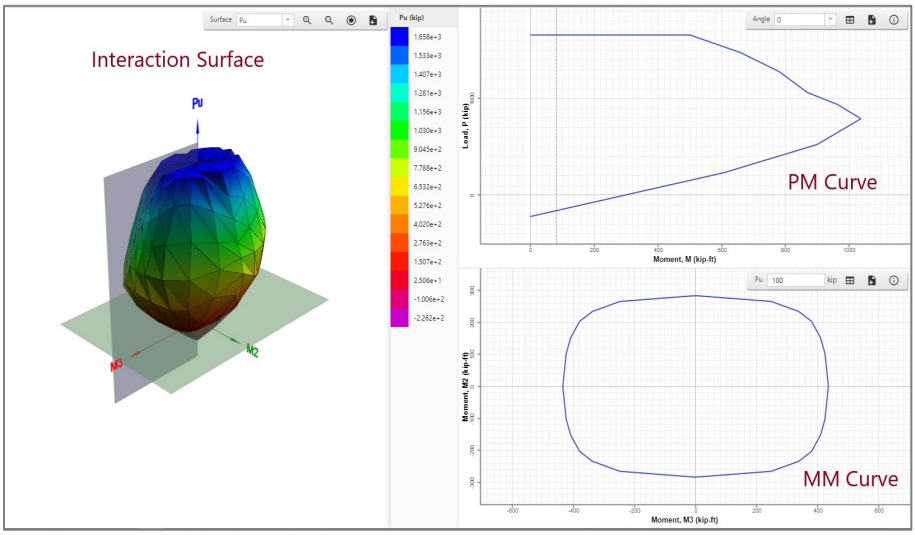
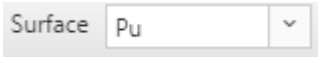


Figure 5.3 Interaction Diagram Screen

- Interaction Surface
  The variation of all three parameters governing the section capacity of a column section is plotted in a color-coded 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_3 and M_2 along two orthogonal horizontal axes. The associated toolbar has the following options:
  - Interaction Surface drop-down list 

: Use the options on this drop-down list to set the interaction surface display color for P, M_2 or M_3. A color-coded key is displayed at the extreme right of the section to help interpret the values of the selected surface.
  - Zoom In button  : Click the Zoom In button to zoom in (make the image larger). Zooming in can also be done by moving the scroll button on the mouse upwards.
  - Zoom Out button  : Click the Zoom Out button to zoom out (make the image smaller). Zooming out can also be done by moving the scroll button on the mouse downwards.
  - Rest and Refresh View button  : Use this button to restore the original default

display of the interaction surface.

- Add to Report button  : Click on this button to add the interaction surface to the detailed report.

- P-M Curve
  P-M curves are outlines obtained when a vertical plane cuts the interaction surface at different specified section rotations showing the relationship between the axial load and the moment at the specified neutral axis rotation. The associated toolbar has the following options:

  - Angle drop-down list 

: This list displays the neutral axis angle for the currently displayed P-M curve, and it also displays the user-specified load combinations for capacity checks. All the defined load combinations are listed at the end of the drop-down list. If the section is adequate for the selected loading condition, the vector lies within the bounds of the P-M plot. This gives an idea of the reserve capacity of the section. Select the neutral axis angle or load combination for which you want to view the P-M Curve. The curve display will be updated automatically. Move the cursor/mouse pointer over the curve to read the values for any point on the curve.
  - Tabulated Output button  : Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
  - Add to Report button  : Click on this button to add the P-M diagram to the detailed report.
  - Interaction Curve Details button  : Click on this button to access the Interaction Curve Details form which displays details regarding the interaction diagram such as angle, maximum capacity in compression and tension and maximum moment capacity.

- 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_2-M_3 surface plots. The associated toolbar has the following options:

  - Value of Pu edit box 

: This edit box displays the axial load level for the currently displayed M-M curve. Type in the axial load level (Pu) for which you want to view the M-M Curve. The curve display will be updated automatically.
  - Tabulated Output button  : Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
  - Add to Report button  : Click on this button to add the M-M diagram to the detailed report.

- Interaction Curve Details button : Click on this button to access the Interaction Curve Details form which displays details regarding the interaction diagram such as maximum and minimum moment capacities in both axis 2 and 3.

5.6. Animate Interaction

Click on the *Results Menu> Capacity> Animate Interaction* button or the *Project Explorer> Columns> Results> Capacity> Animate Interaction* to view the animation of capacity surface generation for different neutral axis angles. The associated toolbars have the following options:

- *Play/Stop* button : Click on this button to start the animation. This button changes to the *Stop* button while the animation is playing.
- *Zoom In* button : Click the Zoom In to zoom in (make the image larger. Zooming in can also be done by moving the scroll button on the mouse upwards.
- *Zoom Out* button : Click the Zoom Out button to zoom out (make the image smaller). Zooming out can also be done by moving the scroll button on the mouse downwards.
- *Rest and Refresh View* button : Use this button to restore the original default display of the interaction surface.

5.7. Capacity Point Details

Use the *Results Menu> Capacity> Capacity Point Details* button or the *Project Explorer> Columns> Results> Capacity> Capacity Point Details* to view the capacity point details at several points along the P-M diagram at selected neutral axis angle with respect to X-axis or for a specified load combination.

The working area is divided into two sections with associated toolbars. The left section is the P-M curve section and the right section is the stress/strain contour section. As you hover along the curve points in the P-M curve section, parameters such as Point Status, axial load, major moment, force and moment on concrete and rebar, strain, etc are displayed in the context-sensitive *Capacity Point Details* grid whereas the corresponding stress/strain contour is displayed in the stress/strain contour section.

