ConcreteSection 3.0

User's Guide

ConcreteSection 3.0 User's Guide

Table of Contents

1 Help Topics

1.1 Welcome to ConcreteSection 3.0

ConcreteSection will help you design complex reinforced concrete sections. This powerful program takes care of the modeling details while you specify the location and size of steel reinforcement. The software will analyze your concrete model to determine the 3D interaction diagram, concrete stresses and strains, and reinforcement stresses. ConcreteSection also performs design checks using ACI or CSA strength reduction factors by calculating a moment unity value based on the applied loading and the interaction surface.

Getting Started

- Use **File | Open Example Project** to see sample projects.
- Feature List
- Program Layout
- Upgrade Guide (what's new)
- FAQ Answers at iesweb.com for business, licensing, installation issues.

Help Notation

Menu items appear like this: **File | New**.

Keystrokes or mouse commands appear like this: ***Shift+Click***.

Disclaimer

ConcreteSection is a proprietary computer program of Integrated Engineering Software (IES, Inc.) of Bozeman, MT. This product is intended for use by licensed, practicing engineers who are educated in structural engineering, students in this field, and related professionals (e.g. Architects, Building Inspectors, Mechanical Engineers, etc.). Although every effort has been made to ensure the accuracy of this program and its documentation, IES, Inc. does not accept responsibility for any mistake, error, or misrepresentation in, or as a result of, the usage of this program and its documentation. (Though we will make every effort to ensure that problems that we can correct are dealt with promptly.) The results obtained from the use of this program should not be substituted for sound engineering judgment.

License and Copy Restrictions

By installing ConcreteSection on your computer, you become a registered user of the software. The ConcreteSection program is the copyrighted property of IES, Inc. and is provided for the exclusive use of each licensee. You may copy the program for backup purposes and you may install it on any computer allowed in the license agreement. Distributing the program to coworkers, friends, or duplicating it for other distribution violates the copyright laws of the United States. Future enhancements and technical support depend on your cooperation in this regard. Additional licenses and/or copies of ConcreteSection may be purchased directly from IES, Inc.

IES, Inc.

Integrated Engineering Software, Inc.
3740 Equestrian Ln Unit 1
Bozeman, MT 59718
Sales or Licensing: 406-586-8988, sales@iesweb.com
Technical Support: support@iesweb.com

ConcreteSection 3.0 User's Guide

1.2   Key Features

Modeling

- Quick-start page with common geometries
- Create any geometry by sketching boundaries
- Import from CAD for complex and detailed boundaries
- Create new model objects by copying existing items
- Ability to create a shape with one or more holes
- Reinforcement can be a variety of shape, including hot-rolled steel
- Reinforcement arrays for easy placement

Loading

- Load may be placed in multiple service load cases (e.g. Dead, Live, etc.)
- Includes IBC, ASCE 7, and NBC Load Combinations (customizable)
- Automatic generation of Building Code Load Combinations
- Create custom load combinations in any project
- Apply axial loads and moments in two directions (X & Y)
- Copy and scale loads to other load cases

Analysis

- Column 3D Interaction Diagrams
- Adjustable number of interaction points and quadrant slices
- Adjustable concrete strain limits (crushing and ultimate)
- Stresses calculated for each reinforcement "bar" in the section based on the applied loads
- Reinforcing is assumed to be linear-elastic then purely plastic after yield
- Concrete stress-strain behavior follows the model as discussed on the Analysis page
- Automated "background" analysis is fast
- Advanced error-checking and reporting

Design

- Unity Checks calculated based on the applied loads and the Column Interaction Diagram
- Ability to use strength reduction factors (phi) per ACI and CSA Specifications
- Limited maximum nominal compressive strength, $\phi Pn,max$

Reporting

- IES Advanced Reporting System
- Custom reporting to include just the information you need
- Print Preview mode while working with reports
- Paste any graphics into your report
- Customizable page margins, fonts, colors
- Use your own company logo in report page headers
- Print to any printer including PDF

- Export to text clipboard or save to other formats like .xlsx

General

- 3D Graphics
- Simple, standard Windows interface for easy navigation
- Unlimited Undo & Redo commands
- Work in any unit system, perform math on input, use custom unit 'styles'
- Program is self-documenting with tooltips on commands and input parameters
- Numerous preference settings for better defaults
- Free training videos provided for learning efficiency
- Free technical support email with fast, friendly turnaround

Limitations

- Concrete strength must be the same for all boundaries in the project
- Does not produce structural drawings
- Curved boundaries are approximated with series of straight segments
- Cannot model a system of disconnected boundaries

Be a Squeaky Wheel

If you need a new feature, please let us know! We are always looking for ways to improve products in ways that you desire. See Technical Support.

# 1.3   Program Layout

The best way to learn ConcreteSection is to use and explore the program to get to know what is available under each button or menu. Several Tutorial Videos are also available which explain many features of the software.

Screen Layout

The image below introduces the program terminology used in this help file and the training videos. Panels may be resized by dragging their dividers or repositioned by dragging their title bars or right-clicking on the title. Use the "pushpin" icon to collapse panels temporarily to gain more space for working. Hold the mouse pointer over the screen image below for information about each area of the program.

