sp mats

Version 10.00

This Computer program (including software design, programming structure, graphics, manual, and on-line help) was created and published by STRUCTUREPOINT, formerly the Engineering Software Group of the Portland Cement Association (PCA) for the engineering analysis and design of concrete foundation mats, combined footings, and slabs on grade.

While STRUCTUREPOINT has taken every precaution to utilize the existing state-of-the-art and to assure the correctness of the analytical solution techniques used in this program, the responsibilities for modeling the structure, inputting data, applying engineering judgment to evaluate the output, and implementing engineering drawings remain with the structural engineer of record. Accordingly, STRUCTUREPOINT does and must disclaim any and all responsibility for defects or failures of structures in connection with which this program is used.

Chapter 1: PROGRAM INTRODUCTION

Chapter 2: SOLUTION METHODS

Chapter 3: PROGRAM INTERFACE

spmats

Chapter 4: MODELING METHODS

Chapter 5: MODEL DEVELOPMENT

Chapter 6: MODEL SOLUTION

Chapter 7: MODEL OUTPUT

APPENDIX

CHAPTER

1

PROGRAM INTRODUCTION

spMats is an engineering software program 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, columns, or translational and rotational nodal restraints. Slaved degrees of freedom may also be applied to selected nodes. The model is analyzed under static loads that may consist of uniform (surface) and concentrated loads. The resulting deflections, internal forces, soil pressure, and reactions are key parts of the program output. In addition, the program computes the required area of reinforcing steel in the slab based on the provisions of the design code selected.

spMats uses the thin plate-bending theory and the Finite Element Method (FEM) to model the behavior of the mat or slab. The soil supporting the slab is assumed to behave as a set of unidirectional (compression-only) translational springs (Winkler foundation). During the analysis, if loading/support conditions or the mat shape causes any uplift and induces tension in a spring, 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 equilibrium is reached. The uplift threshold along with other solver options are controlled by the user to achieve the required model.

1.1   Program Features

- Support for ACI 318-14/11/08/05/02 and CSA A23.3-14/04/94 design standards

- Object-based modeling of foundation slab systems with a full featured graphical interface

- Structural grids may be utilized to facilitate structural member placement from plan

- Export of column and/or pile sections as CTI files to be analyzed by spColumn

- Import of grids, loads, and load cases & load combination information from text files to facilitate model generation

- Export of grids, loads, and load cases & combination information from text files to facilitate model generation

- Four-noded, prismatic, thin plate element with three degrees-of-freedom per node

- Material properties (concrete and reinforcing steel) may vary from slab object to slab object

- Soil may be applied uniformly over slab objects or concentrated and applied at nodes using nodal spring supports

- Default definitions and assignment of model properties are provided to facilitate model generation

- Nodes may be restrained for vertical displacement and/or rotation about X and Y axes

- Nodes may be slaved to share the same displacement and/or rotation

- Applied loads may be uniform (vertical force per unit area) or concentrated (P_z, M_x, and M_y)

- Load combinations are categorized into service (serviceability) and ultimate (design) levels

- The self-weight of the slab is automatically computed and may optionally be included in the analysis

- Result envelopes (maximum and minimum values) for deflections, pressures, and moments

- Design moments include contribution of twisting moments via Wood-Armer formulas

- Isometric (3D) view of the modeled slab with ability to view grids, loads and other typical model features in typical CAD environment in multi view ports with up to 6 concurrent views

- Contour plots to visualize results of analysis and design

- U.S. Customary or SI (metric) units

- Checking of data as they are input for validation

- User-controlled screen display settings including a full color pallet

- Ability to save defaults and settings for future input sessions

1.2 Program Capacity

- 255 X-grid lines

- 255 Y-grid lines

- 1000 Nodes in total

- 25 Nodes per linear generation

- 36 Nodes per circular generation

- 24, 36, 48 Circle segments for meshing circular foundations

- 255 Thickness definitions

- 255 Concrete definitions

- 255 Soil definitions

- 255 Nodal spring definitions

- 255 Slaved nodes definitions

- 255 Pile definitions

- 26 Load cases

- 255 Load combinations (service plus ultimate)

- 64,500 Mesh elements

1.3 System Installation Requirements

spMats is a 32-bit Windows application. spMats solver has a 32-bit and a 64-bit version. The proper version is selected automatically by the installation program based on the target computer architecture. Any computer running Microsoft Windows 10 operating system is sufficient to run the spMats program provided that .NET Framework v4 is installed. If it is not detected by the installation program then it will be installed automatically.

PROGRAM INTRODUCTION

The actual program capacity depends on system resources available on the computer on which spMats is running. To solve models with the maximum number of nodes and load combinations, a 64-bit operating system with at least 8GB of RAM is required.

spMats output results/retrieval may be delayed or seem to freeze after analysis completion when running larger models on a network or cloud drive. It is recommended to run the model locally for fastest response.

Instructions on how purchasing, downloading, installation, licensing, and troubleshooting issues, please refer to support pages on the StructurePoint website at *StructurePoint.org*.

1.4   Terms & Conventions

The following terms are used throughout this manual. A brief explanation is given to help familiarize you with them.

Windows	Refers to the Microsoft Windows environment as listed in System Requirements
[]	Indicates metric equivalent
Click on	Means to position the cursor on top of a designated item or location and press and release the left-mouse button (unless instructed to use the right-mouse button).
Double-click on	Means to position the cursor on top of a designated item or location and press and release the left-mouse button twice in quick succession.
Marquee select	Means to depress the mouse button and continue to hold it down while moving the mouse. As you drag the mouse, a rectangle (known as a marquee) follows the cursor. Release the mouse button and the area inside the marquee is selected.

Various styles of text and layout have been used in this manual to help differentiate between different kinds of information. The styles and layout are explained below:

Bold	All bold typeface makes reference to either a menu or a menu item command such as **File** or **Save**, or a tab such as **Description** or **Grid**.
`Mono-space`	Indicates something you should enter with the keyboard. For example, type "c:*.txt".

KEY + KEY

Indicated a key combination. The plus sign indicated that should press and hold the first key while pressing the second key, then release both keys. For example, "ALT + F" indicated that should press the "ALT" key and hold it while you press the "F" key, then release both keys.

SMALL CAPS

Indicates the name of an object such as a dialog box or a dialog box component. For example, the OPEN dialog box or the CANCEL or MODIFY buttons.

SOLUTION METHODS

2.1   Introduction

spMats uses the Finite Element Method for the structural modeling and analysis of reinforced concrete slab systems or mat foundations subject to static loading conditions.

The slab is idealized as a mesh of rectangular elements interconnected at the corner nodes. The same mesh applies to the underlying soil with the soil stiffness concentrated at the nodes. Slabs of irregular geometry will be idealized to conform to geometry with rectangular boundaries. Even though slab and soil properties can vary between elements, they are assumed uniform within each element.

The three degrees of freedom are considered at each node are the vertical translation and two rotations about the two orthogonal axes. An external load can exist in the direction of each of the above degrees of freedom, i.e., a vertical force and two moments about the Cartesian axes.

2.1.1   Foundation Slab Mat Systems

spMats can be used to model, analyze, and design foundation systems such as mat foundations, spread and combined footings, soil-supported foundations, slabs on grade, pile-supported foundations. Samples of such systems are illustrated below:
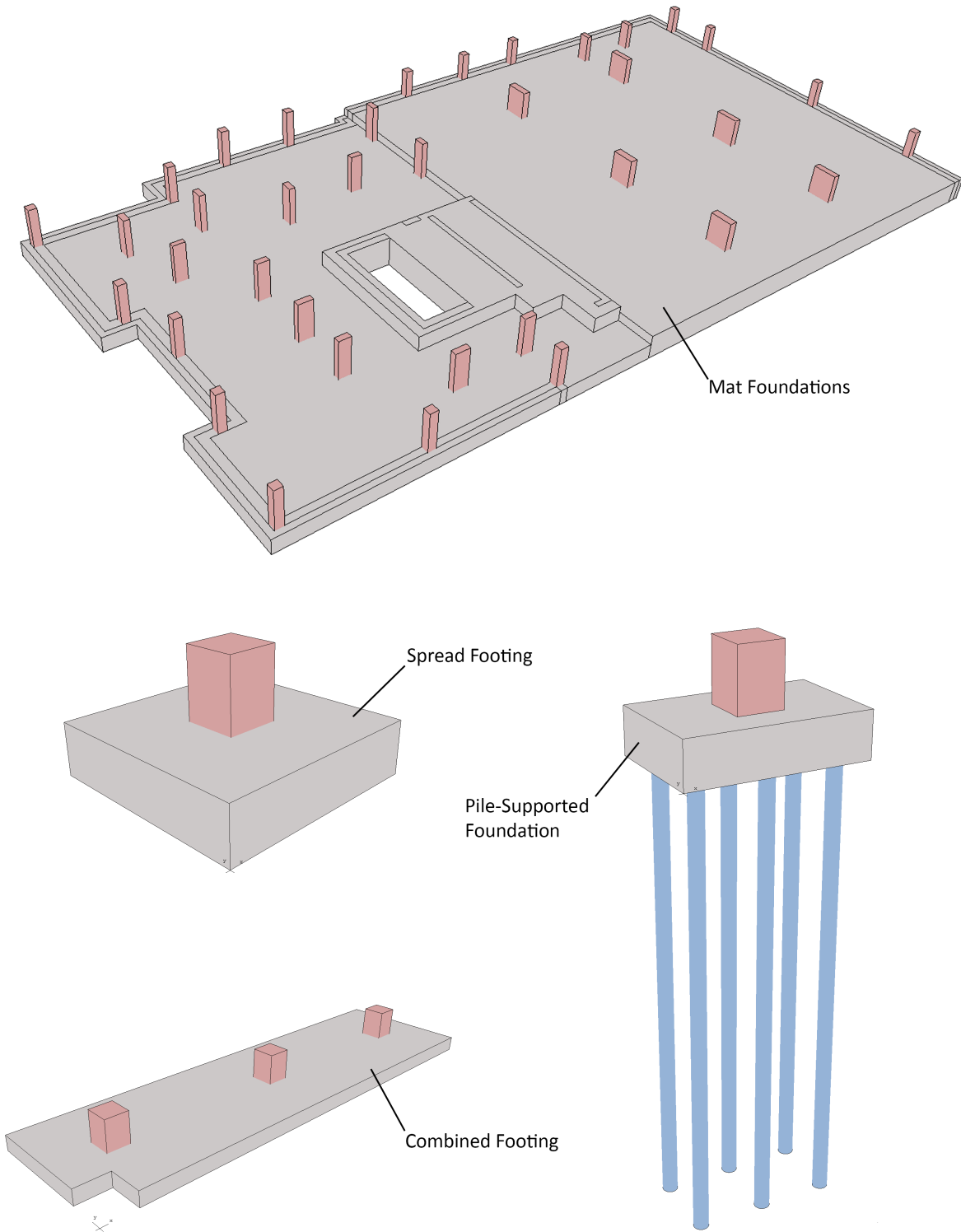
Mat Foundations

Spread Footing

Pile-Supported
Foundation

Combined Footing

Figure 2-1 Foundation Systems

2.1.2   Coordinate Systems

Global Coordinate System

The mid-surface of the slab lies in the XY plane of the right-handed XYZ orthogonal coordinate system shown in Figure 2-2. The slab thickness is measured in the direction of the Z-axis. Looking at the display monitor, the origin of the global coordinate system is located in the bottom left corner of the screen. The positive X-axis points to the right, the positive Y-axis points upward towards the top of the monitor, and the positive Z-axis points out of the screen. Thus, the XY plane is defined as being the plane of the display monitor.
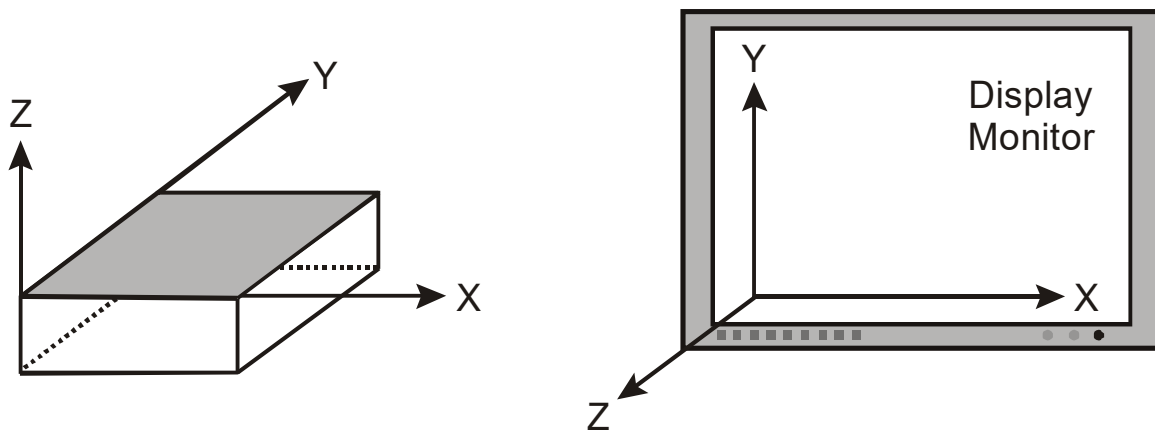
Figure 2-2 Global Coordinate System

Local Coordinate System

There is no local coordinate system requirement in spMats.

2.2   Codes and Standards Provisions

2.2.1   Code Checks

Geometry Considerations

The Program does not have any code checks regarding geometry considerations.

Material Considerations

The Program does not have any code checks regarding material considerations.

Loading Considerations

External loads are applied as concentrated nodal loads and/or surface loads according to the sign convention shown in Figure 2-3.

A concentrated nodal load consists of a vertical load, Pz, and two concentrated moments about the X and Y axes, Mx and My. It should be noted that a positive vertical load is applied upward (in the positive Z-direction).
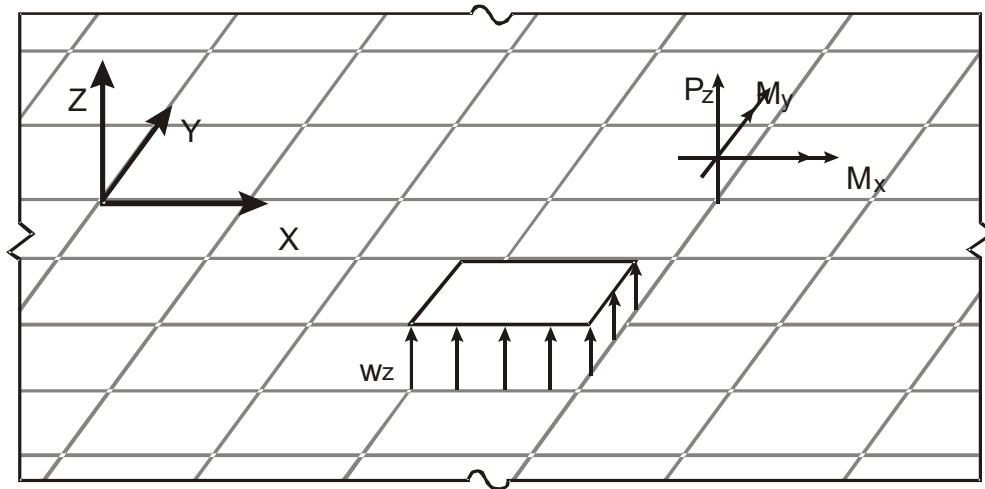
Figure 2-3 Applied Loads

The uniform element surface load, wz, applied over an element is internally discretized by the program into equivalent nodal loads as shown in Eq. 2-1:

$$\begin{Bmatrix} P_{iz} \\ M_{ix} \\ M_{iy} \\ P_{jz} \\ M_{jx} \\ M_{jy} \\ P_{kz} \\ M_{kx} \\ M_{ky} \\ P_{lz} \\ M_{lx} \\ M_{ly} \end{Bmatrix} = w_z \cdot \begin{Bmatrix} 1/4 \\ -b/24 \\ a/24 \\ 1/4 \\ b/24 \\ a/24 \\ 1/4 \\ -b/24 \\ -a/24 \\ 1/4 \\ b/24 \\ -a/24 \end{Bmatrix} \cdot a \cdot b \qquad \text{Eq. 2-1}$$

where a and b are the element dimensions.

The self-weight of the slab is computed internally based on the assigned concrete unit weight and the thickness of each element. The self-weight is treated like a surface load and may optionally be considered in the analysis under the dead load case.

Analysis Options

The Program utilizes the Finite Element Method as a method of analysis as permitted in ACI 318-14, 6.2.3 (e). The analysis solver provides the user with options to control foundation model behavior and impose physical restrictions to limit parameters such as maximum displacements and minimum contact area.

Design Options

The Program utilizes the Finite Element Method results to determine the required slab flexural reinforcement in accordance with American (ACI 318) and Canadian (CSA A23.3) design codes. In addition to minimum reinforcement percentage and bar location, spMats provides options to design to either the maximum moment or the average moment within an element. The user can stipulate a number of criteria for generating the finite element mesh before analyzing the model. Options include choices of mesh density and aspect ratio are particularly useful for non-rectangular foundation slabs.

Detailing Options

The Program provides capability for setting the base reinforcement ratio within Design Parameters Tab under Definitions | Properties Section.

The user can utilize reinforcement density contours to detail rebar size and spacing to provide the required strength as required by the selected design code.

2.2.2   Geometric Checks

Input Phase

The Program ensures that nodes with assigned properties exist within the slab. If a node with properties such as columns, piles, restraints or loads is assigned outside the slab, the following Error message is displayed.
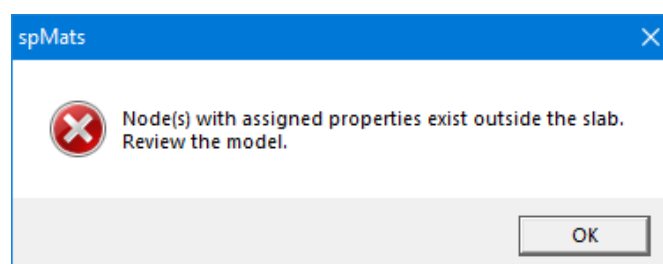
Output Phase

There are no geometric checks performed by the Program during output phase. However, any warning pertaining to model stability, contact area or exceedance of limits is reported. Detailed information on the solution can be found in the **Solver Messages** Dialog under **Tables** Window.

2.3   Analysis Methods

2.3.1   Overview of Finite Element Method (FEM) of Analysis

The finite element method is used in spMats for analysis of foundation slabs. During analysis, spMats converts the object-based model into a finite element model. The user defines the mesh used in the analysis by inputting maximum allowed mesh size and maximum allowed aspect ratio. Additional meshing is automatically introduced at slab boundaries, columns, piles, and nodes with assigned properties such as restraints, and point loads.

Definitions and Assumptions

The rectangular plate finite element[1] used in spMats has four nodes at the corners and three degrees of freedom (D_z, R_x and R_y) per node, as shown in Figure 2-4. This element considers the thin plate theory, which makes use of the following Kirchhoff hypotheses:

1. Plane sections initially normal to the mid-surface remain plane and normal to that surface after bending.

2. The stress component normal to the mid-plane is small compared to other stress components and is neglected.

3. The deflection of the mid-surface is small compared to the thickness of the plate.

4. The mid-plane remains unstrained subsequent to bending.

The element material is homogeneous, isotropic, and obeys Hooke's law. Constant thickness and constant material properties are assumed within an element. Cracking effects or changes in the slab elevation are not taken into account in the model.

Note that when deflections are not small, the bending of plates is accompanied by strain in the mid-plane. Further, for thick slabs, shear deformations (which are not considered by the program) may be significant, and a finite plate element based on the more general Mindlin's Theory may be required.

---

1. Reference [5]
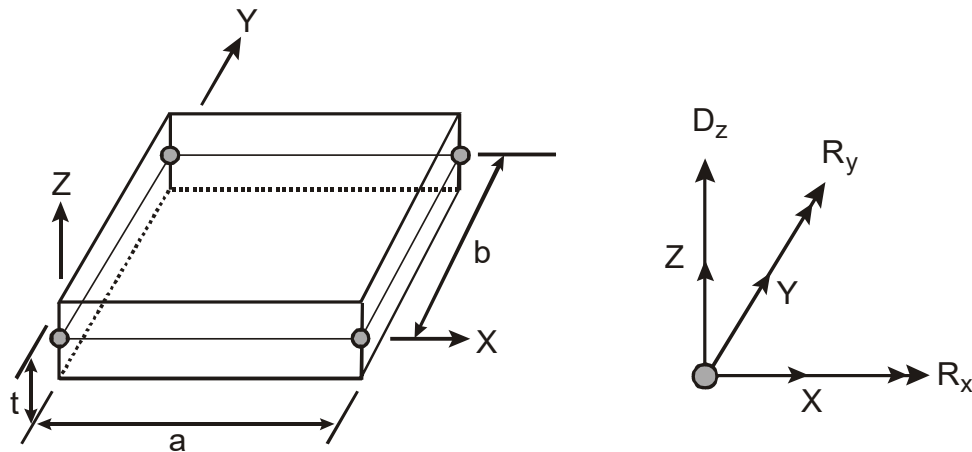
Figure 2-4 Plate Finite Element

2.3.2   Modeling of Supports

The model can be supported by soil assigned to a slab and/or by piles, nodal restraints, nodal springs, and slaved degrees of freedom that can assigned to nodes. Each support option is discussed below.

Soil Support - Winkler's Foundation

The soil supporting the slab is modeled as a group of linear uncoupled springs (Winkler type) concentrated at the nodes. The soil element is tensionless, weightless, and has one degree of freedom, which is the displacement in the **Z** direction (**D$_z$**). The contribution of each element node to the soil spring stiffness is equal to the nodal tributary area (1/4 the element area) multiplied by the soil subgrade modulus, **K$_s$**, under the element. The common nodes of adjacent elements undergo the same displacement. Therefore, if the adjacent elements have dissimilar soil properties, the soil pressures at the common nodes of these elements will differ in proportion to their respective soil subgrade modulus values.

The contact pressure, **P$_z$**, under each element node is proportional to the nodal displacement, **D$_z$**.

$$P_z = K_s D_z$$

<div align="right">Eq. 2-2</div>
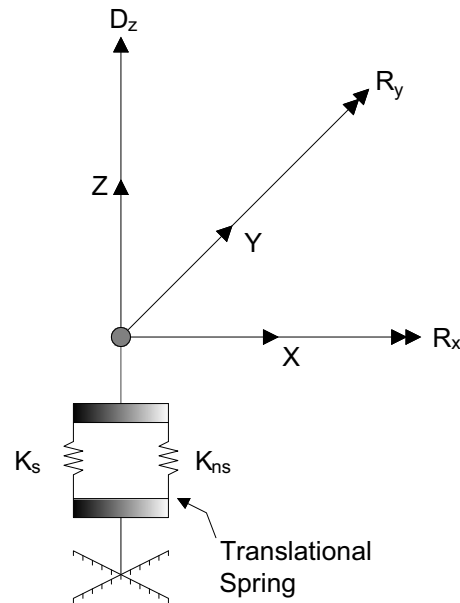
Figure 2-5 Soil and Nodal Springs

Usually, several factors are considered in the determination of the subgrade modulus: the size and shape of the footing, soil type below the footing and deeper, type and duration of loading, footing stiffness, and superstructure stiffness. The program does not perform any correction on the input subgrade modulus to account for these or any other factors.

Additional nodal springs may be applied in parallel to the Winkler's springs, as shown in Figure 2-5. Accordingly, their linear stiffness, K_{ns}, is added to the equivalent spring constant.

The nodal spring reaction at a particular node is proportional to the nodal displacement, D_z

$$F_z = K_{ns}D_z \qquad \text{Eq. 2-3}$$

Piles

Piles are modeled as springs connected to the nodes of the finite element model.

The spring constant, K_p, for a pile is calculated from the formula:

$$K_p = \frac{Q_u}{S} \qquad \text{Eq. 2-4}$$

where Q_u denotes the load applied to the pile and S is the corresponding settlement of the pile. Assuming soil allowable pressure, P_{all}, acting on the pile base, Q_u equals $P_{all} = A_p$, where A_p is

pile cross sectional area. Neglecting long-term effects, the settlement of pile is estimated from the empirical formula for a single pile in cohesionless soil[2]:

$$S = \frac{D}{100} + \frac{Q_u L}{A_p E_p}$$

Eq. 2-5

where D is pile diameter, L is pile length, and E_p is modulus of elasticity of pile material. The above formula is units independent as long as all of its terms have consistent units. For noncircular piles, an effective diameter is calculated from the formula:

$$D = \sqrt{\frac{4A_p}{\pi}}$$

Eq. 2-6

Nodal Restraints

All nodal degrees of freedom (DOF) are assumed to be initially released (i.e., free to move). Mathematically speaking, each DOF implies an equilibrium equation; however, nodal DOFs may be fully restrained against displacement and/or rotation.

Nodal Springs

Partial restraint in the Z direction is possible with the use of translational springs.

Slaved Degrees of Freedom

Slaved degrees of freedom may be assigned to a group of nodes to share the same displacement or rotation. Slaving enforces uniform deformation modes at selected nodes that can help in modeling stiff structural elements such as walls and pedestals. Applying a full restraint and slaving of a node (or nodes in a group) for the same degree of freedom is not allowed. Either only restraining all nodes (zero displacement) in a group or only slaving of all nodes in a group (same non-zero displacement) should be applied.