The capacity point details provide important information regarding the various points in the P-M diagram. By looking at point status details, it can be determined whether the load point is in compression-controlled zone or tension-controlled zone. In the compression-controlled zone, failure is initiated by the crushing of concrete at the highly compressed edge while in tension-controlled zone, failure is initiated by the yielding of the outermost longitudinal bars in the tension side of the neutral axis. The rebar strain and rebar force values are useful in determining the type of bar lap splicing (compression lap splices or tensile lap splices) required in different zones of the P-M diagram.

The associated toolbar in the P-M curve section has the following options:

- *Angle* drop-down list 

: This list displays the neutral axis angle for the currently displayed P-M curve, and it also displays the user-specified load combinations for capacity checks. All the defined load combinations are listed at the end of the drop-down list. If the section is adequate for the selected loading condition, the vector lies within the bounds of the P-M plot. This gives an idea of the reserve capacity of the section. Select the neutral axis angle or load combination for which you want to view the P-M Curve. The curve display will be updated automatically. Move the cursor/mouse pointer over the curve to read the values for any point on the curve.

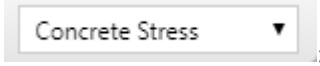
- *Tabulated Output* button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- *Add to Report* button 
: Click on this button to add the capacity point details to the detailed report.
- *Interaction Curve Details* button 
: Click on this button to access the *Interaction Curve Details* form which displays details regarding the interaction diagram such as angle, maximum capacity in compression and tension and maximum moment capacity.

The associated toolbar in the stress/strain contour section has the following options:

- 2D button 

: Use this button to view the stress/strain contour in 2D view.
- 3D button 

: Use this button to view the stress/strain contour in 3D view.
- Type of Contour drop-down list 

: Use this drop-down list to select which contour is to be displayed. The options are concrete stress, bar stress, concrete strain and bar strain.

5.8. Multiple Curves

The *Multiple Curves* feature gives you an idea of the extent of the effect of various parameters such as compressive strength of concrete, tensile strength if rebar or the rebar sizes on the capacity of the cross-section. It also makes it easier for the designer to determine the most effective way to increase the capacity of the section if the section is found to be inadequate for the given load combinations.

Click on the *Results Menu> Capacity> Multiple Curves* button to access the *Multiple Curves* form that allows the user to plot multiple P-M or M-M diagrams in the same graph with varying values for either Compressive Strength of Concrete (fc'), Steel tensile strength (fy) or rebar sizes. Select the curve type from the following options:

- P-M3 (Angle=0): For plotting a P-M diagram at a neutral axis angle of 0 degrees.
- P-M2 (Angle=90): For plotting a P-M diagram at a neutral axis angle of 90 degrees.
- P-Mn, for specified neutral axis angle: For plotting a P-M diagram at a specified neutral axis angle. Selecting this option requires the user to choose the desired neutral axis

angle from a drop-down list.

- P-Mn, for specified Load Combination: For plotting the P-M diagram for the specified load combination. Selecting this option requires the user to choose the desired load combination from a drop-down list. All the load combinations defined earlier will be available in the drop-down list.
- M3-M2, for specified value of axial load (Pu): For plotting the M-M curve at a specified value of axial load. Selecting this option requires the user to input the desired axial load (Pu) into the input box.

Select the parameter you want to vary from the *Variable Type* list and set the other options as required. Click on the *Refresh* button to view the multiple curves.

5.9. Design Loads

Use the *Results Menu> Results> Capacity> Design Loads* button or the *Project Explorer> Columns> Results> Capacity> Design Loads* to access the *Final Design Loads* form where the final loads considered for the design can be viewed. The final design loads are different from the user input loads in case of slender columns as they need to be modified to account for second order effects. Select the loading direction (Along X or Along Y) by clicking on the respective tab to view the loading in the desired direction. The moments under Along X and Along Y tab correspond to the moment about the X-axis and the Y- axis respectively.

5.10. Design Loads

Use the *Results Menu>Results>Design Loads* button or the *Project Explorer>Columns>Results>Design Loads* to access the *Design Loads* form where the final loads considered for the design can be viewed. The final design loads are different from the user input loads in case of slender columns as they need to be modified to account for second order effects. Select the loading direction (Along X or Along Y) by clicking on the respective tab to view the loading in the desired direction. The moments under Along X and Along Y tab correspond to the moment about the X-axis and the Y- axis respectively.

5.11. Capacity Ratio Calculation

After a section has been defined (geometry and material properties), it is checked against all specified load combinations for adequacy on the basis of the capacity ratio. The capacity ratio is the ratio between the applied actions to the corresponding capacity of the cross-section. The capacity ratio is easy to define when there is one independent action and the corresponding capacity as in the case of a beam. But for a column cross-section with axial loads and moments about 2 axes acting simultaneously on it, the capacity and the capacity ratio are not easy to define. Several capacities and capacity ratios can be defined and computed. CSiCol calculates the capacity ratios by the following different methods:

- Moment Sum at P: It is defined as the sum of individual ratios between the applied moment and the maximum moment capacity about each axis for the given axial load (P_u) as seen in Figure 5.4. The moment capacity is determined by cutting the capacity interaction surface at the given axial load to get a M-M interaction curve.

