

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.

**Work quickly.  
Work simply.  
Work accurately.**

**StructurePoint's Productivity Suite of powerful software tools  
for reinforced concrete analysis & design**

**spwall**  
Finite element analysis & design of reinforced, precast ICF & tilt-up concrete walls

**spbeam**  
Analysis, design & investigation of reinforced concrete beams & one-way slab systems

**spcolumn**  
Design & investigation of rectangular, round & irregularly shaped concrete column sections

**spslab**  
Analysis, design & investigation of reinforced concrete beams & slab systems

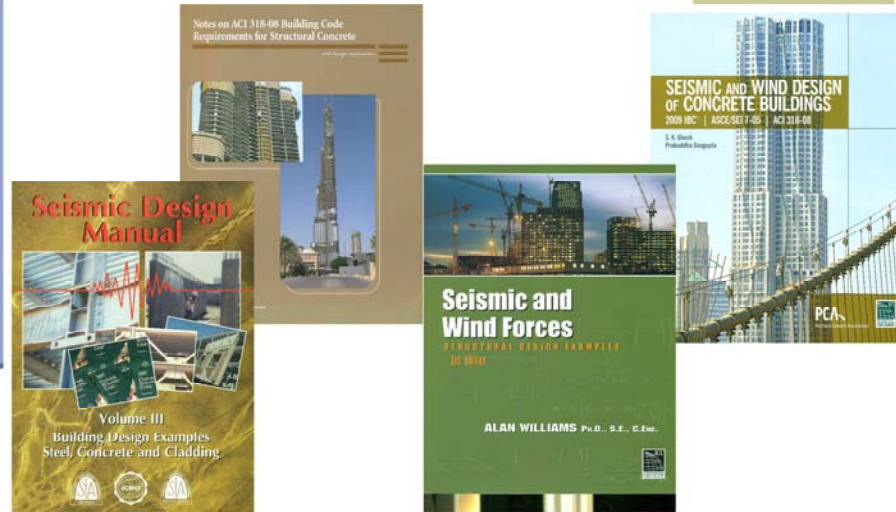
**spmats**  
Finite element analysis & design of reinforced concrete foundations, combined footings or slabs on grade

StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

2

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.

# The Industry Standard

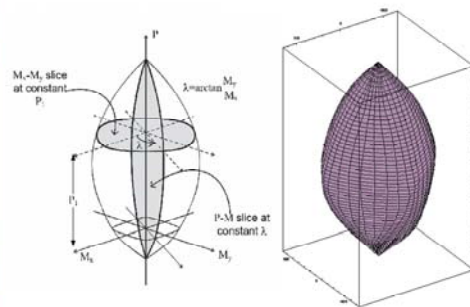
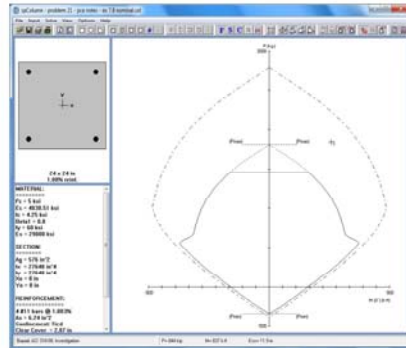


StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

3

Use of the StructurePoint software can be found in many publications regarding reinforced concrete design and analysis.

- Design and investigation of rectangular, round, or irregular concrete sections including slenderness effects



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

# Options



- **Codes:** ACI 318-11/08/05/02  
CSA A23.3-04/94
- **Units:** English or Metric
- **Run Axis:** X, Y or Biaxial
- **Run Options:** Design or Investigation
- **Slenderness:** Yes or No

General Information

Labels:  
Project: \_\_\_\_\_  
Column: \_\_\_\_\_ Engineer: \_\_\_\_\_

Units:  
 English  
 Metric

Run Axis:  
 About X-Axis  
 About Y-Axis  
 Biaxial

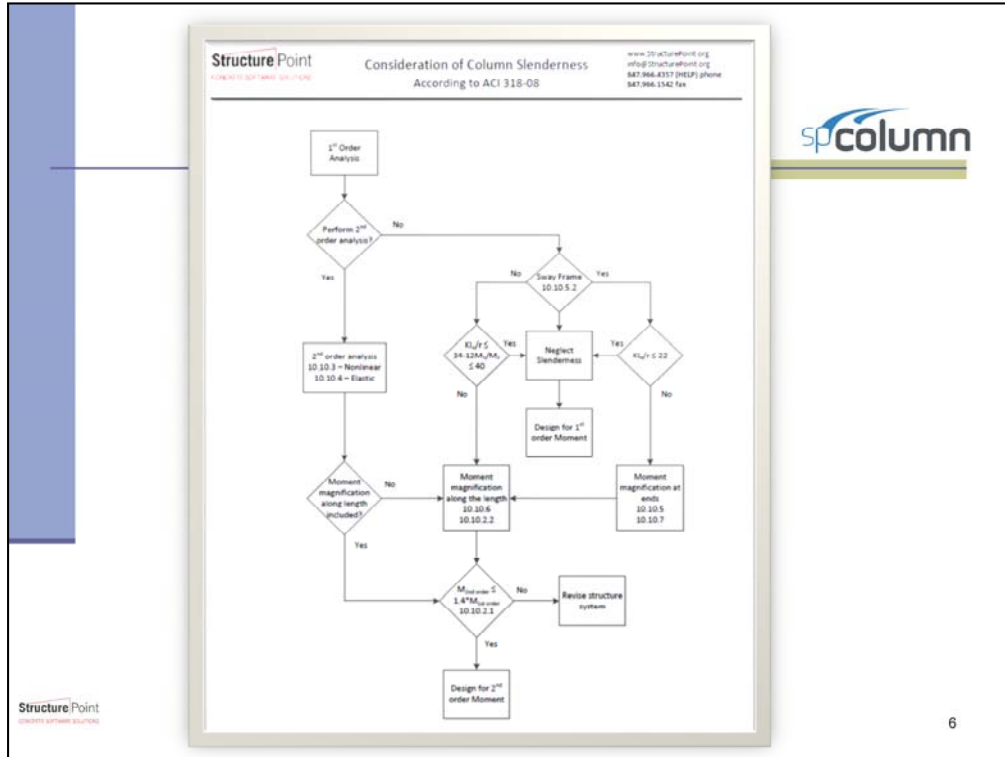
Run Option:  
 Investigation  
 Design

Design Code:  
 ACI 318-11  
 ACI 318-08  
 ACI 318-05  
 ACI 318-02  
 CSA A23.3-04  
 CSA A23.3-94

Consider slenderness?  Yes  No

OK Cancel

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.

# Slenderness



<p> <b>Design Column</b></p> <p>Slenderness related to the column being considered</p>		<p> <b>Beams</b></p> <p>Slenderness related to beams</p>	
<p> <b>Columns Above and Below</b></p> <p>Slenderness related to columns above and below the design column</p>		<p> <b>Factors</b></p> <p>Factors that affect slenderness calculations</p>	

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.

# Reinforcement Options



## ■ Confinement

The Confinement dialog box contains the following fields and options:

- Confinement: Tied (dropdown menu)
- Capacity Reduction Factors: Phi (checkbox)
- Axial compression (a): 0.8 (text input)
- Tension-controlled failure (b): 0.9 (text input)
- Compression-controlled failure (c): 0.65 (text input)
- Tie Sizes:
  - #3 ties with #10 bars or smaller. (dropdown menu)
  - #4 ties with larger bars. (dropdown menu)

## ■ Design Criteria

The Design Criteria dialog box contains the following fields and options:

- Column Type:
  - Structural (radio button, selected)
  - Architectural (radio button)
  - User-defined (radio button)
- Reinforcement Ratio:
  - Minimum: 1 % (text input)
  - Maximum: 8 % (text input)
- Bar Selection:
  - Minimum number of bars (radio button, selected)
  - Minimum area of steel (radio button)
- Minimum clear spacing between bars: 1.5 in (text input)
- Design/Required ratio: 1 (text input)

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.



# Material Properties



## ■ Concrete:

$f'_c$ ,  $E_c$ ,  $f_c$ ,  $\beta_1$ , and  $\epsilon_c$

## ■ Steel:

$f_y$  and  $E_s$





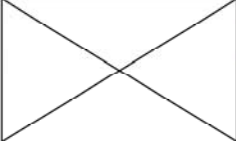
The dialog box titled "Material Properties" contains two columns of input fields. The left column is labeled "Concrete" and includes fields for Strength,  $f_c$ : 5 ksi; Elasticity,  $E_c$ : 4062.0 ksi; Max stress,  $f_c$ : 3.9914 ksi; Beta(1): 0.8838; and Ultimate strain: 0.0035. The right column is labeled "Reinforcing Steel" and includes fields for Strength,  $f_y$ : 60 ksi and Elasticity,  $E_s$ : 29000 ksi. Below these fields is a checked checkbox labeled "Precast" and two buttons: "OK" and "Cancel".

Precast (CSA only)

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.

# Section

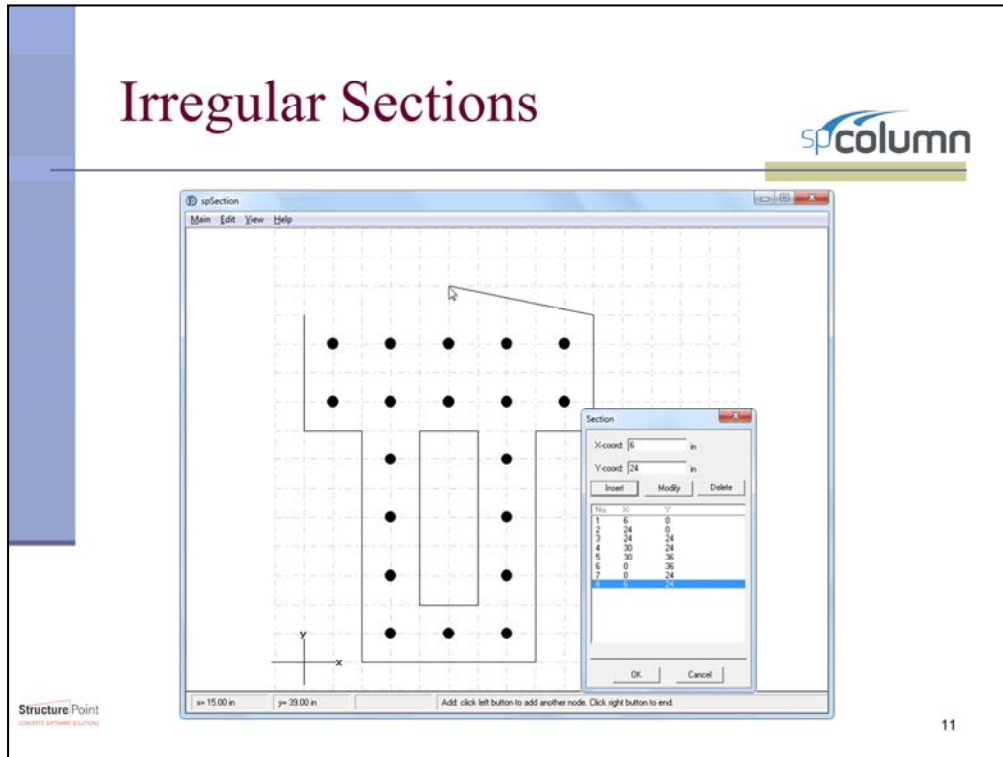


	Design	Investigation
<input type="checkbox"/> <b>Rectangular:</b>		
<input type="radio"/> <b>Circular:</b>		
<input type="checkbox"/> <b>Irregular:</b>		<ul style="list-style-type: none"><li>Irregular Section Editor</li><li>Import Geometry</li></ul>

With the material properties defined, the next step is to create the section dimensions. For the investigation mode this is straight forward. The user can choose between a rectangular section, a circular section, or their own specified irregular section.

