

1 Welcome



ConcreteSection

Cracked analysis and capacity of reinforced shapes.

A New Commercial Version is Available.

IES has upgraded ConcreteSection 2.0. The latest release can be found on our website at: www.iesweb.com/downloads

ConcreteSection will help you design complex reinforced concrete sections. ConcreteSection will take 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

- [How-To Tutorial Videos](#)
- Use File | Open Example to see sample projects.
- [Feature List](#)
- [Program Layout](#)
- [Release History](#)
- [FAQ Answers](#) at iesweb.com for business, licensing, installation issues.
- Downloadable [PDF Help](#) file.

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.

ConcreteSection 2.0 User's Guide

519 E. Babcock St.
Bozeman, MT 59718

Sales or Licensing: 406-586-8988, sales@iesweb.com

Technical Support: support@iesweb.com

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 may be any shape, including hot-rolled steel
- Reinforcement arrays for easy placement and editing

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 (ϕ) per ACI and CSA Specifications
- Limited maximum nominal compressive strength, $\phi P_n, \max$

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

ConcreteSection 2.0 User's Guide

- 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](#).

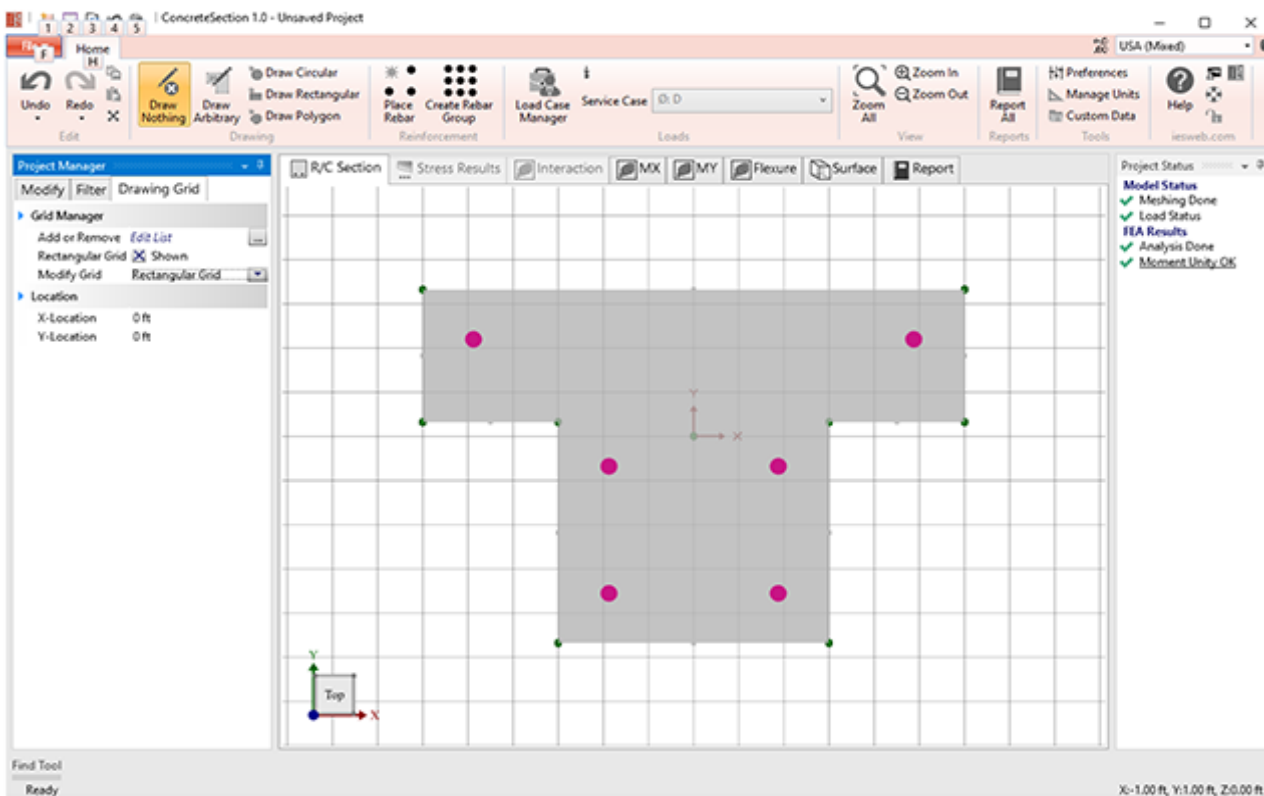
3 Program Layout

Explore

The best way to learn ConcreteSection is to poke around and try things yourself. Get to know what is available under each button or menu. Also try the [tutorial videos](#).