ConcreteSection 3.0 User's Guide
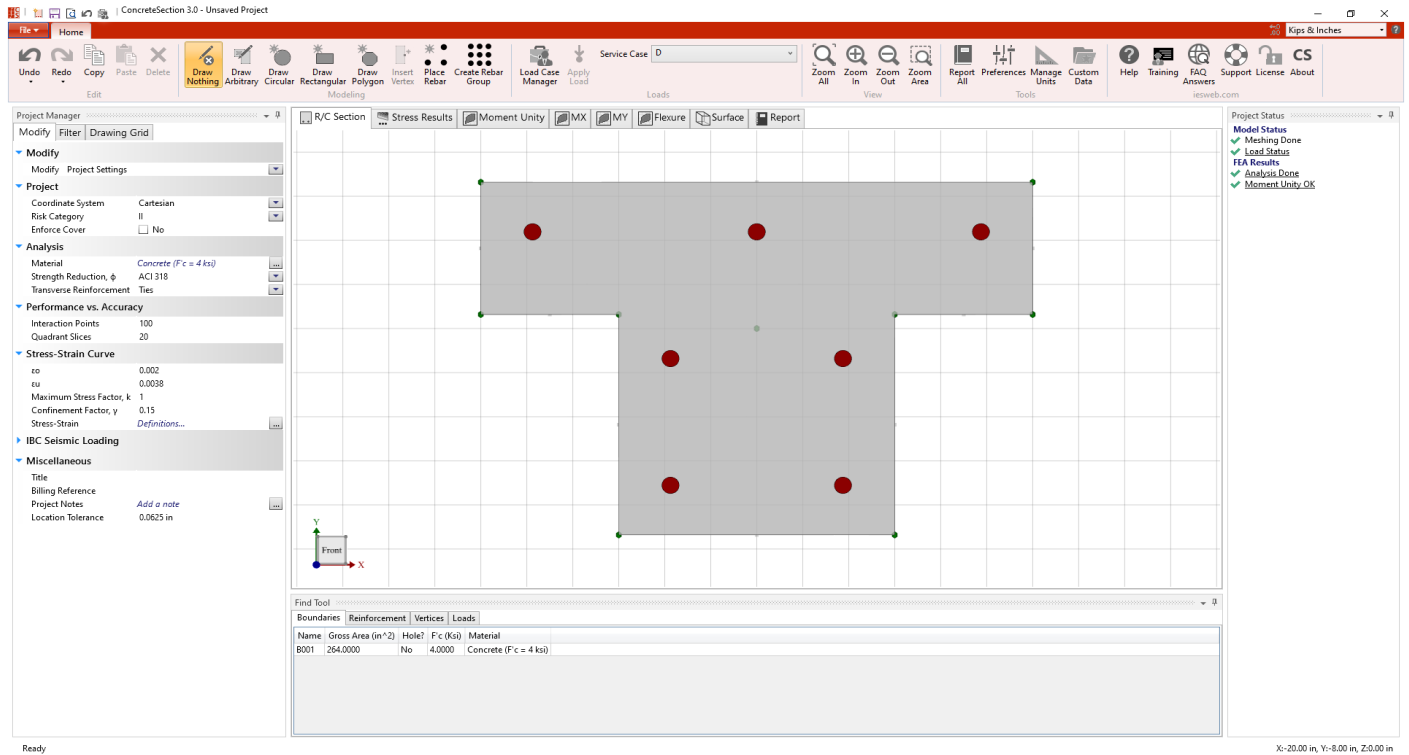
Main Menu / Toolbar

The main menu, Toolbar, or Ribbon, contains various commands to direct ConcreteSection. Each is organized within a group to help locate them quickly. Each command has a description which appears when the mouse pointer is hovered over it. Many have hot-key shortcuts.

Project Manager

The Project Manager provides immediate access to frequent operations in ConcreteSection. This tool is docked on the left side of the window by default and displays various tabs depending on the active window. This window can be docked on the left or made to float independently if more space is needed to work. Alternatively, drag the side border to make it wider or narrower.

- The **Modify** tabs are used to change the project settings or the properties of selected objects in the R/C Section view.
- The **Filter** tab is used to control what is shown or hidden in the active view.
- The **Drawing Grid** tab is used to control the Sketch Grid to aid in drawing models in the R/C Section view.
- The **Result** tab replaces the Modify tab when the Stress Result View is active. This tab provides key result information for the active load case.
- The **Tables** tab is used to add available tables to the report.
- The **Report Filter** tab is used to define the report settings, apply the model filters, modify the parameters of the table selected in the report, and select which table columns to display.

Graphic Views

These views provide a way to view the reinforced concrete section, stress results, interaction diagrams, and reports. Each tab displays different options and will provide different information in the Project Manager and Find Tool. Some Graphic tabs will only appear based on your model, such as the Stress Results view.

Project Status

This panel provides a quick update on what is done, what is in-progress, and whether things are working or failing in your model. Click on any item that is underlined for more information or a report, or a dialog containing quick actions.

Pipeline Status

Shows background meshing and analysis progress. Background processing is done on a separate thread of your processor so you may continue working while the program runs. The only time you need to wait for the program is when the mouse cursor changes into an hour-glass or if you wish to view the stress results that are currently in-progress.

Find Tool

The Find Tool provides an efficient way to view, select, and edit boundaries, reinforcement, loads, etc. This tool is docked on the bottom of the window by default. Use *F7* or the push-pin icon to auto-hide this panel. When docked, drag the side border to make the panel larger or smaller. The Find tool allows you to find, select, edit, and delete objects even if they are not visible in the active window. ***Double-click*** on an element (boundary, reinforcement, vertex, etc) and the graphics window will zoom-in to show that element, if it is visible. Lists shown in the Find tool can be sorted by clicking on a column header (***click*** again to reverse the order). Select items just like any list in Windows using the ***Shift*** and ***Ctrl*** keys to select a range or to toggle individual items.

Units & Precision

Above the toolbar on the far right is the Units drop-down for selecting the way physical quantities are displayed. Change the number of decimal places or significant digits using the icon to the left of the unit selector. Go to **Home | Manage Units** to create custom unit styles or edit existing unit styles.

Data Entry: Physical Quantities

Enter values in any unit style. Enter any number or math expressions followed by a known abbreviation. Length units may be entered in "ft-in-16ths" notation as well. Entered values are converted and then redisplayed in the current 'display' units.

