StructurePoint is a software company that provides concrete design solutions. Formerly the engineering software group of the Portland Cement Association (PCA), StructurePoint (SP) is located in Chicago and does business all around the world with clients in North America, the Middle and Far East. SP has representatives in India, Thailand, Saudi Arabia, Lebanon and the UAE. Formerly PCA products, the SP product line include design and analysis software for reinforced concrete beams, columns, mats, walls, slab systems, and frame analysis. These six programs make up the SP Suite. The software programs can be purchased as the Suite or individually to meet your specific needs over a large business computer network or as single standalone serving one laptop.
The SP Suite has the capability to design an entire concrete structure from foundation to roof. These programs are based on the methods, equations, and procedures found in ACI 318 and CSA 23.3 in English and Metric units. Due to the schedule of updating the concrete codes, the five code driven software are given a major upgrade every three years along with annual updates. The SP suite is designed to allow the user to work quickly, simply and accurately. In essence, you can get to a final design solution fast with confidence and little training and wasted time.
Use of the StructurePoint software can be found in many publications regarding reinforced concrete design and analysis.
spColumn has been the flagship of the SP Software Suite. It is a reputable program used for the design and investigation of rectangular, round, and irregular concrete columns. In addition to allowing any section geometry, spColumn is capable of analyzing any reinforcing arrangement. After taking the section material properties, geometry, and design specifications the program produces the corresponding interaction diagrams and failure surfaces along with a number of useful design results.
spColumn’s many design options allow it to produce accurate results for any situation. Whether working within the US or Canadian code the program can handle both English and metric units. spColumn does not limit the user to just the most recent code release either. This feature allows the user to move through time and determine if a column designed to ACI 318-05 was adequate and then with the click of the mouse determine if it is compliant with the 2008 code requirements. Beyond choosing the code edition, the program allows for both the analysis and the design of columns subjected to axial loading, uniaxial and biaxial bending as well as the consideration of slenderness. All of these program options add up to one thing; spColumn is universally applicable and regardless of the project spColumn will be a very useful tool.
One of the most useful capabilities of this program is its ability to perform slenderness column/wall analysis and moment magnification. The moment magnification can account for either sway or nonsway frames and will perform a series of calculations and checks to ensure that the column is properly designed. This figure demonstrates the methodology used to for consideration of slender columns within ACI 318-08/11.
This complexity of slenderness boils down to four inputs by the user: The column to be designed, the surrounding columns, the surrounding beams, and slenderness factors. For simplicity, spColumn allows for the user to define their own value for the effective length factor $k$, otherwise spColumn will calculate the exact value of $k$ for you.
The user options continue with allowing the type of section confinement to be specified. spColumn accommodates tied confinement, spiral confinement, and user inputted confinement. These options dictate what strength (phi) factors are used in the section design and analysis.

There are four design criteria that the user can specify. The first is column type. Choosing to design the column as either a structural or architectural column will pre-set the range of reinforcing ratio. On the other hand, indicating a user-defined column type will allow the user to manually adjust the minimum and maximum reinforcing ratios. The bar selection option allows the user to indicate to the program whether the ideal reinforcing is a minimum number of bars or a minimum area of steel. The user can also specify the clear spacing between bars as well as the ratio of provided to required reinforcement.
spColumn will design for concrete and steel material properties in the ranges available to the construction markets. Each property can be input individually, but for your convenience when the concrete or steel strength is entered, the rest of the variables will be automatically calculated. The program also handles precast models designed according to the CSA code.
With the material properties defined, the next step is to create the section dimensions. For the investigation mode this is straightforward. The user can choose between a rectangular section, a circular section, or their own specified irregular section.