Screen Layout

The image below introduces the program terminology used in the help or 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 "push pin" icon to collapse panels temporarily to gain more space for working. Hold your mouse over the screen image below for information about each area of the program.



Main Menu / Toolbar

Each command on the menu or toolbar is accessed with a click. They have helpful descriptions or tool-tips available by hovering your mouse over the command.

Project Manager

- Modify Tab: Use this tab to change properties of selected objects.
- Filter Tab(s): Use this tab to control what is shown or hidden in the active view.
- Drawing Grid: Define and show or hide drawing grids. These help you place or sketch objects in the Model and Load view.

Graphic Views

The graphic views provide a way to view the model, analysis results, interaction diagrams, and text reports. Each tab

ConcreteSection 2.0 User's Guide

displays different options and will provide different information in the Project Manager and Find Tool. Some Graphic tabs will only be available 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, a report, or a dialog containing quick actions.

Status Bar

Shows background meshing/analysis/design progress. Background processing is done on a separate thread of your processor so you may continue working while they run. 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 analysis or design checks that are currently in-progress.

Find Tool

The Find Tool window provides an alternate way to view properties or results and to select items that may be difficult to select graphically. You can sort the lists by clicking on header names. If you do not want or need this tool you may collapse it using the "push pin" icon in the upper right corner.

Units & Precision

ConcreteSection can display physical quantities in a variety of unit-systems, including your own custom setup. Select the unit system to use for all displayed values, and adjust the precision (number of decimal places) using the controls in the upper right corner.

Data Entry: Physical Quantities

You may enter values in any unit: enter any number or math expression 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)
- ***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)
- ***Click*** in the 'whitespace' of a view to unselect everything and access Project Settings.

Zoom:

- Use the ***mouse wheel***, position the mouse over the point you wish to zoom in to or out from.
- ***Ctrl+*** (***plus***) and ***Ctrl-*** (***minus***) keys.
- ***Ctrl+Home*** for zoom all/ extents

Pan:

- Hold the ***mouse wheel*** down (like a button, not scrolling it) and ***drag*** the mouse.
- ***Shift+Arrow*** keys will also pan.

Rotate:

- Hold the **mouse wheel** down (like a button, not scrolling it) with the **Ctrl** key and **drag** the mouse.
- **The Cube** in the lower-left corner of the graphics: **Click** on a face, edge or corner to orient 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.

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.

4 Release History

- Version 2.0 released October 2018
- Version 1.0 released December 2017

Version 2.0 Features

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 (ϕ) 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

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 (ϕ)
- 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

5 Preferences

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 History Files

We create copies of a project-file when it is saved. These are stored in the *C:\Users\your.login\AppData\Local\IES\Customer\HistoryProjects* folder. If you have data corruption or lost a project, you might be able to recover from one of these. The number of days these files are kept is set in the Project section of the preferences. You can turn this feature off (to save disk space or use your own backup mechanisms) by setting the value to zero.

Crash Protection

In addition to history files, for crash protection we will save a copy of your project every few minutes while you are working. The number of minutes used for this auto-save is also shown in the Project section of the preferences. In the event of a program crash the next time the program is started a message indicating a recovery project is available and you have the option of opening this saved file. You can turn this feature off by setting the value to zero.

Project, Next Inspector Field on Enter

By default, in the Project Manager, Modify area, when you press the Enter key it sets your input data but does not change focus to the next entry field. The Tab key lets you move from one field to another. You can enable the movement to the next field on the Enter key with this option.