Mouse and Keyboard Commands

Selection:

- ***Click*** to select (mouse hover indicates what object will be selected)
- ***Click*** in the 'whitespace' of a view to unselect everything and access Project Settings
- ***Ctrl+Click*** to toggle object selection without affecting other objects
- ***Shift+Click*** to select all objects of a given type
- ***Shift+Drag*** draw a selection box (left-to right selects fully enclosed objects, right-to-left selects any partially enclosed objects)
- ***Shift+Ctrl+Click*** to select items of one type with the same Name Prefix as the item clicked on

Zoom:

- ***Scroll Mouse Wheel*** with the pointer over the point to zoom in or out
- ***Double Click Mouse Wheel*** to Zoom All
- ***Ctrl+*** (plus) and ***Ctrl-*** (minus) keys
- ***Ctrl+Home*** for zoom all/extents
- ***Ctrl+End*** to enable the **Home | Zoom Area** command then ***Drag*** to create the Area

Pan:

- *Drag Mouse Wheel* to pan
- *Shift+Arrow* keys will also pan

Rotate:

- *Ctrl+Drag Mouse Wheel* to rotate the view
- *Click* on a face, edge or corner of the Cube in the lower-left corner of the graphics to rotate the view
- *Ctrl+Arrow* keys will also rotate

Context Menu:

- *Right-Click* the mouse for a short menu of relevant commands based on the view and what is selected
- *Shift+F10* also display the context menu

Hot Keys:

- *Alt* will expose the hot-keys in the main menu
- *F1* Help
- *F7* Show or hide the Find Tool
- *Esc* Cancel the Graphic drawing and enter the Draw Nothing mode
- *Delete* the Graphic selection
- *Ctrl+C* Copy graphic image to clipboard
- *Ctrl+V* Generate copies, or paste graphics in Report View

Miscellaneous:

- *Drag* in the R/C Section view to move reinforcement.
- *Double Clicking* in the Stress Result will generate a Text Report for the object. Double-clicking on an element or node in the Find Tool will Zoom to that item.

Context Menu:

Right-Click the mouse for a short menu of relevant commands based on the view and what is selected.

Middle-Mouse "Button" in Windows

Depending on your system, you may need to go into Control Panel, Hardware, Mouse, and set the wheel button to behave like a "middle button click". Some mouse utility programs may override that setting or it may not be set up on some versions of Windows.

1.4   Upgrade Guide

Version 3.0 (May 2022)

Watch the **Upgrade Guide Video** to see the new features of ConcreteSection 3.0 in action.

New Features

- Added reinforcement strain to the section results, reports, and interaction diagrams
- Improved interaction diagram filters and annotations
- Updated concrete design specifications to ACI 318-19 and CSA A23.3:19
- Added reinforcement minimum stress to the section results and report
- Created new embedded steel shape template projects
- Enhanced the Create Rebar Group dialog to include confinement and clear cover
- Improved drawing grids
- Load Case Manager columns now retain their size and order
- Improved filtering of table extremes
- Individual column justification added in reports (right click on column)
- Preferences created for justification of text and data in reports

Fixes & Minor Changes

- Improved Help File documentation
- Crash recovery file improvements
- Updated the c++ runtimes to the current standards

1.5   Release History

Prior Releases

- [Version 2.0 released October 2018](#)
- [Version 1.0 released December 2017](#)

Version 2.0 Features (October 2018)

General

- Improved start screen (thumbnail views)
- Graphic wire frame on model objects
- Project Manager: Categories remember last open/collapsed state
- Project Manager: drop-lists are activated by clicking anywhere, not just on arrow
- Improved Print Preview display for graphics
- Graphics performance is **up to 20x faster**
- Removed 'memory leaks', which slowed program over time
- Multiple selection in Load Case Manager

Modeling

- Steel is no longer required to be contained in the concrete boundary, allowing the analysis of composite steel cross sections
- Select and drag reinforcing graphically using **Alt+Drag** on the reinforcement
- Section can now have a "cover" value the reinforcement must be inside of
- Each steel reinforcement object can have its own Fy value
- Ability to insert a vertex along an area side
- DXF import shows bounds and an option for centering at the origin

- Polygon boundary defined by side length in addition to radius

Loading

- Added support for NBC load combinations

Design

- **CSA A23.3** support for strength reduction factors (phi) when generating Interaction Diagrams

Reporting

- **Name filters** are now enabled for both graphic and report filters
- Unavailable tables shown disabled with reason for not being available
- Table drop position used to locate when adding new tables in Text Reports

Version 1.0 Features (December 2017)

Introductory version.

General

- 64-bit implementation
- Multiple-threaded architecture uses all processor cores
- UI, Ribbon toolbar, consistent with other IES tools
- 3D representation of section
- [Preference](#) settings (fonts, colors, sizes, options, etc.)
- History Flies (automatic daily backups of a project-file, see preferences)
- Your Logo in a Text Report (see preferences)

Modeling

- No merge/intersection necessary for shape boundaries
- Easy to edit shapes
- Ability to create a shape with one or more holes
- Any Reinforcement Shape (triangulation into "bars")
- Automatic shape dimensioning

Loading

- Loading with Service Case source and Building Code Combinations
- Implements IBC and ASCE 7 load combinations

Analysis

- Load Level Interaction Diagram, including ACI strength reduction factors (phi)
- User adjustable concrete strain limits (crushing and ultimate)
- Automated behind-the-scenes
- Much faster analysis (using all processor cores)
- Accurate results

Design

- Unity Checks based on the applied forces and the Column Interaction Diagram

Reporting

- Powerful Report Viewer
- Many tables and options
- Saved reports in project files
- Paste graphics into reports
- Complete project report

1.6   Property Definitions

Concrete Shape Terminology

Fy	The yield stress of the reinforcing bars.
f'c	The 28-day concrete compressive strength.
f"c	The maximum concrete compressive stress.
ε_u	The ultimate concrete strain.
ε_o	The concrete strain at the maximum compressive stress.
ε	Extreme compression fiber strain under the loading specified.
Maximum Stress Factor, k	Multiplier for the compressive strength, f'c, to obtain the maximum compressive stress, f"c.
Confinement Factor, γ	The confinement factor used to calculate the compressive stress at the ultimate concrete strain.