Slaving of degrees of freedom produces a stiffer slab system and reduces the number of equations to be solved. It should be noted that slaved degrees of freedom (SLDOF) are assigned by grouping of nodes. A group of nodes can be designated to share the same D_z, R_x, or R_y. If a group of nodes should share all three degrees of freedom, three different SLDOF groups (one for each DOF) must be defined. It should also be noted that a node can belong to more than one SLDOF group as long as these groups are slaved for different degrees of freedom. The external load corresponding to a SLDOF group corresponds to the sum of loads applied to all slaved nodes in the groups.

---

2. Reference [8]

2.3.3   Determination of Internal Forces

The Program determines the element internal moments as shown below:

Element Internal Moments

The bending moments, M_{xx} and M_{yy}, and the twisting moment, M_{xy}, are computed at the corner nodes of each element. Figure 2-6 shows the element moment sign convention used by the program. Note that unlike in beams and columns, the traditional plate and shell theory convention is that M_{xx} denotes the moment along (not about) the X-axis and M_{yy} denotes the moment along the Y-axis. Both moments are positive when they produce tension at the top.

The principal moments, M_{r1} and M_{r2}, and the principal angle (see Figure 2-7), are computed from the general moment transformation equations:

$$M_{r1} = M_{xx}\cos^2\theta + M_{yy}\sin^2\theta + M_{xy}\sin(2\theta)$$

Eq. 2-7

$$M_{r2} = M_{xx}\sin^2\theta + M_{yy}\cos^2\theta - M_{xy}\sin(2\theta)$$

Eq. 2-8
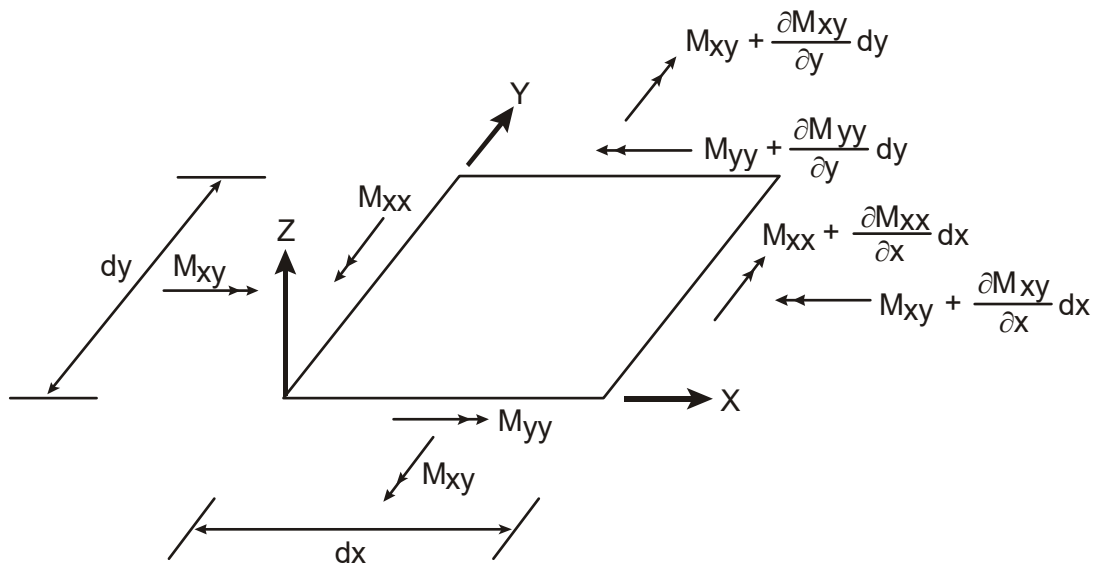
Figure 2-6 Element Nodal Moments

Note that since M_{r1} and M_{r2} are principal moments, the twisting moment associated with the r_1-r_2 axes (M_{r12}) is zero:

$$M_{r12} = \frac{M_{yy} - M_{xx}}{2} \sin(2\theta) + M_{xy} \cos(2\theta) = 0 \qquad \text{Eq. 2-9}$$

and the angle θ is:

$$\theta = \frac{1}{2} \tan^{-1}\left(\frac{2M_{xy}}{M_{xx} - M_{yy}}\right) \qquad \text{Eq. 2-10}$$

AB = 1
BC = sin (Θ)
AC = cos(Θ)
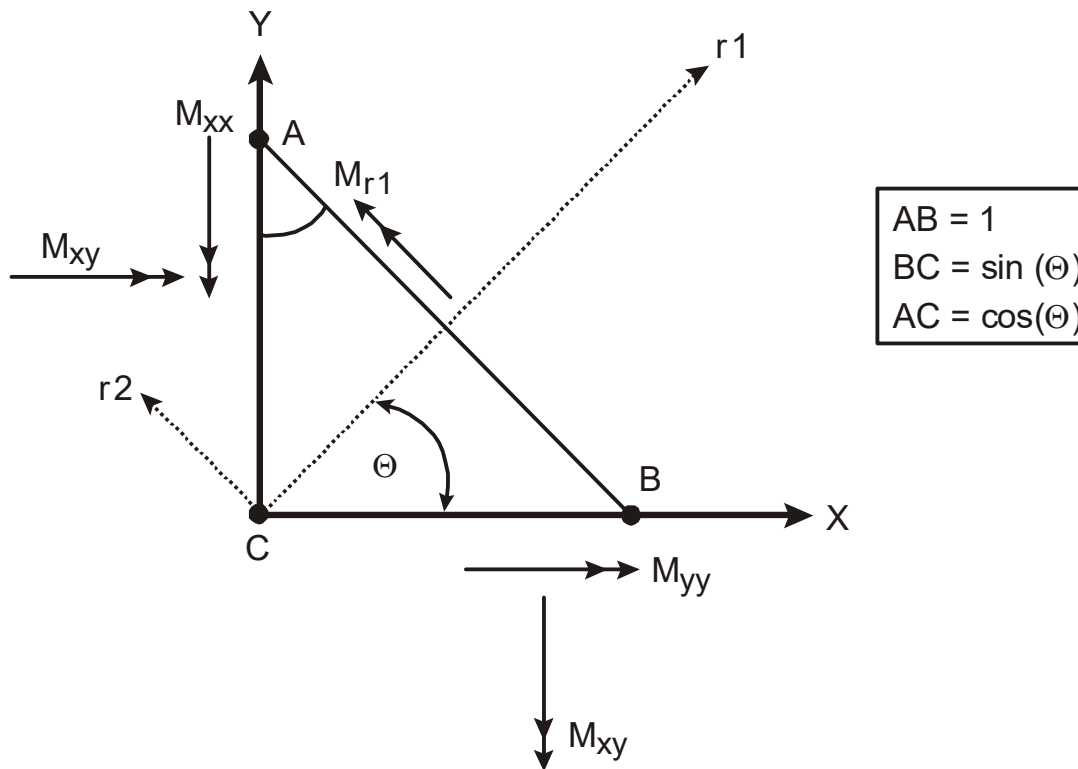
Figure 2-7 Element Principal Moment

2.3.4   Displacements and Pressures

Displacements

The Program calculates the displacements, namely displacement, $\mathbf{D_z}$, X-Rotation, $\mathbf{R_x}$, and Y-Rotation, $\mathbf{R_y}$, at all four nodes of an element for both service load and ultimate load combinations. Service-level nodal vertical displacement, $\mathbf{D_z}$, envelopes are also reported by the Program.

During analysis, if a node undergoes upward vertical displacement, the Program disconnects that node from the analysis model and iterates the solution. The maximum allowed service vertical downward displacement value can be set by the user under Solve Option and its default value is 11 in.

Under Solve Options when "Uplift occurs when displacement exceeds" user-input is entered as positive value; the Program permits an upward vertical displacement of a node up to that value without disconnecting the node. However, this user-input must be set to zero if a particular model contains a soil-supported node. An ideal use of this option is for foundations supported by piles where piles have tensile capacity. The soil, however stiffness it has, under the foundation must be ignored in modeling in such a model for accuracy of calculations.

Pressures

The Program calculates the soil pressures at all four nodes of an element for both service load and ultimate load combinations. The calculated soil pressures are compared with the allowable pressure values during the analysis. If the calculated soil pressure exceeds the allowable pressure specified by the user, the Program displays a warning message when analysis is completed and elements with this conditions are indicated in the graphical pressure contours view.

2.4 Design Methods

2.4.1 Flexural Design

The Program utilizes the FEM analysis results in order to calculate the flexural reinforcement according to American (ACI 318) and Canadian (CSA A23.3) design codes. For each element in the analysis model, spMats processes the results to determine applicable design moments are consequently used to calculate the required flexural reinforcement per the selected code edition.

Element Design Moments

The Principal of Minimum Resistance[3] is used by the program to obtain values for the design moments, which include the effects of the twisting moment.

The equivalent design bending moments, **M_{ux}** and **M_{uy}**, for the design of reinforcing steel respectively in the X and Y direction are computed as follows:

- For top reinforcement where positive moments produce tension:

$$M_{ux} = M_{xx} + \left| M_{xy} \right|$$
Eq. 2-11

$$M_{uy} = M_{yy} + \left| M_{xy} \right|$$
Eq. 2-12

However, if either **M_{ux}** or **M_{uy}** is found to be negative, the negative value of the moment is changed to zero and the other moment is given as follows:

$$\text{if } M_{ux} < 0, \text{ then } M_{ux} = 0 \text{ and } M_{uy} = M_{yy} + \left| \frac{M_{xy}^2}{M_{xx}} \right|$$
Eq. 2-13

$$\text{if } M_{uy} < 0, \text{ then } M_{uy} = 0 \text{ and } M_{ux} = M_{xx} + \left| \frac{M_{xy}^2}{M_{yy}} \right|$$
Eq. 2-14

- For bottom reinforcement where negative moments produce tension:

---

3. References [1], [2], [3], and [4]

$$M_{ux} = M_{xx} - \left| M_{xy} \right| \qquad\qquad \text{Eq. 2-15}$$

$$M_{uy} = M_{yy} - \left| M_{xy} \right| \qquad\qquad \text{Eq. 2-16}$$

However, if either $\mathbf{M_{ux}}$ or $\mathbf{M_{uy}}$ is found to be positive, the positive value of the moment is changed to zero and the other moment is given as follows:

$$\text{if } M_{ux} > 0, \text{ then } M_{ux} = 0 \text{ and } M_{uy} = M_{yy} - \left| \frac{M_{xy}^2}{M_{xx}} \right| \qquad\qquad \text{Eq. 2-17}$$

$$\text{if } M_{uy} > 0, \text{ then } M_{uy} = 0, \text{ and } M_{ux} = M_{xx} - \left| \frac{M_{xy}^2}{M_{yy}} \right| \qquad\qquad \text{Eq. 2-18}$$

Flexural Reinforcement

The Program reports the area of flexural reinforcement per unit width [in^2/ft (US Customary Units) or mm^2/m (Metric Units)]. The total area of reinforcement in an element, then, can be obtained by multiplying the reported area of reinforcement by the width of the element.

$\mathbf{A_{sx}}$ reinforcement is placed along X-direction and calculated based on the greater of the design moment, $\mathbf{M_{ux}}$ or the minimum reinforcement ratio specified by the user under Design Parameters Input Menu.

Similarly, $\mathbf{A_{sy}}$ reinforcement is placed along Y-direction and calculated based on the greater of the design moment, $\mathbf{M_{uy}}$ or the minimum reinforcement ratio specified by the user under Design Parameters Input Menu.

For computation of the required flexural reinforcement, the Program offers two options under Solve Options. These are:

- Compute required reinforcement based on **maximum moment** within an element.

- Compute required reinforcement based on **average moment** within an element.

The assumptions in determination of required flexural reinforcement are based on the design moment conform to the design specifications based on the accepted Strength Design Method and Unified Design Provisions. These are:

1. The reinforcement is computed based on a rectangular section with no compression reinforcement and one layer of tension reinforcement.

2. The maximum usable strain at the extreme concrete compression fiber is 0.003 for ACI standards and 0.0035 for CSA standards.

3. The rectangular concrete stress block is assumed with the block depth equal to:

$$a = \beta_1 c \qquad\qquad \text{Eq. 2-19}$$

where c is the distance from the extreme compression fiber to the neutral axis and factor β_1 equals

$$0.65 \leq \beta_1 = 1.05 - 0.05 f_c' \leq 0.85 \qquad\qquad \text{Eq. 2-20}$$

for ACI standards and

$$0.67 \leq \beta_1 = 1.05 - 0.025 f_c' \qquad\qquad \text{Eq. 2-21}$$

for CSA standards

4. To compute the stress in the steel layer, the elastic-perfectly plastic stress strain distribution is used. The required area of reinforcing steel is calculated as:

$$A_s = \rho\, b\, d \qquad\qquad \text{Eq. 2-22}$$

which reinforcement ratio, ρ, equal to

$$\rho = g(1 - \sqrt{1 - m}) \qquad\qquad \text{Eq. 2-23}$$

where factors **m** and **g** are calculated for ACI standards as

$$m = \frac{2M_u}{0.85\, \phi\, f_c'\, bd^2} \qquad\qquad \text{Eq. 2-24}$$

$$g = 0.85\frac{f_c'}{f_y} \qquad\qquad \text{Eq. 2-25}$$

with strength reduction factor $\varphi = \mathbf{0.9}$ for tension-controlled sections[4]. For CSA standards, factors **m** and **g** are equal to

$$m = \frac{2M_f}{\alpha_1 f_c' bd^2 \phi_c} \qquad\qquad \text{Eq. 2-26}$$

$$g = \alpha_1 \frac{\phi_c f_c'}{\phi_s f_y} \qquad\qquad \text{Eq. 2-27}$$

where $\boldsymbol{\alpha_1}$ is defined as

$$0.67 \le \alpha_1 = 0.85 - 0.0015 f_c' \qquad\qquad \text{Eq. 2-28}$$

and steel resistance factor[5] $\boldsymbol{\phi_s} = \mathbf{0.85}$ and concrete resistance factor[6], $\boldsymbol{\phi_c}$, that takes value of $\boldsymbol{\phi_c} = \mathbf{0.60}$ for CSA A23.3-94, $\boldsymbol{\phi_c} = \mathbf{0.65}$ for CSA A23.3-04 and CSA A23.3-14, and $\boldsymbol{\phi_c} = \mathbf{0.70}$ in case of precast concrete for CSA A23.3-04 and CSA A23.3-14 standards.

α1 is the ratio of the average stress in the rectangular compression block to the specified concrete strength. It equals 0.85 for ACI Code[7] and **0.85 − 0.0015f_c'** but not less than 0.67 for the CSA Standard[8].

Maximum Reinforcement

For the ACI Code, the maximum reinforcement ratio is derived from the condition[9] that the net tensile strain at nominal strength is not less than 0.004.

---

4. ACI 318-14, 21.2; ACI 318-11, 9.3.2; ACI 318-08, 9.3.2; ACI 318-05, 9.3.2; ACI 318-02, 9.3.2
5. CSA A23.3-14, 8.4.3; CSA A23.3-04, 8.4.3; CSA A23.3-94, 8.4.3
6. CSA A23.3-14, 8.4.2, 16.1.3; CSA A23.3-04, 8.4.2, 16.1.3; CSA A23.3-94, 8.4.2
7. ACI 318-14, 22.2.2.3; ACI 318-11, 10.2.6; ACI 318-08, 10.2.6, 10.2.7; ACI 318-05, 10.2.6, 10.2.7; ACI 318-02, 10.2.6, 10.2.6
8. CSA A23.3-14, 10.1.1; CSA A23.3-04, 10.1.1; CSA A23.3-94, 10.1.1
9. ACI 318-14, 9.3.3.1; ACI 318-11, 10.3.5; ACI 318-08, 10.3.5; ACI 318-05, 10.3.5; ACI 318-02, 10.3.5 2. CSA A23.3-14, 10.5.2; CSA A23.3-04, 10.5.2; CSA A23.3-94, 10.5.2;

For the CSA Standard, the area of tension reinforcement is such that the neutral axis-to-depth ratio is:

$$\frac{c}{d} < \frac{700}{700 + f_y}$$

Eq. 2-29

When the required area of steel exceeds the maximum allowed by the code, the program provides a warning message during solution stating that steel design of some elements failed.

Minimum Reinforcement

The Program does not automatically check the minimum reinforcement requirement per design code. Instead, the Program allows the user to input minimum reinforcement ratio % per layer under **Design Parameters** Menu.

The minimum amount of reinforcement in each layer is computed as the minimum reinforcement ratio % defined by the user multiplied by the gross area. Since minimum reinforcement area is calculated by the program separately for each of the two layers, the user should provide half of the minimum reinforcement ratio stipulated by design standards for total reinforcement in order to meet the standards requirements.

For ACI, the minimum total reinforcement amount required in foundations is equal to[10] $\mathbf{0.002A_g}$ for steel Grade 40 or 50, $\mathbf{0.0018A_g}$ for steel Grade 60, and $\mathbf{(0.0018 \cdot 60,000/f_y)A_g}$ for reinforcement with yield stress exceeding 60,000 psi.

For CSA standards[11], the minimum reinforcement requirement is equal to $\mathbf{0.002A_g}$.

---

10. ACI 318-14, 13.3.4.4, 8.6.1.1; ACI 318-11, 15.10.4, 7.12.2.1; ACI 318-08, 15.10.4, 7.12.2.1; ACI 31805, 10.5.4, 7.12.2.1; ACI 318-02, 10.5.4, 7.12.2.1
11. CSA A23.3-14, 15.4.1, 10.5.1.2(a), 7.8.1; CSA A23.3-04, 15.4.1, 10.5.1.2(a), 7.8.1; CSA A23.3-94, 10.5.1.2(a), 7.8.1

2.5   Detailing Provisions

2.5.1   Reinforcement Selection

The Minimum Reinforcement Ratio percentage (%) input under **Define | Design Parameters** Menu can be utilized by the user for base reinforcement selection. Once the reinforcement selection is expressed with this input, the Program shall select that amount as a minimum reinforcement in the output.

For example, for 24" thick mat foundation, the user intends to place #5 @ 12" reinforcement Top & Bottom. Note that "Min Reinf. Ratio % input is per layer. With the given data, this input shall be (0.31) * 100 / (12 * 24) = 0.1076 % which will be equal to 0.31 in^2 / ft reinforcement per layer as shown below.
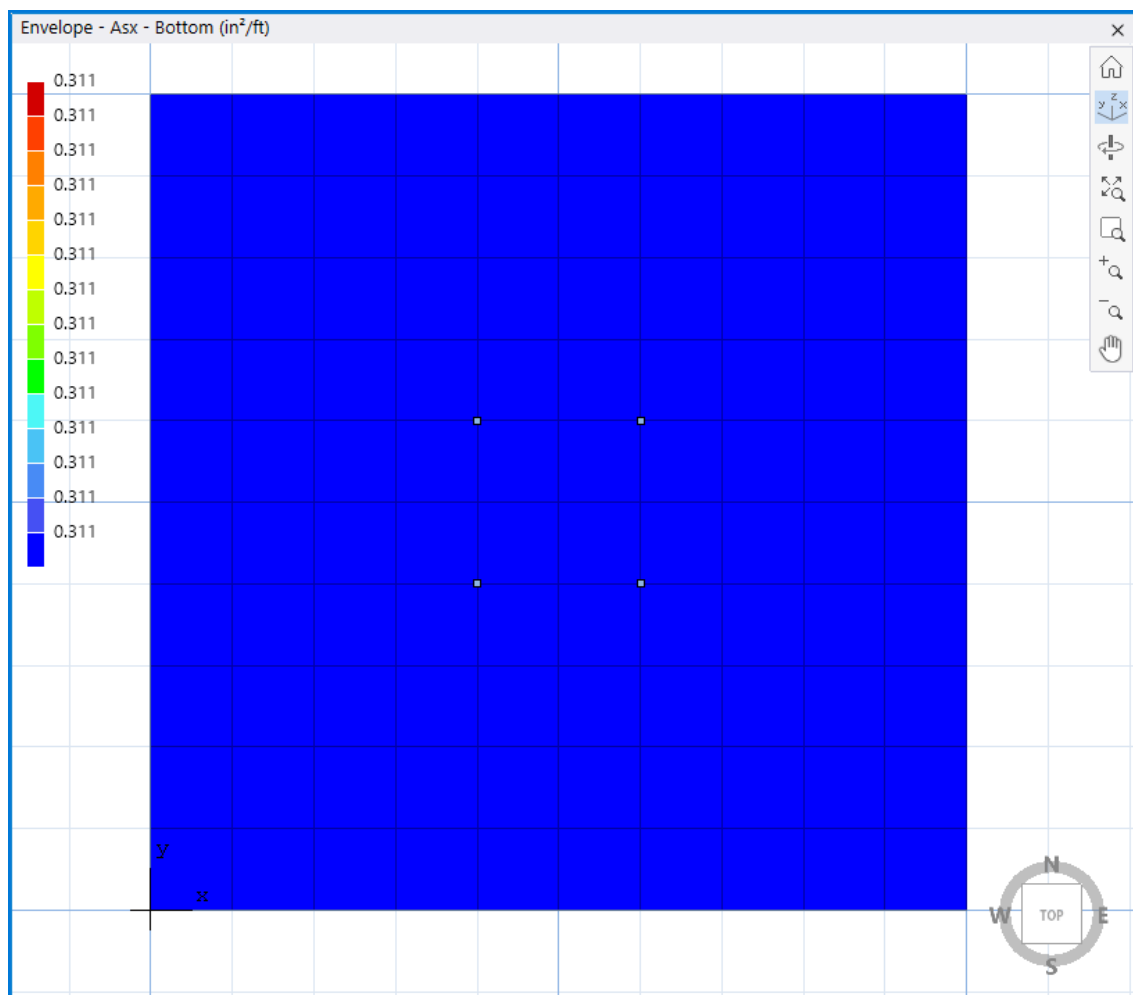
If by analysis, the minimum base reinforcement is insufficient in some parts of the model, this condition shall be displayed as shown below.
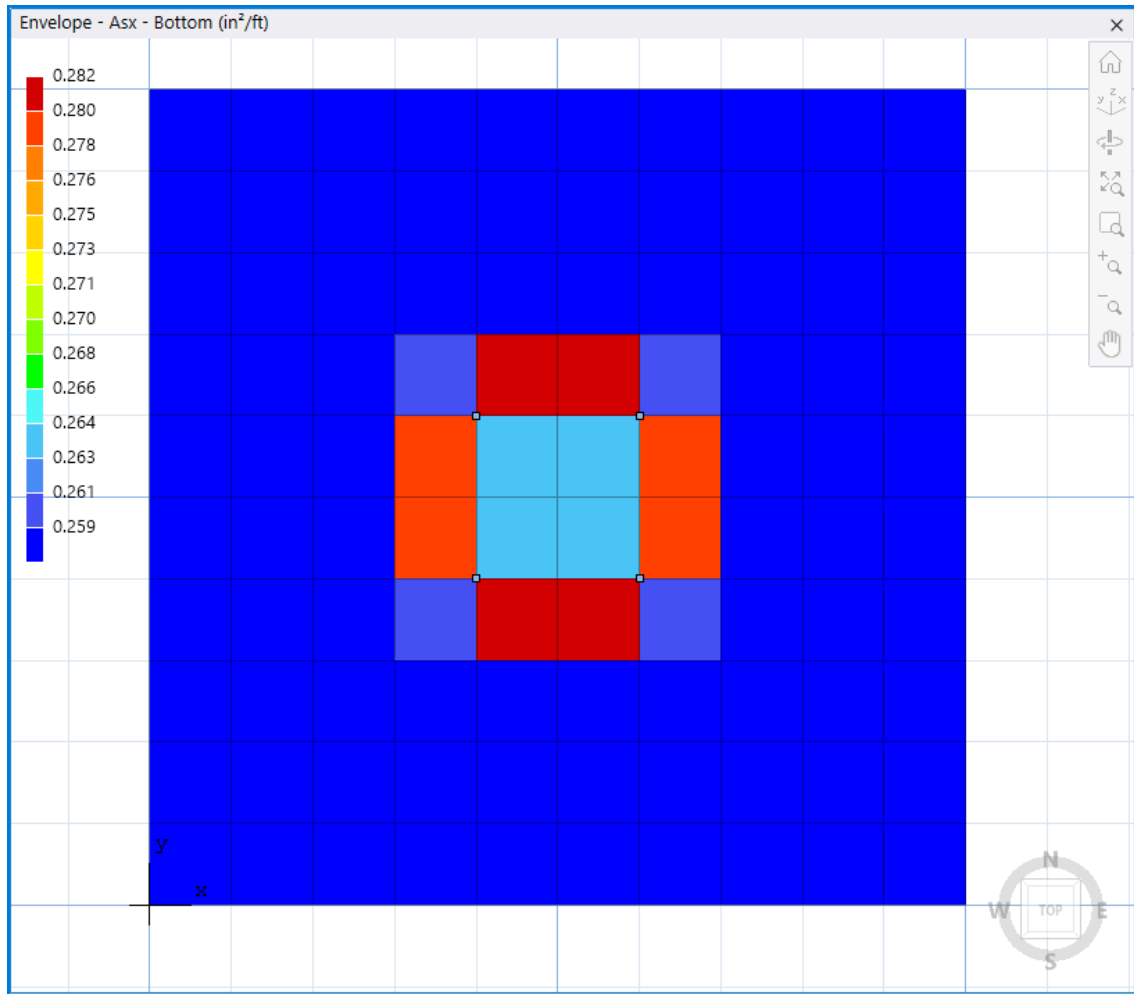
2.6   Special Topics

2.6.1   Single Layer of Reinforcement Modeling

In order to simulate single layer reinforcing, top and bottom reinforcement covers may be inputted such that they are at the same plane per direction. It is important to note that the Min. Reinf. Ratio (% of A_g per layer) is to be kept equal to the design code minimum (i.e. 0.18%) since the governing reinforcement per element will then be selected manually as the maximum of the top and bottom reinforcement per direction. i.e. For X-direction, MAX[A_{sx} Top, A_{sx} Bottom], Y-direction, MAX[A_{sy} Top, A_{sy} Bottom].