Reports, Customer Logo

You can use your own company logo in text reports (Report Viewer). Ideally you'll use a .bmp, .jpeg, or .png that is a reasonable shape and size. Most printers print at 300 dpi, so a 2" logo might be 600 pixels square. We will scale your image to fit the header area of the report. You can use a logo anywhere on your machine, but if you drop one in the *IES\Customer\ReportLogo.jpg* location, we'll find it automatically.

6 Load Combination Criteria

Load Combination Sets

Strength (LRFD) load combinations should be used in ConcreteSection if you are using it to check the section capacity using either ACI or CSA strength reduction factors. Several sets of Building Code Combinations are built into the program to assist you with creating the applicable combinations.

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

In order 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.

7 Loading

To perform a stress analysis of the concrete section, loads must first be applied using the *Apply Load* command available from the **Ribbon | Home** tab or with the *right-click* context menu. Only one load can be applied to each Service Case. 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, with a positive axial load causing compression of the section. The applied loads are assumed to act at the centroid of the section.

Once loads have been applied to the section, use the **Load Case Manager** to generate Load Combinations. Analysis results will be available from the **Stress Results** view and through the [Text Reports](#).

Note that loads are not required to be applied to the cross-section if you only want to view the interaction diagrams.

8 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. Because of this, 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 deformed to the degree that it "grips" the concrete so that both concrete and steel strain the same amount at the interface between the two (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

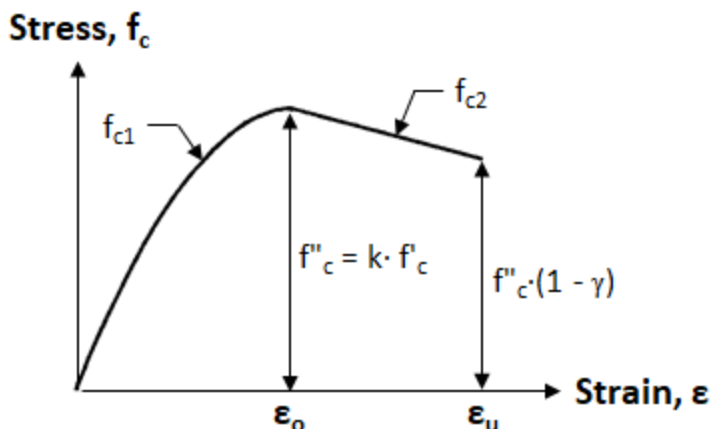
Steel Constitutive Model

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

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 $(1 - \gamma)$. 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:

$$f_{c1} := f''_c \cdot \left[\frac{2\epsilon}{\epsilon_o} - \left(\frac{\epsilon}{\epsilon_o} \right)^2 \right] \text{ and } f_{c2} := f''_c \cdot \left[1 - \gamma \cdot \left(\frac{\epsilon - \epsilon_o}{\epsilon_u - \epsilon_o} \right) \right]$$



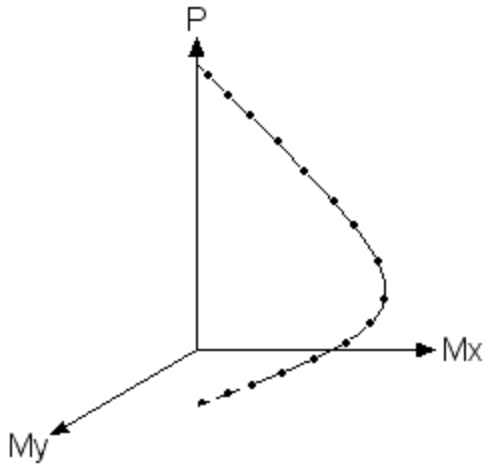
Interaction Diagrams

In ConcreteSection, the interaction diagram is determined using equilibrium, strain compatibility, and the concrete and steel constitutive models. Therefore, the more refined approach that ConcreteSection uses to calculate capacities might produce results that vary slightly from the approximate methods that use the Whitney rectangular stress distribution.

ConcreteSection 2.0 User's Guide

The interaction diagram parameters control how many results are calculated. For a given value of axial force P and bending moment resultant (M_x , M_y) a single point is plotted on a theoretical interaction surface. For a given ratio of M_x/M_y we can calculate a series of points which are viewed as a slice through the interaction surface. You control how many points are used in the slice using the **Project Manager**.