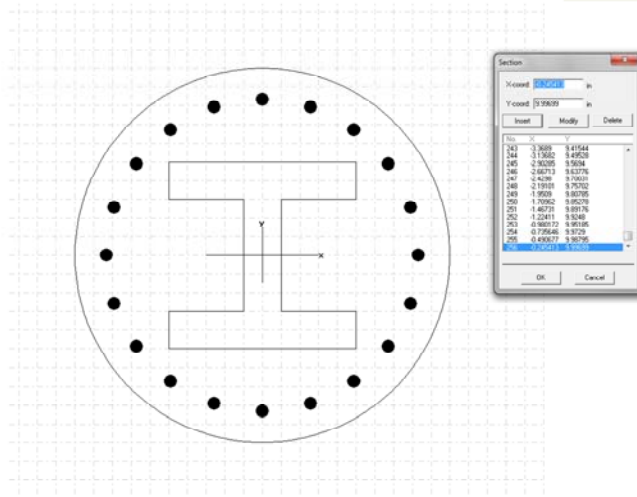
For column design, the section input becomes little more involved. spColumn designs a column by starting with the smallest cross section allowable as specified by the user. The program then steps through each possible cross section at user defined increments until a suitable column size and reinforcement is determined. This process is the same for both rectangular and circular columns.

# Irregular Sections



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.

# Irregular Sections



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.

# Irregular Sections



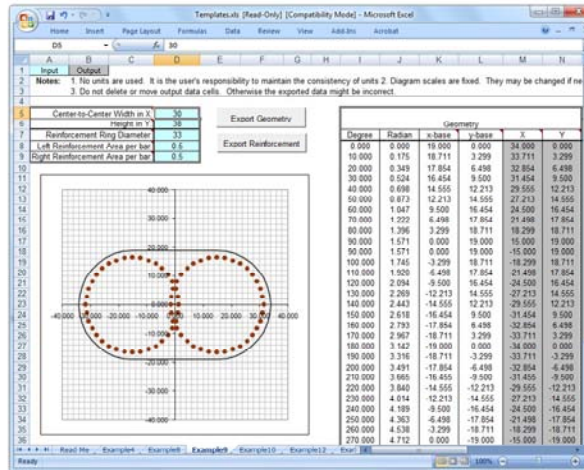
Import Data (from text file)

Geometry  
 Reinforcement  
 Service loads  
 Factored loads

OK Cancel

**Geometry Data:**

No\_Of\_Section\_Nodes  
 Xs1 Ys1  
 Xs2 Ys2  
 .  
 .  
 Xsn Ysn  
 No\_Of\_Opening\_Nodes  
 Xo1 Yo1  
 Xo2 Yo2  
 .  
 .  
 Xon Yon



StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

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.

# Reinforcement



	Design	Investigation
<b>All Sides Equal:</b>		
<b>Equal Spacing:</b>		
<b>Sides Different:</b>		
<b>Irregular:</b>		

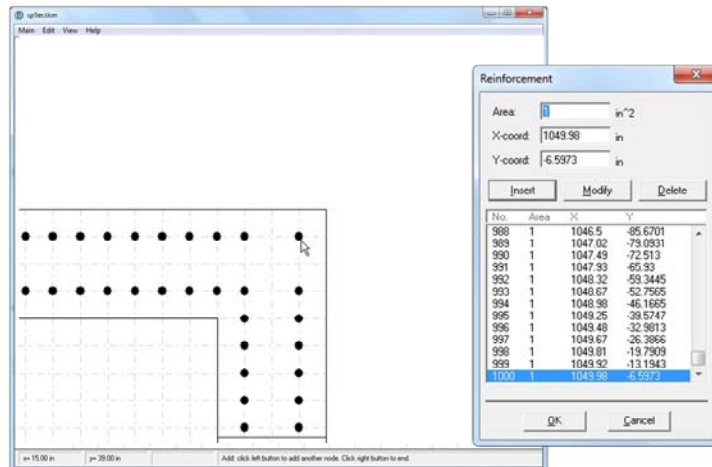
StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

14

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.

# Irregular Reinforcement



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.

# Irregular Reinforcement



Import Data (from text file)

Geometry  
 Reinforcement  
 Service loads  
 Factored loads

OK Cancel

**Reinforcement Data:**

No. Of Bars  
 A1 X1 Y1  
 \*  
 An Xn Yn

Template.xls [Read-Only] [Compatibility Mode] - Microsoft Excel

Notes: 1. No units are used. It is the user's responsibility to maintain the consistency of units. 2. Diagram scales are fixed. They may be c  
 3. Do not delete or move output data cells. Otherwise the exported data might be incorrect.

Input	Output
Center-to-Center Width in X	40
Height in Y	30
Reinforcement Ring Diameter	25
Reinforcement Area per Bar	0.5

Export Geometry  
Export Reinforcement

Degree	Radian	x-base	y-base	X	Y
0.000	0.000	15.000	0.000	35.000	0.000
10.000	0.175	14.772	2.665	34.772	2.665
20.000	0.349	14.095	5.130	34.095	5.130
30.000	0.524	12.990	7.500	32.990	7.500
40.000	0.698	11.491	9.642	31.491	9.642
50.000	0.873	9.642	11.491	29.642	11.491
60.000	1.047	7.500	12.990	27.500	12.990
70.000	1.222	5.130	14.095	25.130	14.095
80.000	1.396	2.665	14.772	22.665	14.772
90.000	1.571	0.000	15.000	20.000	15.000
90.000	1.571	0.000	15.000	20.000	15.000
100.000	1.745	-2.665	14.772	-22.665	14.772
110.000	1.920	-5.130	14.095	-25.130	14.095
120.000	2.094	-7.500	12.990	-27.500	12.990
130.000	2.269	-9.642	11.491	-29.642	11.491
140.000	2.443	-11.491	9.642	-31.491	9.642
150.000	2.618	-12.990	7.500	-32.990	7.500
160.000	2.793	-14.095	5.130	-34.095	5.130
170.000	2.967	-14.772	2.665	-34.772	2.665
180.000	3.142	-15.000	0.000	-35.000	0.000
180.000	3.142	-15.000	0.000	-35.000	0.000
190.000	3.316	-14.772	-2.665	-34.772	-2.665
200.000	3.491	-14.095	-5.130	-34.095	-5.130
210.000	3.665	-12.990	-7.500	-32.990	-7.500
220.000	3.840	-11.491	-9.642	-31.491	-9.642
230.000	4.014	-9.642	-11.491	-29.642	-11.491
240.000	4.189	-7.500	-12.990	-27.500	-12.990
250.000	4.363	-5.130	-14.095	-25.130	-14.095

StructurePoint  
 CONCRETE SOFTWARE SOLUTIONS

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.



# Loads



## F Factored Loads

Import Data (from text file)

Geometry  
 Reinforcement  
 Service loads  
 Factored loads

OK Cancel

Factored Loads

Load	U Moment	V Moment
1:20	500	0
2:40	0	0
3:60	0	0
4:80	0	0
5:100	0	0
6:120	0	0
7:140	0	0
8:160	0	0
9:180	0	0
10:200	0	0

OK Cancel

## S Service Loads

Mandatory for slenderness

Import Data (from text file)

Geometry  
 Reinforcement  
 Service loads  
 Factored loads

OK Cancel

Service Loads

Load	U Moment	V Moment	U Moment	V Moment
1:20	500	0	0	0
2:40	0	0	0	0
3:60	0	0	0	0
4:80	0	0	0	0
5:100	0	0	0	0
6:120	0	0	0	0
7:140	0	0	0	0
8:160	0	0	0	0
9:180	0	0	0	0
10:200	0	0	0	0

OK Cancel

## A Axial Loads

Uniaxial, short column investigation only

Axial Loads

Load	Final load	Increment
1:20	500	0
2:40	0	0
3:60	0	0
4:80	0	0
5:100	0	0
6:120	0	0
7:140	0	0
8:160	0	0
9:180	0	0
10:200	0	0

OK Cancel

## Load Factors

For service loads only

Load Combinations

Combo	Dead	Live	Wind	EQ	Other
1:1	1.0	0.0	0.0	0.0	0.0
2:2	1.0	0.0	0.0	0.0	0.0
3:3	1.0	0.0	0.0	0.0	0.0
4:4	1.0	0.0	0.0	0.0	0.0
5:5	1.0	0.0	0.0	0.0	0.0
6:6	1.0	0.0	0.0	0.0	0.0
7:7	1.0	0.0	0.0	0.0	0.0
8:8	1.0	0.0	0.0	0.0	0.0
9:9	1.0	0.0	0.0	0.0	0.0
10:10	1.0	0.0	0.0	0.0	0.0

OK Cancel

## C Control Point

Investigation Only

$$\phi P_{n,max}/0.85 \quad \phi P_{n,max} \quad f_s = 0.0 \quad f_s = f_y/2 \quad \epsilon_s = f_y/E_s \quad \epsilon_s = 0.005 \quad P = 0 \quad \phi P_{n,min}$$

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



Import Data (from text file)

Geometry  
 Reinforcement  
 Service loads  
 Factored loads

OK Cancel

Factored Loads

Load	X-Moment	Y-Moment
-120	582	0
(kip)	(k-ft)	(k-ft)

Insert Modify Delete

No.	P	Mx	My
18	-780	1048.8	0
19	790	885	0
20	790	-1656	0
21	480	468	0
22	480	-942	0
23	190	129	0
24	190	-132	0
25	-120	-288	0
26	-120	582	0

OK Cancel

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.