The designer may consider entering different cover values for X and Y direction reinforcement to simulate actual rebar placement.

Refer to the Technical Article for additional details: Industrial Floor Slab on Grade with Single Layer of Reinforcement

2.6.2   Fiber Reinforced Slabs on Grade Modeling

Ground supported slabs in industrial and residential floors may be specified with fiber reinforcement in lieu of a single layer of reinforcing or welded wire fabric. Such slabs are referred to as membrane slabs, floating slabs, or filler slabs and range in thickness from as little as 4" to 8" depending on the supported loads. In warehouses and storage facilities such slabs may be subjected to concentrated point loads from storage rack posts or forklift wheel loads.

Refer to the Technical Article for additional details: Fiber Reinforced Industrial Floor Concrete Slabs on Grade

2.6.3   Plain Unreinforced Concrete Slabs on Grade Modeling

Ground supported slabs with light loading and residential flooring are frequently designed without reinforcing (unreinforced). Such plain concrete slabs are referred to as membrane, floating, or filler slabs and range in thickness from as little as 4" to 8" depending on the supported loads. In warehouses and storage facilities such slabs are subjected to concentrated point loads from storage rack posts or forklift wheel loads.

Refer to the Technical Article for additional details: [Plain Unreinforced Concrete Slabs on Grade](#)

2.6.4 Pile Cap Design Considerations

Thick concrete mat with piles are commonly referred as pile caps and require detailed consideration of pile location and spacing. Two technical articles provide additional guidance for completing pile cap models.

[Pile Supported Foundation (Pile Cap) Analysis and Design](#)

[Pile Reaction Distribution in Pile Cap Foundations](#)

2.6.5 Punching Shear Analysis

The punching shear in spMats has been eliminated in version 10.00. A more detailed and comprehensive treatment of punching shear around columns and piles in spMats models is being developed and will be featured in a future release of spMats.

The provisions used for punching shear analysis in spMats v8.50 and prior are provided in this section for reference.

Punching Shear Analysis

The punching shear in spMats is checked where columns in conjunction with concentrated loads and where piles are assigned as supports.

ACI Standard

For ACI standards, the following condition is used:

$$v_u \leq \phi \, v_n \qquad \text{Eq. 2-30}$$

where

v_u = factored shear stress,

v_n = nominal shear resistance of slab,

ϕ = shear resistance factor equal to 0.75.

The nominal shear resistance, v_n, is a sum of nominal shear resistance provided by shear reinforcement, v_s, and nominal shear resistance, v_c, provided by concrete. In spMats, v_s is assumed to be zero and v_c is taken as the smallest of v_{c1}, v_{c2}, and v_{c3}, which are respectively equal to[12]:

$$v_{c1} = \left(2 + \frac{4}{\beta_c}\right) \lambda \sqrt{f'_c}$$ Eq. 2-31

$$v_{c2} = \left(2 + \frac{\alpha_s d}{b_0}\right) \lambda \sqrt{f'_c}$$ Eq. 2-32

$$v_{c3} = 4 \lambda \sqrt{f'_c}$$ Eq. 2-33

with:

β_c = the ratio of long side to short side of the column (or the pile),

α_s = 40 for interior, 30 for edge, and 20 for corner columns or piles,

b_0 = perimeter of the critical section,

d = average effective depth of the critical section segments (less pile embedment, if any),

$\sqrt{f'_c}$ = square root of compressive strength of concrete,

$$\lambda = \begin{cases} 1.0, \; if \; w_c \geq 135 \, pcf \left(2155 \, kg/m^3\right) \\ 0.85, \; if \; 115 \, pcf \left(1840 \, kg/m^3\right) < w_c < 135 \, pcf \left(2155 \, kg/m^3\right) \\ 0.75, \; if \; w_c \leq 115 \, pcf \left(1840 \, kg/m^3\right) \end{cases}$$
  Eq. 2-34

CSA Standard

Similarly, for the CSA standards, factored shear stress, v_f, is checked against factored shear resistance, v_r, which only takes into account concrete shear resistance calculated as the minimum of the following three values:

$$v_{c1} = \left(0.2 + \frac{0.4}{\beta_c}\right) \phi_c \lambda \sqrt{f'_c}$$ Eq. 2-35

$$v_{c2} = \left(0.2 + \frac{\alpha_s d}{b_0}\right) \phi_c \lambda \sqrt{f'_c}$$ Eq. 2-36

---

12. ACI 318-14, 22.6.5.2, 22.6.5.3; ACI 318-11, 11.11.2.1; ACI 318-08, 11.11.2.1; ACI 318-05, 11.12.2.1; ACI 318-02, 11.12.2.1

$$v_{c3} = 0.4\,\phi_c\,\lambda\,\sqrt{f'_c}$$
Eq. 2-37

for the CSA A23.3-94 Standard[13], and

$$v_{c1} = \left(0.19 + \frac{0.38}{\beta_c}\right)\phi_c\,\lambda\,\sqrt{f'_c}$$
Eq. 2-38

$$v_{c2} = \left(0.19 + \frac{\alpha_s d}{b_0}\right)\phi_c\,\lambda\,\sqrt{f'_c}$$
Eq. 2-39

$$v_{c3} = 0.38\,\phi_c\,\lambda\,\sqrt{f'_c}$$
Eq. 2-40

for the CSA A23.3-04 and CSA A23.3-14 standards[14]. Factor α_s equals 4 for interior columns, 3 for edge columns, and 2 for corner columns for all CSA standards. Factor λ accounts for low density concrete and is equal to:

$$\lambda = \begin{cases} 1.0, \ if \ w_c \geq 2150\,kg\,/\,m^3\,(134.2\,pcf) \\ 0.85, \ if \ 1850\,kg\,/\,m^3\,(115.5\,pcf) < w_c < 2150\,kg\,/\,m^3\,(134.2\,pcf) \\ 0.75, \ if \ w_c \leq 1850\,kg\,/\,m^3\,(115.5\,pcf) \end{cases}$$
Eq. 2-41

Also, for interior column and piles, the value of effective depth, **d**, in Eq. 2-38 through 2-40 will be multiplied[15] by factor **1300/(1000+d)**.

---

13. CSA A23.3-94, 13.4.4
14. CSA A23.3-14, 13.3.4; CSA A23.3-04, 13.3.4
15. CSA A23.3-14, 13.3.4.3; CSA A23.3-04, 13.3.4.3

CHAPTER
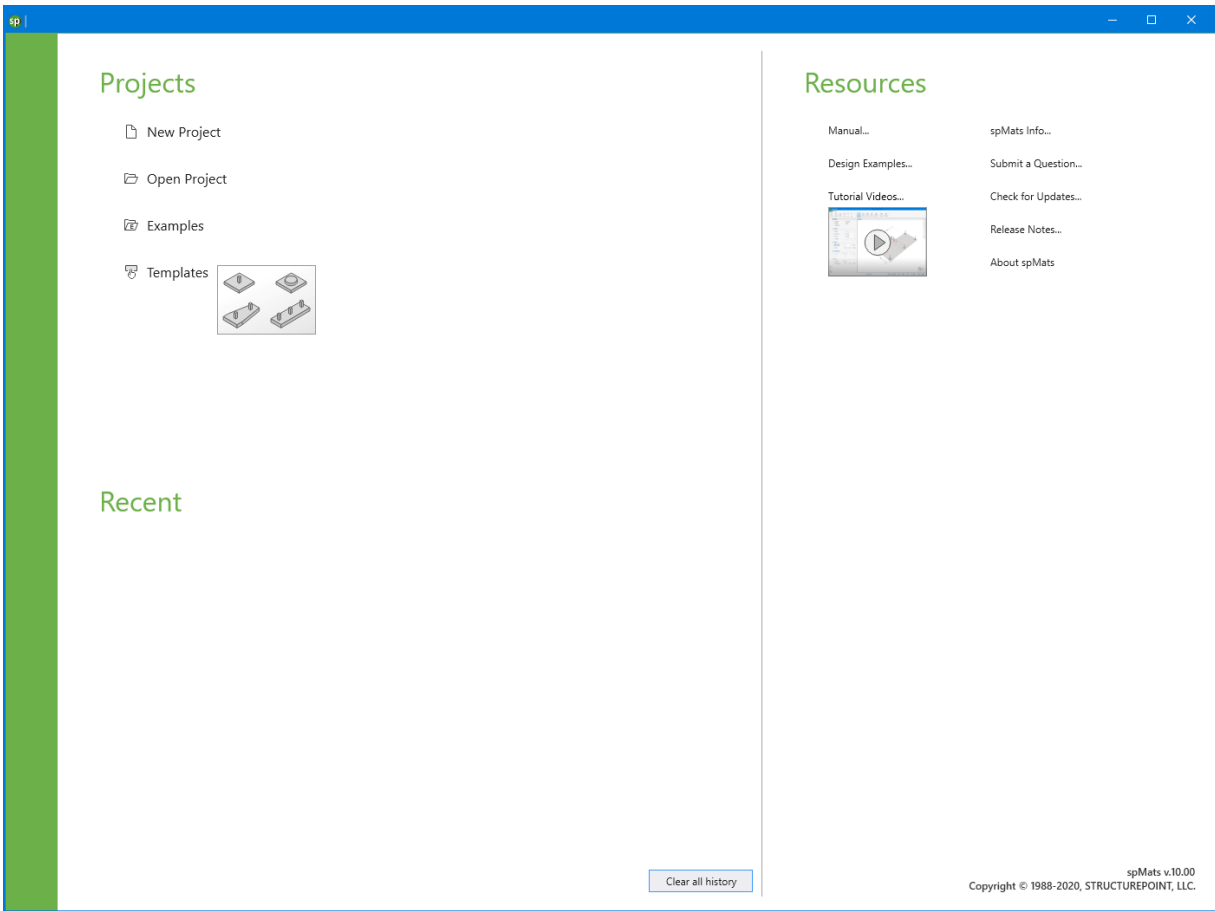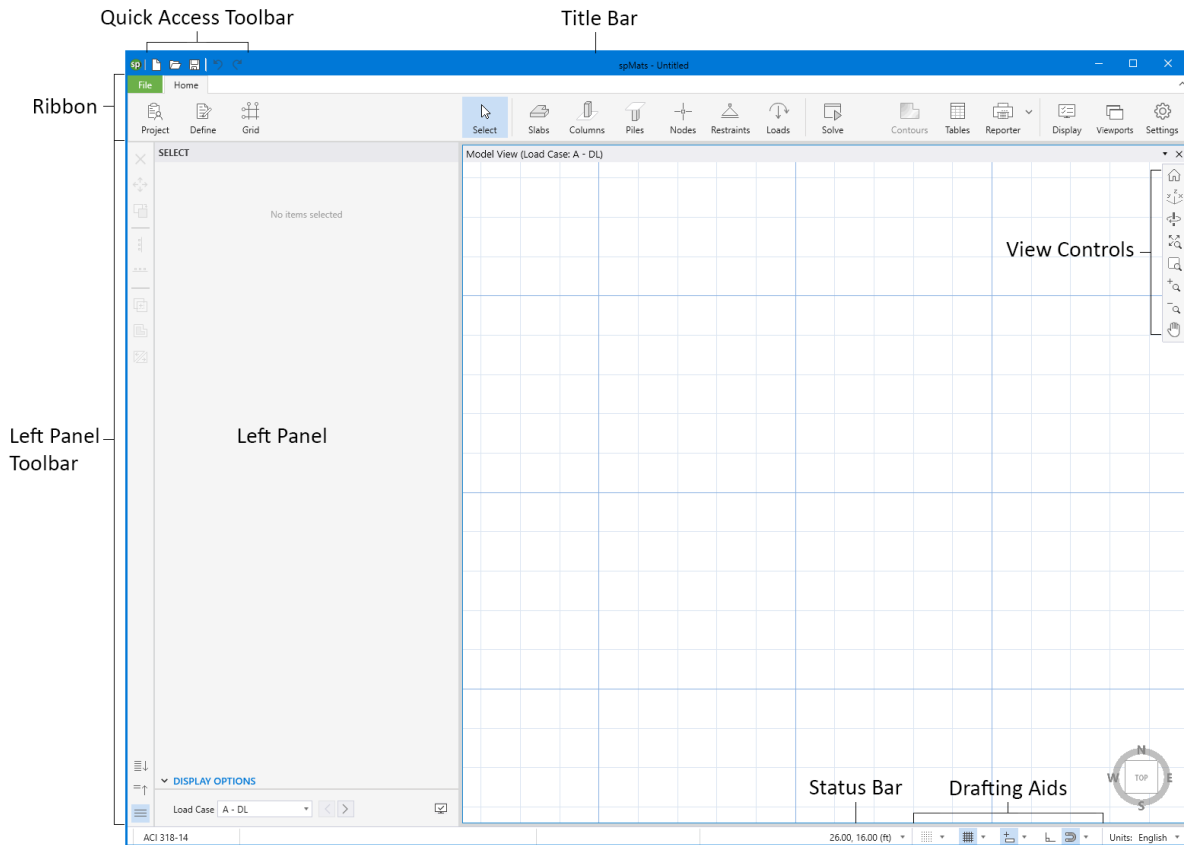
3

PROGRAM INTERFACE

3.1   Start Screen

When the Program is launched, a start screen appears as shown below. The Start Screen consists of options to start **New Project**, **Open** existing **Project**, open **Examples** folder, open **Templates**, links to available program **Resources** and a list of **Recent** files. The program name and copyright information are located in the bottom right of the start screen.

Projects

- New Project
- Open Project
- Examples
- Templates

Recent

Clear all history

Resources

Manual...   spMats Info...

Design Examples...   Submit a Question...

Tutorial Videos...   Check for Updates...

                             Release Notes...

                             About spMats

spMats v.10.00
Copyright © 1988-2020, STRUCTUREPOINT, LLC.

3.2   Main Program Window

The **Main Program Window** shown above consists of the following:

Quick Access Toolbar

The **Quick Access Toolbar** includes **New**, **Open**, **Save** and **Undo** and **Redo** commands.

Title Bar

The Title Bar displays the name of the program, along with the filename of the current data file in use. If the file is new and has not yet been saved, the word "Untitled" is displayed in the **Title Bar**. It also displays "(Modified)" if the file has been changed and not saved yet.

Ribbon

The **Ribbon** consists **File** and **Home** tabs.

File Tab consists of commands to go **Back** to **Home** Tab, create **New** file, **Open** an existing file, **Save** a file, **Save as**, **Import**, **Export** and **Exit**. In addition, the entire **Start Screen** is present under the **File** Tab.

Home Tab gives quick access to commands which are needed to complete the task of creating a model, executing it and analyzing solutions. These commands are:

Project – enables to enter DESIGN CODE, UNIT SYSTEM, and PROJECT DESCRIPTION.

Define – enables to define **Slab**, **Column** and **Piles** objects; **Soil**, **Concrete**, **Reinforcement**, and **Design Parameters** properties; **Nodal Springs** and **Slaved Nodes** restraints; **Load Cases**, **Service** and **Ultimate Load Combinations**.

Grid – enables to add new or edit existing grids

Select – enables to select various model items.

Slabs – enables to create Rectangular, Circular or Polygon shaped slabs.

Columns – enables to add columns into the model. Columns can also be defined through the left panel.

Piles – enables to add piles into the model. Piles can also be defined through the left panel.

Nodes – enables to create a single node or multiple nodes in a linear or circular pattern.

Restraints – enables to assign nodal springs, slaved nodes, and D_z, R_x, and R_y restraints to the model.

Loads – enables to assign area and point loads to the model.

Solve – enables to specify solver options, mesh generation options, and solve the model.

Contours – after a successful run, enables to view graphical results such as design moments, required reinforcements, soil pressures, and displacements.

Tables – enables to open Tables Module to view tabular input and output.

Reporter – enables to open Reporter Module to view the Report.

Display – enables to toggle on/off model items.

Viewports – enables to select from a predefined viewport configuration.

Settings – enables to modify various Program settings.

Left Panel

The properties of active commands under **Home** Tab or the properties of items selected in the **Viewport** are displayed in the **Left Panel** which can then be used to execute the commands or edit

the selected items. After execution **Left Panel** also displays various solutions which can be explored using the **Viewport**.

Left Panel Toolbar

The **Left Panel Toolbar** contains commands that can be used to edit various items in the **Viewport**.

Viewport

The **Viewport** covers the majority of the main program window. It is the space where models can be created and graphical results can be viewed. Up to 6 **Viewports** can be used at once.

View Controls

The **View Controls** contains various commands which can be used to adjust the views of **Viewport** both during modeling or viewing the graphical results.
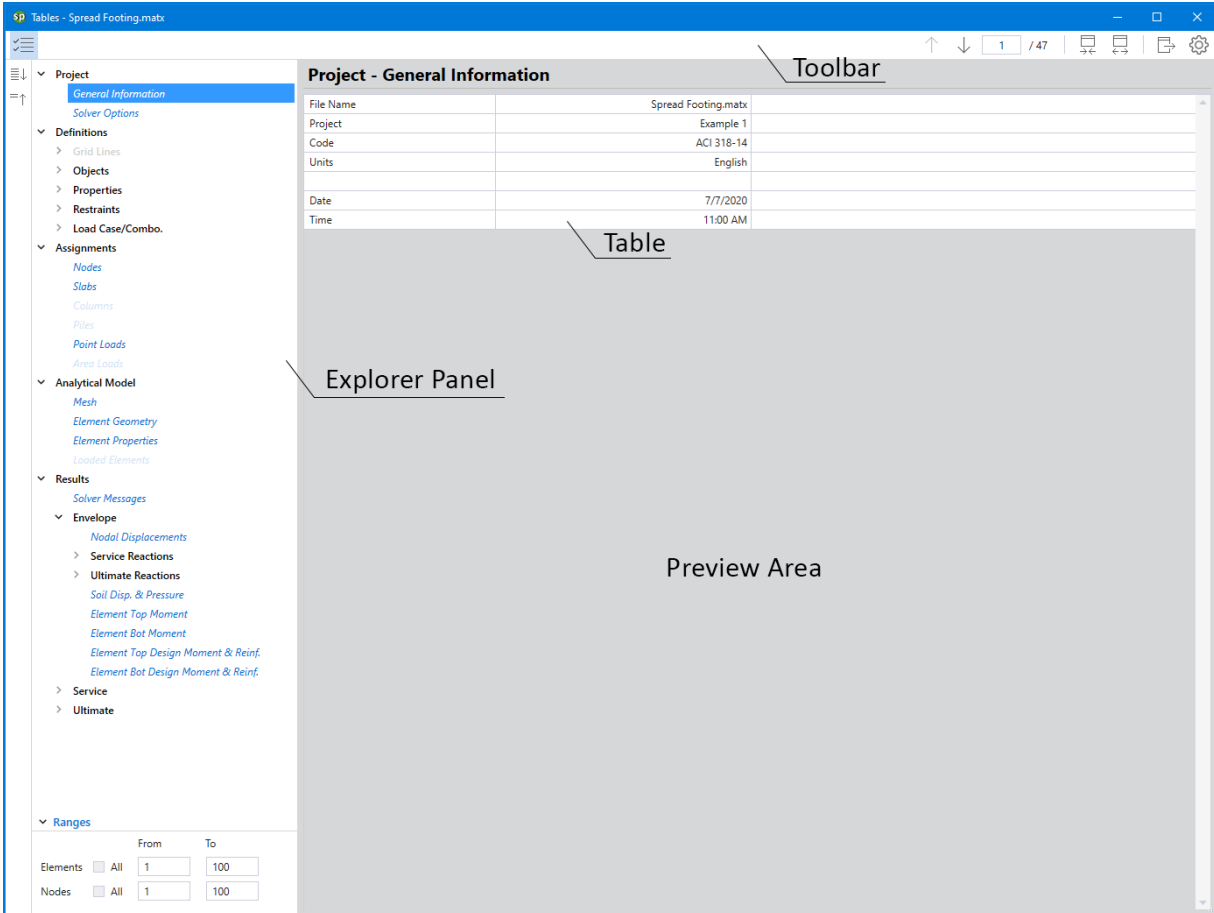
Drafting Aids

The **Drafting Aids** provides access to various **viewport** and **grid** parameters and commands which can facilitate in drafting a model.

Status Bar

The **Status Bar** displays important information such as the design code being used, cursor position and current units. It also houses the drafting aid commands.

3.3 Tables Window

The **Tables Module** interface shown above enables the user to view program inputs and outputs in tables and export them in different formats.

The **Tables Module** is accessed from within the Main Program Window by clicking the **Tables** button from the **Ribbon**. Alternatively, **Tables Module** can also be accessed by pressing the F6 key. If the model has not been executed yet, then the **Tables Module** will only contain a list of input data tables. When a model has been successfully executed, the **Tables Module** will also display the output data tables.

Toolbar

The **Toolbar** contains commands which can be used to navigate through various Tables

Previous table

  Displays the previous table.

Next table

Displays the next table.

Table number box

Displays the table with the table number entered in the box.

Auto fit column width to view area

When toggled on always fits the width of table to the **Preview Area** width.
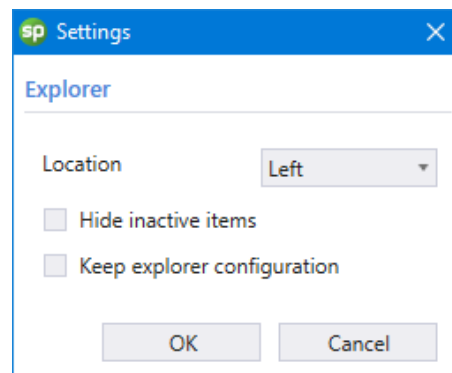
Maintain maximum column width

Restores all table columns to their default maximum width.

Export current table

Exports the table being viewed in the selected format.

Settings

Contains settings for the **Explorer Panel**.

- LOCATION - displays **Explorer Panel** on the left or right side of screen depending on selection.

- HIDE INACTIVE ITEMS - hides unused tables from the explorer view.

- KEEP EXPLORER CONFIGURATION - saves the explorer configuration i.e. information about selected tables and opened/closed sections so that it is available the next time user opens Tables Module.

Explorer

Shows or hides the **Explorer Panel**.

Explorer Panel

The **Explorer Panel** consists of all the available items of the inputs and results classified into sections and arranged hierarchically. Any item in the **Explorer Panel** can be clicked on to display the corresponding table in the **Preview Area**.

Expand all

Expands item list.

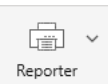**Collapse all**

Collapses item list.

Ranges

The RANGES feature allows you to explore the results for a selected range of ELEMENTS and/or NODES. To view element and nodal results for the entire model simply check the ALL checkbox. Alternatively, to view element and nodal results within a range, make sure that the ALL checkbox is unchecked and enter the desired range in the FROM and TO text boxes.

3.4   Reporter Window
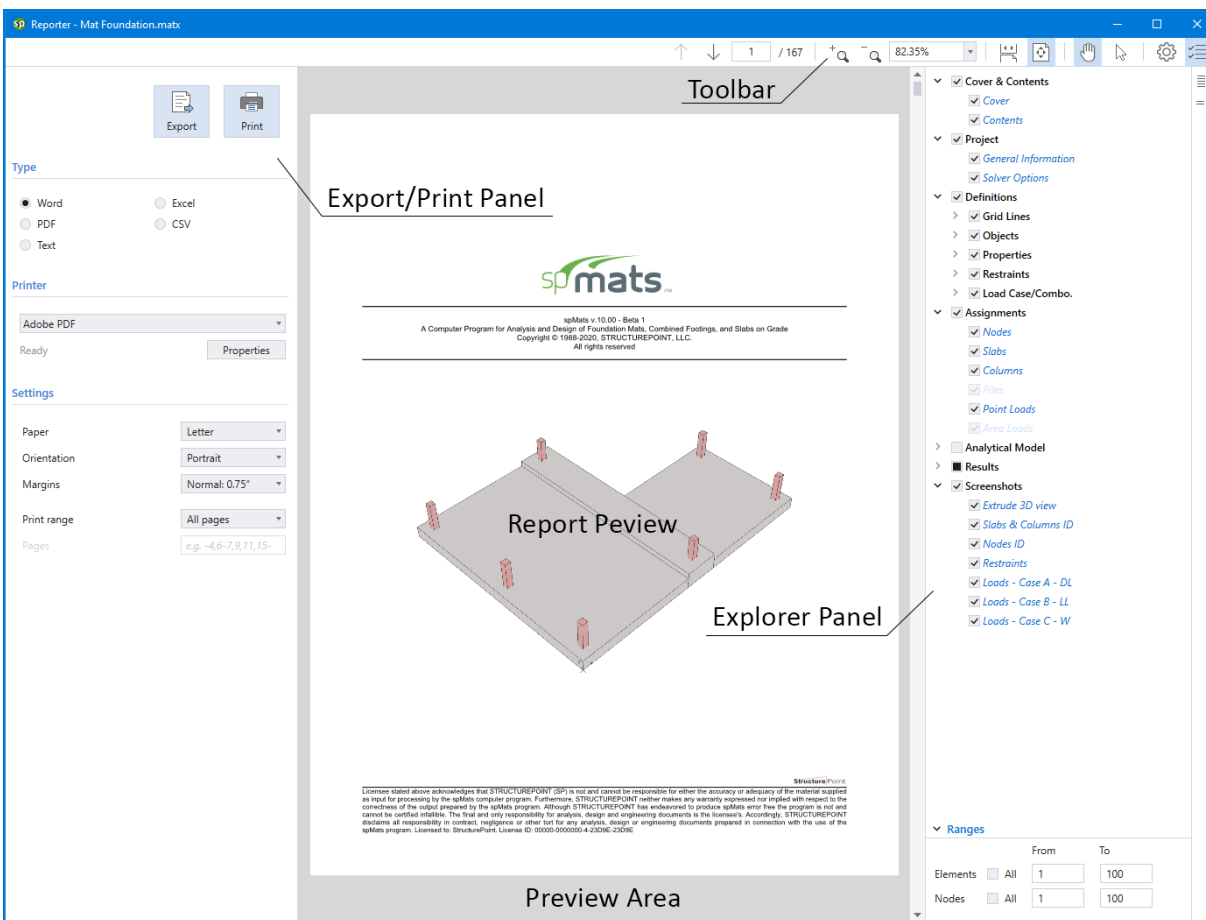
The **Reporter Module** interface shown above enables the user to view, customize, print and export reports in different formats.