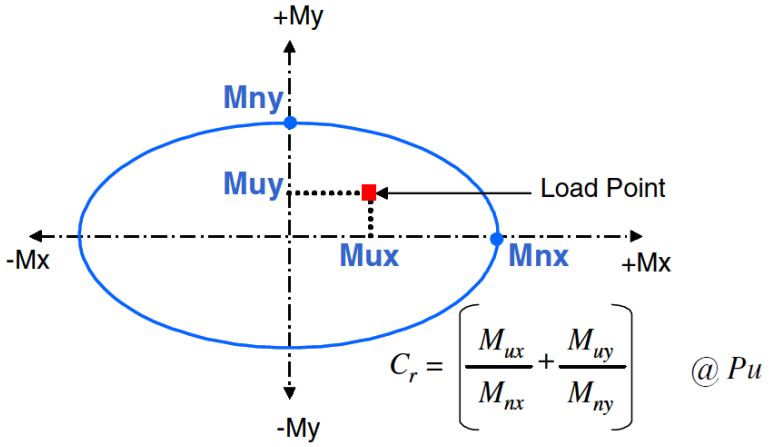


Figure 5.4 Capacity Ratio Based on Moment Sum at P

- Moment Vector at P: It is defined as the ratio of the length of moment vector to the moment capacity vector at the axial load (P_u) as seen in Figure 5.5. The moment capacity is determined by cutting the capacity interaction surface at the given axial load to get a M-M interaction curve.

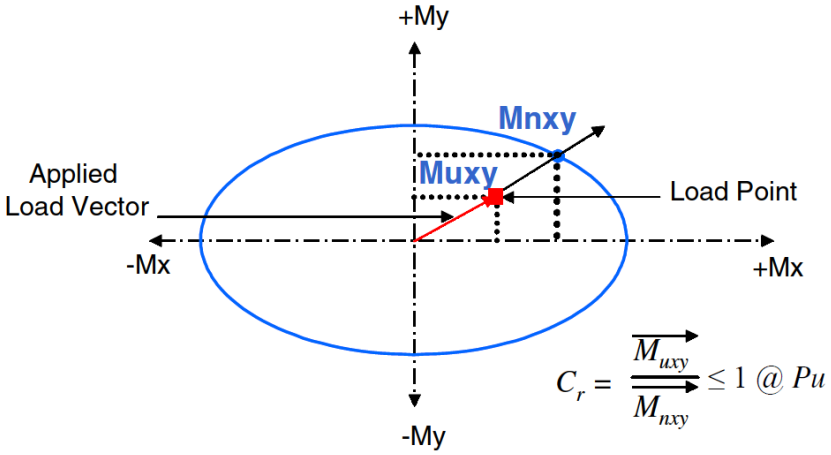


Figure 5.5 Capacity Ratio Based on Moment Vector at P

- Axial P Compression: It is defined as the ratio between the compressive axial load and the maximum compressive axial load capacity in the absence of any moments.

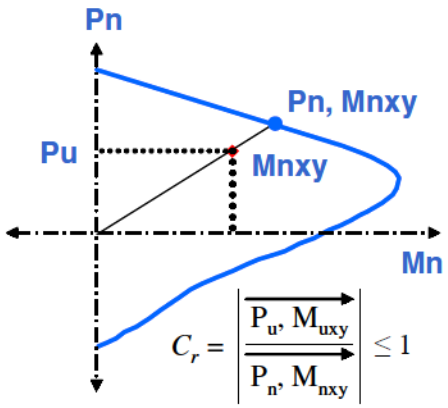
$$Capacity\ Ratio = \left(\frac{P_u}{\phi P_{no}}\right)_{compression}$$

- Axial P Tension: It is defined as the ratio between the tensile axial load and the maximum tensile axial load capacity in the absence of any moments.

$$Capacity\ Ratio = \left(\frac{P_u}{\phi P_{no}}\right)_{tension}$$

- PM Vector: It is defined as the ratio of the length of applied action vector to the capacity vector along the same direction as seen in Figure 5.6

Figure 5.6 Capacity Ratio Based on PM Vector



5.12. Capacity Ratios Results

The results are displayed in terms of capacity ratio. 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 5.7.

Display the capacity calculation ratio as follows:

1. Define the column cross-section for which you want to check the capacity.
2. Click the *Results Menu> Results> Capacity> Capacity Ratios* button to access the *Capacity Calculation Result* form. The form may also be accessed using the *Project Explorer> Columns> Results> Capacity> Capacity Ratios*

The maximum of the capacity ratios calculated from different methods for the bottom and top ends of the section are displayed in two separate tabs. It also displays whether the section is adequate for each load combination or not. For maximum capacity ratios greater than one, the software reports the section as inadequate. The output can be exported to an Excel file by clicking on the Export button Export or copied to the clipboard by clicking on the *Copy* button.