Uncracked Concrete Transformed Section Properties

Modular Ratio, n	The ratio of the modulus of elasticity of steel (Es) to the modulus of elasticity of concrete (Ec).
Area	Area of the cross section.
Xc, Yc	X and Y coordinates of the cross-section's centroid relative to the global origin.
Ix, Iy	Moment of inertia about the centroidal x and y-axes
Ixy	Product of inertia about the centroidal x and y-axes
Sx+, Sx-, Sy+, Sy-	Elastic section modulus about the centroidal axis (x or y) with respect to the extreme fiber in the positive or negative direction.
fcr	The Modulus of Rupture

$$f_{cr} = 7.5\sqrt{f'_c}$$

Note: Applicable properties are transformed to concrete.

Mcr Top Mcr Bottom Mcr Right Mcr Left	The cracking moment for bending about the x and y axes. The Mcr Right corresponds to the right side of the section being in tension, etc. $$M_{cr} = f_{cr} \cdot S$$

Note: Applicable properties are transformed to concrete.

Cracked Concrete Section Properties

N.A. Offset	Perpendicular distance from the neutral axis to the centroid under the loading specified.
N.A. Angle	Neutral axis inclination angle measure ccw from the X-axis under the loading specified.
Icr X, Icr Y	Moments of inertia of the cracked section about the X and Y-axes.

Interaction Surface Points

Po, To	The axial only capacity of the section. Calculated based on a load that causes crushing of the concrete (compression) and/or yielding of the reinforcement (tension, compression).
Mox, Moy	The uniaxial moment capacity of the section for bending about the x and y-axes.
Pbx, Pby, Mbx, Mby	The balance points (i.e. transition points) on the Interaction Diagram for Axial-Moment interaction caused by bending about the x and y-axes.

Note: These extreme points on the Interaction Diagrams are modified by the Strength Reduction Factors (φ) when specified

1.7   Preferences

ConcreteSection preferences are default settings that primarily affect the behavior of new projects. These are not project-specific settings, which are found in the Project Manager. The preference settings can be adjusted through **Home | Preferences**. Some settings do not take effect until a new project is created or until the program is restarted. Use the Restore All Defaults button to restore the ConcreteSection preference settings to their original state. While most of the preference settings are self-explanatory, a few are documented below. Preference settings are saved on your machine in the IES folder: *C:\Users\<your.login>\AppData\Local\IES\Customer*.

Project

The project preferences affect new projects, and do not affect the current project. For current projects, use the settings in **Project Manager**.

- **History Files** - Set how many once-per-day backup files ConcreteSection should keep. Files are located in **Home | Custom Data** in the History Projects folder.
- **Next Inspector Field On Enter** - In the Modify tab, 'Enter' can simply accept changes or also advance like a 'Tab' to the next row.

Fonts

Change the character size and styles used to display text in graphic views and reports.

Reports

- **Customer Logo** - Specify the location of a logo to use for the reports. If left blank, AppData\Local\IES\Customer\ReportLogo.jpg will be tried.
- **Logo Alignment** - Select the alignment of the logo in the header.
- **Header Height** - Set the height of the header in the report.
- **Justification, Text Data** - Specify the justification of the text data in the report tables.
- **Justification, Physical Data** - Specify the justification of the physical data in the report tables.

Graphics

- **Graphic Sizes** - Change how large objects are drawn in graphic views.
- **Rotate, Pan, & Zoom -** Control how much or how fast the view changes with mouse-wheel or arrow keys.
- **Default Snap Points** - Set the default number of evenly spaced internal points that can be snapped to.
- **Print Resolution** - Set the DPI (dots per inch) precision to be used when displaying graphics views on the printer or when placing graphics information on the clipboard.

Colors

Change the colors of objects in Graphic Views and Reports. Every visible object type shown in graphic views have a default color (e.g. rebar is red by default).

1.8 Load Combination Criteria

Load Combination Sets

Several sets of building code load combinations are built into ConcreteSection. Custom load combinations can also be created in the Load Combinations tab of the Load Case Manager. Service level load combinations can be added to look at the results for the service case. Strength (LRFD) load combinations should be used in ConcreteSection to check the section capacity using either ACI or CSA strength reduction factors.

Importing Load Combinations

Custom factored load combinations can be imported from the clipboard using the **Import From Clipboard** button in the Load Combinations tab in the Load Case Manager. Text must be tab delimited and copied to the clipboard in the following format:

> {ComboName} {Factor} {ServiceCaseName} {Factor} {ServiceCaseName2} ...

For example:

ComboName	1.2	D	1.6	L	0.5	Lr
MyCombo	0.9	D	1.3	W		

Seismic Criteria

To correctly generate load combinations that contain seismic loads, several additional parameters are required. Please refer to ASCE 7, Section 12.4, for how these parameters are used in generating load combinations. It is important to note that these parameters (such as SDS and SD1) are only used to generate load combinations. For example, ASCE 7 says that Seismic Category A does not require the combined orthogonal direction combinations (e.g. X+30%Y), but Category

D does. ConcreteSection does not automatically generate any loads, just the combination cases.

1.9 Loads

In ConcreteSection, loads are applied to the model in a service load case. The appropriate service case can be selected in the Service Case drop-down list in the **Ribbon | Home** tab. Loads may be easier to see and select by rotating the view to get a 3D perspective of the model.