The **Reporter Module** is accessed from within the Main Program Window by clicking the

Reporter button [Reporter] from the **Ribbon**. Alternatively, **Reporter Module** can also be accessed by pressing the F7 key. If the model has not been solved then the **Reporter Module** will only contain a list of input data reports. When a model has been successfully executed, the **Reporter Module** will also display the output data reports. Immediately after opening the **Reporter Module**, you can export and/or print the default report by pressing **Export/Print** button. Various options to customize the report before printing and/or exporting it are also provided. Once the work in **Reporter Module** is complete click the close button in the top right corner to exit **Tables** window.

Previous page

Displays the previous page of the report.

Next page

Displays the next page of the report.

Page number box

Displays the page with the page number entered in the box.

Zoom in

Zooms in on the report (Ctrl + Mouse wheel up).

Zoom out

Zooms out on the report (Ctrl + Mouse wheel down).

Zoom box

Zooms on the report preview to the extent typed in the box or selected from the dropdown list.

Fit to window width and enable scrolling

Fits the width of report to the preview space width and enables scrolling.

Fit one full page to window

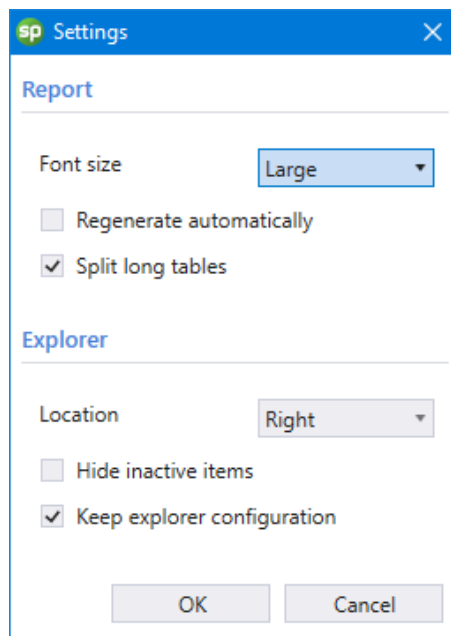Fits one full page in the preview space.

Pan

When toggled on and report if bigger than preview window, enables panning the report.

Text selection

When toggled on enables selecting text in the report.

Settings

Modifies settings for report and **Explorer Panel**.

REPORT SETTINGS

- FONT SIZE: Provides the options to use small, medium or large font sizes in the report.

- REGENERATE AUTOMATICALLY: Enables automatic regeneration of report when content selection is modified by the user.

- SPLIT LONG TABLES: Displays table headings in all pages when tables are split along several pages.

EXPLORER SETTINGS

- LOCATION: Displays **Explorer Panel** on the left or right side of screen   depending on selection

- HIDE INACTIVE ITEMS: Hides unused tables from the explorer view

- KEEP EXPLORER CONFIGURATION: Saves the explorer configuration i.e. information about selected tables and opened/closed sections so that it is available the next time user opens **Reporter**.

Explorer

Shows or hides the **Explorer Panel**.

Export / Print panel

EXPORT

Exports the report in the selected format.

PRINT

Prints the report in the selected format when the option is available.

TYPE

Provides 5 format options to print and/or export the reports

- WORD: produces a Microsoft Word file with .docx extension.

- PDF: produces an Adobe Acrobat file with .pdf extension.

- TEXT: produces a Text file with .txt extension.

- EXCEL: produces a Microsoft Excel file with .xlsx extension.

- CSV: produces a Comma Separated file with .csv extension.

PRINTER

Provides the option to select available printers and change printer properties.

SETTINGS

Provides the options to modify print settings.

- PAPER: Provides the options to select from available paper sizes.

- ORIENTATION: Provides the options to select between landscape or portrait paper orientation.

- MARGINS: Provides the options to use narrow, normal, wide or custom margins to the report.
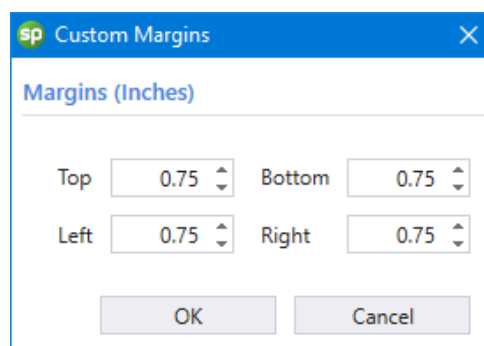
- PRINT RANGE: Provides the options to select the pages to print and/or export.

Explorer Panel

The **Explorer Panel** consists of all the available report items classified into sections and arranged hierarchically. Each item listed in the **Explorer Panel** is preceded by a checkbox. The user can check/uncheck the checkbox to include or exclude from the report, the items or sections.

Expand all

　　　Expands item list.

Collapse all

　　　Collapses item list.

Ranges

The RANGES feature allows you to explore the results for a selected range of ELEMENTS and/or NODES. To view element and nodal results for the entire model simply check the ALL checkbox. Alternatively, to view element and nodal results within a range, make sure that the ALL checkbox is unchecked and enter the desired range in the FROM and TO text boxes.

3.5 Print/Export Window
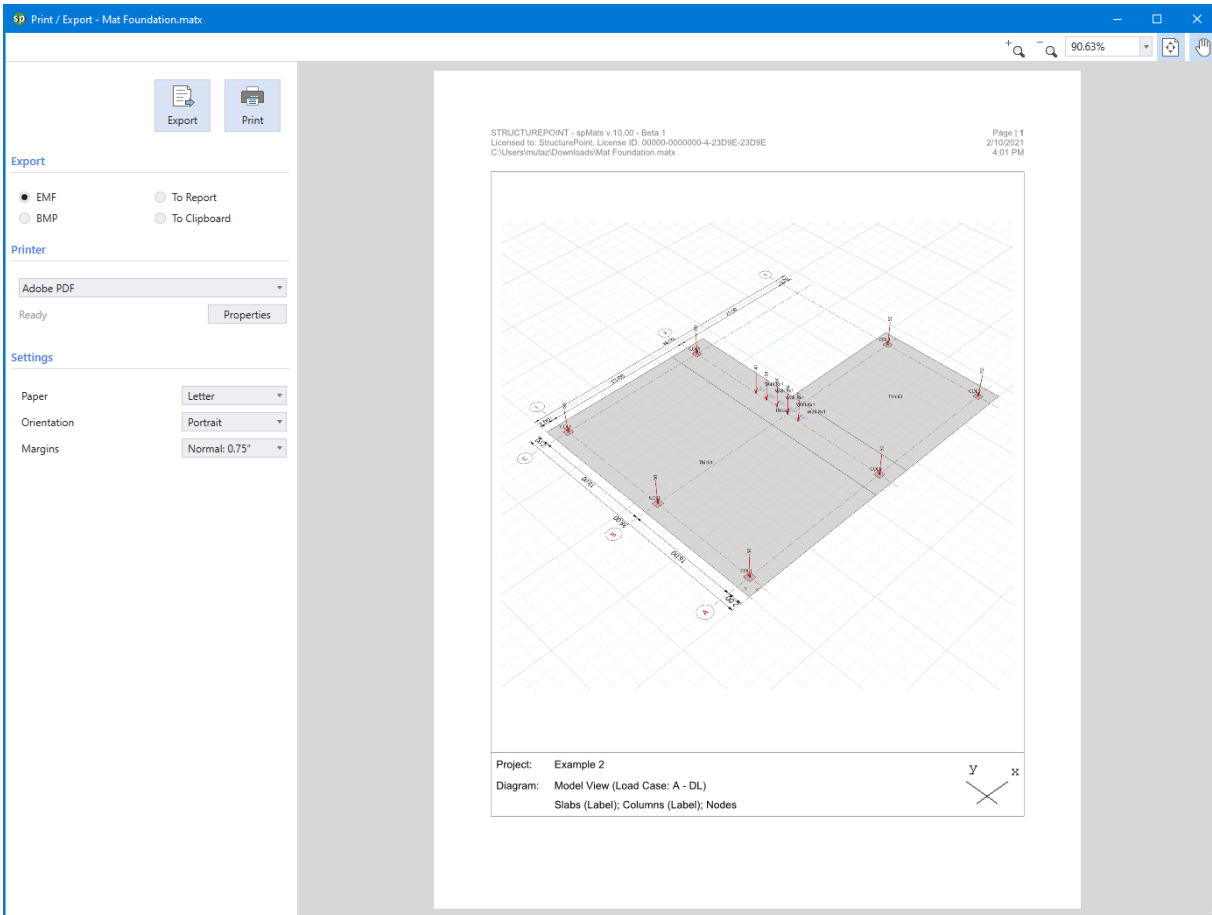
Print/Export Module interface shown above enables the user to view, customize, print and export diagrams in different formats.

The **Print/Export Module** is accessed from within the **Main Program Window** by using the **Right Click Menu** or from the **Reporter Submenu** in the **Ribbon**.
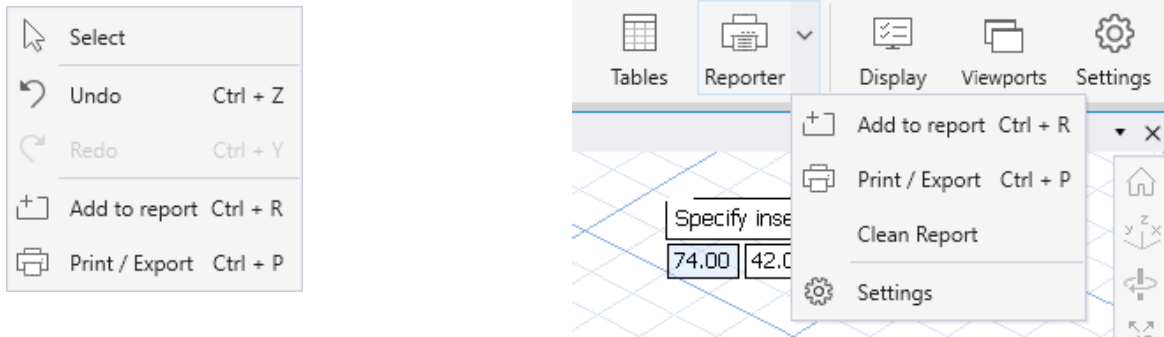
Alternatively pressing the "CTRL + P" also opens the **Print/Export Module**. Once the module is open the rest of the program is locked until the **Print/Export Module** is closed.

Immediately after opening the **Print/Export Module**, you can export and/or print the generated diagram by pressing **Export/Print** button. Options to customize the diagram orientation, paper size and margins are provided. Once the work in **Print/Export Module** is complete click the close button in the top right corner to exit the module.

Toolbar

Zoom in

Zooms in on the report (Ctrl + Mouse wheel up).

Zoom out

Zooms out on the report (Ctrl + Mouse wheel down).

Zoom box

Zooms on the report preview to the extent typed in the box or selected from the dropdown list.

Fit one full page to window

Fits one full page in the preview space.

Pan

When toggled on and report if bigger than preview window, enables panning the report.

Export / Print panel

Export

Exports the report in the selected format.

Print

Prints the displayed diagram.

Type

Provides 4 format options to export the reports

- EMF produces a file with .emf extension

- BMP produces a file with .bmp extension

- TO REPORT adds the diagram to the report

- TO CLIPBOARD copies the diagram to clipboard to be pasted elsewhere

Printer

Provides the option to select available printers and change printer properties.

Settings

Provides the options to modify print settings.

- PAPER: Provides the options to select from available paper sizes.

- ORIENTATION: Provides the options to select between landscape or portrait paper orientation.

- MARGINS: Provides the options to use narrow, normal, wide or custom margins to the report.

CHAPTER

4
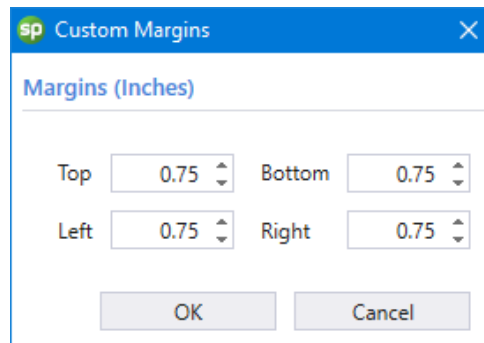
MODELING METHODS

4.1 Model Creation Concepts

The key to successfully implementing spMats in a project is to understand the unique and powerful approach the program takes in modeling, analysis, and designing slabs. This chapter provides an overview of some of the special features and their associated terminology.

Physical Modeling Terminology

In spMats, reference is often made to objects, members, and elements. Objects represent the physical structural members in the model. Elements, on the other hand, refer to the finite elements used internally by spMats to generate the stiffness matrices. In many cases, objects and physical members will have a one-to-one correspondence, and it is these objects that the user draws in the spMats interface. Objects are intended to be an accurate representation of the physical members. Users typically need not concern themselves with the meshing of these objects into the elements required for the mathematical, or analysis finite element model. For example, a single area object can model an entire slab, regardless of the number of spans and variety of loads. With spMats, both model creation, as well as the reporting of results, is achieved at the object level.

This differs from a traditional approach in previous versions of the program, where the user is required to define a sub-assemblage of finite elements that comprise the larger physical members. In spMats, the objects, or physical members drawn by the user, are automatically meshed internally prior to the analysis, into the greater number of finite elements needed for the analysis model, without user input. Because the user is working only with the physical member-based

objects, less time is required both to create the model and interpret the results. The user, however, can dictate several meshing criteria after examination of the automatic mesh proposed by the program.

It is extremely important that you grasp the concept of objects in a structural model as it is the basis for creating models in spMats. After you understand the concept and have worked with it for a while, you should recognize the simplicity of physical object-based modeling, the ease with which you can create models using objects, and the power of the concept when editing and creating complex models.

Structural Objects

The spMats program uses objects to represent physical structural members. When creating a model, start by drawing the geometry using drawing area common CAD tools and then assign properties and loads to completely define the slab structure.

The following object types are available, listed in descending order of geometrical dimension:

- **Slab/Area objects** are used to model slabs, openings, soil supports, and surface loads
- **Column objects** are used to model columns supported on the slab from above
- **Pile objects** are used to model piles supporting the slab from below
- **Node objects** are automatically created at the corners or ends of all other types of objects and also can be added anywhere in the model. Node objects are used to model point loads as well as for applying point restraints and springs.

As a general rule, the geometry of the object should correspond to that of the physical member as much as possible. This simplifies the visualization of the model and reduces the chances of input error. However, engineers can omit small changes in shape and geometry where added model accuracy or complexity is not consequential to the analysis & design results. A great deal of engineering judgment is involved in the conversion of a physical structure into an analytical model. However, significant gains can be achieved by keeping model simple & practical to the extent possible.

Properties

Properties are assigned to each object to define the structural or soil support behavior of that Slab/ Area and/or Pile object in the model. Properties under the **Definitions window**, namely soil, concrete, reinforcement, and design parameters properties, are named entities that must be specified before assigning them to objects. If a property is assigned to an object, for example a design parameters property, any changes to the definition of the property will automatically apply to the slab objects with this property assigned. A named property has no effect on the model unless it is assigned to an object.

Soil subgrade support properties may be assigned to slab/area objects, and for these properties, spMats generates spring elements at each mesh node location.

Input Preparation

The first step in preparing the input is to draw a scaled plan view of the slab. The plan should include the boundaries of the slab, variations in the slab thickness and material properties, openings within the slab, and any variations in the soil sub-grade modulus. All superimposed loads applied on the slab should also be added. Structural grids and drawing area tools should be used to speed input preparation.

The next step is to select suitable mesh criteria including the maximum allowed mesh size, the maximum allowed aspect ratio and the number of segments the circumference of a circular slab (if any) is to be divided in. Based on these parameters and location of columns, piles, slab boundaries and point loads the program automatically creates the most suitable mesh to use for the FEM analysis.

The user can increase or decrease the mesh density by changing the maximum allowed mesh size. A well-graded mesh will produce results which will effectively capture the variations of the displacements and element forces. While the use of finer meshes will generally produce more accurate results, it will also require more solution time, computer memory, and disk space. Elements with aspect ratios (length/width) near unity are generally expected to produce accurate results for regions having gradual changes of curvature. For slab regions where heavy concentrated forces are applied and where drastic changes in geometry exist, the use of finer element meshes may be required. Thus, in order to obtain a practical as well as accurate analytical solution, engineering judgment must be used.

The member nodal incidences are internally computed by the program. All nodes and members are numbered from left to right (in the positive X-direction) and from bottom to top (in the positive Y-direction), as shown in Figure 4-1. When the reference grid system and/or assembling of elements is modified, the program internally renumbers all nodes and elements. In order to save solution time, memory, and disk space, it is recommended to position the side of the slab with fewer nodes (i.e., fewer degrees of freedom) parallel to the X-direction.
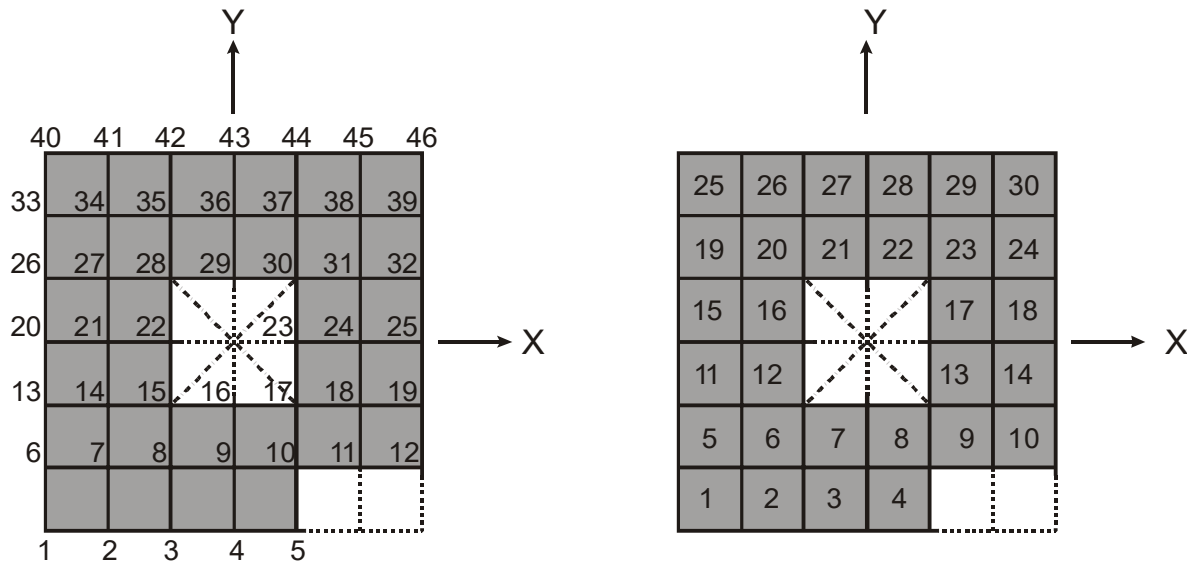
Figure 4-1 Node and Element Numbering in Analytical Model

4.2 Model Editing Concepts

During the course of creating a model, it may be necessary to edit the model. This can be done through the **Select** button in the **Ribbon**. Then, any object that is present on the active **Viewport** can be selected and edited by the tools available within the **Left Panel Toolbar** and **Left Panel**. The editing tools at the **Left Panel Toolbar** can also be invoked by right-clicking the mouse button in the active **Viewport**. The editing tools that are available at the **Left Panel Toolbar** per Objects are:

Slab Object: Slab objects can be deleted, moved, duplicated, merged, offset or split.

Column Object: Column objects can be deleted, moved or duplicated.

Pile Object: Pile objects can be deleted, moved or duplicated.

Node Object: Node objects can be deleted, moved, duplicated, aligned vertically or aligned horizontally.
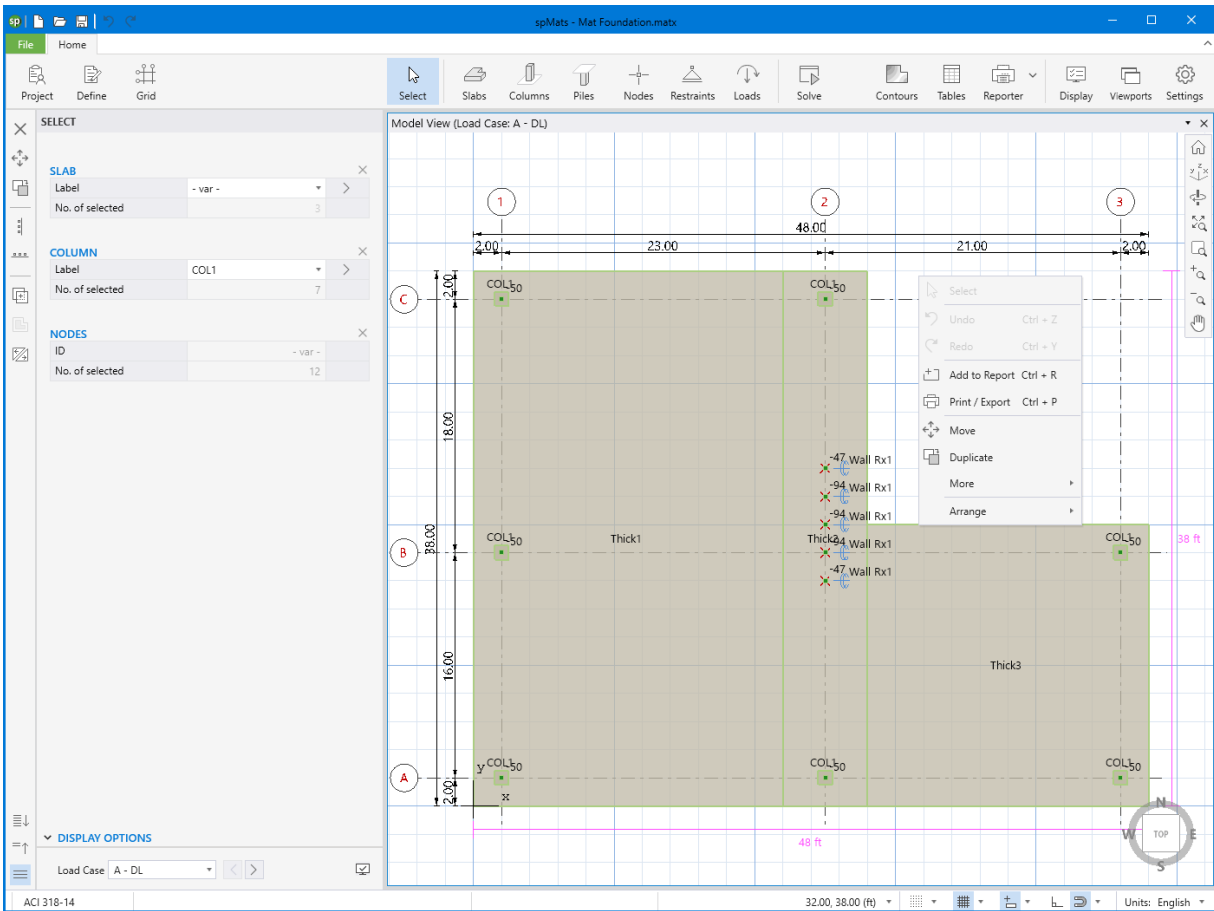
The **Left Panel** can also be utilized to edit an Object further:

Slab Object: In addition to the editing tools provided in **Left Panel Toolbar**, the slab label, section type and thickness of section, properties, position size and area loads can be edited.

Column Object: In addition to the editing tools provided in **Left Panel Toolbar**, the column label, section type, dimension, and location coordinates can be edited.

Pile Object: In addition to the editing tools provided in **Left Panel Toolbar**, the pile label, section type, dimension, properties, and location coordinates can be edited.