For column design, the section input becomes little more involved. spColumn designs a column by starting with the smallest cross section allowable as specified by the user. The program then steps through each possible cross section at user defined increments until a suitable column size and reinforcement is determined. This process is the same for both rectangular and circular columns.
As mentioned, spColumn is capable of analyzing irregular sections as defined by the user. There are two methods to create an irregular section with the first one being a graphical creation tool. This picture is the irregular section editor. The section editor functions by creating a grid or mesh, of which the user can specify the spacing, and drawing lines between grid nodes. One of the foremost reasons for the irregular section editor is to allow for the inclusion of a void or opening within the column. These voids are created in the same way as the rest of the section.
The irregular section editor is not limited to just rectangular sections or the tedious method of approximating circular sections with straight lines. Circular sections that are defined before entering the section editor will be carried into the section editor as exact circles. From here the section can be edited and voids can be created to model the space where a steel I beam is included as shown.
If a section is more complicated than the irregular section editor can handle, the program can model sections by importing a table of section data points. The spColumn software includes a design assistant excel worksheet that creates the geometry of 7 different sections based on the users specifications. These 7 sections are based on the program’s included examples and the default dimensions are set to these examples. If these sections are not exactly the required section, any section can be manually created using this section input method. Microsoft Excel or any other spreadsheet editor capable of performing this function.
After the section is defined, spColumn moves to defining the reinforcement. Oriented to speed and efficiency, spColumn has preset reinforcement arrangements and the capability of analyzing irregular spaced reinforcing.