# Service Loads



The screenshot displays three overlapping windows from a structural analysis software. The 'Service Loads' window is in the background, showing input fields for Dead, Live, Wind, EQ, and Snow loads, categorized by Area Load (k/ft²), Moments (ft-k), and Sustained Load (%). The 'Load Combinations' window is in the foreground, showing a table of combinations (U1-U13) with columns for Dead, Live, Wind, EQ, and Snow. The 'Import Data (from text file)' window is partially visible on the right, with radio buttons for Geometry, Reinforcement, Service loads, and Factored loads.

Combo	Dead	Live	Wind	EQ	Snow
U1	1.4	0	0	0	0
U2	1.2	1.6	0	0	0
U3	1.2	1	0	0	0
U4	1.2	0	0.8	0	0
U5	1.2	1	1.6	0	0
U6	0.9	0	1.6	0	0
U7	1.2	0	-0.8	0	0
U8	1.2	1	-1.6	0	0
U9	0.9	0	-1.6	0	0
U10	1.2	1	0	1	0
U11	0.9	0	0	1	0
U12	1.2	1	0	-1	0
U13	0.9	0	0	-1	0

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.

# Axial Loads



- For non-slender uniaxial loading only

The screenshot shows a dialog box titled "Axial Loads" with a close button (X) in the top right corner. At the top, there are three input fields: "Initial load (kip)" with the value -120, "Final load (kip)" with the value 582, and "Increment (kip)" with the value 0. Below these fields are three buttons: "Insert", "Modify", and "Delete". The main part of the dialog is a table with the following data:

No.	Initial	Final	Inc.
18	-780	1048.8	0
19	790	885	0
20	790	-1656	0
21	480	468	0
22	480	-942	0
23	190	129	0
24	190	-132	0
25	-120	-288	0
26	-120	582	0

At the bottom of the dialog are "OK" and "Cancel" buttons.

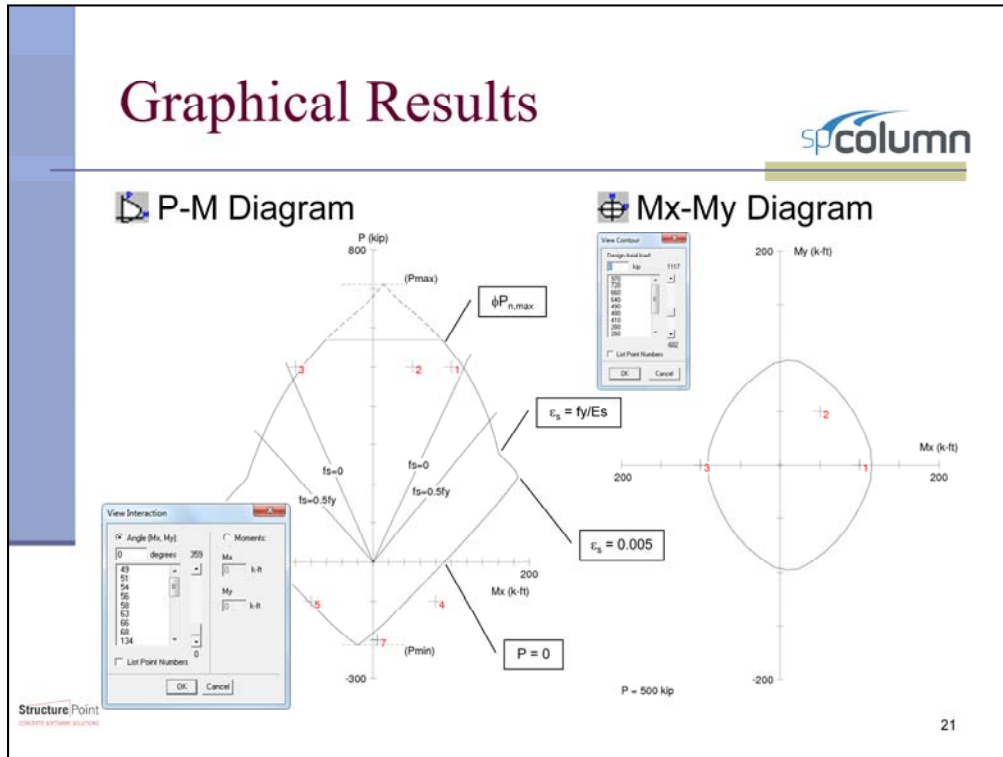
The option for axial load is intended for the engineers use to determine the corresponding allowable moment value for an existing axial force. This option does not show any points within the interaction diagram because this option essentially tells the user the values of the interaction diagram.

# Graphical Results



## P-M Diagram

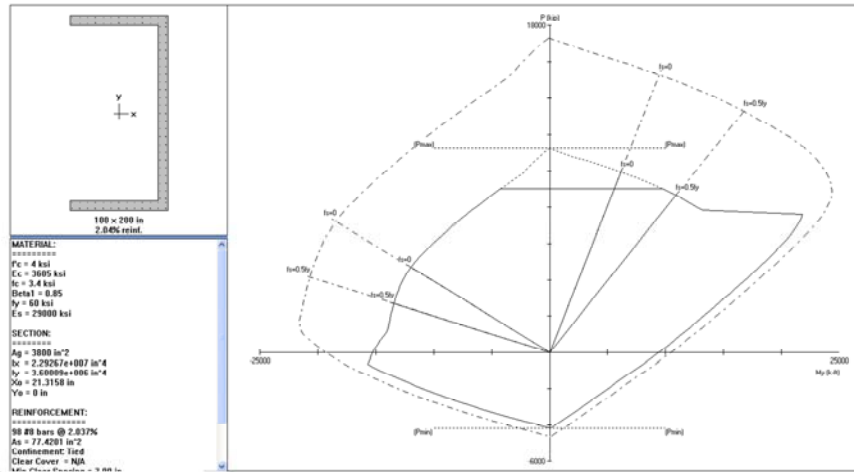
## Mx-My Diagram



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.

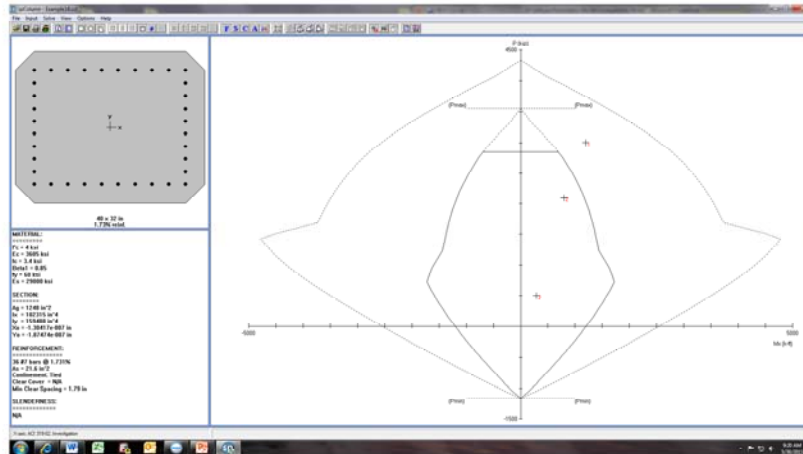
# Graphical Results

## ■ Nominal Interaction Diagram



The program can also provide the nominal interaction diagram to show the column's capacity before it is factored.

# Superimposing Diagrams



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.

# Text Results



```

Factored Loads and Moments with Corresponding Capacities:
=====
NOTE: Each loading combination includes the following cases:
First line - at column top
Second line - at column bottom
Load      Pu      Mu      My      P1Mom    P1Mom    P1Mom/P1Mo  HA  depth  Dc  depth  epx_s  Phi
No.  Combo  kip      k-ft     k-ft     k-ft     k-ft     k-ft              in      in      in      in      %
-----
1 1 U1    280.00    49.00    61.60    244.88    307.47    4.991    11.75    24.43    0.00325  0.751
2      280.00   -218.40   -47.60   -301.43   -46.70    1.380    14.59    27.32    0.00260  0.696
3 1 U2    640.00    78.80    124.80    136.96    216.90    1.738    19.64    26.81    0.00900  0.650
4      640.00   -284.40   -60.00   -238.04   -85.67    0.928    19.64    27.25    0.00116  0.650 #
8 1 U3    490.00    66.00    97.80    142.03    243.79    2.493    16.99    24.14    0.00141  0.650
6      490.00   -229.20   -82.80   -264.31   -60.89    1.153    17.41    27.21    0.00169  0.650
7 1 U4    480.00    62.80    122.40    130.13    263.69    2.972    17.13    26.88    0.00171  0.650
8      480.00   -223.20   -85.60   -260.09   -99.75    1.165    17.25    27.13    0.00172  0.650
9 1 U5    970.00    106.60    237.00   -----Pu > Pmax-----#
10     970.00   -301.20   -142.40   -----Pu > Pmax-----#
11 1 U6    660.00    79.10    178.80    91.76    224.49    1.255    20.32    26.80    0.00096  0.650
12     660.00   -212.40   -120.20   -215.88   -122.00    1.015    20.12    25.80    0.00086  0.650
13 1 U7    0.00      0.00     21.20    -16.80    306.46   -242.85    14.456    7.18    23.47    0.00683  0.900
14     0.00     -191.20    4.00     -356.79    9.44     2.360    10.96    27.06    0.00469  0.879
15 1 U8    10.00     23.40   -41.40    192.49   -340.56    8.226    8.82    25.44    0.00565  0.900
16     10.00   -167.20    36.80   -364.94    83.66    2.271    10.43    27.09    0.00480  0.883

STRUCTUREPOINT - spColumn v4.60 (TM)                                     Page 6
Licensed to: StructurePoint, License ID: 00000-0000000-4-28191-28191    05/30/11
C:\Program Files (x86)\StructurePoint\spColumn\Examples\Example11.col    09:33 AM

17 1 U9   -300.00   -10.10   -99.60   -26.22   -288.59    2.596    6.14    26.94    0.01018  0.900
18     -300.00   -68.40    89.00   -172.76   149.87    2.540    4.21    23.25    0.01858  0.900
19 1 U10  720.00    59.00    134.80    136.31    185.60    1.377    20.41    24.06    0.00500  0.650
20     720.00   -285.20   -117.80   -206.40   -85.25    0.724    21.63    26.38    0.00666  0.650 #
21 1 U11  410.00    65.50    76.60    211.57    247.42    3.230    14.49    24.28    0.00203  0.650
22     410.00   -196.40   -98.60   -265.84   -128.43    1.343    16.20    26.83    0.00199  0.650
23 1 U12  260.00    31.00    60.80    158.09    310.06    5.100    13.33    26.89    0.00299  0.729
24     260.00   -173.20    12.20   -306.78    21.61    1.771    14.31    27.07    0.00267  0.702
25 1 U13  -50.00     2.50     2.60   -283.86   244.02    101.945    4.48    23.26    0.00777  0.900
26     -50.00   -84.40    34.40   -337.08    137.39    3.994    8.80    26.19    0.00595  0.900

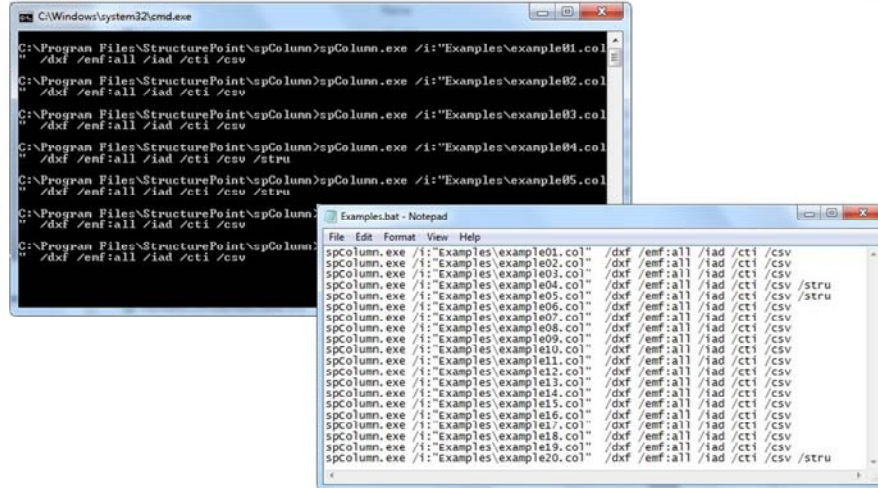
# Section capacity exceeded. Revise column!
Pmax = 853.93 kip
*** End of output ***
    
```