Node Object: In addition to the editing tools provided in **Left Panel Toolbar**, the node location coordinates, point loads, and restraints can be edited.

When multiple different object types are selected, all applicable editing tools will be populated at the **Left Panel Toolbar**. The object types can be unselected by clicking the x button next to the Object Type at the **Left Panel**.
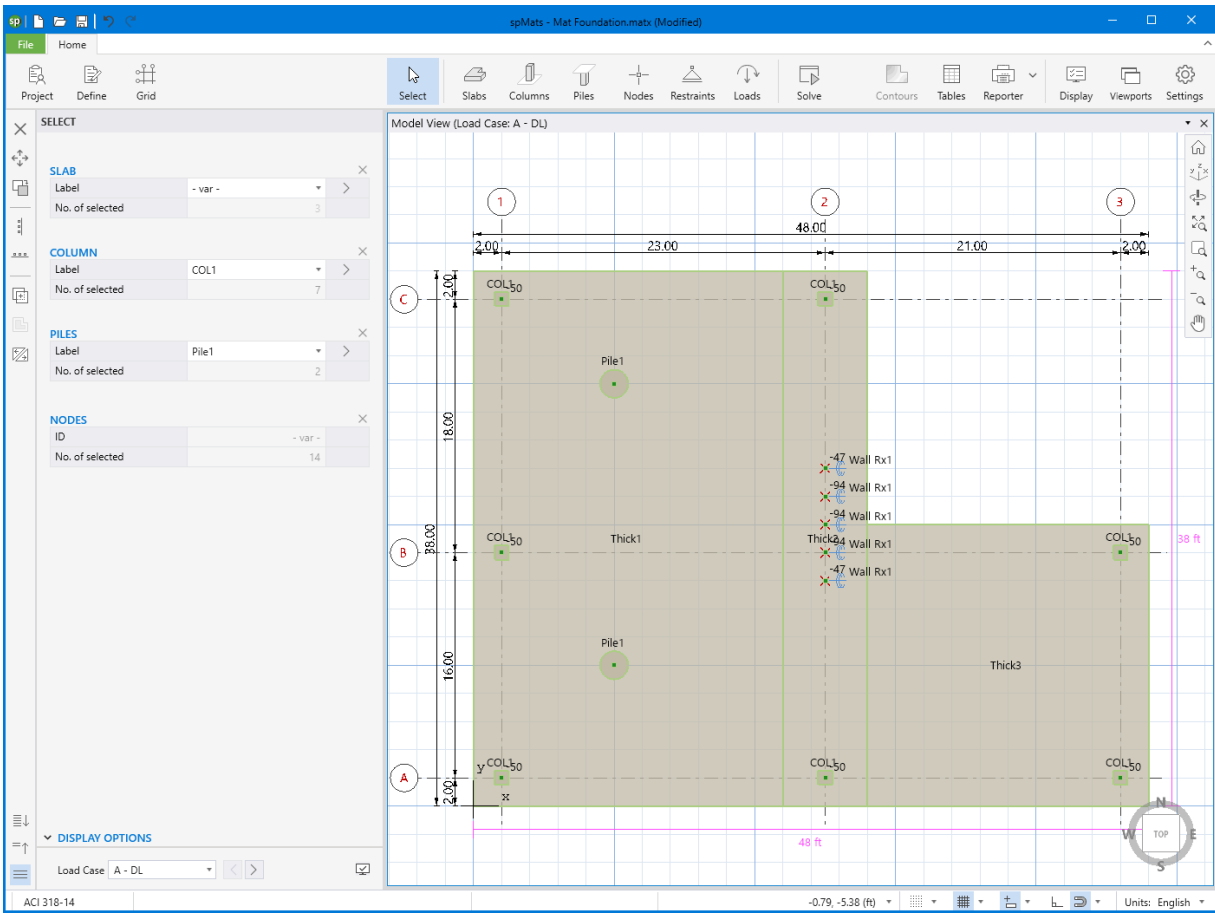
Press any white space in the **Viewport** in order to undo the selection.

CHAPTER

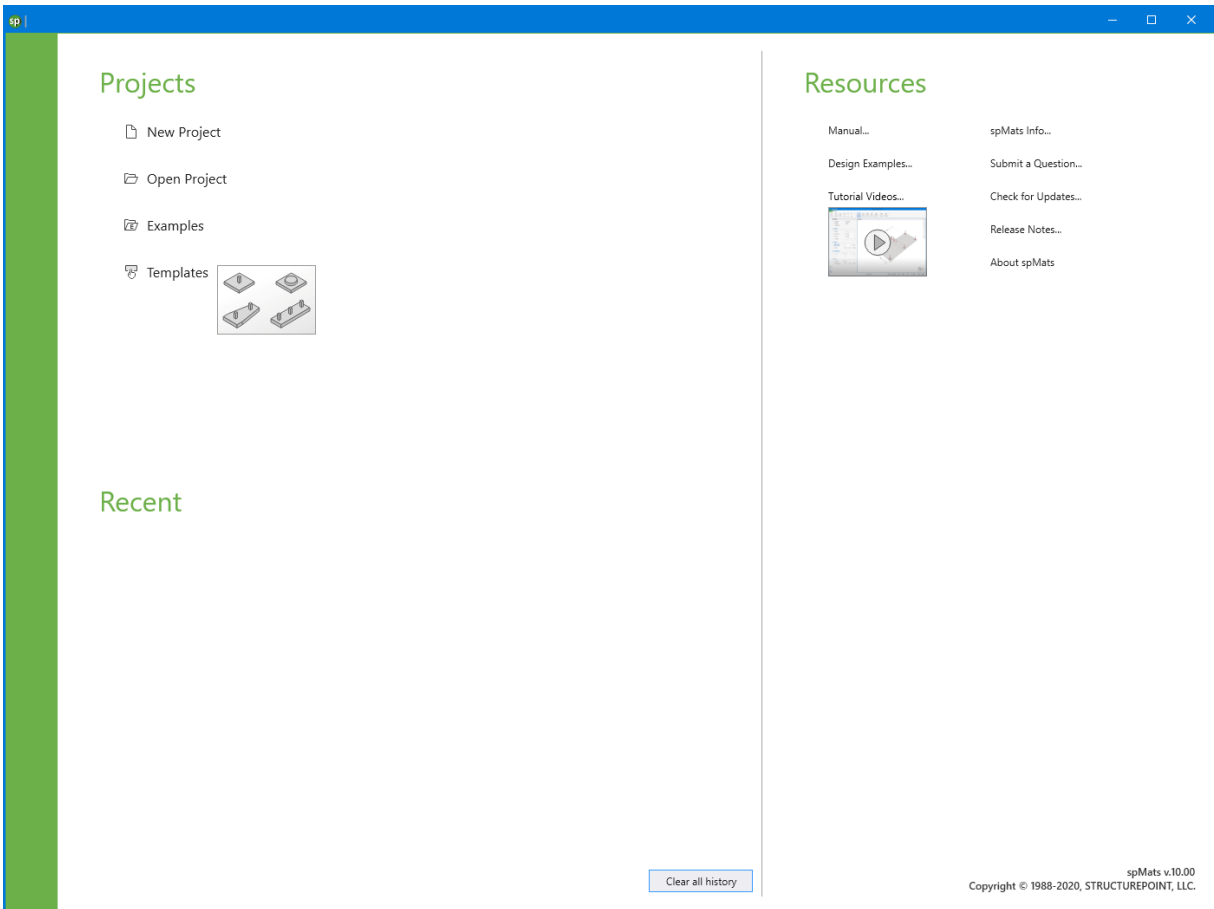# 5

MODEL DEVELOPMENT

5.1   Model Creation

In spMats, models can be started by utilizing one of the four methods under Projects within **Start Screen**. These are namely; **Open Project**, **New Project**, **Templates**, and **Examples**.

5.1.1   Opening Existing Models

In the **Start Screen** under **Projects** select the OPEN PROJECT option and browse to the folder that contains an existing spMats input file. The input files created in spMats v8.xx (.ma8) and in spMats v10 can be opened. The input files for the prior versions of the Program require to be saved in consecutively newer version until .ma8 file is obtained. Then, that ma8 file can be opened in v10. There is no backward compatibility in spMats which means the input files for newer versions of the Program cannot be opened by a previous version.

5.1.2   Creating New Models

In the **Start Screen** under **Projects** select the NEW PROJECT option. The model development process may require general input regarding a specific project. Project Information is entered through the **Project** command button , and Structural Grids are entered through the **Grid** command button .

Project Information

The project information regarding to DESIGN CODE, UNIT SYSTEM, PROJECT NAME, PROJECT DESCRIPTION, PROJECT DATE, and PROJECT TIME can be entered into the model through the PROJECT **Left Panel**. The Program supports American (ACI 318) and Canadian (CSA A23.3) Design Codes, and English and Metric unit systems.

Structural Grids

The structural grids can be created or imported in order to facilitate the input of structural elements such as columns, walls, piles in a specific structural plan view.

Working with Grids

You can select the **Grid** command button from the **ribbon**. The corresponding GRIDS **Left Panel** provides various tools and options for effectively working with grids.

Generating Grids

- Click the **Generate** ⊹ icon to display the GENERATE GRID LINES dialog box

- To create multiple grid lines at once enter:

  number of grid spaces @ grid interval in the GRID SPACING input box

- Entering a single number in the GRID SPACING input box will create a single grid line at the given spacing from either the start coordinates or the last existing grid line in that direction

- You can create grid lines at multiple intervals by separating the grid intervals by space
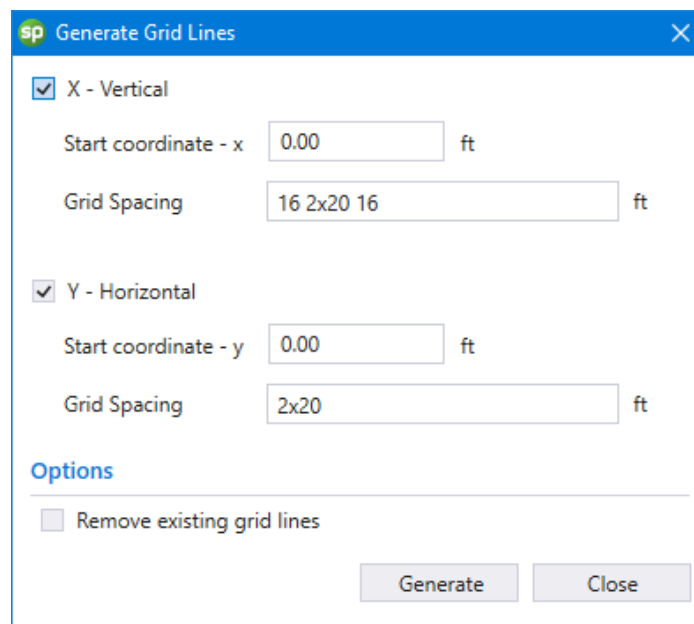
Adding Grids

- Click the **Add-H** ⊙± or **Add-V** ⊕+ commands to add a single HORIZONTAL or VERTICAL grid.

Using the Grid Table

You can use the Grid table to change the LABEL, COORDINATE or SPACING of the grid. The table can also be used to add or delete specific grids.

∨ **VERTICAL**   ↓↑ + ×

Label	Coordinate-X	Spacing
	ft	ft
1	3.00	3.00
2	33.00	30.00
3	63.00	30.00
4	93.00	30.00
6	103.00	10.00

- Click on the + button to add a gridline in the active direction.

- Click on the × button to delete the selected gridline in the table.

- Click on the ↓↑ button to reorder the labels if they have been shuffled.

- To change a grid LABEL, click on the desired grid in the table and click on its label. Leaving a grid label field empty will remove the label bubble from the gridline in the model.

- COORDINATE or SPACING of the grids can also be changed by clicking on the respective field and typing in the desired value.

- Changing the coordinate of the first gridline in the table displaces the entire grid system in the respective direction.

- Except the first gridline in the table, other gridlines cannot be assigned a negative spacing value.

Working with labels

You can use the SETTINGS / GENERAL / GRIDS change the DIRECTION, ORDER & TYPE and LOCATION grid labels. You can also change how far the grid labels are located from the grid lines by changing the EDGE EXTENSION value.
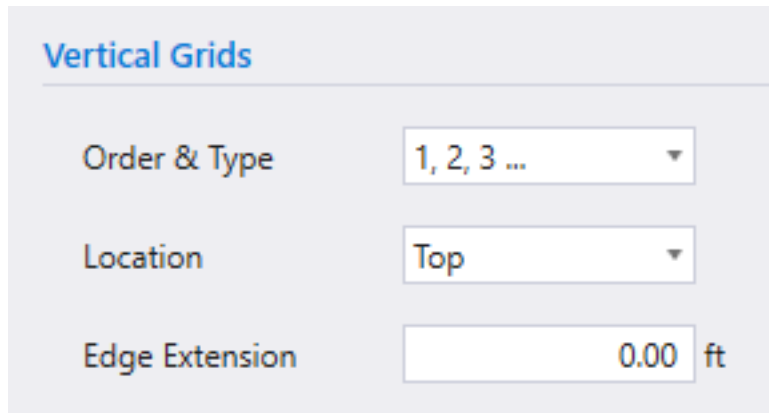
Grid Display Options

- Use the checkboxes to toggle the display of various grid items in the model.

- You can use the slider to adjust the SIZE of the grid LABELS, UNITS and DIMENSIONS displayed.
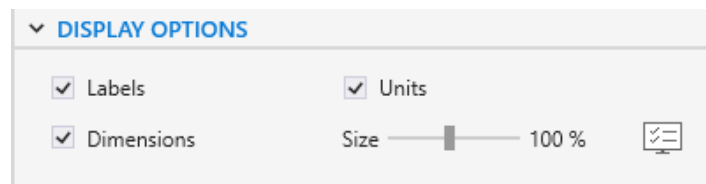
In the **Start Screen** under **Projects** select the **New Project** option. This will take you to the main program window where **Project** command is selected by default. In the **Left Panel**, the DESIGN CODE, UNIT SYSTEM, PROJECT NAME, and PROJECT DESCRIPTION can be entered. Next, select the **Define** command and the DEFINITIONS Dialog Box will appear.
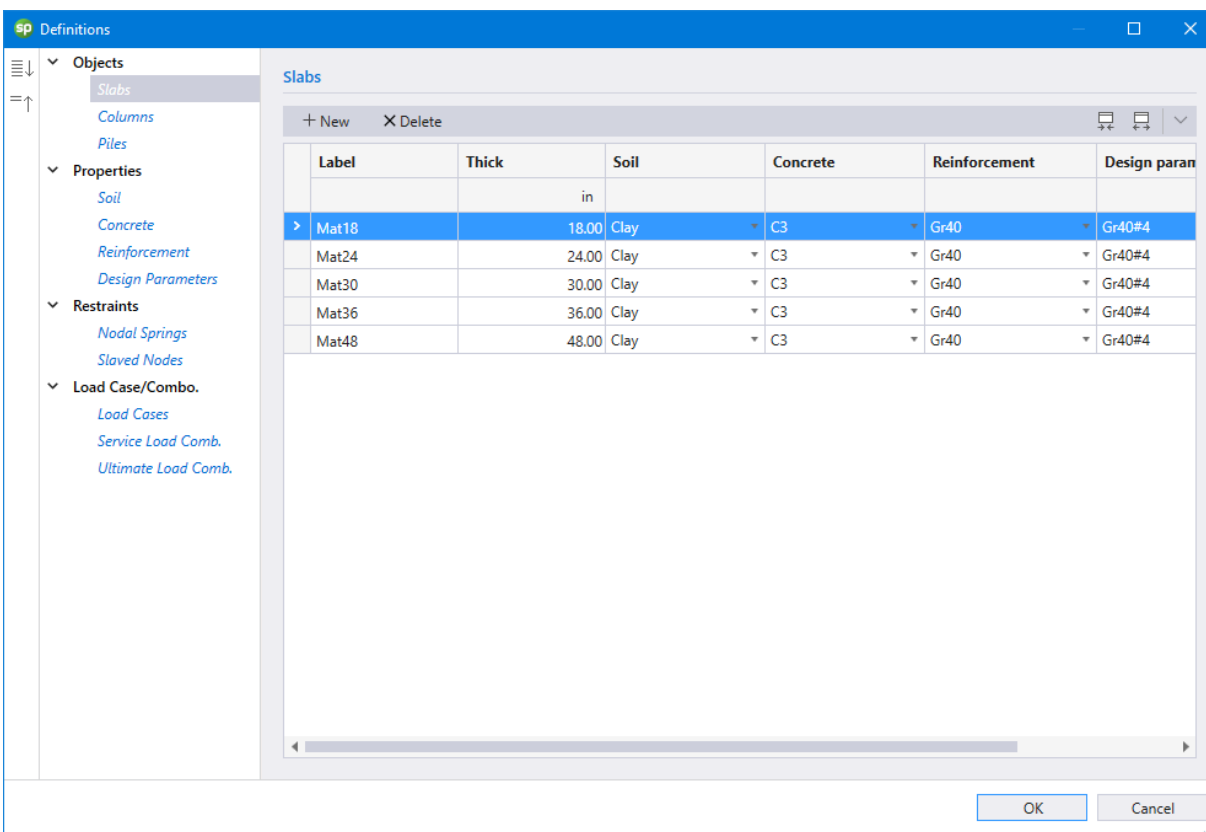
5.1.2.1   Generating Definitions

The DEFINITIONS dialog box contains predefined labels for various OBJECTS, PROPERTIES, RESTRAINTS, and LOAD CASE/COMBO. you can enter additional definitions per your project requirements by clicking the + New button. Similarly, an existing definition can be removed by clicking the × Delete button. either start by creating definitions to be used in the program in the DEFINITIONS dialog box or simply start creating the model with one of the several command buttons available in the **Ribbon** using the default program definitions.

Items like PROPERTIES, RESTRAINTS and LOAD CASES/COMBO are available during modeling only after they have been first defined in the DEFINITIONS dialog box. Objects i.e. slabs, columns and piles can also be defined in the DEFINITIONS dialog box to be used during modeling. Conversely, Object definitions are automatically added to the list of definitions as they are created and used during modeling process.

Clicking the dialog launcher > next to an item in the Left Panel will open the DEFINITIONS dialog box.

Objects

The OBJECTS that can be defined are: SLABS, COLUMNS, and PILES.

Slabs

The SLABS Object definition consists of the LABEL and THICKNESS, the PROPERTIES and whether the Slab label is ASSIGNED to a model or not.
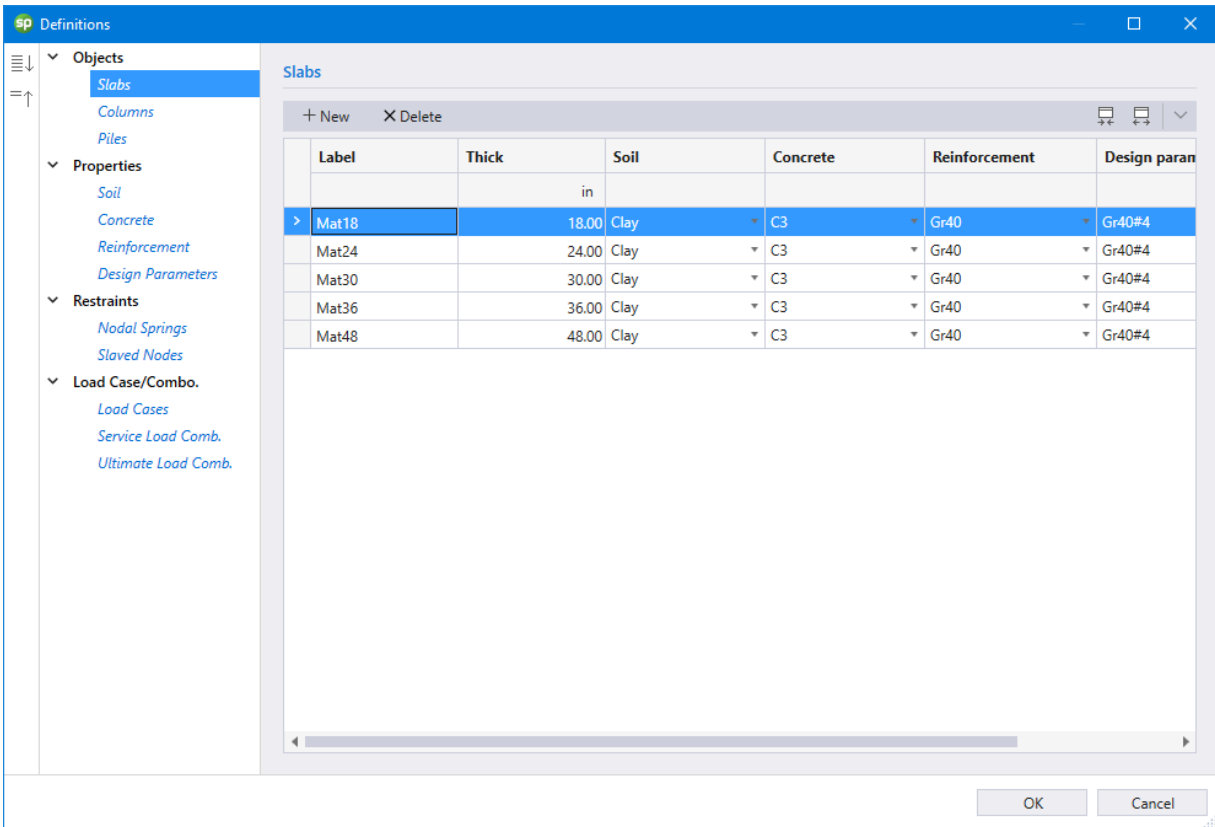
Columns

The COLUMNS Object definition consists of the LABEL, TYPE, DIMENSIONS, and whether the Column label is ASSIGNED to a model or not.
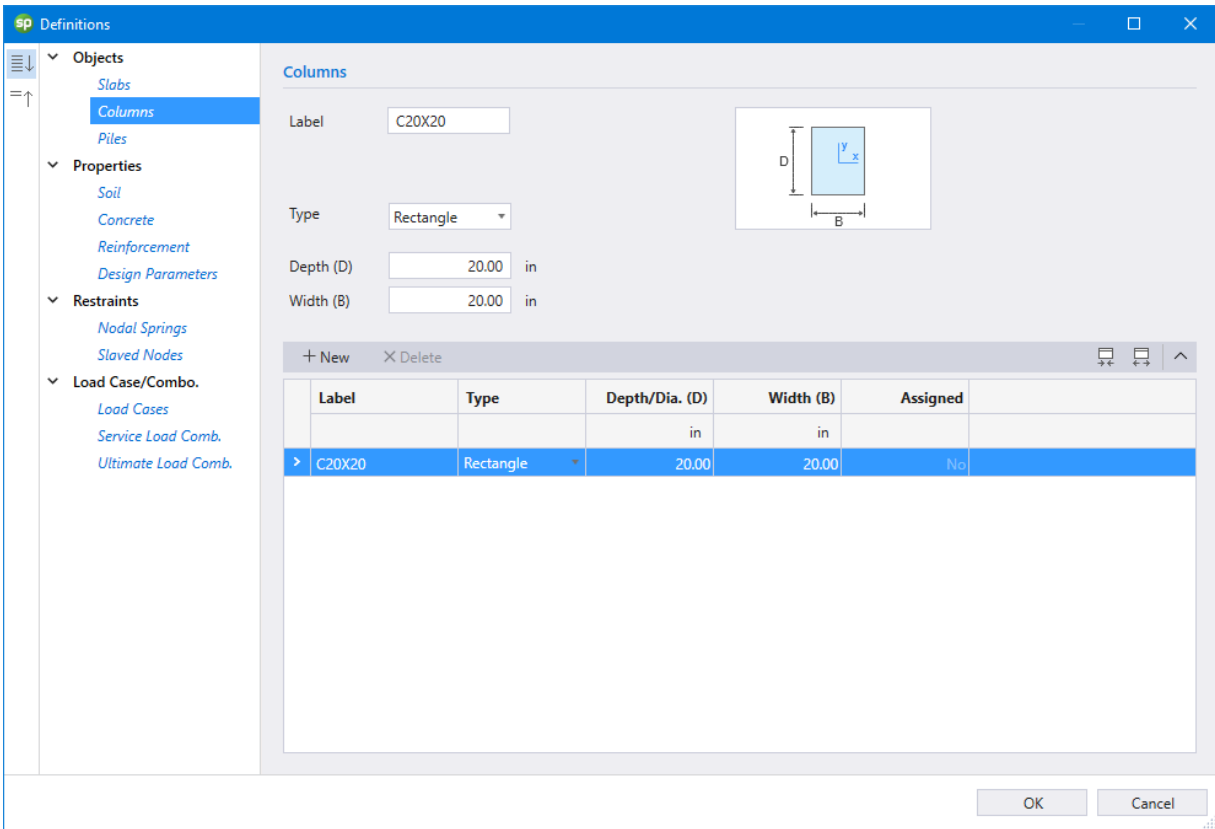
Piles

The PILES Object definition consists of the LABEL, TYPE, DIMENSION, and CHARACTERISTICS OF PILE, SUPPORTING SOIL, and whether the Pile label is ASSIGNED to a model or not.
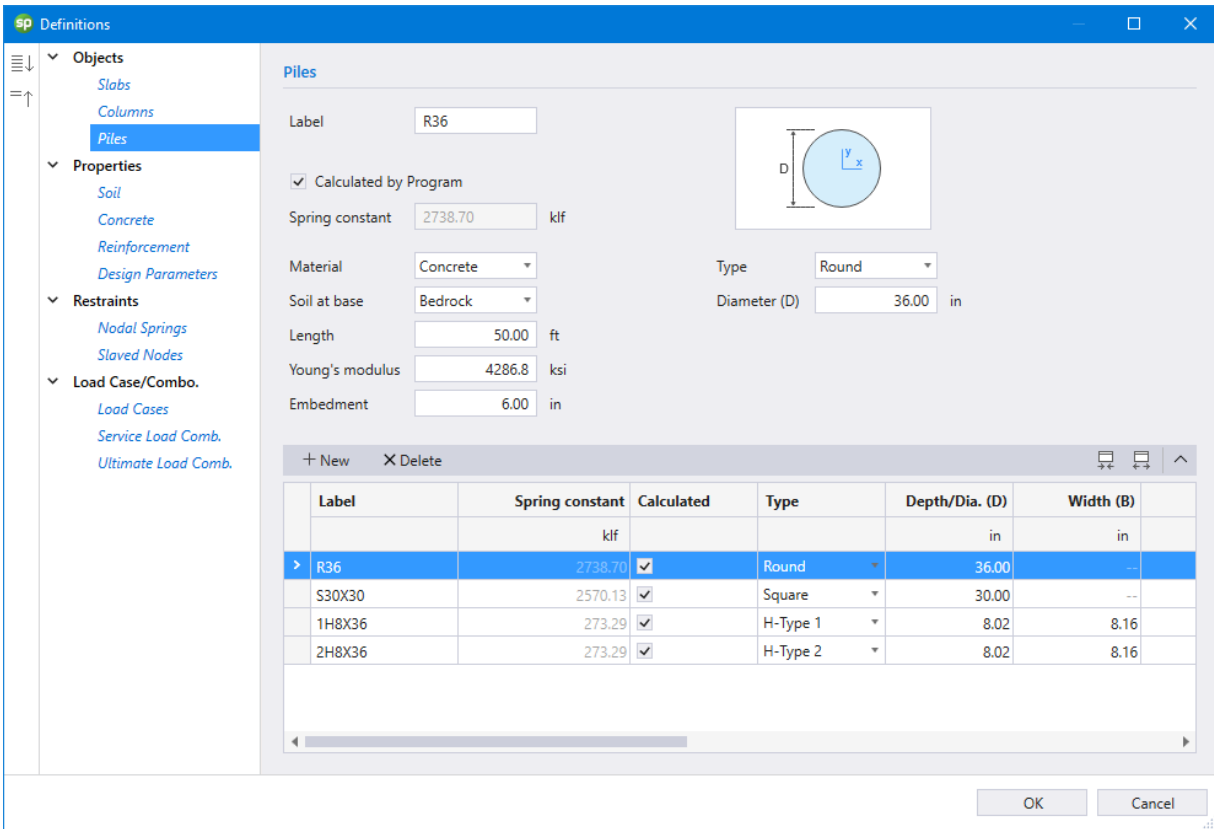
Properties

The PROPERTIES that can be defined are: SOIL, CONCRETE, REINFORCEMENT, and DESIGN PARAMETERS.