For the design of reinforcing, the user specifies both the range of acceptable number of bars as well as sizes permitted by the user. spColumn uses the minimum spacing requirements, which were specified earlier, to increase the reinforcing within the smallest allowed section until the maximum amount of steel is placed in the section. The program then moves to the next section size and repeats the process until the right combination of section size and reinforcement is found.
Similar to creating irregular sections, spColumn can define reinforcing using the irregular section editor. This ability allows reinforcing to be assigned using the cursor or, for the more complex reinforcing, by defining the exact coordinates. Assigning reinforcing in this manner is required for irregular sections, but it can also be used for rectangular or circular sections.
Reinforcing can also be assigned using the same method of inputting a data table of reinforcing locations. If the section was defined in the same manner, this may be the easiest way input the reinforcing. The same 7 design assistant worksheets provided for sections will create the reinforcing placement.
spColumn has many load options available to the user. Loads can be entered as factored loads or service loads depending on what the user is provided. The service loads and axial loads option are intended more for investigative purposes.
Factored loads can be either loaded from a text file or manually entered into
the program. Using the factored load option of choice if the applied loads are
already provided by an analysis software like spFrame.
Similar to the factored loads, service loads can be loaded from a text file or inputted manually. The choice between inputting the loads as service or factored loads is entirely based on what values the engineer is provided and his or her preference.
The option for axial load is intended for the engineers use to determine the corresponding allowable moment value for an existing axial force. This option does not show any points within the interaction diagram because this option essentially tells the user the values of the interaction diagram.
With input completed, spColumn produces the interaction diagrams for easy visual identification of a column’s capacity. These interaction diagrams are interactive in that they can be stepped through by either the axial force or the angle depending on which diagram is active. If a specific point number is needed to be seen, the program can list individual point numbers so a specific point’s interaction diagrams can be found among the rest. spColumn will also copy these interaction diagrams to the computer’s clipboards so they can be easily inserted into column design reports.
The program can also provide the nominal interaction diagram to show the column’s capacity before it is factored.
One unique feature of spColumn is its ability to superimpose these interaction diagrams on each other. The superimposing feature might have its most useful application in comparing different section types. The column being analyzed is represented by the solid line while the superimposed interaction diagram is represented by the dotted line. In this case, the original column was found to be underdesigned. The superimposed interaction diagram demonstrates that this alternate design is acceptable for the current loading as well as how much more capacity is achieved than with the original design.
In addition to producing the graphical results, spColumn outputs a text report of the design or analysis process. The report is designed to provide all the necessary information an engineer needs to evaluate the designed column, but at the same time allow the user to easily identify the critical load combinations. In this example Factored Loads number 4, 9, 10, and 20 exceed the section capacity and require the column to be revised.
The batch mode feature of the program is a great automation tool to streamline the column investigation process. Intended for integration with 2D/3D structural analytical models, the batch mode takes text files that include all of the column properties and loading conditions and produces results like interaction diagrams, interaction diagram data, column section dxf files, and all graphical results. The process of setting up the batch files to run is where the time in this process is invested, but with intelligent methods the total required time can be greatly reduced.
The text files that are used by the batch mode are spColumn Text Input (CTI) files. This is an example of what one looks like. This file contains all the information that spColumn needs to produce results. Luckily, there is no need to create these CTI files from scratch. Start by creating a typical column using the spColumn program. When everything has been defined, save the file as a CTI file. This will save the format which can then be used as a template for all future CTI files. Also, the spColumn user manual has a detailed section designed to help users modify and develop their CTI files for batch mode analysis. Many users devised in-house scripts to further leverage this feature and connect with other software tools and BIM programs.
Formerly pcaSlab and ADOSS, spSlab was created for the analysis and design of reinforced concrete beams and slab floor systems. Two-way slab systems are analyzed using the Equivalent Frame Method. Beams and frames of up to 22 spans can be analyzed and designed. In addition to the design option, spSlab has the capability of investigating existing beams and slab systems. spSlab includes provisions for slab band systems as well as punching shear check and deflection calculations using either cracked or gross sections. For beams, moment redistribution as well as combined shear and torsion design are available. In addition to the required area of reinforcing steel at the critical sections, spSlab provides a complete bar schedule that includes the number of bars, bar sizes, and lengths.
Floor systems are intricate with many choices available to an engineer. These choices are reflected in spSlab’s General Information tab. From here the user has the option to choose which code and what edition to follow, the units, if compression reinforcement is to be used, what percentage of live load is sustained for long term deflections, and if transverse or longitudinal beams are included in the support design to name a few.
With the number of supports defined under General Information, spSlab automatically populates the Span Data dialog box with spans between supports and, if specified, cantilevered spans on the ends. The Span Data window allows each span to be given unique section properties. The Span Control window is a convenient way to edit entire spans. It allows for new spans to be inserted and existing spans to be duplicated, moved, or deleted.
The supports dimensions entered into spSlab are used to determine the torsional stiffness of the system and to check the punching shear capacity of the slab system. The boundary conditions are used to simulate the conditions that the slab will experience at the supports. Adjusting these boundary conditions can be quite useful in creating a more realistic slab system than could otherwise be created. For example, since transverse beams can only be specified at the support locations this may cause a problem if there are transverse beams between columns that should be providing torsional stiffness to the system. The problem with this situation is that even if columns are given zero dimensions they still prevent vertical deflection. To bypass this problem, the supports can be given spring stiffnesses of negligible magnitudes. This will allow the system to act as if there is no column while including the torsional resistance of the transverse beams.
The choices for reinforcing change depending on the mode of spSlab being used. For the design mode the program takes user inputs for clear cover, the range of bar sizes, the minimum and maximum spacing between bars, as well as the minimum and maximum reinforcing ratio.

For the investigation mode, the bar size, number of bars, and their length are inputted as well as their location within the slab. For the horizontal position, the user can choose to either make the bars continuous across the bottom or top of the span, specific to just the left or right top of the span, or discontinuous through the bottom.