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.

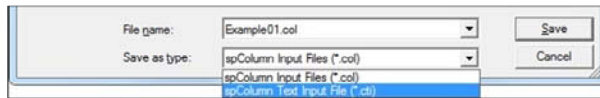


# Batch Mode



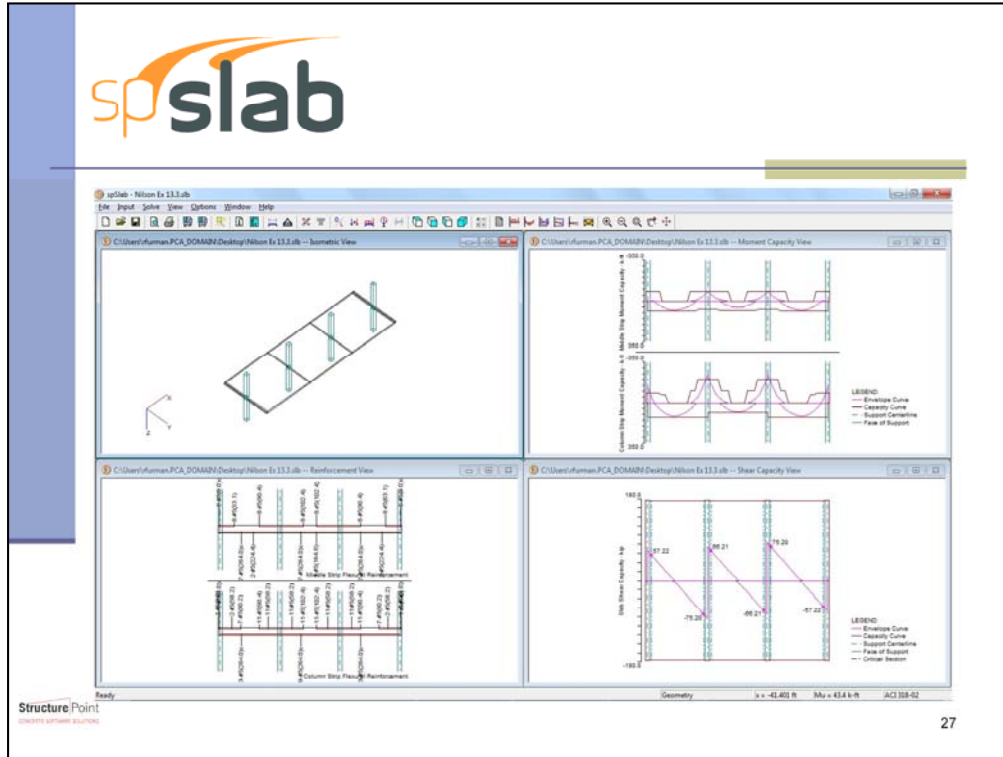
The batch mode feature of the program is a great automation tool to streamline the column investigation process. Intended for integration with 2D/3D structural analytical models, the batch mode takes text files that include all of the column properties and loading conditions and produces results like interaction diagrams, interaction diagram data, column section dx 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.

# CTI Files



```
example01.cti - Notepad
File Edit Format View Help
#spColumn Text Input (CTI) File
[spColumn version]
4.600
[Project]
spColumn Manual Example 1
[Column ID]
PCANotes 6.4
[Engineer]
SP
[Investigation Run Flag]
15
[Design Run Flag]
9
[slenderness Flag]
0
[User Options]
0,0,4,0,0,0,0,0,0,0,0,2,-1,0,-1,4,2,0,5,0,0,0,0,0,0,0,0,13
[irregular Options]
-2,0,0,1,0.790000,50.000000,50.000000,-50.000000,-50.000000,0.000000,0.000000,5.000000,5.000000
[Tries]
0,1,7
[Investigation Reinforcement]
4,2,0,0,5,5,5,5,1.500000,1.500000,1.500000,1.500000
[Design Reinforcement]
0,0,0,0,0,0,0,0,0,0.000000,0.000000,0.000000,0.000000
```

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.

# Options



General Information | Span Control | Solve Options |

Labels  
Project: Demonstration Design  
Frame:  
Engineer:

Options  
Design code: ACI 318M-08  
Reinforcement: ASTM A615M

Run mode  
 Design  
 Investigation

Floor System  
 Two-Way  
 One-Way/Beam

Other  
 Left cantilever  Right cantilever  
 Distance location as ratio of span

OK Cancel Help

## Two-way systems

General Information | Span Control | Solve Options |

Design Options  
Live load pattern ratio: 100 %

Compression Reinforcement  User Slab Strip Widths  
 Decremental Reinf. Design  User Distribution Factors  
 One-way Shear In Drop Panels  Beam T-Section Design  
 Distribute Shear to Slab Strips  Long. Bm. Supt. Design  
 Trans. Bm. Supt. Design

Critical section for punching shear  
Ignore side on a free edge if within 4 times the slab thickness from the face of the support.

Deflection calculation options  
Sections to use in deflection calculations are:  
 Gross (included)  Effective (included)  
In negative moment regions, to calculate I<sub>y</sub> and I<sub>or</sub> use:  
 Rectangular Section  T-Section

Calculate long-term deflections  
Duration of load: 60 months Sustained part of live load: 0 %

OK Cancel Help

## One-way systems

General Information | Span Control | Solve Options |

Design Options  
Live load pattern ratio: 100 %

Compression Reinforcement  User Slab Strip Widths  
 Decremental Reinf. Design  User Distribution Factors  
 Effective flange width  
 Rigid beam-column joint  
 Moment Redistribution

Tension Analysis and Design  
Tension type:  Equilibrium  Strips in Ranges  
 No  Yes  
 Compatibility

Deflection calculation options  
Sections to use in deflection calculations are:  
 Gross (included)  Effective (included)  
In negative moment regions, to calculate I<sub>y</sub> and I<sub>or</sub> use:  
 Rectangular Section  T-Section

Calculate long-term deflections  
Duration of load: 60 months Sustained part of live load: 0 %

OK Cancel Help

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.

# Span Data



## ■ Defining Spans

## ■ Span Manipulation

Span Data dialog box showing input fields for span properties and a table of span data.

Span No.	Location	Length	Thickness	Width-L	Width-R
1	Interior	0.7	7	7	7
2	Interior	18	7	7	7
3	Interior	18	7	7	7
4	Interior	18	7	7	7
5	Interior	0.7	7	7	7

General Information dialog box showing support selection and span control list.

New#	Old#	Sup L/R	Copy
1-CL	1-CL	- / 1	
2	2	1 / 2	
3	3	2 / 3	
4	4	3 / 4	
5-CR	5-CR	4 / -	

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.

# Support Data



## ■ Defining Supports

## ■ Boundary Conditions

Sup.	Stiff%	H.A.	c1A.	c2A.	H.B.	c1B.	c2B.	Shear	Gamma
1	100	9	16	16	9	16	16	Yes	No
2	100	9	16	16	9	16	16	Yes	No
3	100	9	16	16	9	16	16	Yes	No
4	100	9	16	16	9	16	16	Yes	No

Sup. No.	Kz	Ky	Far End - Above	Far End - Below
1	0	0	Fixed	Fixed
2	0	0	Fixed	Fixed
3	0	0	Fixed	Fixed
4	0	0	Fixed	Fixed

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.

# Reinforcement



## ■ Design

Reinforcement Criteria dialog box. The 'Beams' tab is selected. It has sub-tabs for 'Top bars' and 'Bottom bars'. Fields include: Clear (1.5), Bar size (Min: #4, Max: #4), Spacing (Min: 1, Max: 10), and Reinf. ratio (Min: 0.18, Max: 2). A checkbox 'There is more than 12 in of concrete below top bars.' is present. Buttons: OK, Cancel, Help.

## ■ Investigation

Reinforcing Bars dialog box. The 'Beam Bars' tab is selected. Fields include: Bar size (#5), No. of bars (11), Length (ft) (7.5311), Top left position, and Cover (in) (1.125). A 'Span = 22 ft' label is shown. Buttons: Add, Modify, Delete, OK, Cancel, Help.