Applying Loads

To apply loads to the cross-section, click the Apply Loads command in the **Ribbon | Home** tab or *right-click* and select Apply Loads from the context menu. Once the load is applied, an axial force and moments about the X and Y-axes can be entered using the **Project Manager | Modify** tab. A positive axial load causing compression of the section. Note: Only one load can be applied to each Service Case and the applied loads are assumed to act through the centroid of the section. Note: The Stress Results and Moment Unity tabs are not available when no loads are present in the model.

1.10 Analysis

A reinforced concrete section contains two dissimilar materials. Concrete has significant compressive strength and little or no tensile strength, whereas steel has similar strength in both tension and compression. As a result, simple equations are only available for two extreme states: pure compressive capacity and pure tension capacity. Any analysis of behavior between these states will require an iterative solution which satisfies principles of static equilibrium and compatibility of material strains. Regarding strains, it is assumed that plane sections remain plane and the reinforcement is bonded to the concrete so that both concrete and steel have the same strain at their interface (i.e. strain compatibility exists between the steel and the concrete). ConcreteSection solves this iterative process using the methods outlined in the reference at the end of this section.

Constitutive Models

Steel Constitutive Model

An elastic/perfectly plastic stress-strain relationship is used for tension and compression of the steel reinforcement in ConcreteSection.
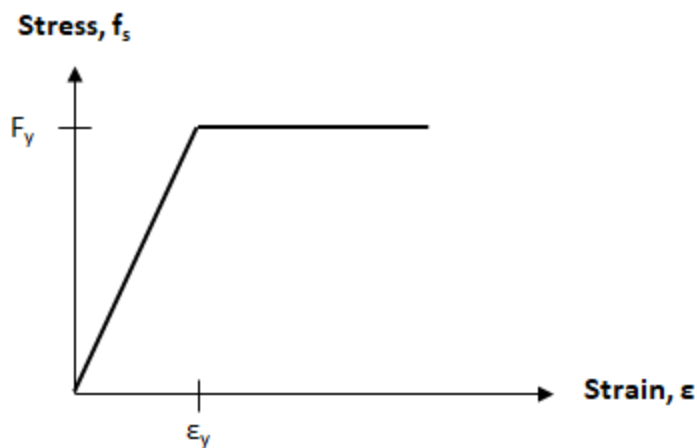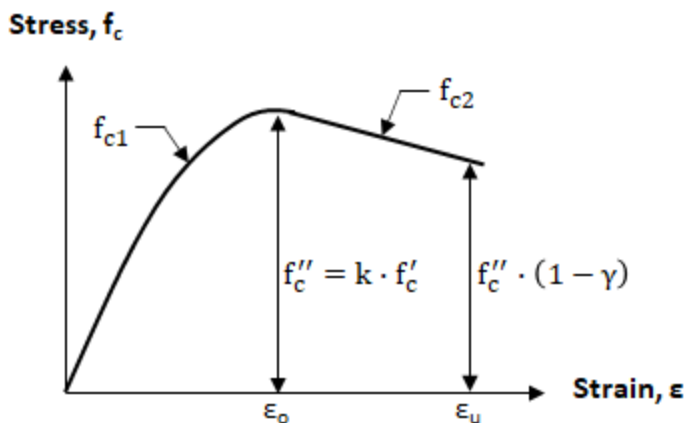

Steel Constitutive Model

Concrete Constitutive Model

In ConcreteSection, the concrete is assumed to have no tensile capacity. In compression, the concrete stress-strain behavior is specified by parameters which include the concrete strain at maximum stress (ε_o), the ultimate concrete strain (ε_u), the maximum stress factor (k), and the confinement factor (γ). The stress vs. strain diagram is shown below. The maximum concrete compressive stress (f"c) is obtained by multiplying the concrete compressive strength (f'c) by k. Note: f"c occurs when the compressive strain is ε_o. The compressive stress at the ultimate concrete strain is obtained by multiplying f"c by (1 - γ). Increasing the confinement factor, γ, reduces the effect of confinement on the section. The values for ε_o, ε_u, k, and γ can be manually adjusted in ConcreteSection. The equations of the two lines shown below are as follows:

$$f_{c1} = f_c'' \cdot \left[\frac{2\varepsilon}{\varepsilon_o} - \left(\frac{\varepsilon}{\varepsilon_o} \right)^2 \right] \quad \text{and} \quad f_{c2} = f_c'' \cdot \left[1 - \gamma \cdot \left(\frac{\varepsilon - \varepsilon_o}{\varepsilon_u - \varepsilon_o} \right) \right]$$

Concrete Constitutive Model

References

1. Rodriguez, J. A. and J. Dario Aristizabal-Ochoa. "Biaxial Interaction Diagrams for Short RC Columns of any Cross-Section" *Journal of Structural Engineering*, Vol. 125, No. 6, June 1999, pp. 672-683

1.11 Stress Results

In ConcreteSection, the Stress Results for each result case are determined using equilibrium, strain compatibility, and the constitutive models for both concrete and steel as discussed in the Analysis section of the help file. Note: The Stress Results view is inactive when no loads are applied to the reinforced concrete section.
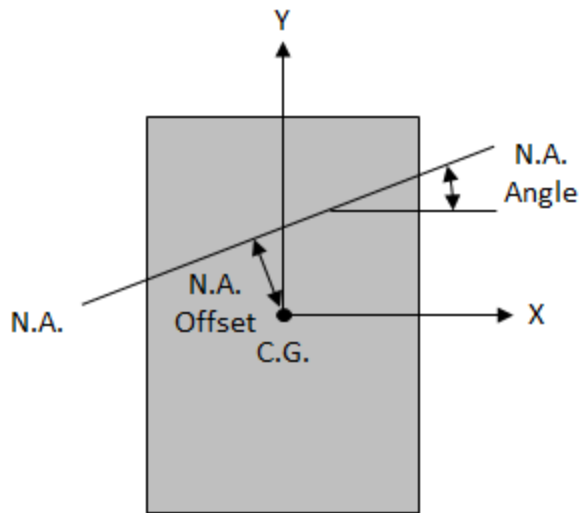
Graphical Results

The Stress Results view graphically shows the results on the cross-section for the result case selected in the **Ribbon | Home** tab. This view shows the cracked neutral axis of the section and the compressive stress or strain in the concrete. Additionally, the force, stress, or strain can be viewed for each bar of reinforcement in the cross-section. Use the **Project Manager | Result Filter** to set which results are displayed graphically for the concrete and reinforcement. Color contour legends are available so that the range of values displayed can be controlled.