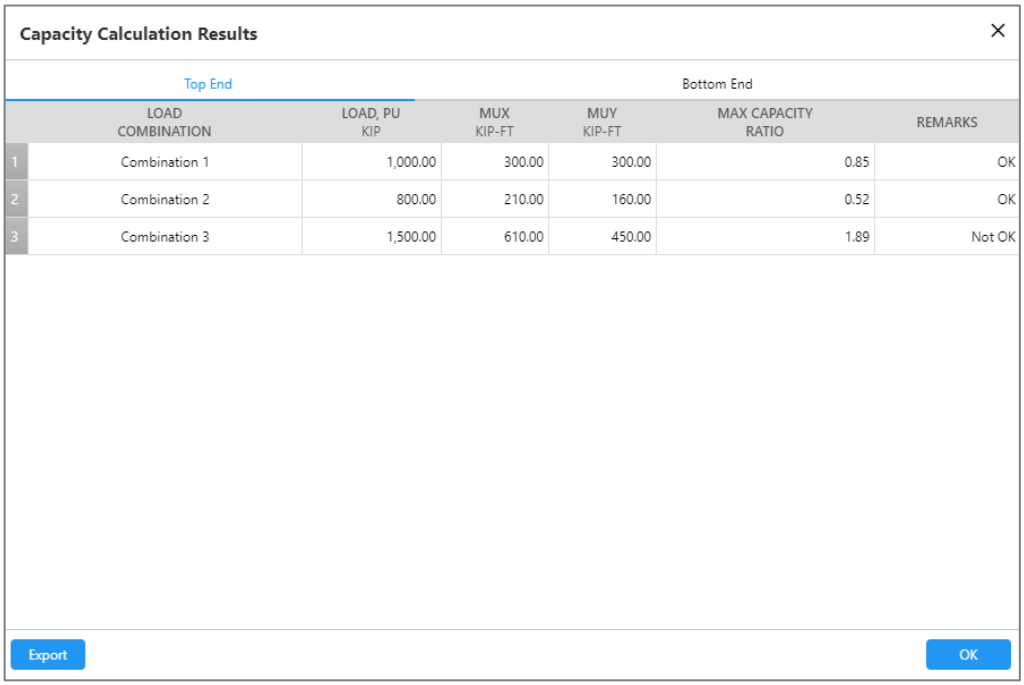


Figure 5.7 *Capacity Calculation Results* Form

Note: The capacity ratio calculated using the "Moment Sum at P" and "PM Vector" methods are excluded while determining the maximum capacity ratio.

5.13. Detailed Results

The detailed results can be accessed by using the *Results Menu> Results> Detailed Results* button or the *Project Explorer> Columns> Results> Detailed Results.* The detailed *Capacity Calculation Results* form (Figure 5.8) displays some additional results besides the maximum capacity ratios such as the resultant moment and resultant moment angle, capacity ratios calculated from all the methods for each load combination. The output can be exported to an Excel file by clicking on the Export button Export or copied to the clipboard by clicking on the *Copy* button.

Figure 5.8 Detailed *Capacity Calculation Results* Form

Note: If the specified axial load is greater than the maximum axial load capacity, the capacity ratios based on "Moment Sum at P" and "Moment Vector at P" show very large values (200 and 100 respectively) indicating that the section is inadequate.

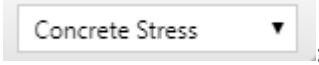
5.14. Stress Strain for Loading

Click on the *Results>Capacity>Stress Strain for Loading* button or the *Project Explorer>Columns>Results>Capacity>Stress Strain for Loading* to view the stress/strain contours in concrete and reinforcement as well as the orientation and the location of the neutral axis for a selected load combination. The associated toolbar has the following options:

- *Load* Combination Drop-down list 

: Select the desired load combination from this drop-down list.
- 2D button 

: Use this button to view the stress/strain contour in 2D view.
- 3D button 

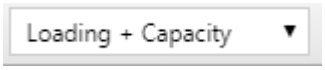
: Use this button to view the stress/strain contour in 3D view.
- Type of Contour drop-down list 

: Use this drop-down list to select which contour is to be displayed. The options are concrete stress, bar stress, concrete strain and bar strain.

5.15. Loading Capacity

Click on the *Results>Capacity>Loading Capacity* or the *Project Explorer Columns>Results>Capacity>Loading Capacity* button to view a visual representation of the loading and capacity of the selected column section in 3D and 2D. The associated toolbar has the following options:

- Load Combination Drop-down list 

: Select the desired load combination from this drop-down list.
- Show Loading/Capacity Drop-down list 

: Use this drop-down list to choose whether to view loading or capacity or both.

5.16. Moment Curvature

Moment curvature diagram is the graphical representation of moment resistance at a section with respect to the curvature. A moment-curvature diagram obtained from the stress resultants are independent of the member geometry or the bending moment diagram and for a specified axial load, it is the property of the cross-section.

The moment-curvature relation is an important input for nonlinear static and nonlinear dynamic analyses of structures including the evaluation of post-elastic behavior and determination of the rotational capacity of plastic hinges formed during high seismic activity. This relationship is also the basis for the capacity-based and performance-based design methods.

The software is capable of plotting moment-curvature curves for a section. Click the *Results Menu> Results> Moment Curvature> Moment Curvature* button or the *Project Explorer> Columns> Results> Moment Curvature> Moment Curvature* to activate the Moment Curvature Diagram in the working area as seen in Figure 5.9. Clicking on this button activates the *Diagram* grid (Figure 5.10) where various settings for the calculation of the moment-curvature diagram such as neutral axis angle, axial load, maximum compression and tension strains, maximum curvature and number of curve points can be adjusted can be adjusted. After the necessary changes have been made in this grid, click on the *Refresh* button to update the diagram. The associated toolbar has the following options:

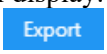
- *Curve Type* Drop-down list 

: Click on this drop-down list to select the type of graph to be displayed. The options available are Moment vs Curvature, Compressive Steel Force vs Curvature, Concrete Force vs Curvature, Maximum Steel Strain vs Curvature, Maximum Concrete Strain vs Curvature and Neutral Axis vs Curvature graphs.

  There is an additional option in the toolbar for Compressive Steel Force vs Curvature, Concrete Force vs Curvature, Maximum Steel Strain vs Curvature and Maximum Concrete Strain vs Curvature diagrams which allows you to view the diagrams considering compression to be negative. CSiCol considers compression to be positive but for the moment curvature diagrams, this option provides a functionality to change the sign convention.
- *Change sign convention* button 

: Click on the +ve sign or -ve sign to consider compression as positive or negative respectively.
  There is an additional option in the toolbar for Neutral Axis vs Curvature diagram which allows you to select whether the neutral axis depth is considered from the top of the section or from the centroid of the section.