Shown below is a single slice. The number of slices are also controlled using the **Project Manager**. All 4 quadrants of the surface will have the same number of slices. With an increased number of points and slices comes a better definition of the interaction surface is created.



*Note: A point on the interaction diagram is calculated assuming a "failure" when the concrete strain reaches ϵ_0 . The interaction diagrams do not take into account **length effects**.*

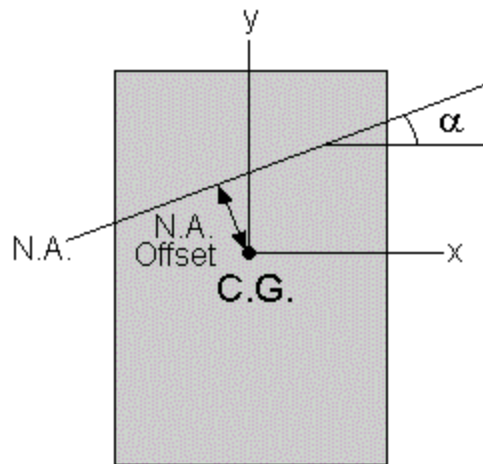
Stress Results View

The **Stress Results** view graphically shows results for a given result combination case. Shown is the cracked neutral axis of the section, the stress or strain in the concrete on the compression side. For reinforcing, the resultant force, average stress, maximum stress or stress variation can be displayed for each bar.

The **Project Manager | Results** tab displays several sets of results including: the applied axial load and bending moment for the result case; concrete ultimate stress and strains; reinforcement force and stress ranges; cracked section properties; uncracked-transformed section properties; and interaction diagram extreme points.

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.

For 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 diagram below.



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

9 Results

ConcreteSection provides both graphical and text result information.

Interaction Diagrams

Four interaction diagrams, which are independent of any applied loading are available:

- MX - Uniaxial bending about the X-axis
- MY - Uniaxial bending about the Y-axis
- Flexure - Bending interaction at zero axial load (pure flexure)
- Moment Unity - The moment unity value at each P_u value. See the [Design](#) page for additional information.
- Surface - A 3D interaction surface for the section plotting axial force, MX and MY

These diagrams show force and moment strength values times the capacity reduction factor (ϕ) when applied. When factored loading exists, the loads will be placed as points on the diagrams. Any loading lying inside the diagrams can be viewed as safe whereas outside indicates the capacity of the section will be exceeded and therefore unsafe.

Stress Results

The stress results view will provide a graphical display of concrete stresses and strains. In addition steel reinforcement result forces, average stresses, and extreme stresses may be displayed. Color contour legends are available so that the range of values displayed can be controlled. Each factored loading combination will have its own set of results and is controlled using the *Result Case* drop down from the **Ribbon | Home** tab.

10 Design

Strength Reduction Factors (ϕ)

ConcreteSection has the ability to apply strength reduction factors to the interaction surface and limit the maximum axial compressive strength ($\phi P_{n,max}$) per the **ACI 318** and **CSA A23.3** specifications. This allows the program to calculate a Moment Unity Value as described below. The parameters for Strength Reduction and the type of transverse reinforcement provided can be set from the **Project Manager | Modify** tab with nothing selected.