Numerical Results

Case Summary

The **Project Manager | Results** tab displays several sets of results in the Case Summary including: the applied axial load and bending moment for the result case; concrete ultimate stress and strains; reinforcement force, stress and strain ranges; cracked section properties; interaction diagram extreme points; and uncracked-transformed section properties. The uncracked geometric properties, such as A, I_x, I_y, S_x, S_y, are calculated using an uncracked section with the steel transformed to an equivalent concrete area. This is done using the modular ratio. The cracked neutral axis offset and angle of rotation are also reported and these values are measured with respect to the centroid and the x-axis respectively, as shown in the figure below.

Cracked Section Neutral Axis

Concrete and Reinforcement

Select the concrete or reinforcement bar(s) to view the numerical results in the **Project Manager | Results** tab for the selected Result Case.

1.12 Interaction Diagrams

In ConcreteSection, the interaction surface is determined using equilibrium, strain compatibility, and the constitutive models for both concrete and steel as discussed in the Analysis section of the help file. The more refined approach that ConcreteSection uses to calculate capacities will produce results that vary slightly from approximate methods that use a rectangular Whitney stress block.

Interaction Surface & Diagram Views

Five different tabs are used to view the interaction surface and diagrams in ConcreteSection:

- Moment Unity - Bending interaction at an applied axial load (Pu)
- MX - Uniaxial bending about the X-axis
- MY - Uniaxial bending about the Y-axis
- Flexure - Bending interaction at zero axial load (pure flexure)
- Surface - A 3D interaction surface for the section plotting axial force, MX, and MY

These diagrams show the capacity of the reinforced concrete section including the Strength Reduction Factors (ϕ) when applied. When factored loading exists, the loads can be viewed as points on the diagrams. The MX and MY diagrams show vertical cuts through the interaction surface, whereas the Moment Unity and Flexure diagrams show horizontal cuts through the interaction surface.

Interaction Surface Development

For a given value of axial force P and bending moment resultant (MX, MY) a single point is plotted on an interaction surface. For a given ratio of MX/MY ConcreteSection calculates a series of points which are viewed as a single slice though the interaction surface (shown in the image below). The number of interaction points used in the slice can be set in the **Project Manager**. The number of slices in each quadrant can also be set in the **Project Manager** (all 4 quadrants of the surface will have the same number of slices). The definition of the interaction surface increases as the number of interaction points and quadrant slices are increased. The points on the surface of the non-factored interaction surface are calculated assuming the maximum concrete strain is at ε_o. Note: The interaction surface does not take into account the effects of slenderness for concrete columns.
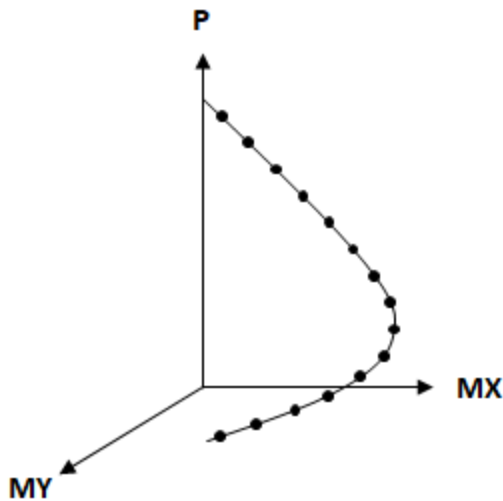
Slice through Interaction Surface

Strength Reduction Factors (ϕ)

In Concrete Section, the Strength Reduction parameter and the transverse reinforcement type can be set in the **Project Manager | Modify** tab. Setting the Strength Reduction parameter to **ACI 318-19** or **CSA A23.3:19** applies both the strength reduction factors and the maximum axial compressive strength limit to the interaction surface per the chosen design specifications. The image below shows how an interaction diagram (i.e. a cut through the surface) is reduced when strength reduction factors are used in the program. For design, when the applied factored loads lie inside the factored interaction surface (P_{u1}, M_{u1}), the capacity exceeds the demand whereas when the applied factored loads lie outside the factored interaction surface (P_{u2}, M_{u2}) the demand exceeds the capacity. For convenience, ConcreteSection provides Moment Unity Values that identify where the factored loads fall in relation to the interaction surface.