- *Neutral Axis Depth* button 

: Click on the YNA button to consider the neutral axis depth from the centroid of the section and the NA button to consider the neutral axis depth from the top of the section.
- Tabulated Output button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button 
: Click on this button to add the Moment Curvature diagram to the detailed report.
- Curve Details button 
: Click on this button to access the Curve Details form which displays details regarding the moment curvature diagram such as cross section angle, axial load capacity, initial stiffness and ductility ratio.

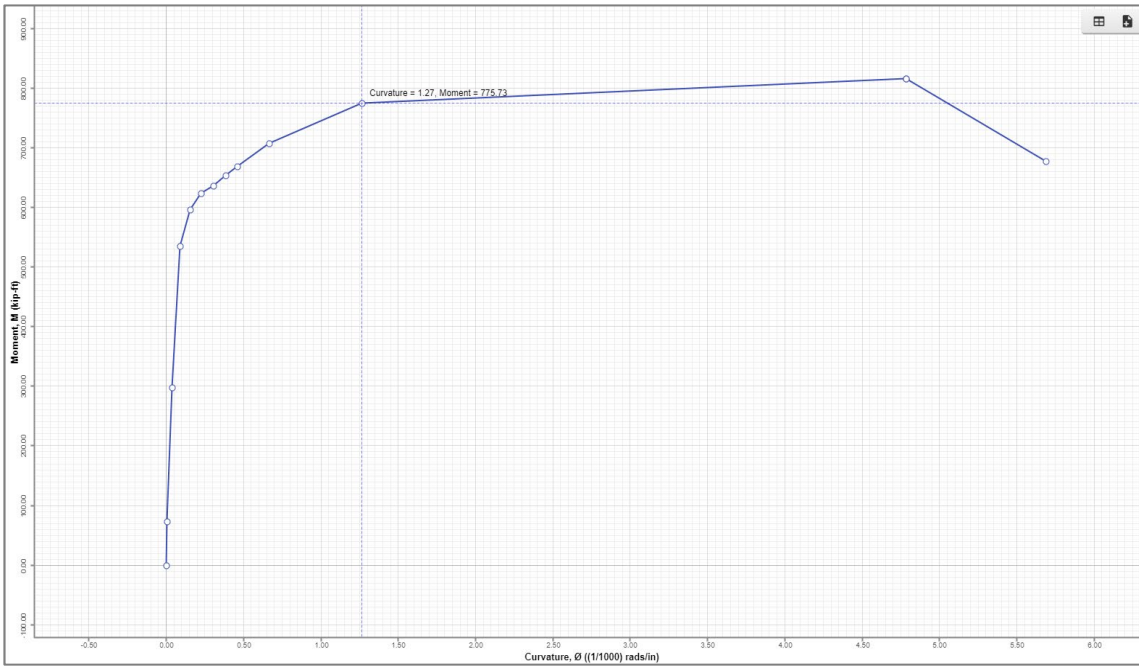


Figure 5.9 Moment Curvature Diagram

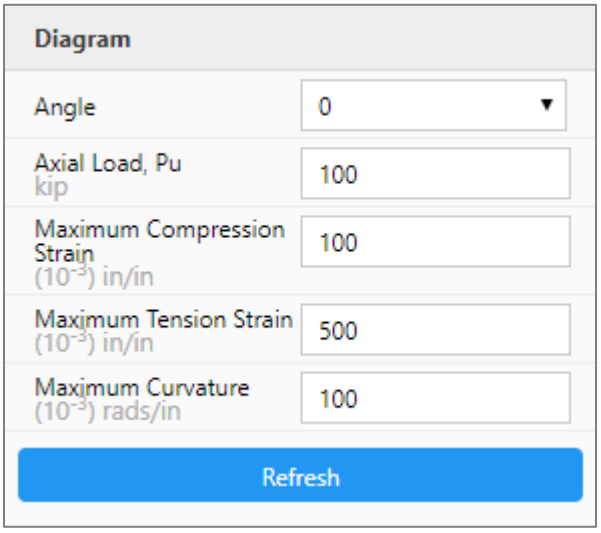


Figure 5.10 *Diagram* Grid

Note: The curves are primarily intended for reinforced concrete sections and composite sections (Concrete sections with steel)

5.17. Moment Curvature Details

Use the *Results Menu> Results> Moment Curvature> Moment Curvature Details* button or the *Project Explorer> Columns> Results> Moment Curvature> Moment Curvature Details* to view moment-curvature details at several points along the curve. The working area is divided into two sections with associated toolbars. The left section is the moment curvature section and the right section is the stress/strain contour section. As you hover along the curve points in the moment curvature section, parameters such as Point Status, force details, moment, curvature, strain values, neutral axis depth and stiffness values are displayed in the context-sensitive grid whereas the neutral axis depth and stress/strain contours are displayed in the stress/strain contour section.

The associated toolbar has the following options:

- *Curve Type* Drop-down list 

: Click on this drop-down list to select the type of graph to be displayed. The options available are Moment vs Curvature, Compressive Steel Force vs Curvature, Concrete Force vs Curvature, Maximum Steel Strain vs Curvature, Maximum Concrete Strain vs Curvature and Neutral Axis vs Curvature graphs.
  There is an additional option in the toolbar for Compressive Steel Force vs Curvature, Concrete Force vs Curvature, Maximum Steel Strain vs Curvature and Maximum Concrete Strain vs Curvature diagrams which allows you to view the diagrams considering compression to be negative. CSiCol considers compression to be positive but for the moment curvature diagrams, this option provides a functionality to change the sign convention.
- *Change sign convention* button 

: Click on the +ve sign or -ve sign to consider compression as positive or negative, respectively.