Soil

The SOIL definition consists of the LABEL, SUBGRADE MODULUS, ALLOWABLE PRESSURE, and whether the Soil label is USED to a model or not.

Concrete

The CONCRETE definition consists of the LABEL, COMPRESSIVE STRENGTH, UNIT WEIGHT, YOUNG'S MODULUS, POISSON'S RATIO, and whether the Concrete label is USED to a model or not.

Reinforcement

The REINFORCEMENT definition consists of the LABEL, YIELD STRENGTH, YOUNG'S MODULUS, and whether the Reinforcement label is USED to a model or not.

Design Parameters

The DESIGN PARAMETERS definition consists of the LABEL, MINIMUM REINFORCEMENT RATIO, REINFORCEMENT CENTERLINE LOCATIONS, and whether the Design Parameter label is USED to a model or not.
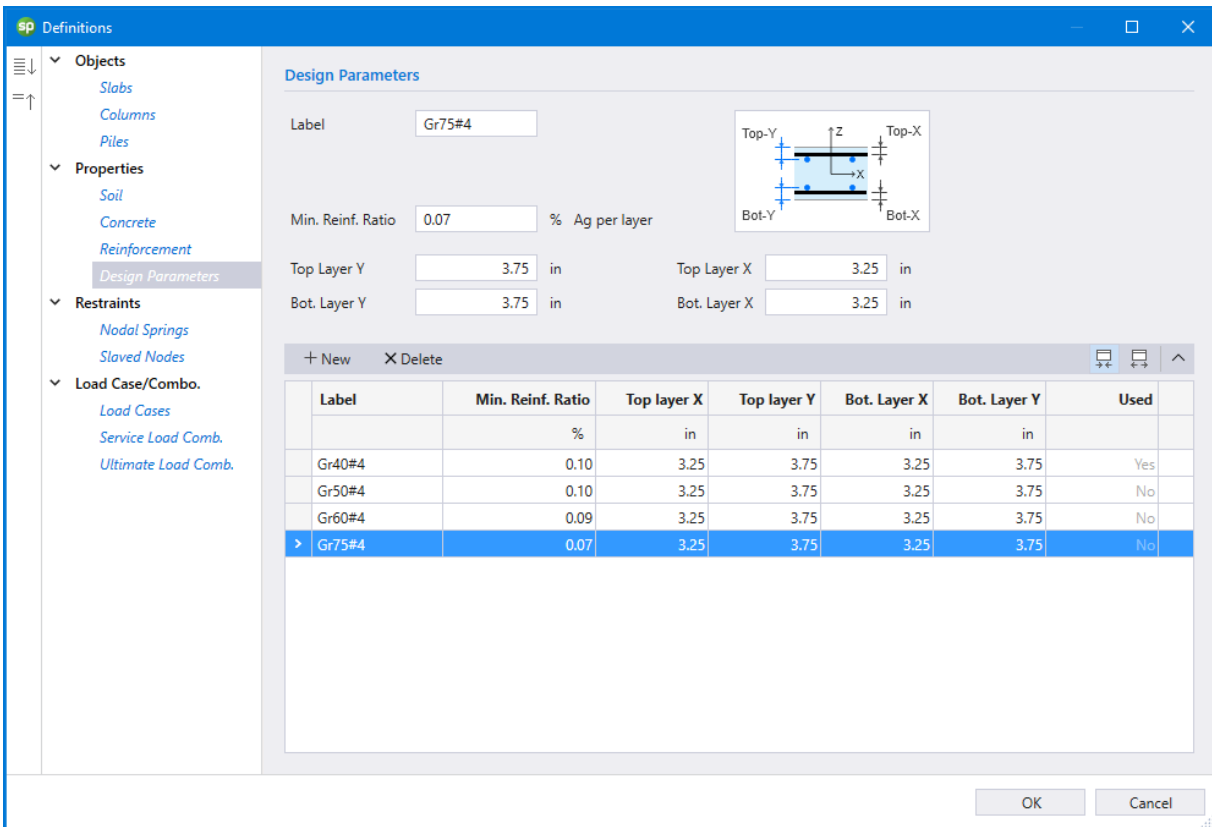
Restraints

The RESTRAINTS that can be defined are: NODAL SPRINGS, and SLAVED NODES.

Nodal Springs

The NODAL SPRINGS definition consists of the LABEL, SPRING CONSTANT, and whether the Nodal Spring label is ASSIGNED to a model or not.
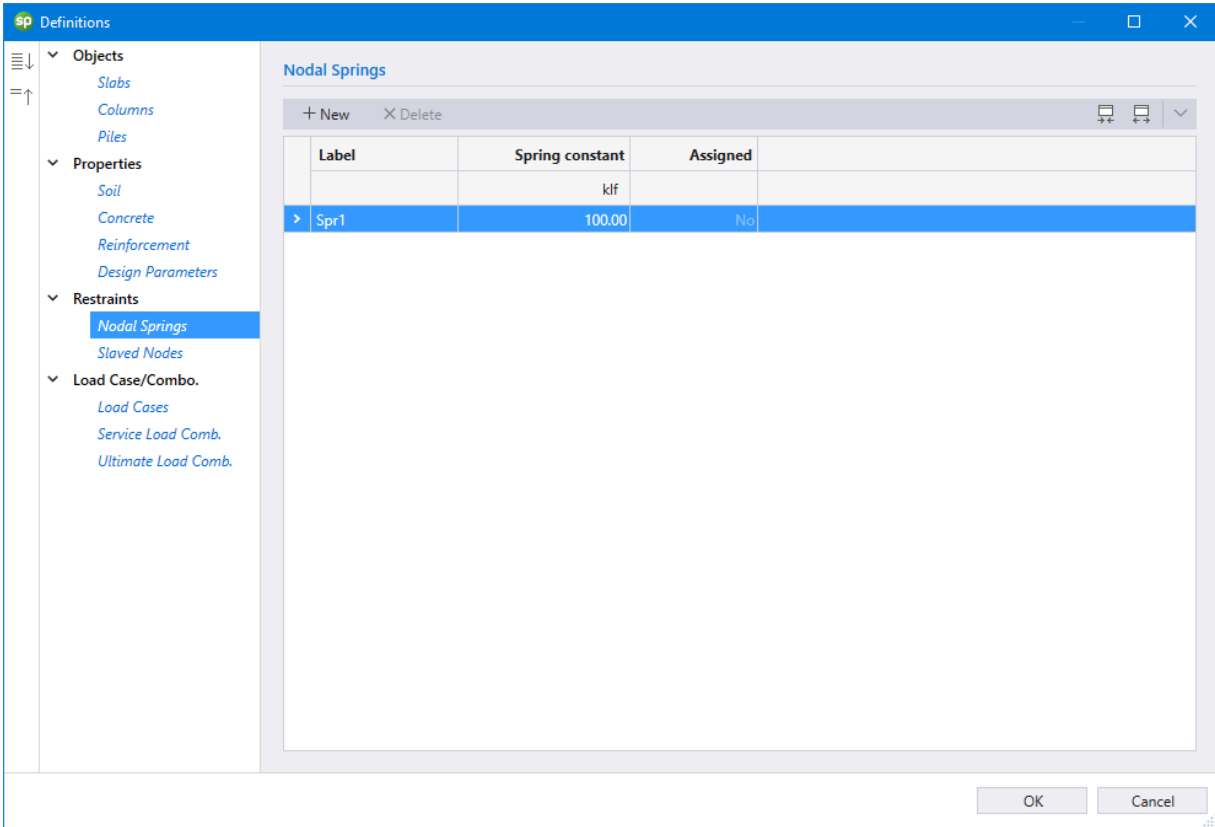
Slaved Nodes

The SLAVED NODES definition consists of the LABEL, DEGREE OF FREEDOM, and whether the Slaved Node label is ASSIGNED to a model or not.
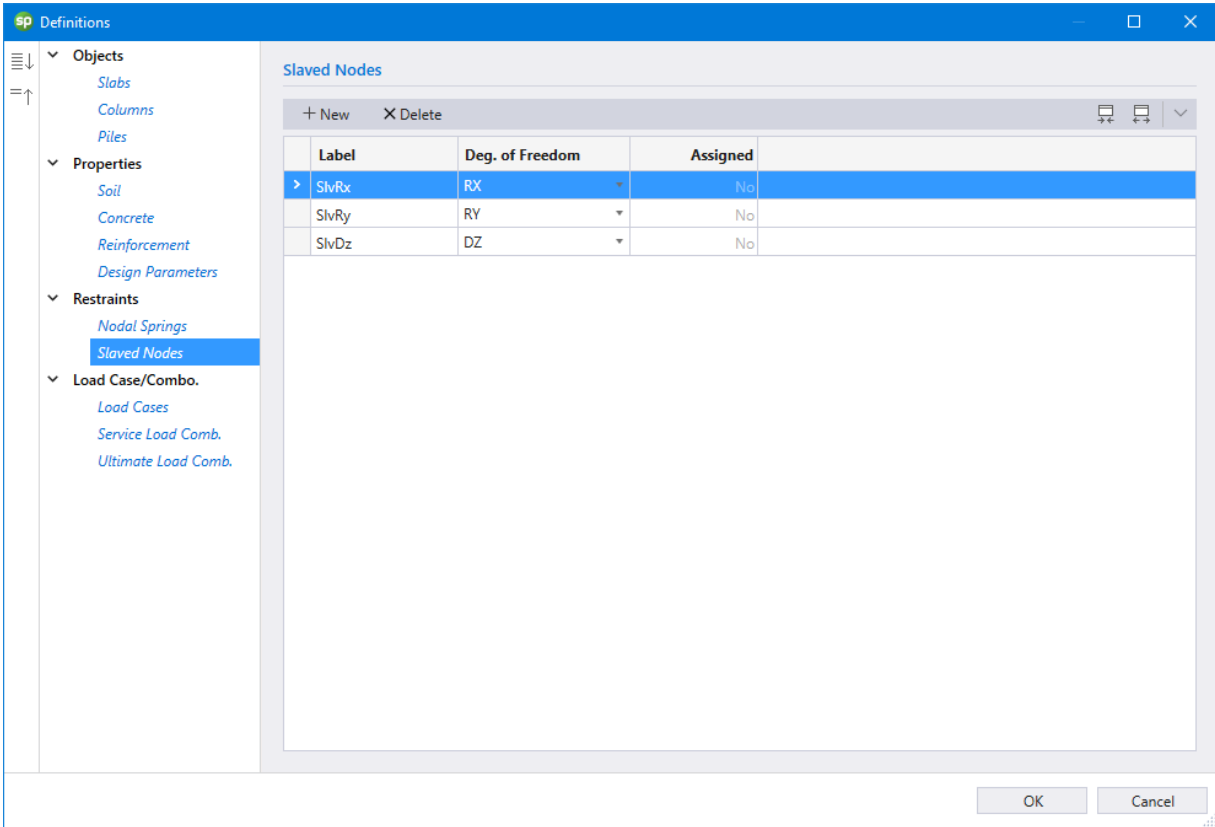
Load Case/Combo

The LOAD CASE/COMBO that can be defined are: LOAD CASES, SERVICE LOAD COMBINATIONS, and ULTIMATE LOAD COMBINATIONS.

Load Cases

The LOAD CASES definition consists of the CASE, TYPE, LABEL, whether the SELF-WEIGHT be included with the load case or not, and whether the Load Case is USED in the model or not.
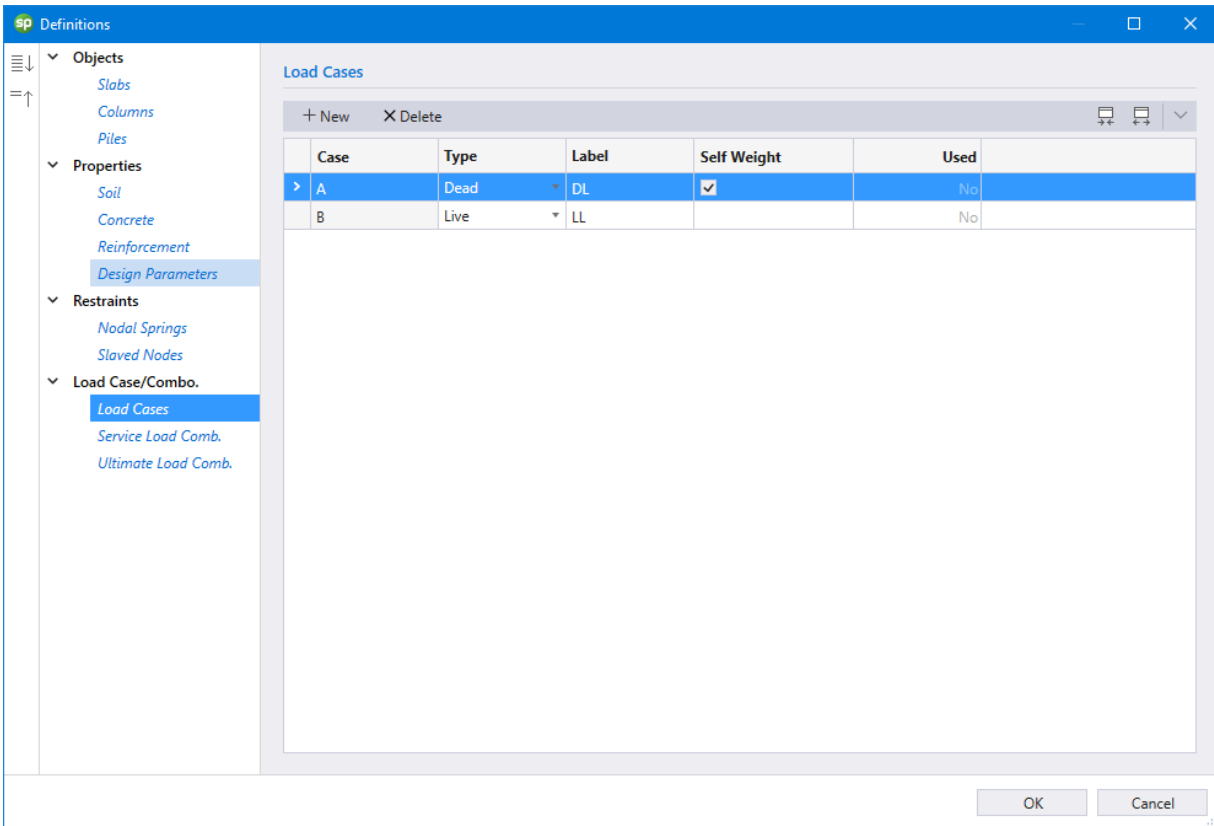
Service Load Combinations

The SERVICE LOAD COMBINATIONS definition consists of the LOAD CASES, TYPE, LOAD COMBINATION NUMBER, LOAD COMBINATION LABEL, and LOAD FACTORS. Load combinations can also be generated automatically based on the design code in the model.
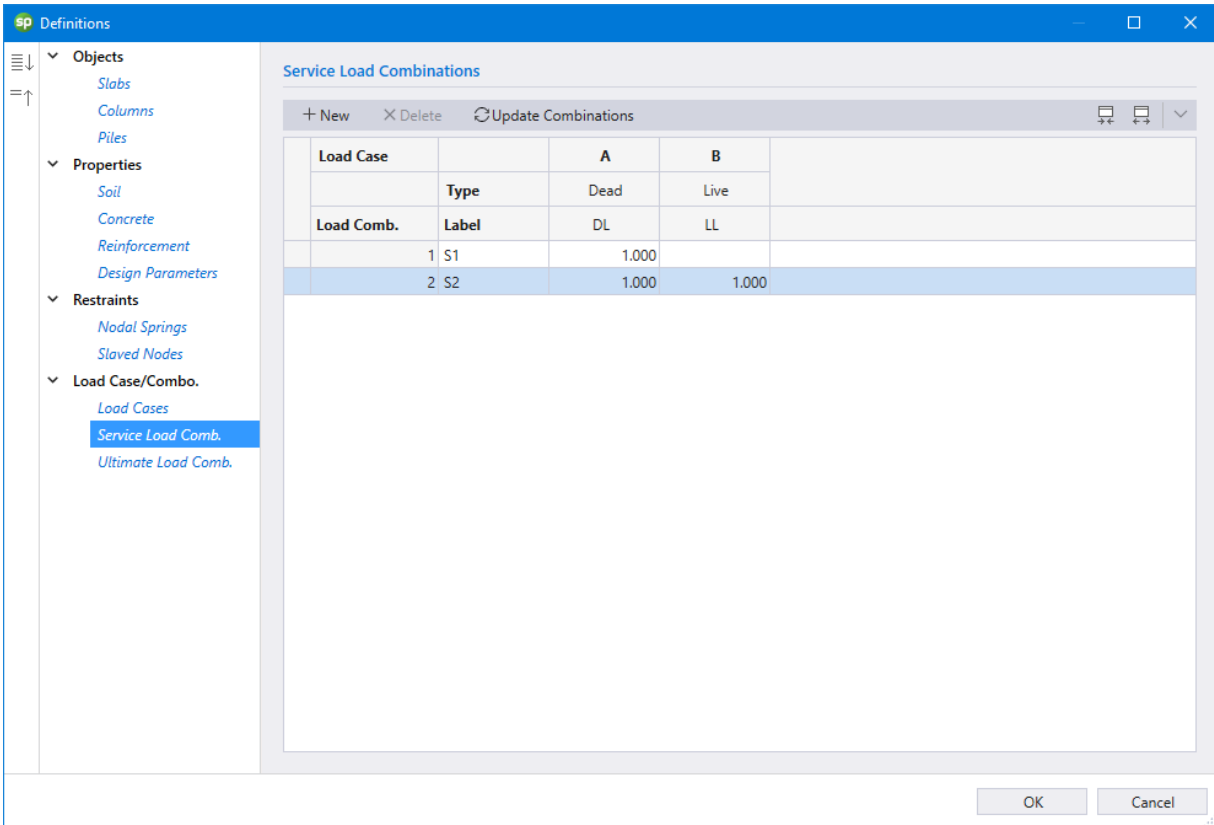
Ultimate Load Combinations

The ULTIMATE LOAD COMBINATIONS definition consists of the LOAD CASES, TYPE, LOAD COMBINATION NUMBER, LOAD COMBINATION LABEL, and LOAD FACTORS. Load combinations can also be generated automatically based on the design code in the model.
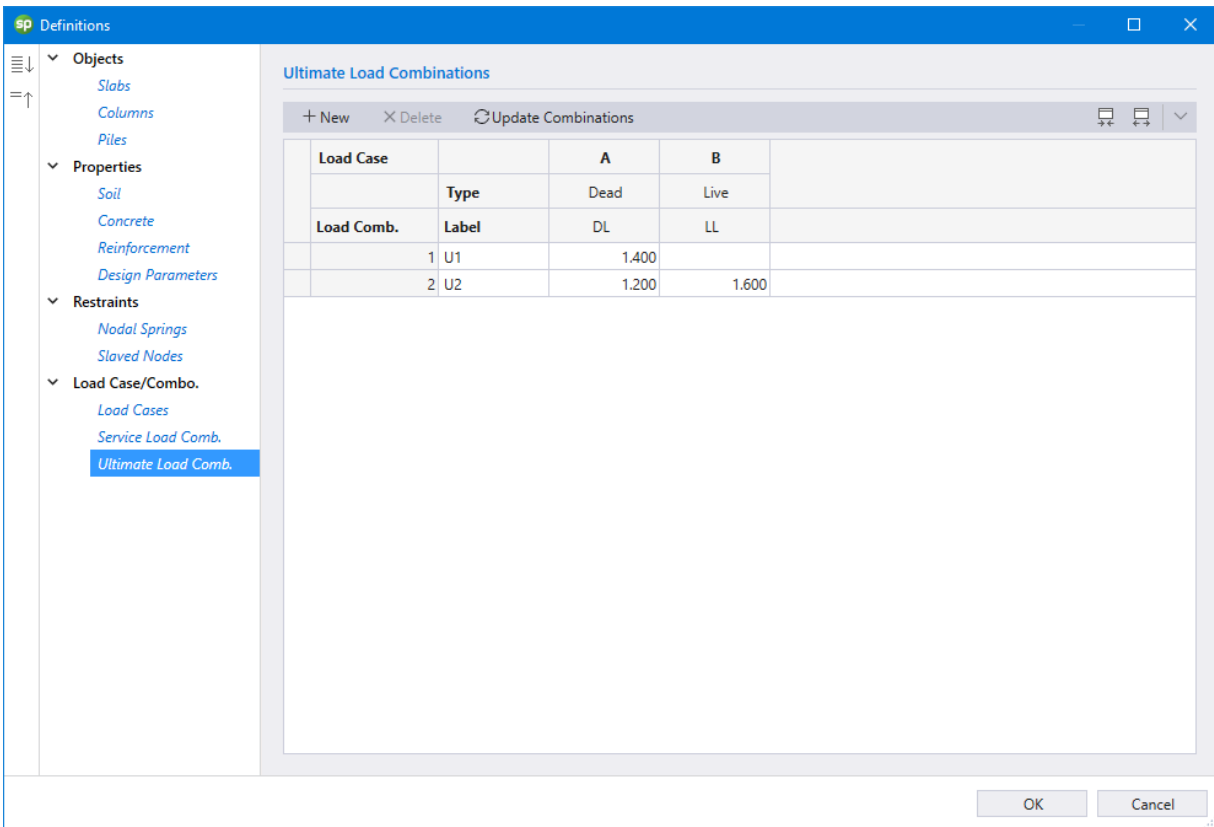
5.1.2.2   Creating Model Objects

The Objects that can be created are: SLABS, COLUMNS, and PILES.

Slabs

You can create **Slabs** using the command from the **Ribbon**.

You can draw RECTANGULAR, CIRCULAR or POLYGONAL slabs by using one of the following three tools in the SLABS **Left Panel**.

If you have already defined a slab type in the DEFINITIONS dialog, select it from the LABEL drop down menu in the **Left Panel** before you start drawing. You can also simply select the SECTION

and PROPERTIES and start drawing, the corresponding definition created will automatically be added to the DEFINITIONS dialog.

SLAB		
Label	Slab1	▾ >
⌄ Section		
Type	Solid	▾
Thickness	48.00	in
⌄ Properties		
Soil	Clay	▾ >
Concrete	C5	▾ >
Reinforcement	Gr60	▾ >
Design Param.	Gr60#8	▾ >

To draw a slab shape using the mouse, first select the desired tool.