ConcreteSection 3.0 User's Guide
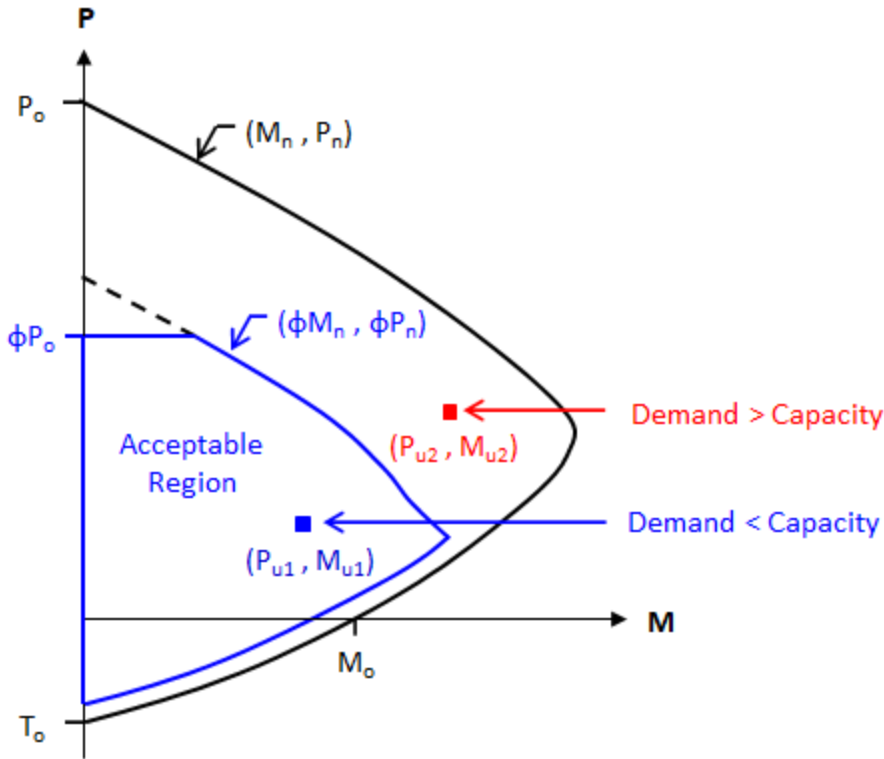
Interaction Diagram for Design

## Moment Unity

Moment Unity View

The moment unity view shows the flexural capacity of the section at the factored axial load for a particular Result Case. This is simply a horizontal cut through the interaction surface located at the factored axial force. If the axial force exceeds the axial load only capacity (ϕP_O) shown in the image above, this surface cannot be displayed. Also, the Moment Unity view is not active when there are no loads applied to the reinforced concrete section.

Moment Unity Value

The moment unity view can be used to visualize how the Moment Unity Values are calculated. ConcreteBending calculates a design level Moment Unity value for each result case as:

$$\text{Moment Unity} = \frac{\text{Moment Demand}}{\text{Moment Capcity}}$$

where the Moment Demand is the distance from the center of the diagram to the load point and the Moment Capacity is the distance from the center to the edge of the unity diagram along the line used to calculate the Moment Demand. These values are shown in the figure below. Note: When Strength Reduction Factors are not used, the Moment Unity calculation is based on the non-factored interaction surface.
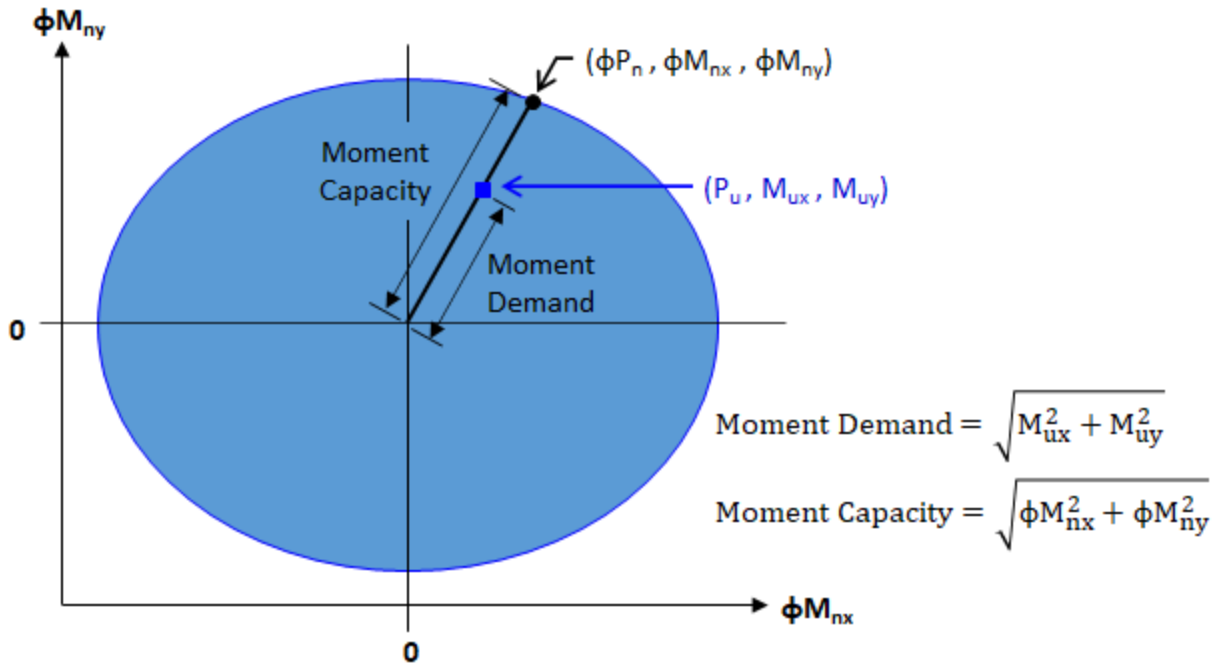
Moment Interaction Diagram at Axial Load, P_u

$$Moment\ Demand = \sqrt{M_{ux}^2 + M_{uy}^2}$$

$$Moment\ Capacity = \sqrt{\phi M_{nx}^2 + \phi M_{ny}^2}$$

1.13 Reports

Reports in ConcreteSection are designed to present information in a clear, concise, and organized fashion. Reports can include both text-based and graphical information that can be printed to paper, to .pdf, or saved in a number of different file formats. Use the Report All button in the ribbon to create a new report with the most relevant project information and graphic images. Note: A description of several of the reported values used throughout ConcreteSection can be found on the Property Definitions page.

Tables

Tables are used to report information in a clear and concise manner. The tables available for the report are listed in the **Project Manager | Tables** tab when the Report View is active. Tables fall into one of four categories (Project, Structure, Load, and Result) and will automatically appear or disappear depending on the items in the model and the available analysis results. Hover the mouse over a table in the list to view its description.