Size	Type	Count	Cover	Length	Start
#5	TopL	11	1.125	7.53112	--
#5	TopL	11	1.125	4.8506	--
#5	TopR	7	1.125	7.51534	--
#5	TopR	2	1.125	4.8506	--
#5	BotC	13	1.125	--	--

StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

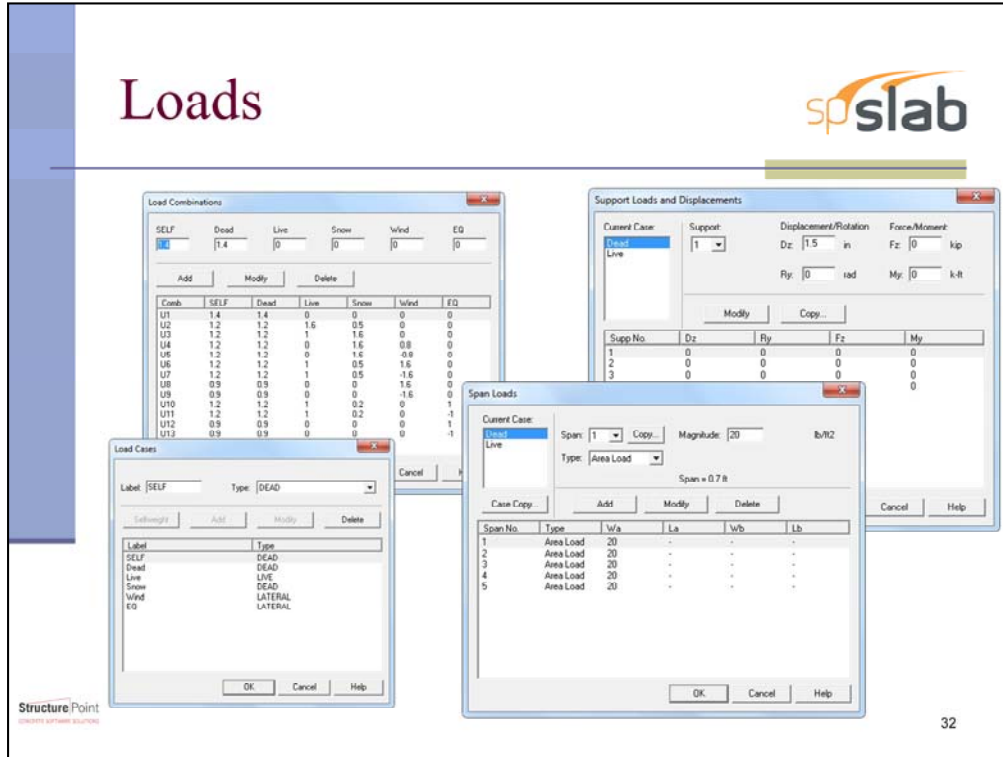
31

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 reinforcing. Assuming that the bar makes contact at the middle of the bend gives a more accurate calculation of reinforcing spacing.

# Loads



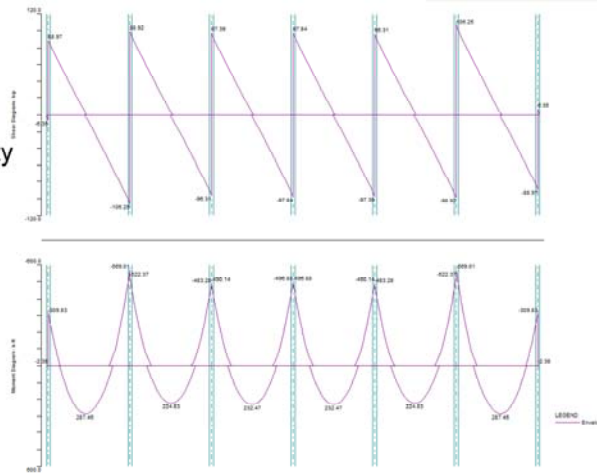
The user has load input choices of area loads, line loads, and point loads. These loads are designated a label to identify them and allow them to be factored. The system can be further modified to include concentrated forces on the supports along with prescribed support displacement and rotation. spSlab accounts for lateral loads as well by modeling them as moments acting on the ends of a span.



# Graphical Output



- Loads
- Internal Forces
- Moment Capacity
- Shear Capacity
- Deflections
- Reinforcement



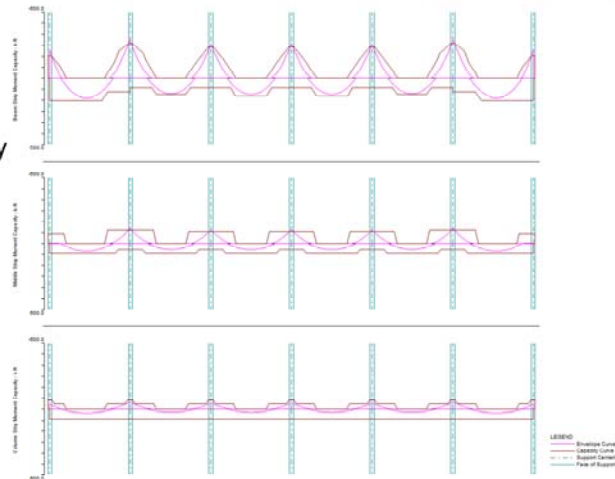
Internal Force Diagrams

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.

# Graphical Output



- Loads
- Internal Forces
- Moment Capacity
- Shear Capacity
- Deflections
- Reinforcement



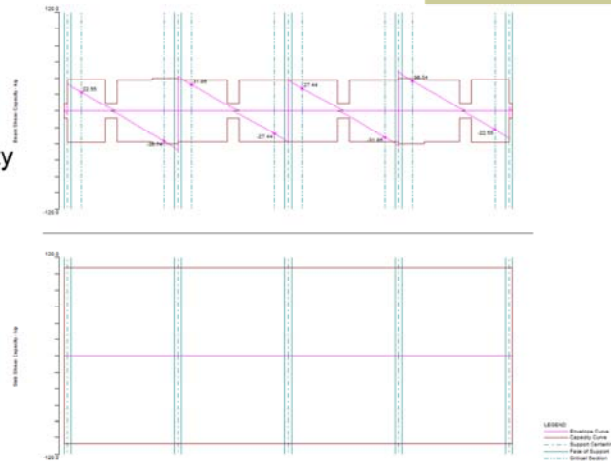
Moment Capacity Diagrams

The moment capacity diagrams include individual results for the beam strip, the column strip, and the middle strip.

# Graphical Output



- Loads
- Internal Forces
- Moment Capacity
- Shear Capacity
- Deflections
- Reinforcement



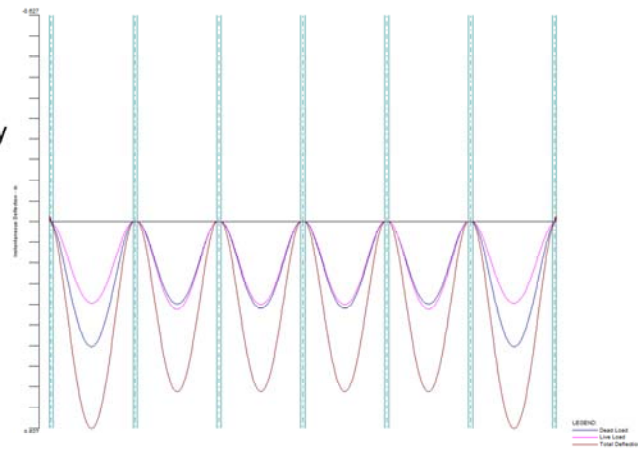
Shear Capacity Diagrams

The shear capacity diagrams include the results for both the beam and the slab.

# Graphical Output



- Loads
- Internal Forces
- Moment Capacity
- Shear Capacity
- Deflections
- Reinforcement



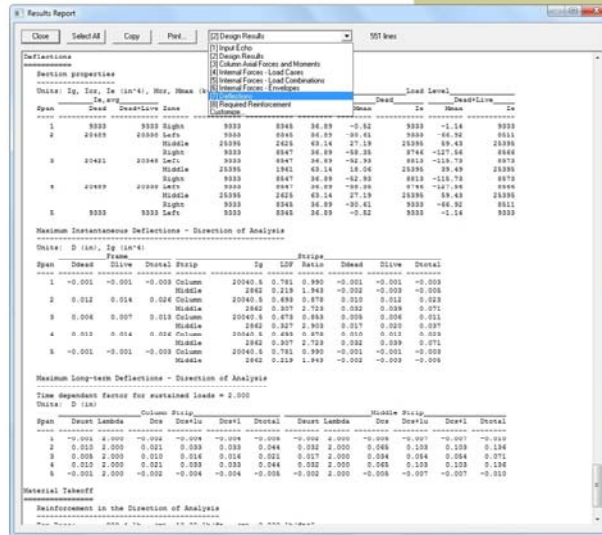
Deflection Diagram

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.

# Text Ouput

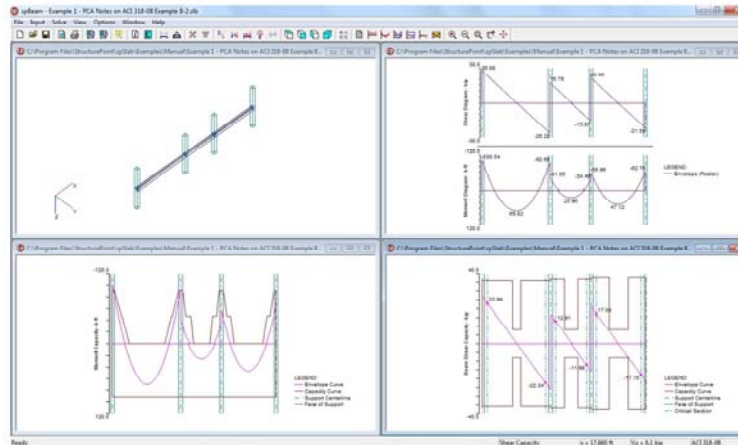


- Input Echo
- Design Results
- Column Forces
- Internal Forces
- Deflections



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.

- Analysis, design, and investigation of R/C beams and one-way slab systems



38

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.

# Options



The image displays two screenshots of the spBeam software's options dialog box. The left screenshot shows the 'General Information' tab, and the right screenshot shows the 'Solve Options' tab.