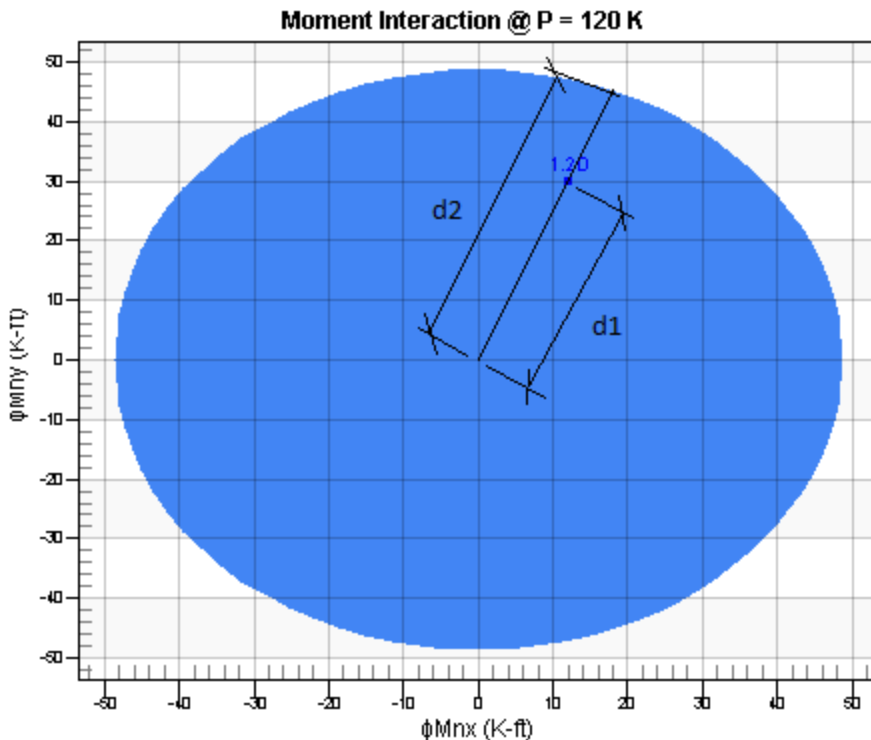
Moment Unity at the Applied Axial Load

The moment unity view shows the flexural capacity of the section at the factored axial load for a particular Result Case. This can be thought of as a horizontal slice cut through the interaction surface where the slice is located at the factored axial force. If the axial force exceeds the axial load only capacity (ϕP_{no}), this surface can not be displayed.

A design level "unity" value can be calculated using this view. The unity is calculated as

- Unity = $d1/d2$

where $d1$ is the distance from the center of the diagram to the load point and $d2$ is the distance from the center to the edge of the unity diagram. These values are shown below:



11 Reporting

Reports are easily created to document your work. Switch to the **Report** tab and drag available tables from the panel on the left onto the report. Use the **Project Manager | Modify** tab to rearrange the tables. Click within a table to adjust table specific settings.

Reports may be filtered by selection, or by names and load cases or combinations. There is a Report All command that will construct a large report, including graphics. Once this report is generated you may adjust or remove items before printing or saving. Export the entire report or individual tables (select, right-click) to a spreadsheet or any text-based program. You may also copy and paste any graphic from within ConcreteSection (or any product) to include in your report.

Tables are included that will allow you to view *Uncracked Sections Properties*, *Cracked Section Results*, *Reinforcement Results*, *Moment Unity Checks*, and more. See the **Report** view for a complete list of the Available Tables. A description of several of the reported values used throughout ConcreteSection can be found on the [Property Definitions](#) page.

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.)

12 Property Definitions

Concrete Shape Terminology

Fy	The yield stress of the reinforcing bars.
f'c	The concrete compressive strength.
f''c	The maximum concrete compressive stress.
Epsu	The ultimate concrete strain.
Eps0	The concrete strain at the maximum compressive stress.
Eps	Extreme compression fiber strain under the loading specified.
Maximum Stress Factor, k	Mutiplier for the compressive strength, f'c, to obtain the maximum compressive stress, f''c.
Confinement Factor, γ	The gamma confinement factor used to calculate the compressive stress at the ultimate concrete strain.

Uncracked Concrete Properties

Modular Ratio, n	Defined as the ratio of the modulus of elasticity of steel (Es) to the modulus of elasticity of concrete (Ec). Normal weight concrete is assumed in the calculation of Ec and is given by the following formula:
------------------	--

$$E_c = 5700\sqrt{f'c}$$

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-	Section modulus values about the centroidal axis (x or y) with respect to the extreme fiber in the positive or negative direction.
fcr	The Modulus of Rupture

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

Cracking Moment <i>Mcr Right</i> <i>Mcr Left</i> <i>Mcr Bottom</i> <i>Mcr Top</i>	The cracking moment for bending about the x and y axes, with fr being the Modulus of Rupture. The Mcr Right value corresponds to having the right side of the section in tension, etc.
---	--

$$M_{cr} = f_{cr} \cdot S$$

* applicable properties are transformed to concrete

Concrete Analysis Results

IcrX, IcrY	Cracked section moments of inertia.
------------	-------------------------------------

ConcreteSection 2.0 User's Guide

N.A. Offset	Perpendicular distance from the neutral axis to the centroid under the loading specified.
N.A. Alpha	Neutral axis inclination angle measure ccw from the X-axis under the loading specified.
Bar Stresses <i>Bar1, Bar2, ... BarN</i>	Tension stress in each reinforcing bar under a given loading. Plotted graphically. Bar numbers correspond to the numbers displayed in the Results View.