Table Types

- **Project Tables** are used to document the project wide information for the model including the Project Settings, Service Load Cases, and Factored Load Combinations.
- **Structure Tables** are used to document the input data for various model objects including Boundaries, Reinforcement, and Vertices. Also, a Model Summary can be reported.
- **Load Tables** are used to document every load applied to the model in each service load case.
- **Result Tables** are used to document the analysis results for the cross-section including Uncracked Section Properties, Cracked Section Properties, Reinforcement Results, Moment Unity Checks, and more.

Adding & Removing Tables

To add a table to the report, simply *drag* the table from the **Project Manager | Tables** tab to the desired location in the

report or **double-click** on the table to insert it at the end of the report. A list of the report's Included Tables is shown in the **Project Manager | Tables** tab which can be rearranged by **dragging** them with the mouse. To remove a table from the report, click the X next to the table in the Included Tables list or **right-click** on the table in the report and select Remove.

Modifying Tables

Tables can be modified using the Report Settings or Model Filters in the **Project Manager | Report Filter** tab. The Report Settings are used to specify which Service Cases and Result Cases to include in the report while the Model Filters are used to filter the items that are included in the report (such as Boundaries, Point Supports, Load Points, Line Stiffeners, etc.). Tables can also be modified by clicking on the tables in the report. **Click** the column header to sort the column, **drag** the column header to rearrange the columns in the tables, or **drag** the column boarders to adjust the column widths.

Selected Table

Click within a table to select the table and activate the Selected Table section in the **Project Manager | Report Filter** tab. In this tab, the Title can be modified, the columns can be sorted, and the page width can be defined. Choose which columns are included in the table under the Columns section and **drag** the columns in this section to rearrange them in the report.

Selected Table Extremes

Certain tables have the Selected Table Extremes option available in the **Project Manager | Report Filter** tab. The following parameters are used to set how the information is filtered in the selected table.

- **Extreme Rows -** Set to show the Extreme Rows Only for the table or to Show All (which can lead to lengthy reports that may need to be filtered by result cases or reported items to be manageable).
- **Included Rows -** Specify how the extreme rows are considered.
    - **Max and Min -** Keep only the max and min values.
    - **Max -** Keep only the max value.
    - **Min -** Keep only the min value.
    - **Max/Min (when opposite sign)** - Keep the max and min values, if different signs, else keep the most extreme.
    - **Extreme -** Keep only the most extreme value, positive or negative.
- **Applies To -** Specify if the extreme rows be kept on a table wide basis or by each item in the table.
- **Consider Zero as Extreme -** Specify if zero should be considered an extreme value.
- **Show All Extreme Rows** - Choose to show all rows with the extreme value or only show the first occurrence of the extreme value.

Column Justification

Right click on a column in the report to set the Column Justification to Left, Center, or Right for an individual column in a table. In the Reports category of the Preferences, the default Justification for the Text data and Physical data can be specified.

Graphics

Graphics that show the reinforced concrete section, the stresses/strains results, interaction diagrams, etc. can be included in the reports. The graphics available for the report are listed in the **Project Manager | Tables** tab when the Report View is active. Graphical information can also be inserted into a report using the **Copy** and **Paste** commands or printed directly using the **File | Print** command. Graphics are added to and removed from reports in the same manner

as adding and removing tables discussed above.

Graphic Types

When graphics are inserted into the report they will reflect the current view set in their respective window. For example, if the concrete and reinforcement stresses are displayed in the Stress Results window, the stresses will be shown in the report when the Stress-Strain Graphic is inserted. Multiple instances graphic types can be included in a report to show different results from the same window (e.g. to show both stresses and strains). The following types of graphics are available to be used in the report:

- **Bending Interaction Diagram -** The bending interaction at zero axial load (pure flexure).
- **Moment-X Interaction Diagram** - Uniaxial bending about the X-axis.
- **Moment-Y Interaction Diagram** - Uniaxial bending about the Y-axis.
- **R/C Section View** - A graphic view of the reinforced concrete section.
- **Stress-Strain Graphics** - A graphic view of the cracked section for each result case.
- **Surface Interaction Diagram** - A 3D interaction surface for the section plotting axial force, MX, and MY.

Custom Report Logo

The report may be customized to include your own (company) logo in the header. All you need to do is create a logo image: ReportLogo.png or ReportLogo.jpg, and place it in the IES\Customer folder, which you can access via the **Tools | Custom Data** toolbar command. The image should be kept to less than 5 times wider than it is tall. It will be scaled to fit in the header area, but wide images may cause other text to start wrapping or get truncated. If the image works you'll see it in the report/preview immediately after restarting the program. (This logo works for many other IES tools as well.)

1.14 Support Resources

Did you Search this Help File?

Take advantage of the help and support built into the software, as described in the Program Layout section of the User's Guide. This document can be searched, and you should try different potential terms, sometimes less is more when searching (use just the unique word or words). A Table of Contents is also available.

Do Not Contact Support For:

- **Licensing/Sales.** Use www.iesweb.com or sales@iesweb.com.
- **Modeling Advice.** Determining how to model a structure is your responsibility as an engineer.
- **Model Validation.** IES cannot validate your model or your results. If you can document a software defect, contact support and we will investigate further and create fixes as necessary.
- **Engineering Theory**. IES is not in the business of educating engineers. There are textbooks referenced in this help file.

Technical Support

- **Support Email:** support@iesweb.com. Replies are usually within 2 business hours, if you don't hear anything within a business day, assume it got spam filtered or lost and follow-up. For best results, be sure to ask a question, indicate exactly which IES product & version you are using, include as much detail as is practical. If relevant, please attach a project file and/or screenshots.
- **Support Telephone:** Not Available. We have found this to be too inefficient for everybody. With email you can attach a screen shot, a project file, and we can better direct your question to the IES expert for that

product or area. Phone tag takes longer than you think.

- **Business Questions:** For any licensing or sales-related questions or issues contact sales@iesweb.com.