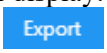
There is an additional option in the toolbar for Neutral Axis vs Curvature diagram which allows you to select whether the neutral axis depth is considered from the top of the section or from the centroid of the section.

- *Neutral Axis Depth* button 

: Click on the YNA button to consider the neutral axis depth from the centroid of the section and the NA button to consider the neutral axis depth from the top of the section.

- *Value of P_u* edit box 

: This edit box displays the axial load level for the currently displayed curve. Type in the axial load level (P_u) for which you want to view the curve and click on the *Refresh* button.
- *Angle* drop-down list 

: This list displays the neutral axis angle for the currently displayed curve. Select the neutral axis for which you want to view the curve and click on the *Refresh* button.
- *Refresh* button  : Click on this button to update the displayed curve after changing the P_u or Angle values.
- Tabulated Output button  : Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button  : Click on this button to add the Moment Curvature diagram to the detailed report.
- Curve Details button  : Click on this button to access the Moment Curvature details form which displays details regarding the moment curvature diagram such as cross section angle, axial load capacity, initial stiffness and ductility ratio.

5.18. Max Curvature Plot

Use the *Results Menu>Moment Curvature>Max Curvature Plot* or the *Project Explorer>Columns>Results> Moment Curvature>Max Curvature Plot* button to view the Maximum Curvature plot.

Clicking on this button activates the *Diagram* grid in the context-sensitive area where various parameters for the max curvature plot such as maximum compression strain, maximum tension strain, maximum curvature, maximum curvature factor and number of curve points can be adjusted. After making the necessary changes in this grid, click on the *Refresh* button to update the diagrams.

The working area is divided into three sections with associated toolbars that can be used to modify the display on each. Each section has been described below with their associated toolbars.

5.18.1. Maximum Curvature Surface

The variation of all three parameters governing the maximum curvature surface (axial load, curvature 2 and curvature 3) is plotted in a color-coded 3D graph. This gives an overall picture of the variation of all three parameters for the given column section. Axial load is plotted along the

vertical axis and curvatures along two orthogonal horizontal axes. A color-coded key is displayed at the extreme right of the section to help interpret the axial load values.

5.18.2. Axial Load vs Curvature Curve

The associated toolbar has the following options:

- *Angle* drop-down list 

: This list displays the neutral axis angle for the currently displayed curve. Select the neutral axis for which you want to view the curve. The curve display will be updated automatically. Move the cursor/mouse pointer over the curve to read the values for any point on the curve.
- Tabulated Output button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

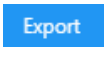
or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button 
: Click on this button to add the axial load vs curvature diagram to the detailed report.

5.18.3. Curvature-Curvature Curve

This view shows a variation of curvature, for a selected value of axial load for various angles of neutral axis.

The associated toolbar has the following options:

- *Value of* P_u edit box 

: This edit box displays the axial load level for the currently displayed curvature-curvature curve. Type in the axial load level (P_u) for which you want to view the curve. The curve display will be updated automatically.
- Tabulated Output button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button 
: Click on this button to add the curvature-curvature diagram to the detailed report.

5.18.4. Maximum Curvature Surface:

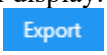
The variation of all three parameters governing the maximum curvature surface (axial load, curvature 2 and curvature 3) is plotted in a color-coded 3D graph. This gives an overall picture of

the variation of all three parameters for the given column section. Axial load is plotted along the vertical axis and curvatures along two orthogonal horizontal axes. A color-coded key is displayed at the extreme right of the section to help interpret the axial load values.

5.18.5. Axial Load vs Curvature Curve

The associated toolbar has the following option:

- *Angle* drop-down list 

: This list displays the neutral axis angle for the currently displayed curve. Select the neutral axis for which you want to view the curve. The curve display will be updated automatically. Move the cursor/mouse pointer over the curve to read the values for any point on the curve.
- Tabulated Output button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button 
: Click on this button to add the axial load vs curvature diagram to the detailed report.

5.18.6. Curvature-Curvature Curve

This view shows a variation of curvature, for a selected value of axial load for various angles of neutral axis.

The associated toolbar has the following options:

- *Value of P_u* edit box 

: This edit box displays the axial load level for the currently displayed curvature-curvature curve. Type in the axial load level (P_u) for which you want to view the curve. The curve display will be updated automatically.
- Tabulated Output button 
: Displays the tabulated output of the curve on display. The output can be exported to an Excel file by clicking on the Export button 

or copied to the clipboard by clicking on the *Copy* button.
- Add to Report button 
: Click on this button to add the curvature-curvature diagram to the detailed report.

5.19. Axial Load Strain Plot

Use the *Results Menu>Moment Curvature>Axial Load Strain* or the *Project Explorer>Columns>Results> Moment Curvature>Axial Load Strain* button to plot axial load vs strain curve for the selected section. Clicking on this button activates the *Diagram* grid in the context-sensitive area where various settings for the calculation of the axial load strain diagram such as maximum strains and number of curve points can be adjusted. After the necessary changes

have been made in this form, click on the *Refresh* button to update the diagram. The associated toolbar has the following options:

- *Add to Report* button  : Click on this button to add the Axial Load vs Strain diagram to the detailed report.

6. Generate a Report

6.1. Overview

This chapter describes the procedure for viewing 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 viewing and printing the report remains unchanged.

6.2. Viewing Report

Report can be viewed by selecting any of the different types of report available under *Project Explorer>Report* or under *Results Menu>Reports.* There are 3 different options available:

- Column Summary
- Simple Report
- Detailed Report

6.3. Column Summary

Use the *Results Menu>Reports>Column Summary* button or the *Project Explorer> Report> Column Summary* to access the *Summary* form (Figure 6.1) which summarizes some critical information regarding all the columns in the *Project File* such as the governing load, the rebar configuration, rebar ratio and the corresponding D/C ratio.