**General Information (Left Screenshot):**

- Labels: Project: spSlab/spBeam Manual, Example 1; Frame: PCA Notes on ACI 318-08, Example 8-2; Engineer: StructurePoint
- Options: Design code: ACI 318-08; Reinforcement: ASTM A615
- Run mode:  Design;  Investigation
- Frame: No. of Supports: 4
- Floor System:  One-Way/Beam;  Left cantilever;  Right cantilever
- Other:  Distance location as ratio of span

**Solve Options (Right Screenshot):**

- Design Options: Live load pattern ratio: 100 %
- Compression Reinforcement;  Effective flange width
- Incremental Reinf. Design;  Rigid beam-column joint
- Moment Redistribution
- Torsion Analysis and Design:  Torsion type;  Equilibrium;  Compatibility;  Stairs in flanges;  No;  Yes
- Deflection calculation options: Sections to use in deflection calculations are:  Gross (uncracked);  Effective (cracked)
- In negative moment regions, to calculate  $I_g$  and  $M_{cr}$  use:  Rectangular Section;  T-Section
- Calculate long-term deflections: Duration of load: 60 months; Sustained part of live load: 0 %

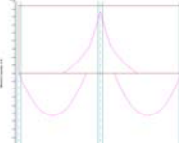
spBeam's options are similar to spSlab's, but notice that it forces the user to design the floor system as a one-way system. Beyond this, the options are identical to spSlab's one-way stem options.

# Moment Redistribution



For ACI 318-08, 05, and 02

$$\delta = \begin{cases} 0, & \text{if } \epsilon_t < 0.0075 \\ 1000\epsilon_t, & \text{if } \epsilon_t \geq 0.0075 \end{cases} \text{ or}$$



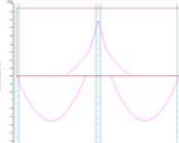
Top Reinforcement

Span Zone	Width	Width	Area (in <sup>2</sup> )	Sp (in)	AsPrime	AsMin	AsMax	SpReq	AsReq	Bars
1 Left	2.08	0.00	0.467	0.000	0.000	1.084	9.792	0.000	3#8	*3
Middle	2.08	3.70	15.400	0.000	0.094	1.084	9.792	0.071	3#8	*3 *1
Right	2.08	33.18	23.333	0.000	0.136	1.084	9.792	0.754	3#8	*3
2 Left	2.08	33.18	0.467	0.000	0.136	1.084	9.792	0.754	3#8	*3
Middle	2.08	3.70	8.400	0.000	0.094	1.084	9.792	0.071	3#8	*3 *1
Right	2.08	0.00	23.333	0.000	0.000	1.084	9.792	0.000	3#8	*3

NOTES:  
 \*1 - Design governed by minimum reinforcement.  
 \*3 - Number of bars governed by maximum allowable spacing.

For ACI 318-99

$$\delta = \begin{cases} 0, & \text{if } (\rho - \rho') > 0.5\rho_b \\ 20\left(\frac{\rho - \rho'}{\rho_b}\right), & \text{if } (\rho - \rho') \leq 0.5\rho_b \end{cases} \text{ or}$$



Moment Redistribution Factors

Span	Side	Orig Mo	Calculated	Factored	Limit (%)	Applied
1	Right	0.00	0	0.00000	0.00	0.00
2	Left	33.18	8	0.20988	9.98	10.00
2	Right	33.18	8	0.20988	9.98	10.00
3	Left	0.00	0	0.00000	0.00	0.00

Top Reinforcement

Span Zone	Width	Width	Area (in <sup>2</sup> )	Sp (in)	AsPrime	AsMin	AsMax	SpReq	AsReq	Bars
1 Left	2.08	0.00	0.467	0.000	0.000	1.084	9.792	0.000	3#8	*3
Middle	2.08	2.01	15.400	0.000	0.082	1.084	9.792	0.038	3#8	*3 *1
Right	2.08	23.44	23.333	0.000	0.136	1.084	9.792	0.418	3#8	*3
2 Left	2.08	23.44	0.467	0.000	0.136	1.084	9.792	0.418	3#8	*3
Middle	2.08	2.01	8.400	0.000	0.082	1.084	9.792	0.038	3#8	*3 *1
Right	2.08	0.00	23.333	0.000	0.000	1.084	9.792	0.000	3#8	*3

NOTES:  
 \*1 - Design governed by minimum reinforcement.  
 \*3 - Number of bars governed by maximum allowable spacing.

For CSA A23.3

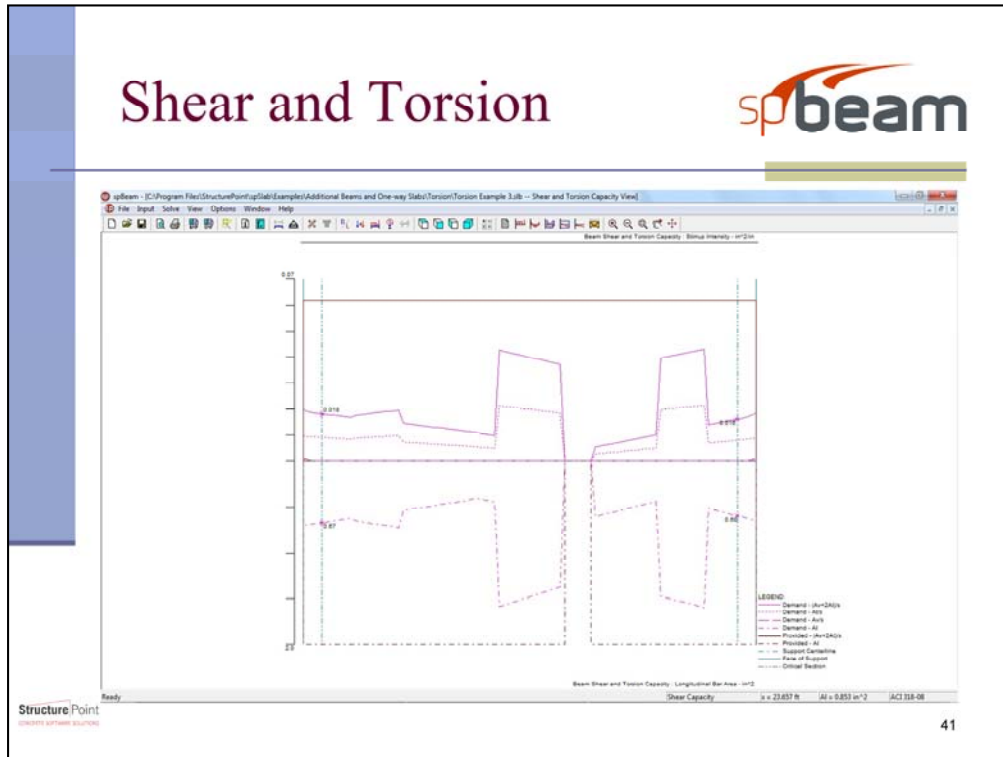
$$\delta = 30 - 50\frac{c}{d}$$



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.

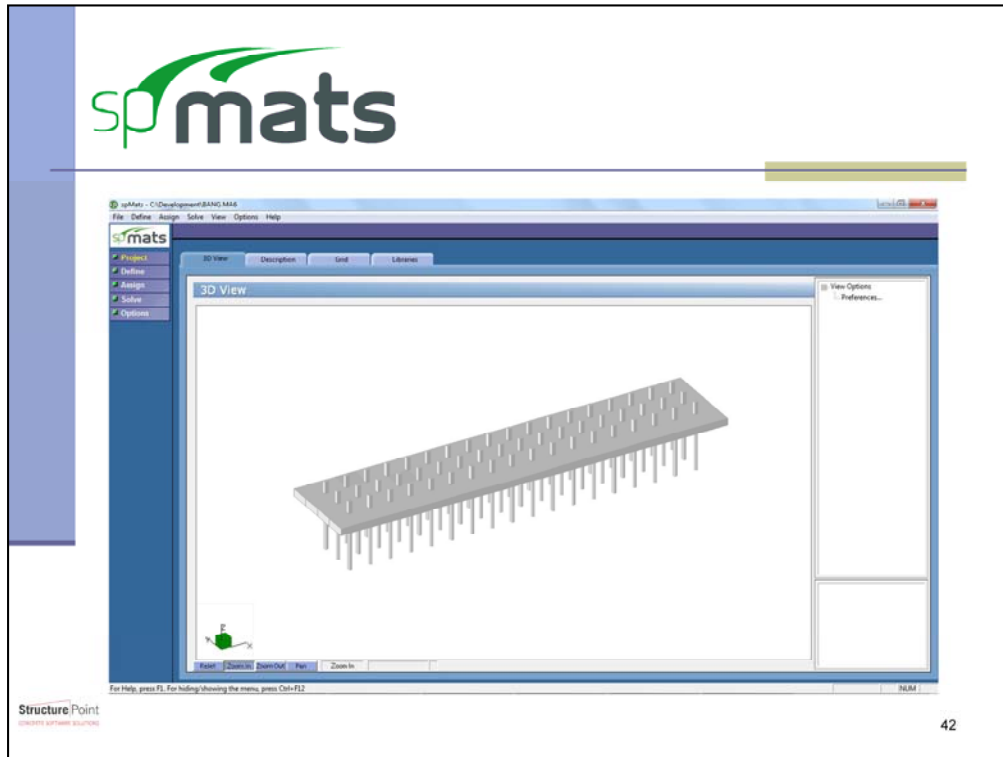


# Shear and Torsion



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.

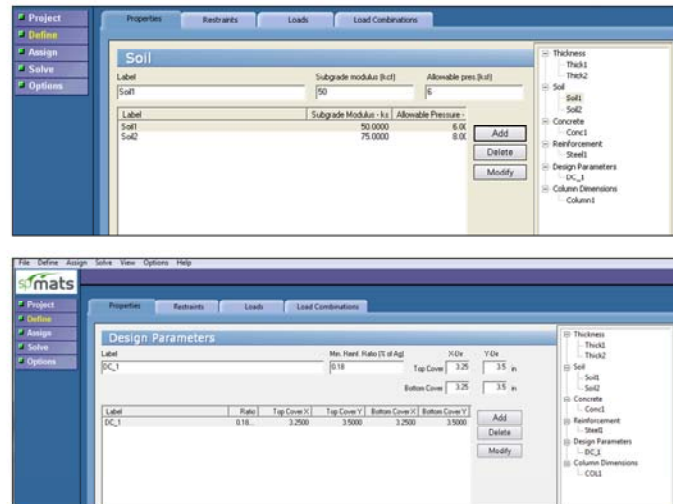
spMats uses the plate-bending theory and the Finite Element Method to model the behavior of the mat or slab. The soil supporting the slab is assumed to behave as a set of one-way compression-only springs. If during the analysis, a loading or the mat shape cause any uplift creating a spring in tension, 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.