When performing the calculation of limits of reinforcing spacing spSlab accounts for the bend in the transverse reinforcement. Assuming that the bar makes contact at the middle of the bend gives a more accurate calculation of reinforcing spacing.
The user has load input choices of area loads, line loads, and point loads. These loads are designated a label to identify them and allow them to be factored. The system can be further modified to include concentrated forces on the supports along with prescribed support displacement and rotation. spSlab accounts for lateral loads as well by modeling them as moments acting on the ends of a span.
With all of the inputs completed spSlab will provide the user with a graphical output of the loads acting on the system, the internal forces, the system’s moment and shear capacity, the deflections, and the required reinforcing.
The moment capacity diagrams include individual results for the beam strip, the column strip, and the middle strip.
The shear capacity diagrams include the results for both the beam and the slab.
The deflection diagram displays the dead load, live load, a total deflections. Due to spSlab's capability to adjust the percent of live load sustained and time duration for long-term deflection calculation, the program allows for quick checks of ACI 9.5 – Control of deflections.
spSlab will also create a text-based Results Report when the file has been solved. The report is split into an Input Echo, Design Results, Column Forces, Internal Forces, and Deflections. Each of these screens can be printed or saved individually or by using the Customize tab some or all of the sections can be combined into a single document.
spBeam is a limited version of spSlab. It includes all the elements of spSlab that apply to beams and one-way slab systems. In this program, beams and slabs up to 22 spans can be analyzed and designed. Moment redistribution as well as torsion and shear design are available for beams. In addition to providing the area of reinforcing steel at the critical sections, spBeam provides a complete bar schedule that includes the number of bars, bar sizes, and lengths.
spBeam’s options are similar to spSlab’s, but notice that it forces the user to design the floor system as a one-way system. Beyond this, the options are identical to spSlab’s one-way stem options.
The moment reduction option is available for all editions of both the CSA and ACI code. By using the equations shown, spBeam will allow for the reduction of negative moments. The results for a one-way system with the moment reduction option enabled displays the calculation procedure including the original moments and the percent factored.
With the torsion option engaged, both shear and torsion contribute to the required transverse reinforcing. The torsional design is based on a thin-walled tube, space truss analogy. Either the equilibrium or compatibility torsion conditions can be applied. The capacity diagram as well as the text report split the required transverse reinforcing into that required for shear, torsion, and total capacity.

The moment redistribution and torsion design options are also available in spSlab.
spMats is used for the analysis and design of concrete foundation mats, combined footings, and slabs on grade. The slab is modeled as an assemblage of rectangular finite elements. The boundary conditions may be the underlying soil, nodal springs, piles, or translational and rotational nodal restraints. The model is analyzed under static loads that may consist of uniform surface and concentric loads. The resulting deflections, soil pressure, and bending moments are outputted. In addition, the program computes the required area of reinforcing steel in the slab and checks the punching shear around columns and piles.

spMats uses the plate-bending theory and the Finite Element Method to model the behavior of the mat or slab. The soil supporting the slab is assumed to behave as a set of one-way compression-only springs. If during the analysis, a loading or the mat shape cause any uplift creating a spring in tension, the spring is automatically removed. The mat is re-analyzed without that or any other tension spring. The program automatically iterates until all tension springs are removed and the foundation stabilizes.
spMats breaks the system up into unique elements and nodes by utilizing a meshing or grid system. The grid can be defined gridline to gridline, or generated by specifying a starting location, the number of grids, and the spacing between each. Once the grid is generated the user can still define new gridlines at nonstandard spacing.
With the elements created, the system requires certain definitions to be assigned to each element. These required definitions are concrete thickness, soil properties, concrete properties, reinforcing properties, and the design parameters concrete cover and minimum reinforcing ratio.
There are three different types of restraints: nodal springs, slaved nodes, and piles.

Nodal springs are useful for situations that involve uneven ground and walls or columns that support the foundation instead of a soil.

Slaved nodes are used to more accurately represent the effect a column or wall has on the concrete mat. Nodes assigned to the same slaved nodes group will share the same displacement or rotation as defined by the user. For example, it can be assumed that a column will experience uniform displacement and rotation across its cross section. So since a column's shadow may lay on top of many nodes these nodes should be slaved together so they accurately represent the movement of the column.