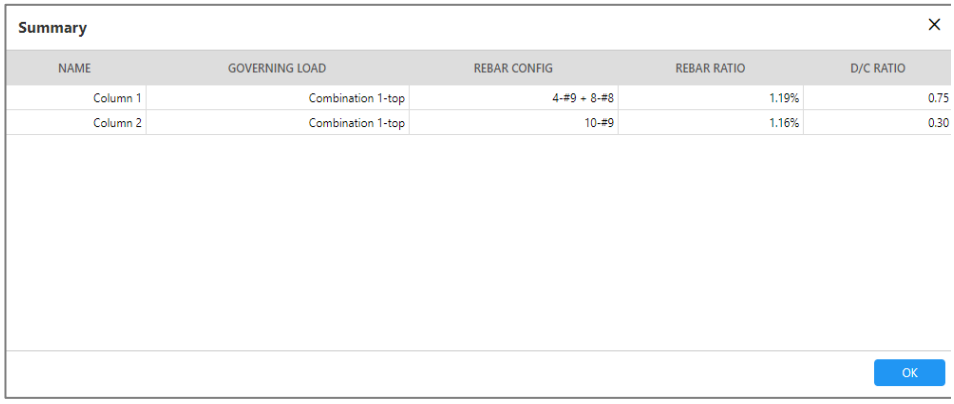


Figure 6.1 *Column Summary* Form

6.4. Simple Report

Use the *Results Menu>Reports>Simple Report* button or the *Project Explorer>Report>Simple Report* to generate a one-page report (Figure 6.2) for each of the columns in the *Project File*. The one-page report contains details regarding the column such as section properties, material properties, rebar details and the governing load combination. It also includes the PM diagram and MM diagram. The various options available in the context-sensitive toolbar of the *Simple Report* are:

- Zoom Level Drop-Down List 

: Select a zoom level from this drop-down list to view the report at the desired zoom level.
- Zoom In button  : Click the Zoom In button to zoom in (make the image larger).
- Zoom Out button  : Click the Zoom Out button to zoom out (make the image smaller).
- Rest and Refresh View button  : Use this button to restore the original default display of the report.
- Print button  : Click on this button to print the Report.

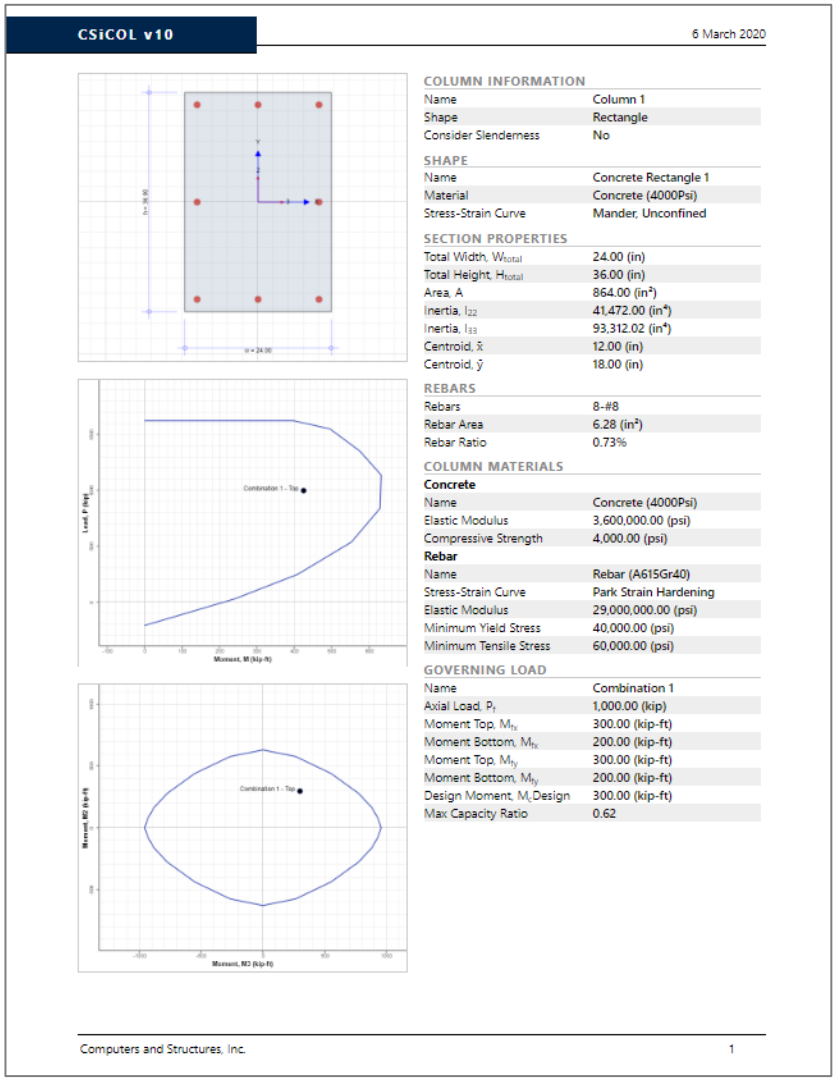


Figure 6.2 *Simple Report*

6.5. Detailed Report

Use the *Home Menu>Results and Reports>Detailed Report* button or the *Project Explorer>Report>Detailed Report* to access the detailed report for all the columns in the Project File. The detailed report includes Project Information, Column Section diagrams, Materials properties, Stress-strain model parameters, Rebar properties, Section properties, Framing (in case of slender columns), Loadings and Design information. The various options available in the context-sensitive toolbar of the *Detailed Report* are:

- Zoom Level Drop-Down List 

: Select a zoom level from this drop-down list to view the report at the desired zoom level.
- Zoom In button  : Click the Zoom In button to zoom in (make the image larger).
- Zoom Out button  : Click the Zoom Out button to zoom out (make the image smaller).