# Meshing



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.

# Define Properties



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.

# Restraints



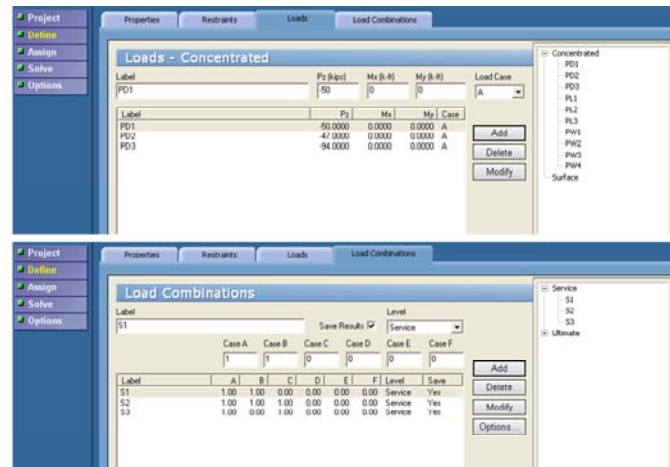
There are three different types of restraints: nodal springs, slaved nodes, and piles.

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.

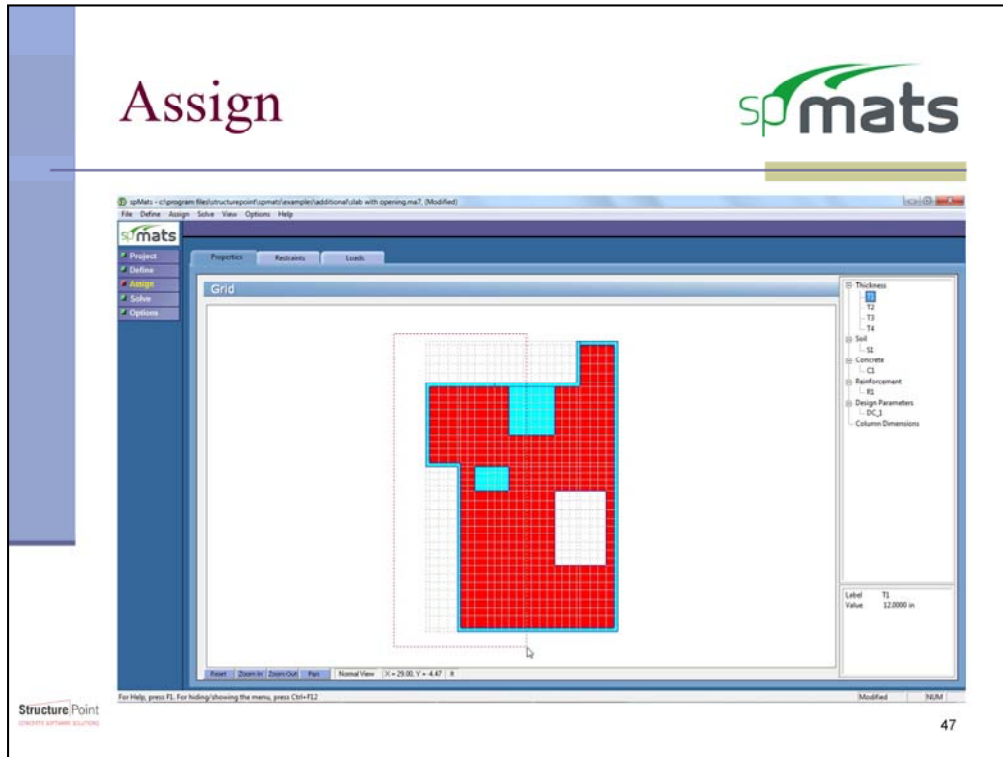
# Defining Loads



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.

# Assign



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.

# Solution



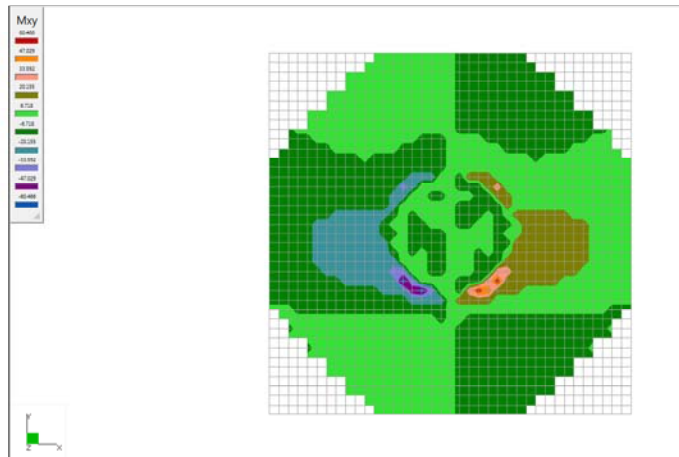
The image displays three screenshots of the SPmats software interface. The top-left screenshot shows the 'Solver' panel with the following settings: Maximum number of iterations: 10; Maximum allowed displacement (in): 12; Minimum contact area (%): 50; Mat uplift occurs for positive nodal displacement exceeding (in): 0; Compute required reinforcement based on:  Maximum moment within an element,  Average moment within an element. The top-right screenshot shows the 'Reports' panel with various report options checked, including 'Design Moment & Steel - Bot'. The bottom screenshot shows the 'View Results' panel with a table of design moment and reinforcement data.

CS0 - ELEMENT BOTTOM DESIGN MOMENT AND REINFORCEMENT:									
Units --> Moment (kip-ft/ft), As (in <sup>2</sup> /ft)									
Flags --> [n] Minimum controls, [s] Exceeds maximum, [x] Cannot...									
Elem Node Ld Comb. Max. M(us) R(sxx) Node Ld Comb. Max. M(uy)...									
1	26	02	-33.59	0.518m	26	02	-35.39	0.518m	26
2	26	05	-38.21	0.518m	26	05	-41.89	0.518m	26
3	3	05	-13.51	0.518m	27	02	-21.81	0.518m	27
4	4	05	-1.70	0.518m	28	05	-16.88	0.518m	28
5	6	-	0.00	0.518m	5	05	-12.33	0.518m	5
6	6	-	0.00	0.518m	6	05	-8.84	0.518m	6
7	7	-	0.00	0.518m	7	05	-8.61	0.518m	7

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.

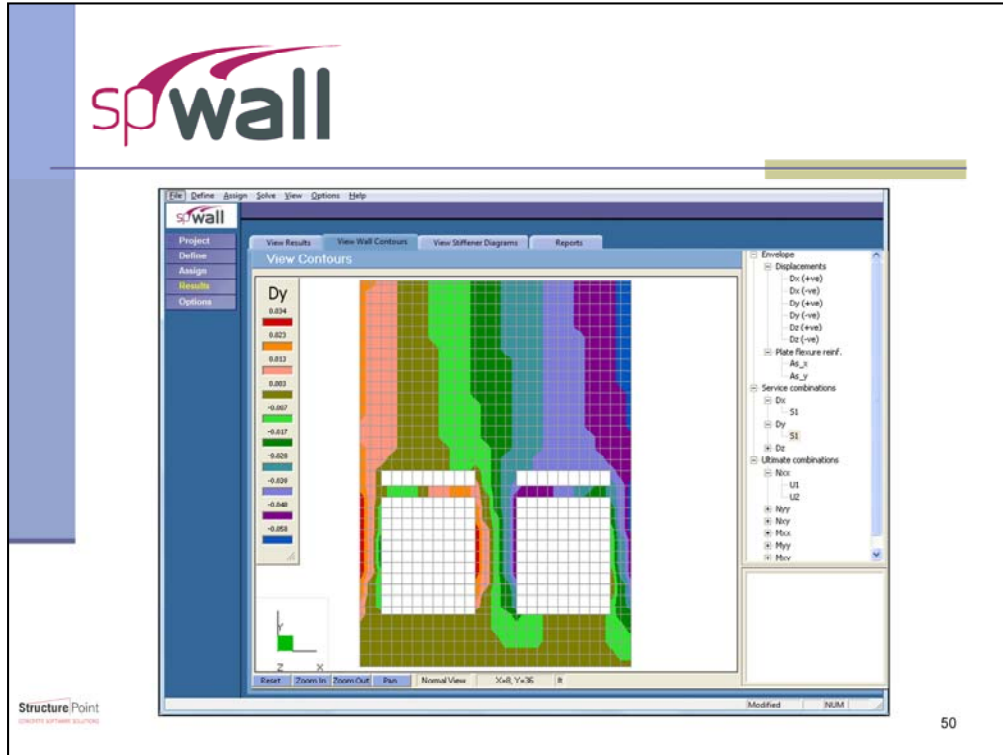


# Contours



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.

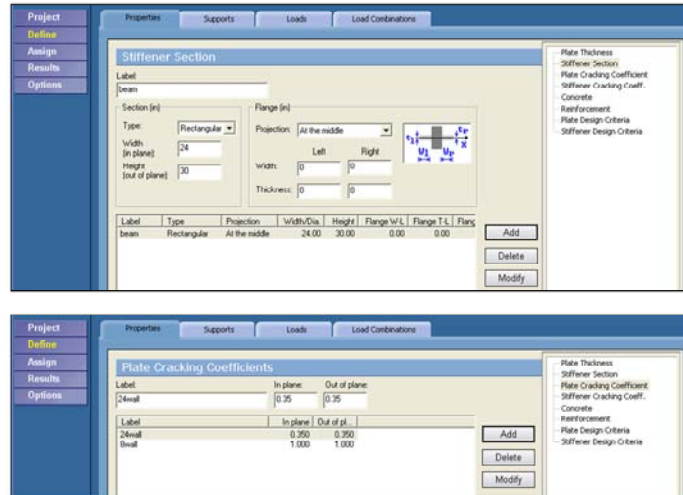
The program designs the steel reinforcing based on the user inputs for design parameters. If the model is unable to produce a reinforcing design that meets the design criteria the model will still run and all the results are available as well as the contour maps. The program flags these failed elements 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.