- Then click on the workspace to begin drawing.

- For circular or rectangular slabs, move your mouse until the desired size is achieved and click again to finish drawing.

- For polygonal slabs keep clicking on the workspace until you create all the desired vertices. You can close a polygon either by clicking the right mouse button or manually clicking on the first vertex point created.

Alternately slab shapes can also be created by using the dynamic input box.

- To display a dynamic input box, press ENTER after you have selected the desired slab tool.
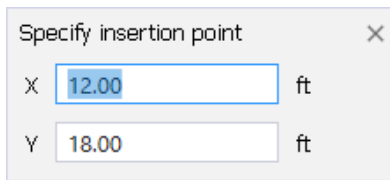
- You will be required to provide the start point (center point for circular slabs).

- Type in the required coordinates and press ENTER.

- Press ENTER once more to display the dynamic input box again. Type in the required quantities i.e. width and height for a rectangular slab or radius for a circular slab and press ENTER to finish drawing.

- For polygonal slabs you will have to keep pressing ENTER to bring out the dynamic input box to provide each vertex. The polygon will be closed once you enter the coordinates of the starting point provided that the shape is geometrically correct.

Columns

You can create **Columns** using the command from the **Ribbon**.

If you have already defined a column type in the DEFINITIONS dialog, simply select it from the LABEL drop down menu in the **Left Panel** to assign it to the model. You can also simply select the section type, enter the dimensions and start assigning, the corresponding column definition created will automatically be added to the DEFINITION dialog.
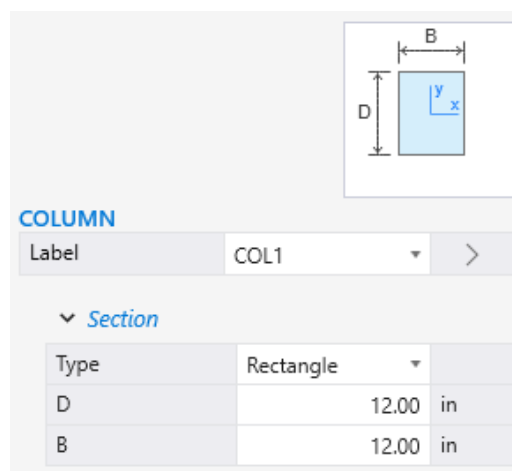
To assign a column using the mouse first decide on the parameters of the column to be assigned. The cursor shape changes to the column selected to be assigned.

- Then click on the workspace to begin assigning.

- You can also marquee select a group of nodes to assign multiple columns at once.

Alternately columns can also be assigned by using the dynamic input box.

- To display a dynamic input box, press ENTER after you have selected the column to be assigned.

- You will be required to provide the insertion point.
- Type in the required coordinates and press ENTER.

Piles

You can create **Piles** using the command from the **Ribbon**.

If you have already defined a Pile type in the DEFINITIONS dialog, simply select it from the Label drop down menu in the **Left Panel** to assign it to the model. You can also simply select the section type, enter the dimensions, select the required properties and start assigning, the corresponding pile definition created will automatically be added to the DEFINITION dialog.

To assign a pile using the mouse first decide on the parameters of the pile to be assigned. The cursor shape changes to the pile selected to be assigned.

- Then click on the workspace to begin assigning.

- You can also marquee select a group of nodes to assign multiple piles at once.

Alternately piles can also be assigned by using the dynamic input box.

- To display a dynamic input box, press ENTER after you have selected the pile to be assigned.

Specify insertion point ✕

X 7.95 ft

Y 11.50 ft

- You will be required to provide the insertion point.

- Type in the required coordinates and press ENTER.

Restraints

The RESTRAINTS that can be assigned are: NODAL SPRINGS, SLAVED NODES, and SUPPORTS. You can create **Restraints** using the ⛉ command from the **Ribbon**.

spMats considers restraints as nodal properties. It is possible to assign more than one kind of restraint to a node at the same time.

- In the SPRING → TRANSLATION - Z and the SLAVE/SUPPORT → TRANSLATION - Z, ROTATION - X and ROTATION - Y boxes, select the required restraint type.

- You can also use the CLEAR RESTRAINTS button to clear any existing restraints and select new ones.

- Next, click on the location you want the restraint to be assigned to.

- You can also marquee select a group of nodes to assign all of them the same restraints.
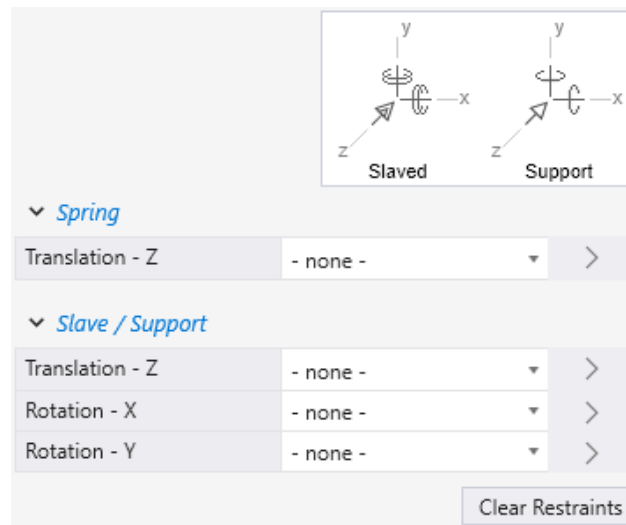
Loads

You can create **Loads** using the command from the **Ribbon**.

You can assign point loads or area loads by using one of the two options that are present in the LOADS **Left Panel**.

Assigning Area Loads

Area loads can only be assigned to **Slabs**.
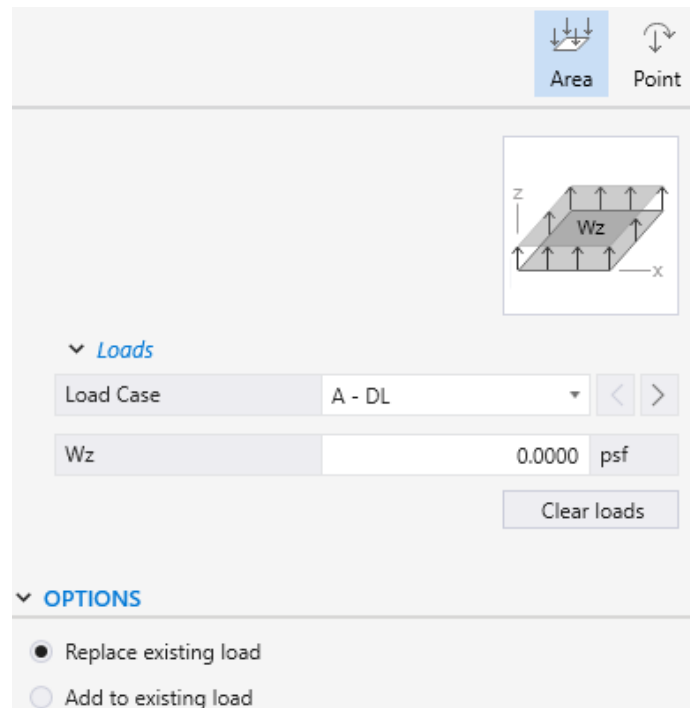
To assign an area load, make sure that the **Area** command in the **Left Panel** is selected. The **Left Panel** should be displaying various **Area Loads** options.

- Select the LOAD CASE you want the area load to belong to. You can always define LOAD CASES in the DEFINITIONS dialog.

- In the Wz box, type in the required load value. Note that downward forces have negative values.

- From the OPTIONS select if you want to ADD TO EXISTING LOAD on the slab or REPLACE EXISTING LOAD completely.

- Next, click on the slab you want the load to be assigned to.

- You can also marquee select a group of slabs to assign all of them the same area load.

Assigning Point Loads

spMats considers point loads as nodal properties.
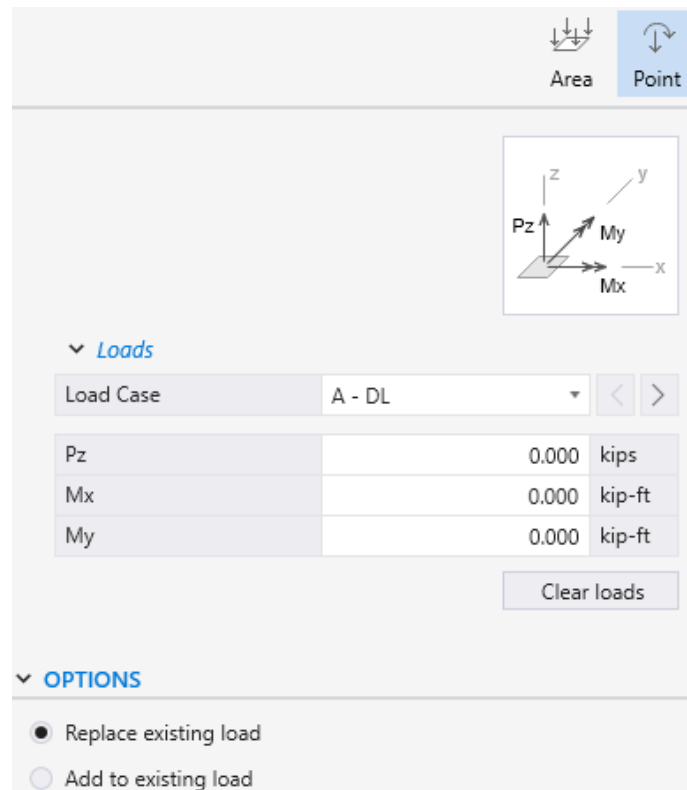
To assign a point load, make sure that the **Point** command ⤵ in the **Left Panel** is selected. The **Left Panel** should be displaying various **Point Loads** options.

- Select the LOAD case you want the Point load to belong to. You can always define LOAD CASES in the DEFINITIONS dialog.

- In the Wz, Mx and My boxes, type in the required load and moment values. Note that downward forces have negative values. To determine the direction of the moments Mx and My, use the right-hand rule.

- From the OPTIONS select if you want to ADD TO EXISTING LOAD on the node or REPLACE EXISTING LOAD completely.

- Next, click on the location you want the load to be assigned to.

- You can also marquee select a group of nodes to assign all of them the same point load.

Nodes

You can create **Nodes** scope using the ─┼─ command from the **Ribbon**.

You can draw a SINGLE node, nodes in LINEAR arrangement or nodes in CIRCULAR arrangement by using one of the following three tools in the NODES **Left Panel**.
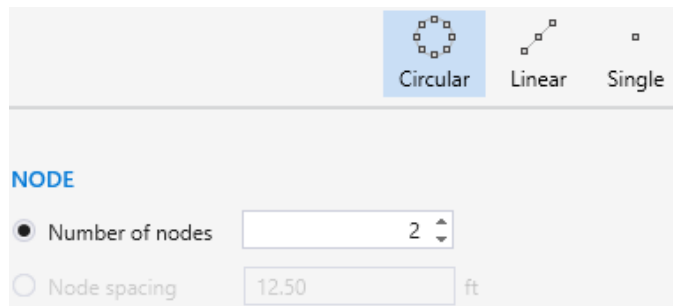
To draw nodes first select the desired NODES Tool.

- Select the NUMBER OF NODES or the NODE SPACING option for the CIRCULAR and/or LINEAR tools and enter the desired number of nodes to be created or the spacing at which the nodes are to be distributed.

- Then click on the workspace to begin drawing. If you have selected the SINGLE node tool then the node is created at the point you click.

- Move your mouse until the desired size is achieved and click again to finish drawing.

- You can also marquee select grid intersections to assign multiple nodes at them.

Alternately you can also use dynamic input box.

- To display a dynamic input box, press ENTER after you have selected the node tool and decided on the number or spacing of the nodes to be created.
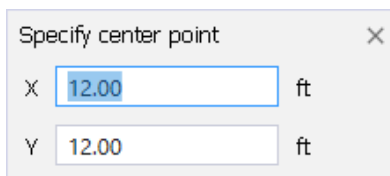
- You will be required to provide the starting point (center point for CIRCULAR node distribution).

- Type in the required coordinates and press ENTER.

- If you have selected the SINGLE node tool then the node is created at the point you specify.

- Press ENTER once more to display the dynamic input box again. Type in the required quantities i.e. X and Y coordinates of the end points for linear node distribution or radius for the circular node distribution and press ENTER to finish drawing.

5.1.2.3   Editing Model Objects

The model can be edited by using the **Left Panel**, **Left Panel Toolbar** or by using right-click at **Viewport**.

Using the Left Panel
Objects

The OBJECTS that can be edited are: SLABS, COLUMNS, and PILES.

Slabs

The corresponding SLABS **Left Panel** provides various tools and options for effectively working with slabs. You must have the **Select** command button toggled on to edit the SLABS.

Section and Properties

To edit a slab section or properties:

  • Click on the slab to select it

  • Then in the **Left Panel**, simply change the desired parameter.

  • If the given set of parameters match the parameters of a pre-defined slab the new slab will automatically be assigned the existing label.

  • If the given set of parameters do not match the parameters of any pre-defined slab the slab will be assigned a new definition.

Columns

The corresponding COLUMNS **Left Panel** provides various options for effectively working with columns. You must have the **Select** command button toggled on to edit COLUMNS.

To edit a single column:

  • Click on the column you want to edit to display its properties and position in the **Left Panel**.

  • Change the desired column parameter.

To edit multiple columns at once:

  • Use marquee select to select multiple columns at once.

  • Notice that if all the selected columns are of the same type then the section properties are displayed in the left panel and you can change the desired parameter as required.

  • If different columns types are selected then only the column label is available to be changed.

Piles

The corresponding PILES **Left Panel** provides various options for effectively working with piles. You must have the **Select** command button toggled on to edit PILES.

To edit a single pile:

- Click on the pile you want to edit to display its properties and position in the **Left Panel**.

- Change the desired pile parameter.

To edit multiple piles at once:

- Use marquee select to select multiple piles at once.

- Notice that if all the selected piles are of the same type then the section properties are displayed in the left panel and you can change the desired parameter as required.

- If different piles types are selected then only the pile label is available to be changed.

Restraints

The corresponding RESTRAINTS **Left Panel** provides various options for effectively working with RESTRAINTS.

spMats considers restraints as nodal properties. Therefore, to edit restraints the corresponding nodes have to be selected.

To edit a single restraint:

- Click on the node containing the restraint you want to edit to display its properties and position in the **Left Panel**.

- Change the desired restraint parameter.

To edit multiple restraints at once:

- Use marquee select to select multiple nodes with desired restraints.

- Change the desired restraint parameter or parameters.

Loads

The corresponding LOADS **Left Panel** provides various options for effectively working with LOADS.

Editing Area Loads

In spMats AREA loads can only be assigned to slabs. Therefore, to edit area loads the corresponding slabs have to be selected.

To edit an area load:

- Click on the slab containing the load you want to edit to display its properties and position in the **Left Panel**.

- Change the load value as desired.

To edit multiple area loads at once:

- Use marquee select to select multiple slabs with desired loads.

- Change the load value as desired.

Editing Point Loads

spMats considers POINT loads as nodal properties. Therefore, to edit point loads the corresponding nodes have to be selected.

To edit a single point load:

- Click on the node containing the load you want to edit to display its properties and position in the **Left Panel**.

- Change the desired load parameter.

To edit multiple point loads at once:

- Use marquee select to select multiple nodes with desired loads.

- Change the desired load parameter or parameters.

Nodes

The corresponding NODES **Left Panel** provides various tools and options for effectively working with nodes. You must have the **Select** command button toggled on to edit NODES.

To edit a single node:

- Click on the node you want to edit to display its properties and position in the **Left Panel**.

- Change the desired node parameter.

To edit multiple nodes at once:

- Use marquee select to select multiple nodes at once.

- Edit the desired nodal properties from the **Left Panel**.

Using the Left Panel Toolbar

You must have the **Select** command button toggled on in order to use the tools available in the **Left Panel Toolbar**. You can use the tools in the **Left Panel Toolbar** to edit various model items.

Delete ✕

The **Delete** command is active only when one or more items are selected.

- Select the item or items you want to remove from the model and click DELETE to remove.

Move ✛

The **Move** command is active only when one or more items are selected.

- Select the item or items you want to move and click the **Move** command.

- Click on the screen to specify the base point from which to start moving. Alternatively, you can also press ENTER and manually enter the coordinates of the base point.

- Drag the selected items to the desired location and click to complete moving. Alternatively, you can also press ENTER and enter the translation vector to move the selected items to their new location.

Copy ▭

The **Copy** command is active only when one or more items are selected.

- Select the item or items you want to make a copy of and click the **Copy** command.

- Click on the screen to specify the base point from which to start the copying process. Alternatively, you can also press ENTER and manually enter the coordinates of the base point.

- Drag the copied items to the desired location and click to create a new instance. Alternatively, you can also press ENTER and enter the translation vector to move the copied instances of items to their new location.

Nodes – Align Vertical ⋮

The **Nodes – Align Vertical** command is active only when one or more nodes are selected.

- Select the node or nodes you want to align vertically in a straight line and click the **Nodes – Align Vertical** command.

- Click on the screen to specify the reference X point at which to vertically align all the selected nodes.

- Alternatively, you can also press ENTER and manually enter the coordinate of the X reference point and press ENTER to complete alignment.

Nodes – Align Horizontal ...

The **Nodes – Align Horizontal** command is active only when one or more nodes are selected.

- Select the node or nodes you want to align horizontally in a straight line and click the **Nodes – Align Horizontal** command.

- Click on the screen to specify the reference Y point at which to horizontally align all the selected nodes.

- Alternatively, you can also press ENTER and manually enter the coordinate of the Y reference point and press enter to complete alignment.

Slabs – Merge

The **Slabs - Merge** command is active only when more than one slabs are selected.

- Select the slabs that you want to merge and click the **Slabs – Merge** command.

  While merging two or more slabs, the program requires you to select a reference slab, the properties of which will be applied to the final merged slab. These include section type and thickness along with properties like soil, concrete, reinforcement, design parameters and any area loads applied.

- Click on slab whose properties you want the final merged slab to contain.

Slabs – Offset

The **Slabs - Offset** command is active only when one slab is selected.

- Select the slabs that you want to offset and click the **Slabs – Offset** command.

  It is possible to offset the slab inwards or outwards.

- Click the side of the slab you want to offset.

- In the input box that appears, specify the offset distance and press ENTER.

Slabs – Split

The **Slabs – Split** command is active only when one or more slabs are selected.

- Select the slabs that you want to split and click the **Slabs – Split** command.

- Click on the screen to specify the start point of the cutting line.

- Alternatively, you can also press ENTER and manually enter the coordinates of the start of the cutting line.

Note that a slab can be split only if the cutting line starts from outside or one edge of the slab and extends to or beyond the other edge of the slab. Starting a cutting line from any point within a slab will not cut that particular slab.

- Next click on the screen to specify the end point of the cutting line.

- Alternatively, you can also press ENTER and manually enter the coordinates of the end point of the cutting line.

Using the Right Click Menu at Viewport

All the tools in the **Left Panel Toolbar** are also available in the **Right Click** Menu at **Viewport** when the **Select** command button is toggled on.
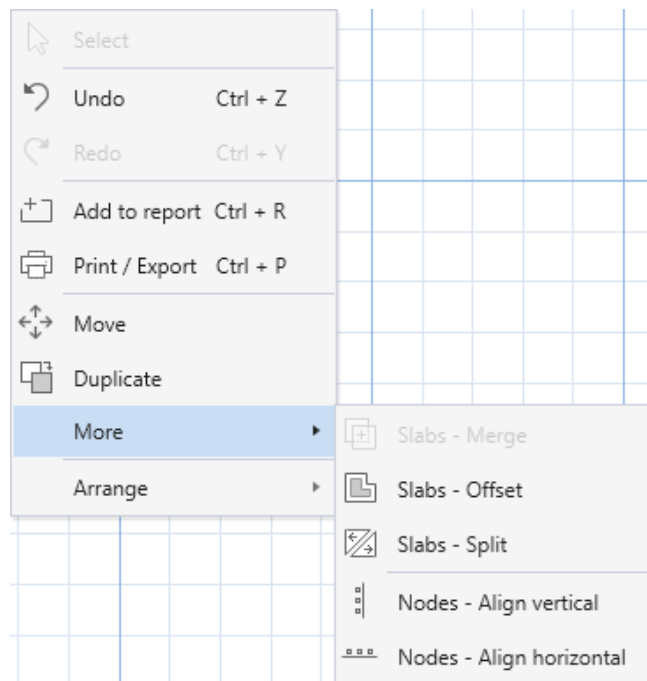
Understanding Slab Layers

While creating models the need may arises to place one slab on top of another. In this case the properties (including the area load applied on the slab) of the slab on top are considered for solving the model. During modeling a slab drawn later is always placed above the slab drawn first.

To move a slab above or below another one:

- Click on the slab to select it.

- Then right click on it to show the **Right Click** Menu.
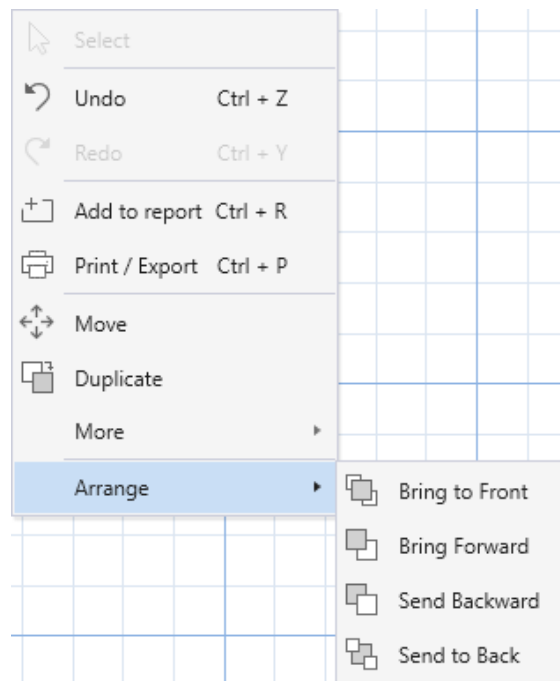
- From the **Arrange** sub menu, select the desired action

It should be noted that openings are always on top regardless of the order they are drawn in.

5.1.3   Modeling with Templates

5.1.3.1   Utilizing Templates

Templates is an option for creating new models in the spMats program. It enables the user to select from a set of pre-defined templates and edit them for quick model generation.