- Rest and Refresh View button  : Use this button to restore the original default display of the interaction surface.
- Print button  : Click on this button to print the report.

6.6. Adding to the Detailed Report

Additional graphics can be added to the report at any time before printing by clicking on the *Add to Report* button  available in the context-sensitive toolbar for all the diagrams in the *Results Menu*.

6.7. Printing or Saving a Report

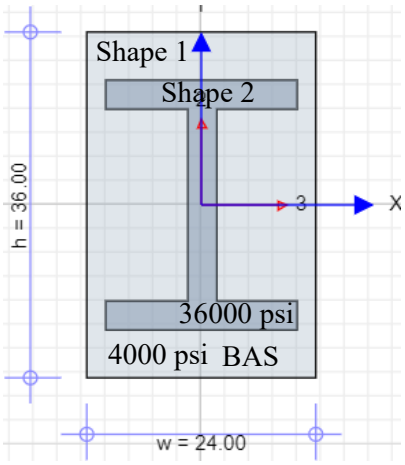
The *Simple Report* and the *Detailed Report* can be printed by clicking on the *Print* button  available in the associated context-sensitive toolbar of the Report. The report can be saved by printing it to PDF format.

Appendix A – Transformed Property Examples

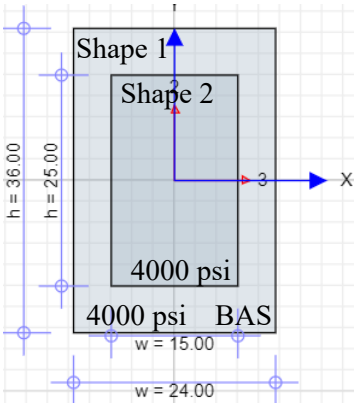
Overview

This Appendix gives examples for common situations where multiple shapes overlap and how the different material properties and shapes are handled automatically in the program calculation.

Transformed Section Properties Calculation

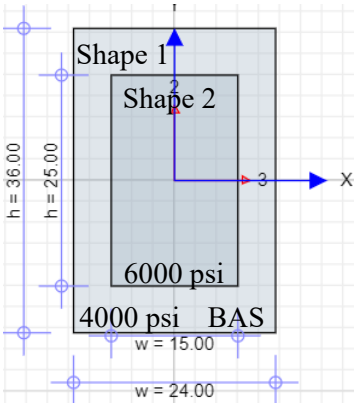
CASE TYPE	DESCRIPTION
Case 1: Fully overlapping Steel shape 	• Overlapping region is accounted for • The overlapping region of steel shape is subtracted from the base concrete shape (Shape 1) and the resulting polygon is added to the main mesh • The steel shape is then added to the main mesh • Since both shapes have different material, modular ratio = 8.06 • Area = 24*36-180 + 8.06*180 = 2134 in^2

Case 2: Fully overlapping concrete shape w/ same material



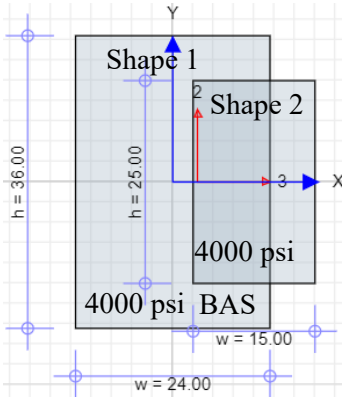
- Overlapping region is accounted for
- The overlapping region is subtracted from the base concrete shape (Shape 1) and the resulting polygon is added to the main mesh
- Shape 2 is then added to the main mesh
- Since both shapes have the same material, modular ratio =1
- Area = = 24*36-15*25 +1*15*25 = 864 in^2

Case 3: Fully overlapping concrete shape w/ different material

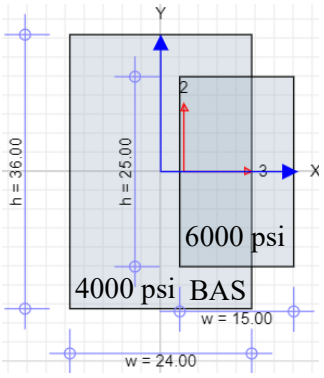


- Overlapping region is accounted for
- The overlapping region is subtracted from the base concrete shape (Shape 1) and the resulting polygon is added to the main mesh
- Shape 2 is then added to the main mesh
- Since both shapes have different material, modular ratio =1.22
- Area = 24*36-15*25 +1.22*15*25 = 949 in^2

Case 4: Partially overlapping concrete shape w/ same material



- Overlapping region is accounted for
- The overlapping region is subtracted from the base concrete shape (Shape 1) and the resulting polygon is added to the main mesh
- Shape 2 is then added to the main mesh
- Since both shapes have the same material, modular ratio = 1
- Area = 24*36-9.5*25 + 1*15*25 = 1001 in^2

Case 5: Partially overlapping concrete shape w/ different material 	• Overlapping region is accounted for • The overlapping region is subtracted from the base concrete shape (Shape 1) and the resulting polygon is added to the main mesh • Shape 2 is then added to the main mesh • Since both shapes have different material, modular ratio =1.22 • Area = 24*36-9.5*25 + 1.22*15*25 = 1086 in^2