# Define

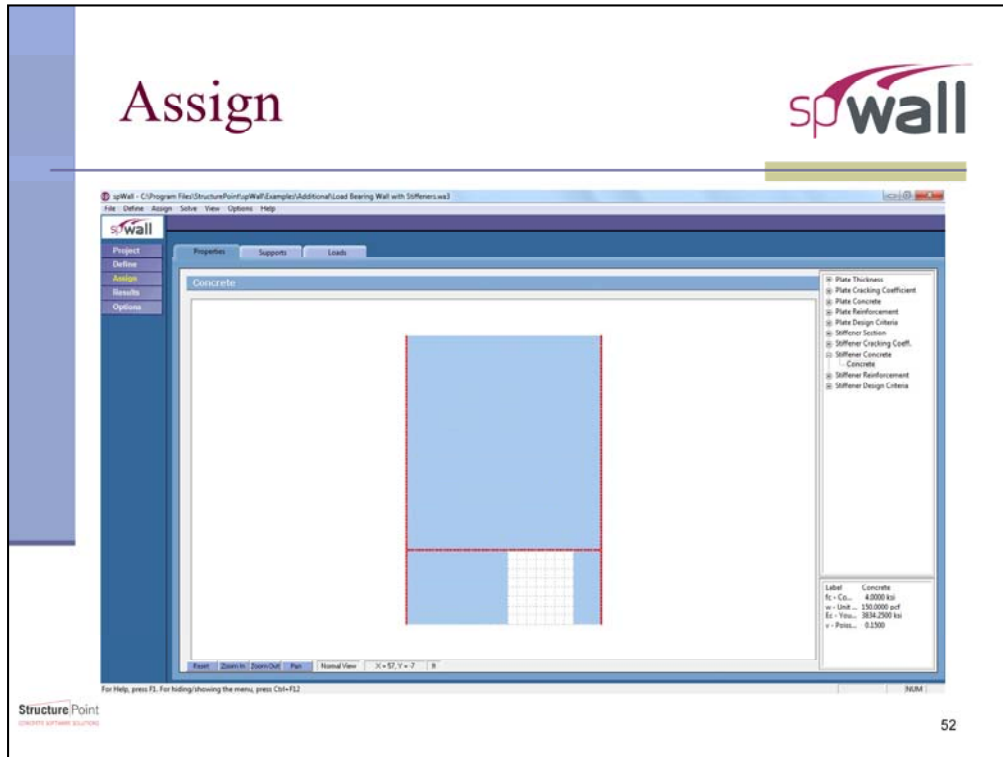


StructurePoint  
CONCRETE SOFTWARE SOLUTIONS

51

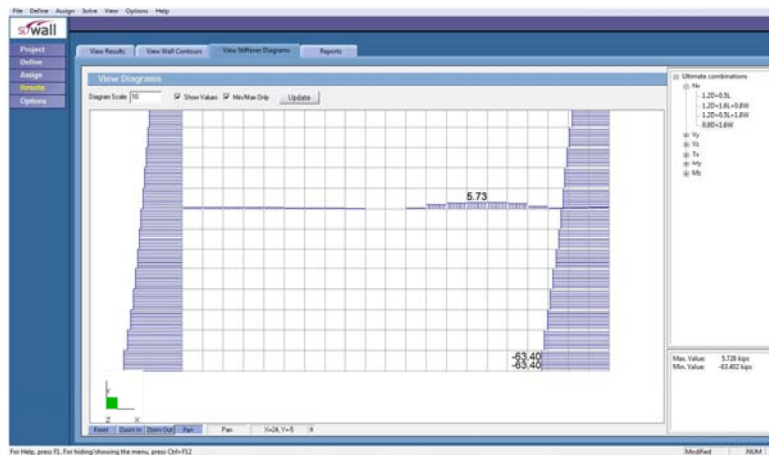
After setting up a meshing system, spWall allows the user to input properties of the wall, properties of wall stiffeners if applicable, supports, and loads. The wall can be designed as either a standard reinforced wall or a precast wall. With both options the user can choose between using a single or double curtain of reinforcing. This reinforcing is allowed to have either uniform or variable cover in relation to the wall faces.

# Assign



The assignment of these definitions takes the form of graphically assigning the properties in the same way as in spMats.

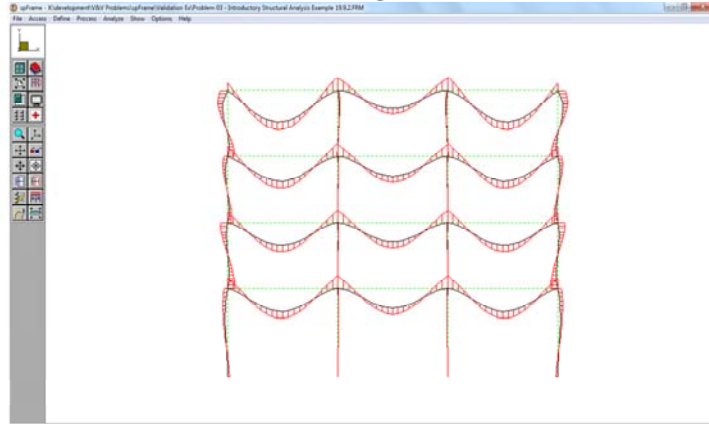
# Solution



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.

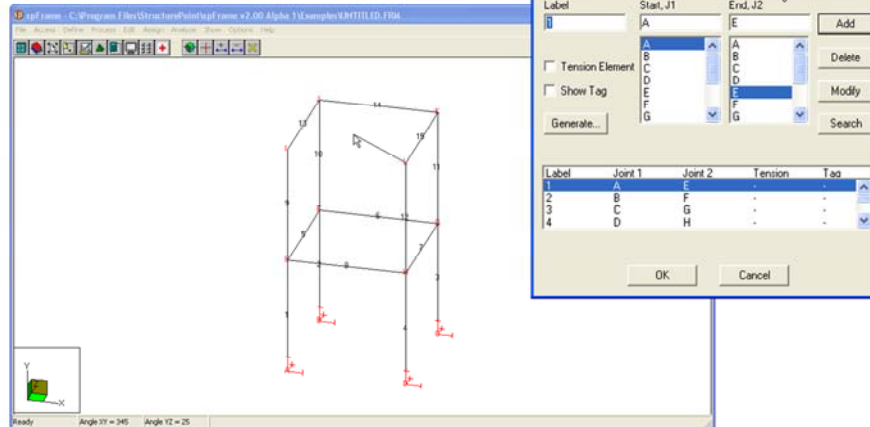
The program designs the steel reinforcing based on the user inputs for design parameters. If the model is unable to produce a reinforcing design that meets the design criteria the model will still run and all the results are available as well as the contour maps. The program flags these failed elements in the required steel reinforcing contour maps.

- General purpose structural analysis of 2-D and 3-D structures subject to static loads



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.

## ■ Creating the Model



The screenshot displays the spFrame software interface. The main window shows a 3D truss model with joints labeled A through H. A 'Members' dialog box is open, allowing the user to define members between joints. The dialog includes fields for 'Label', 'Start, J1', and 'End, J2', along with a list of available joints (A-H) and buttons for 'Add', 'Delete', 'Modify', and 'Search'. A table at the bottom of the dialog shows the current member list:

Label	Joint 1	Joint 2	Tension	Tag
1	A	E	-	-
2	B	F	-	-
3	C	G	-	-
4	D	H	-	-

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.

## ■ Applying the Forces

**Member Loads**

Label: Dist1

Type:  Point Load  Distributed Load  Moment Load  Left Triangle  Right Triangle  Temperature

Intensity (w): 5 kN/m

Distance from J1/Length: 0

Load Length/Length: 1

Orientation:  Local  Global  Projection

Direction:  X  Y  Z

Label	O-D-T	Par1	Par2	Par3
Dist1	L&P	5	0	1
Trn_1	L&S	10	0	0.75
P1	L&P	12	0	-
P2	L&P	20	0	-

OK Cancel

**Known Displacements**

Load case: Settlement

Joint List: A, B, C, D, E, F

Dx: 0 mm Rk: 0 Rad

Dy: 10 mm Ry: 0 Rad

Dz: 0 mm Rz: 0 Rad

Joint	Dx	Dy	Dz	Rk	Ry	Rz
A	0	10	0	0	0	0

OK Cancel

**Joint Loads**

Load case: Wind

Joint List: A, B, C, D, E, F

Fx: 10 kN Mx: 0 kN-m

Fy: 0 kN My: 0 kN-m

Fz: 0 kN Mz: 0 kN-m

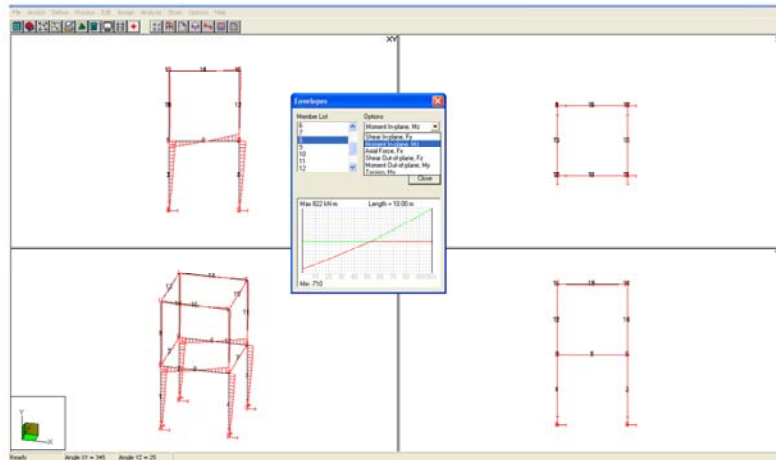
Joint	Fx	Fy	Fz	Mx	My	Mz
F	10	0	0	0	0	0
E	10	0	0	0	0	0
I	15	0	0	0	0	0

OK Cancel


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.



## ■ Viewing Results



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.

The slide features the StructurePoint logo in the center, with the word "Structure" in black and "Point" in a larger black font, separated by a vertical red line. Below the logo, the text "CONCRETE SOFTWARE SOLUTIONS" is written in red. A white rectangular box with a blue border is positioned below the logo, containing the contact information: "Call: +1-847-966-4357" and "Email: info@StructurePoint.org" in red. The slide has a blue vertical bar on the left and a green horizontal bar at the top right.

**StructurePoint**  
CONCRETE SOFTWARE SOLUTIONS

**Call: +1-847-966-4357**  
**Email: [info@StructurePoint.org](mailto:info@StructurePoint.org)**

58

StructurePoint would be glad to hear from you and receive your feed back as well as answer any questions regarding the program features, capabilities, price, and licensing options