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.

* *These are extreme points on the Interaction Diagrams and are modified by ϕ when specified*

13 Support Resources

Did you Search this Help File?

Be sure you make use of the help and support built into the software, as described in the [Program Layout](#) section of the User's Guide. This document may be searched, and you should try various search terms, sometimes less is more when searching -- use just the unique word or words. There is also a logical Table of Contents available.

Do Not Contact Support For

- **Licensing or Sales:** use www.iesweb.com/service or sales@iesweb.com.
- Questions about **how to model** a particular structure. Such questions are your responsibility as an engineer.
- IES cannot **validate your model** or your results. If they "seem" incorrect, please figure out WHY they are incorrect. If you can document a defect, we will be happy to investigate deeper and fix things as necessary, but we cannot afford to check every customer's model.
- Questions about **engineering theory**. IES is not in the business of educating engineers. There are textbooks referenced in this help file and we can provide more guidance as to where to look if you cannot find one.

Technical Support

- **Email Support:** support@iesweb.com (Replies are usually **within 2 business hours**, if you don't hear anything within a 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 or relevant, including attaching a project file. *"I have a problem, can you help?"* is a frequently submitted question, to which the answer is always: *"Maybe, what is the problem?"*).
- **Telephone Support:** No, sorry. 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: sales@iesweb.com.
- **Free Training Videos:** Training videos can be accessed [here](#).

14 Training Videos

Video List

You will find the following videos below, showcasing several features available in ConcreteSection.

- [Introduction](#)
- [Program Layout](#)
- [Working with Reinforcing](#)
- [Working with Loads](#)
- [Stress Results](#)
- [Design Capacity Checking](#)
- [Reporting](#)

Introduction ([BACK TO TOP](#))

ConcreteSection is a specialized tool, developed to assist you in analyzing and designing reinforced concrete sections. The easy-to-use interface allows you to model virtually any shape, and any reinforcement configuration. ConcreteSection then calculates the 3D interaction diagram, concrete stresses and strains, and reinforcement stresses. Loads, if applied, are plotted on the interaction diagram, and unity checks are made to determine if the section can resist the applied loads.

Program Layout ([BACK TO TOP](#))

The layout of ConcreteSection is designed to maximize productivity, making it easy to create reinforced concrete sections, apply loads, view stress results and interaction diagrams, and create reports.

Working with Reinforcing ([BACK TO TOP](#))

ConcreteSection has built in tools to generate reinforcing bar patterns or allows you to manually place the reinforcement. The size and location of the bars can always be modified as needed in your project.

Working with Loads ([BACK TO TOP](#))

In ConcreteSection, service level axial loads and biaxial bending moments can be applied to the centroid of the cross-section. Select from Standard Load Combinations or create your own Custom Load Combinations to compare the results of the factored load combinations with the interaction diagrams.

Stress Results ([BACK TO TOP](#))

ConcreteSection calculates the concrete stresses and strains in addition to the stresses on the reinforcement. The stress results on the cross-section can be viewed for any factored load combination.

Design Capacity Checking ([BACK TO TOP](#))

ConcreteSection computes the 3D interaction surface for the reinforced cross-section displays several planar cuts of the surface (interaction diagrams). The diagrams can be adjusted according to the ACI 318 or the CSA A23.3 design specifications. The results from the factored load combinations are plotted on the diagrams and the unity factors are calculated.

Reporting ([BACK TO TOP](#))

Reports are easily created and modified in ConcreteSection. Either create a complete project report or create custom report by adding and removing the tables and graphics as needed.