Within the program, piles are represented as springs, and as such spMats allows for piles to be defined as nothing more than a spring constant and a name. If the spring constant isn't known for a pile, the program can calculate the spring constant. The formula for the spring constant combines the axial deformation equation with a factor that accounts for the pile’s skin friction and the total force acting on the pile. These piles can be defined as round, square, rectangular, or H-piles. If the properties of a pile are not known but the spring constant is, the user may bypass the spring constant calculation by entering the pile as a nodal spring.
Loads acting on the mat can be modeled as either concentrated point loads or surface loads. These loads can act as a vertical force or as moments acting along the x and y directions. Each of these loads must be assigned to a load case.

Load cases are then combined in the typical way into load combinations for both service and ultimate conditions.
Each element is a unique part of the system that can be given any of the previously defined system definitions. When assigning these definitions, the teal elements represent a part of the meshing that has been assigned a thickness, but not the currently selected definition, red represents that the current definition is assigned, and white indicates unoccupied space. This is to allow for quick visual checks of the system.

Assigning definitions to an individual element is as easy as left clicking on it and assigning definitions to many elements as dragging the cursor around each. Unassigning is the same method but by using the right mouse button.
Before solving the system there are six important inputs required of the user. The maximum number of iterations along with the maximum displacement limit are the controlling factors for most typical models. The computation of reinforcement is based on either the maximum moment or the average moment within the element. This flexibility allows the engineer the ability to use industry standards.
The reason that spMats is such an essential tool for mat design is that it takes the user’s specifications and provides contour maps of the moments, displacements, and reinforcing required of the foundation. These contour maps give the engineer instant recognition of how the foundation will behave and the corresponding required reinforcing.

The program designs the steel reinforcing based on the user inputs for design parameters. If the model is unable to produce a reinforcing design that meets the design criteria the model will still run and all the results are available as well as the contour maps. The program flags these failed elements in the required steel reinforcing contour maps.
spWall is used for the design and analysis of reinforced concrete walls, tilt-up walls, and precast architectural and load-bearing panels. Design provisions in accordance with ACI 318 and CSA A23.3 standards. The program allows for any number of openings and stiffeners, variable design parameters, variable thicknesses, and strength properties. The program uses the finite element method to solve the wall systems and has the capability to account for second order effects.

spWall functions very similarly to spMats due to the similar method of solution. Defining properties, assigning properties, and viewing the results are all identical to spMats.
After setting up a meshing system, spWall allows the user to input properties of the wall, properties of wall stiffeners if applicable, supports, and loads. The wall can be designed as either a standard reinforced wall or a precast wall. With both options the user can choose between using a single or double curtain of reinforcing. This reinforcing is allowed to have either uniform or variable cover in relation to the wall faces.
The assignment of these definitions takes the form of graphically assigning the properties in the same way as in spMats.
Using the FEM, spWall determines the forces acting throughout the system. These results are reported both graphically as seen here and numerically in a text file. Similar to spMats, spWall produces contour maps for quick checks of required internal forces, required reinforcing, and deflections. The program also provides the user with values for the wall’s provided shear strength.

The program designs the steel reinforcing based on the user inputs for design parameters. If the model is unable to produce a reinforcing design that meets the design criteria the model will still run and all the results are available as well as the contour maps. The program flags these failed elements in the required steel reinforcing contour maps.
spFrame is a structural analysis tool for two and three dimensional truss or frame systems. Multi-purpose structural modeling and analysis software for three-dimensional buildings and structures with robust, quick yet simple interface. spFrame calculates internal forces, reactions, rotations, and displacements using linear structural analysis.
Frames and trusses can be quickly modeled in spFrame using either the graphical interface to create members between joints or by assigning members by assigning beginning and ending joints.
spFrame models load as member loads in the form of distributed, point load, or temperature loads. The program can also handle joint loads and loads induced by settlement or temperature effects.
The internal forces of a structure can be analyzed by viewing the shear and moment diagrams by load combinations or by viewing the member envelopes. These results can be seen graphically side by side as well as in a tabular format.
StructurePoint would be glad to hear from you and receive your feedback as well as answer any questions regarding the program features, capabilities, price, and licensing options.