To begin go to the **Start Screen** under **Projects** and select the TEMPLATES option. This will take you to the TEMPLATE selection Dialog Box.
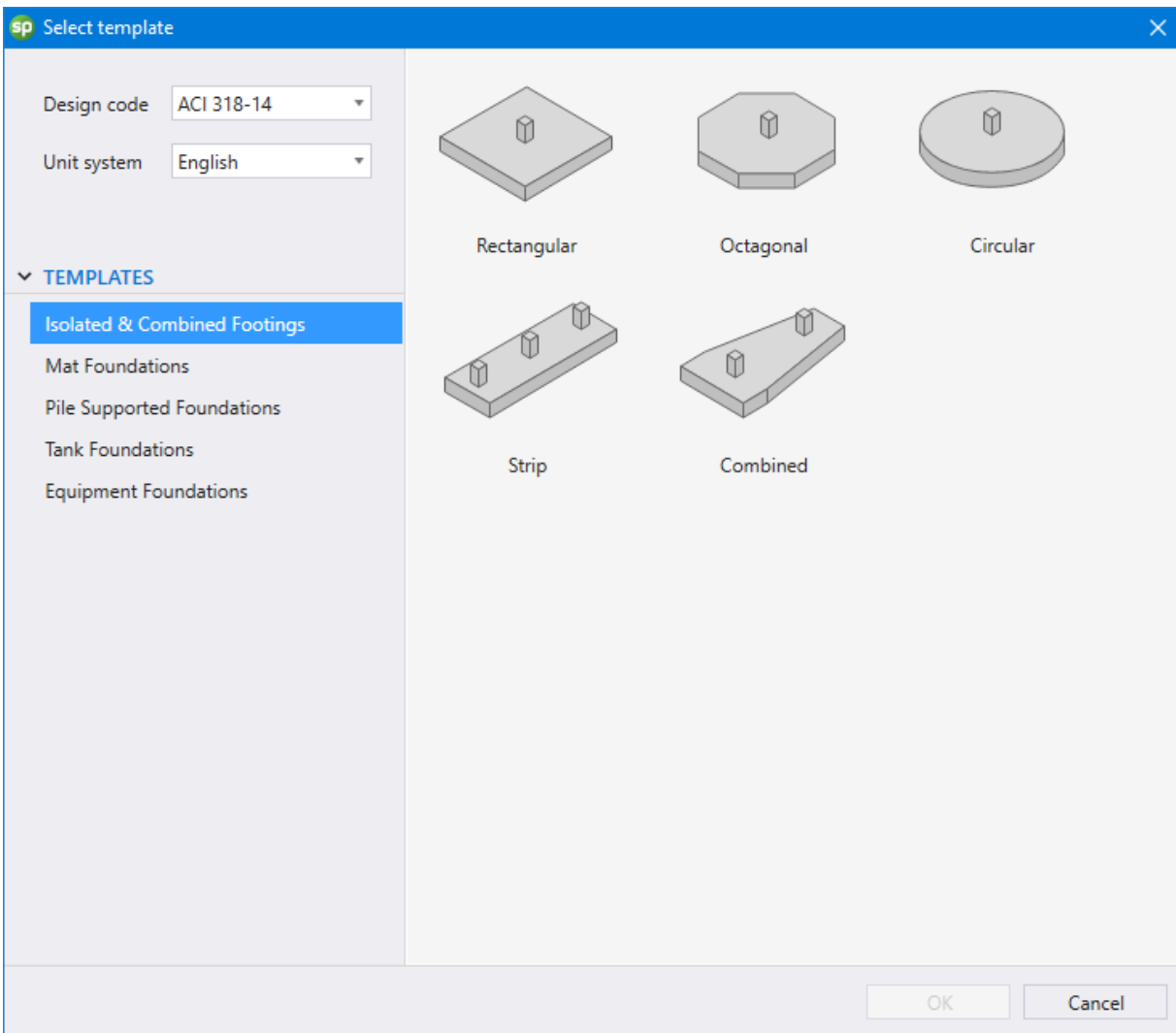
Here you can select the desired template along with the DESIGN CODE and UNIT SYSTEM. Double clicking on a template image or selecting a template image and clicking OK will open the **Template Module** and load the selected Template.
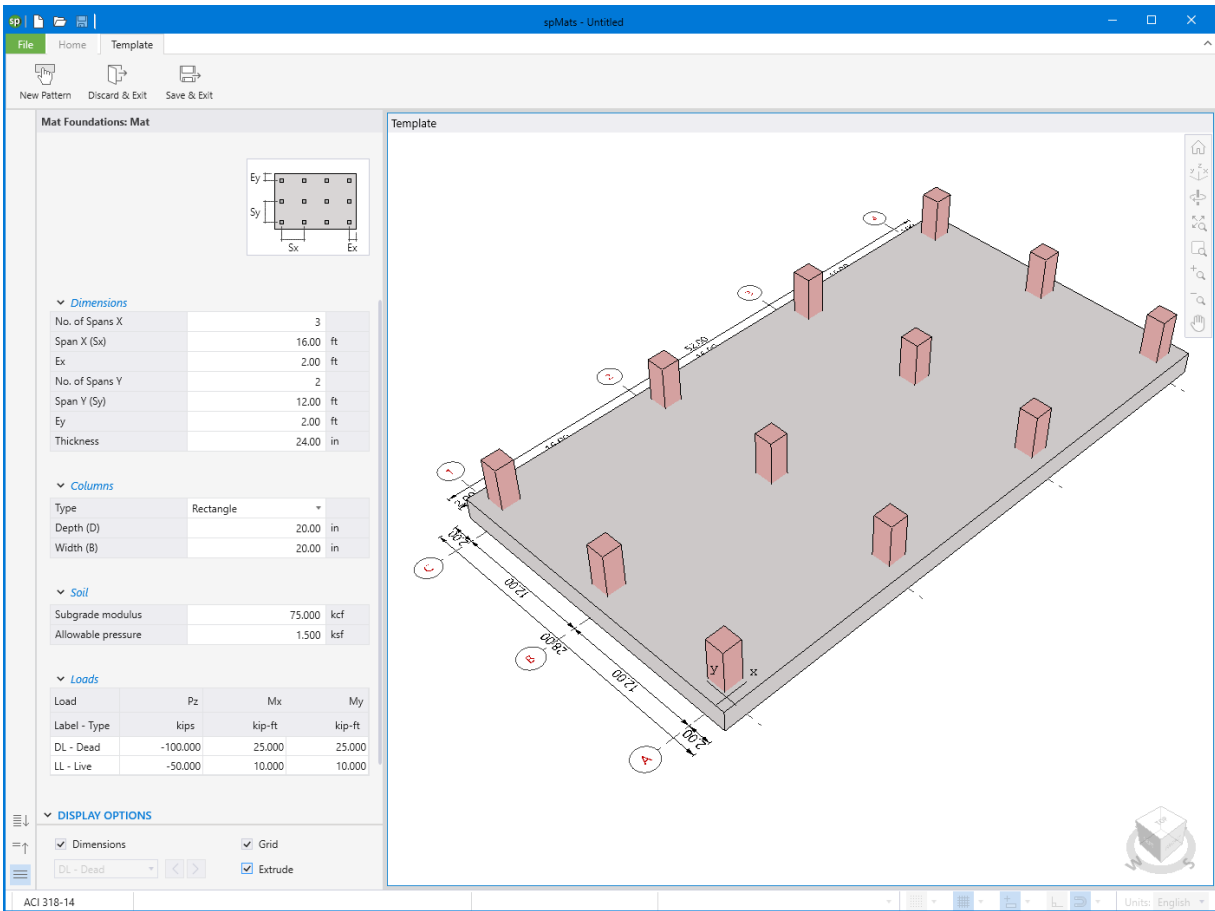
Once you are done editing the template to reflect your project criteria, you can click the SAVE & EXIT button to take it to spMats for further modification or execution.

5.1.3.2 Template Ribbon

The **Template** ribbon provides the following options:

New Pattern

> Opens the TEMPLATE Selection Dialog Box. Selecting a New template will discard the old template and load the new one.

Discard & Exit

> Discards the current template and exits to spMats home screen.

Save & Exit

> Imports the current template to spMats.

Left Panel

The **Left Panel** lists various template properties that can be modified by the user. The bottom part of the **Left Panel** consists of **Display Options**. You can use these to toggle on/off several **Viewport** items and also switch between displaying Load Cases.

5.1.3.3   Types of Templates

Isolated and Combined Footings
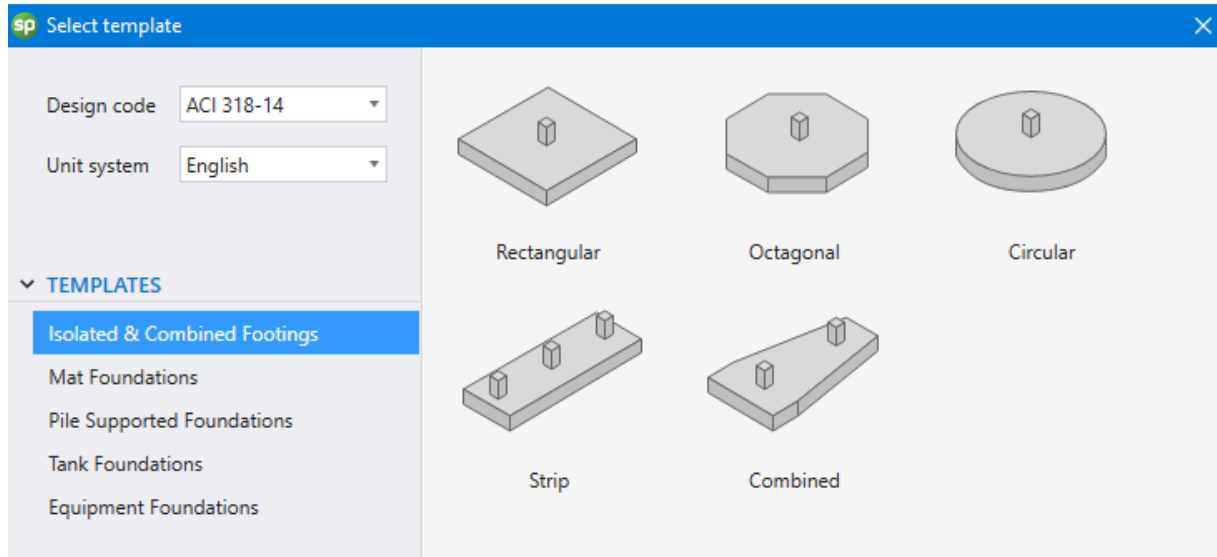
Mat Foundations
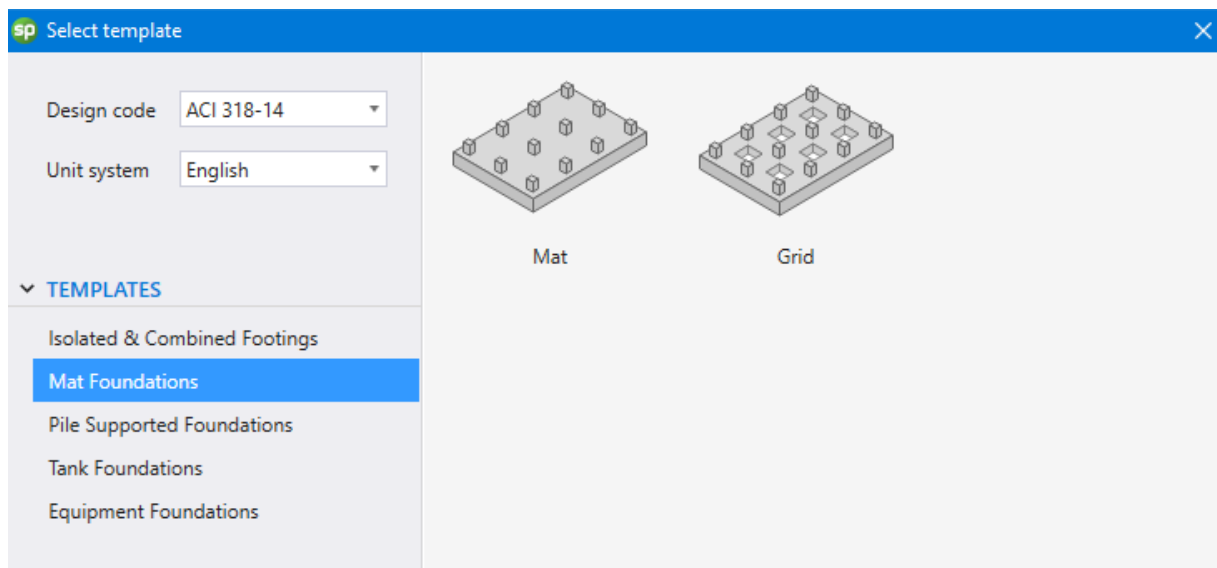
Pile Supported Foundations
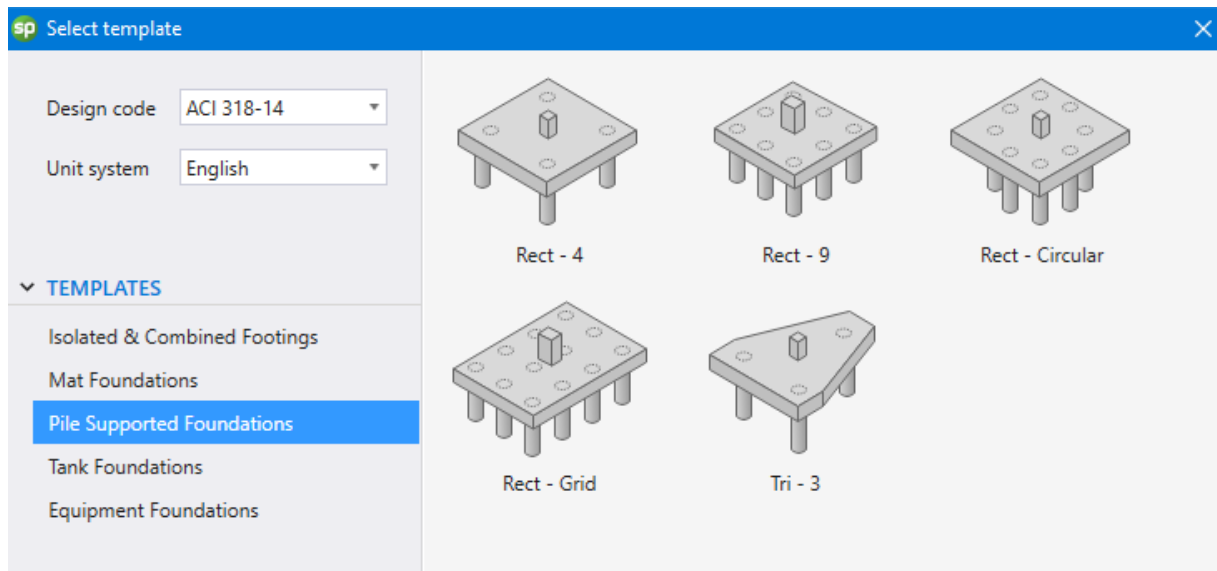
Tank Foundations
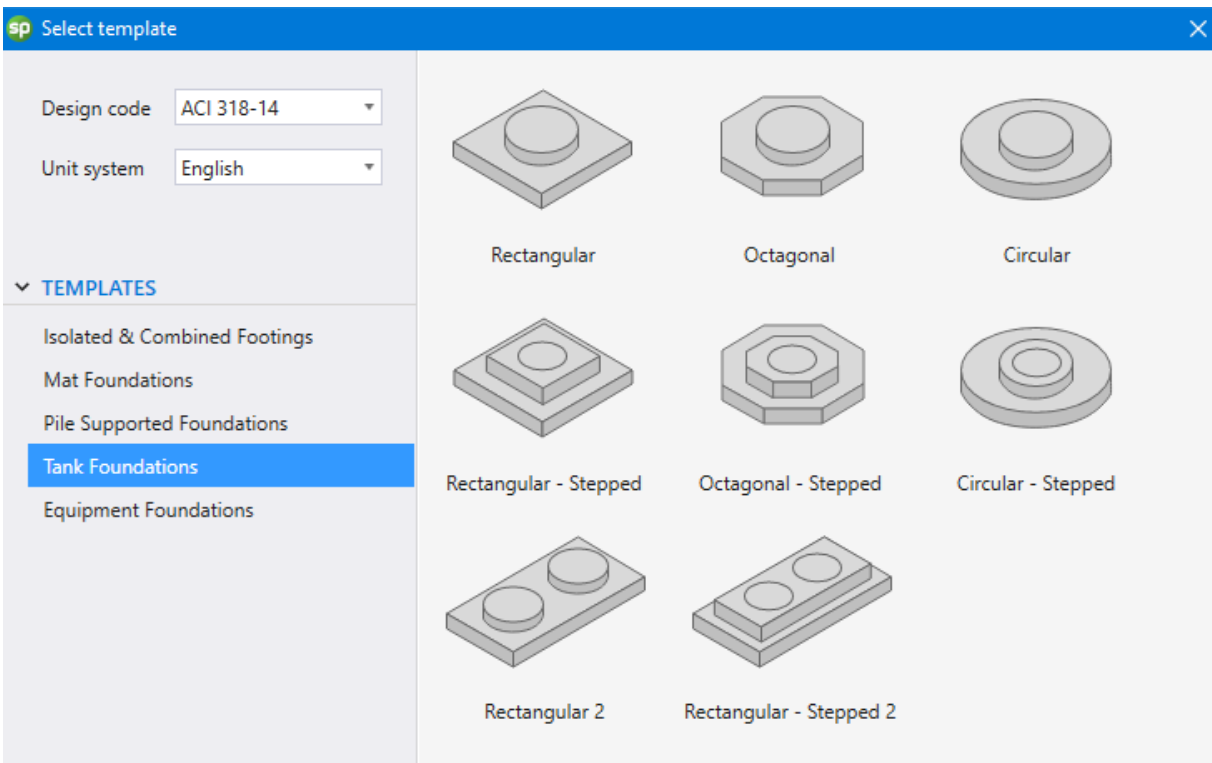
Equipment Foundations
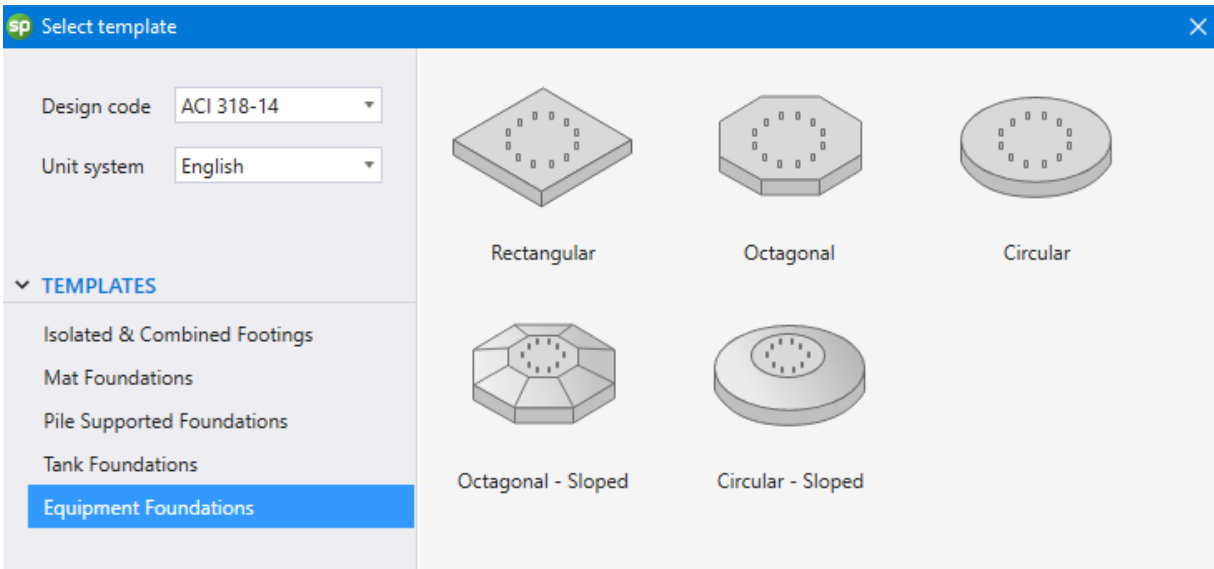
5.1.4   Utilizing Predefined Examples

In the **Start Screen** under **Projects** select the EXAMPLES option. This will take you to the EXAMPLES folder under the spMats installation folder.
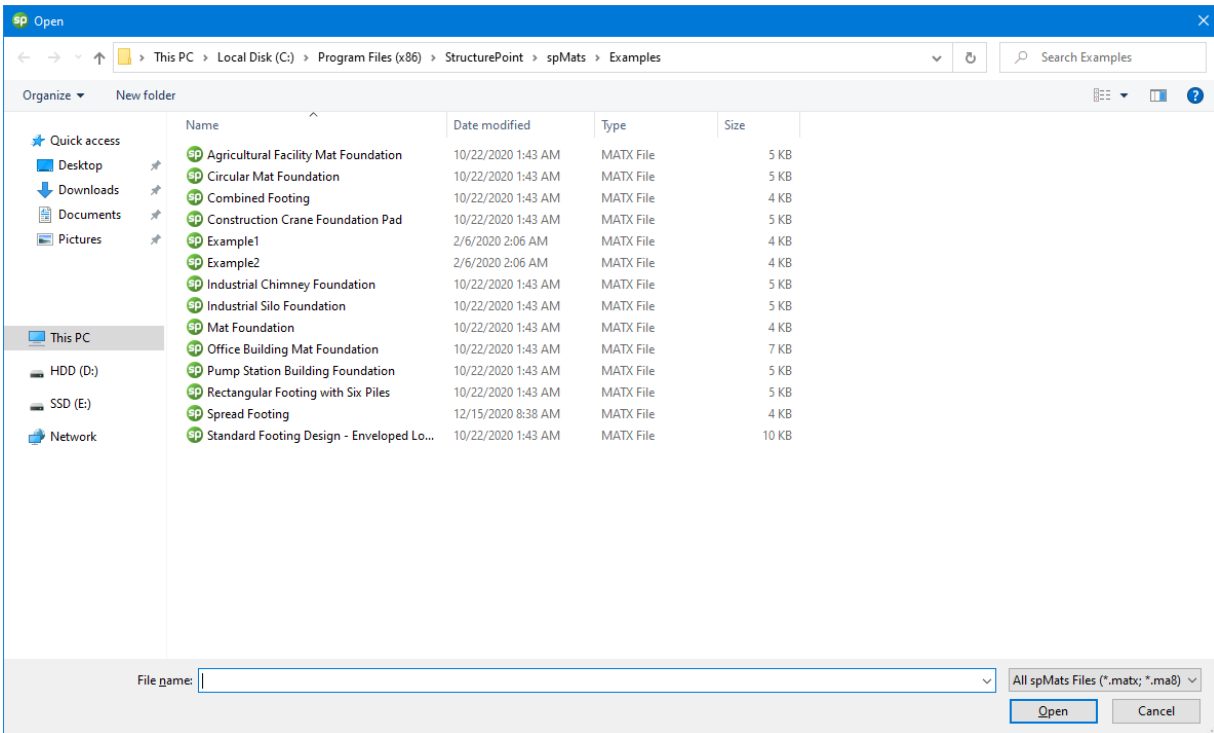
The EXAMPLES folder contains predefined foundation models that can be further utilized by the user.

5.1.5   Importing Model Data

Importing Data

spMats provides the options to import GRID, LOAD and LOAD CASES & COMBINATIONS data from a text file. The data import options can be obtained from **File | Import**.
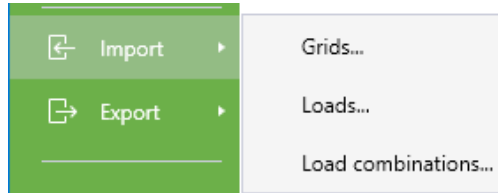
It should be noted that importing a data set will complete replace existing data of that particular set in the program. For details on the import file formats please refer to the **Appendix**.

5.1.6   Exporting Model Data

Exporting Data

Grids, Loads & Load Cases and Combinations

spMats provides the options to export Grid, Load and Load Cases & Combinations data to a text file. The data export options can be obtained from **File | Export**.
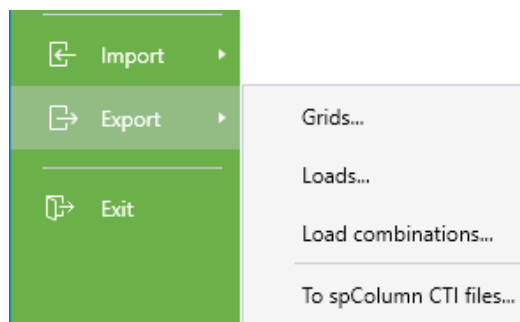
The exported data format will be the same as the import data format provided in the **Appendix**.

5.1.7   Exporting to spColumn CTI files

spMats provides the option to export columns and/or pile sections, used in the foundation model, as spColumn Text Input (CTI) files for analysis by spColumn. The loads coming on the sections

after analysis are also included in the exported files. This export can only be done when the model has been executed and results generated.

- From the FILE menu, select EXPORT | TO SPCOLUMN CTI FILES. The first dialog box among the following two appears.
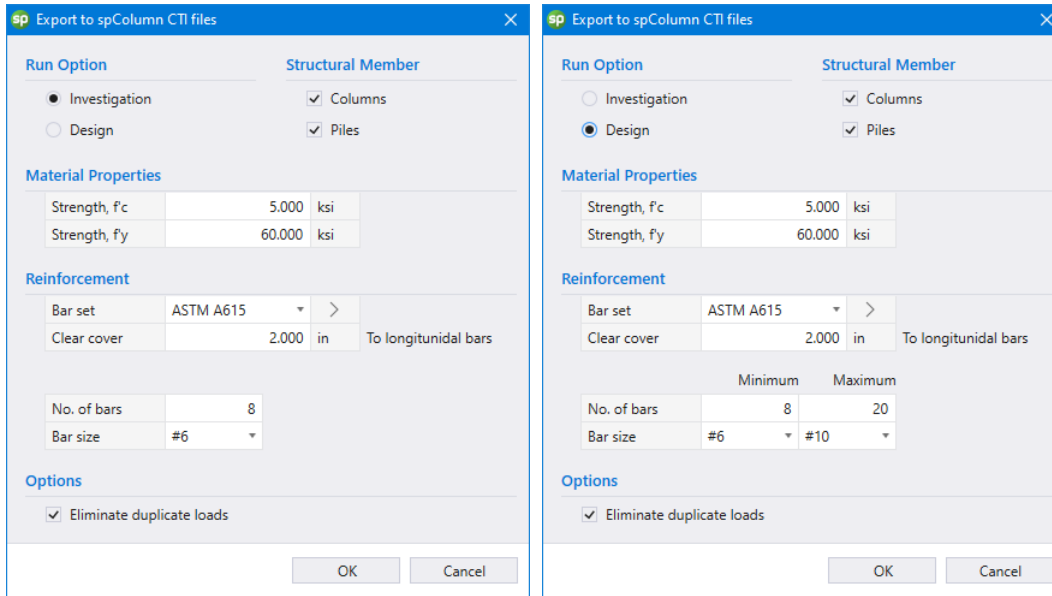
- In the RUN OPTION group, select if you want to export the selected sections for INVESTIGATION or DESIGN by spColumn. By default INVESTIGATION is selected. If you select DESIGN the REINFORCEMENT group changes to that as shown in the second dialog box image.

- In the STRUCTURAL MEMBER group, select if you want to export columns or piles. By default both are selected.

- In the MATERIAL PROPERTIES group, provide the required concrete and steel strengths to be applied to the exported items in spColumn.

- In the REINFORCEMENT group select the BAR SET you want spColumn to use.

- Depending on the RUN OPTION mode you have selected, enter the BAR NUMBER and BAR SIZE information. For DESIGN spColumn iterates the section starting with the minimum number of bars and minimum bar size until it arrives at a section that meets the requirements of the loads applied.

- Provide the CLEAR COVER. The cover provided acts as cover for longitudinal bars.

- Checking the ELIMINATE DUPLICATE LOADS checkbox eliminates multiple instances of the same load magnitude being exported to spColumn factored loads list.

- Choose the OK button.
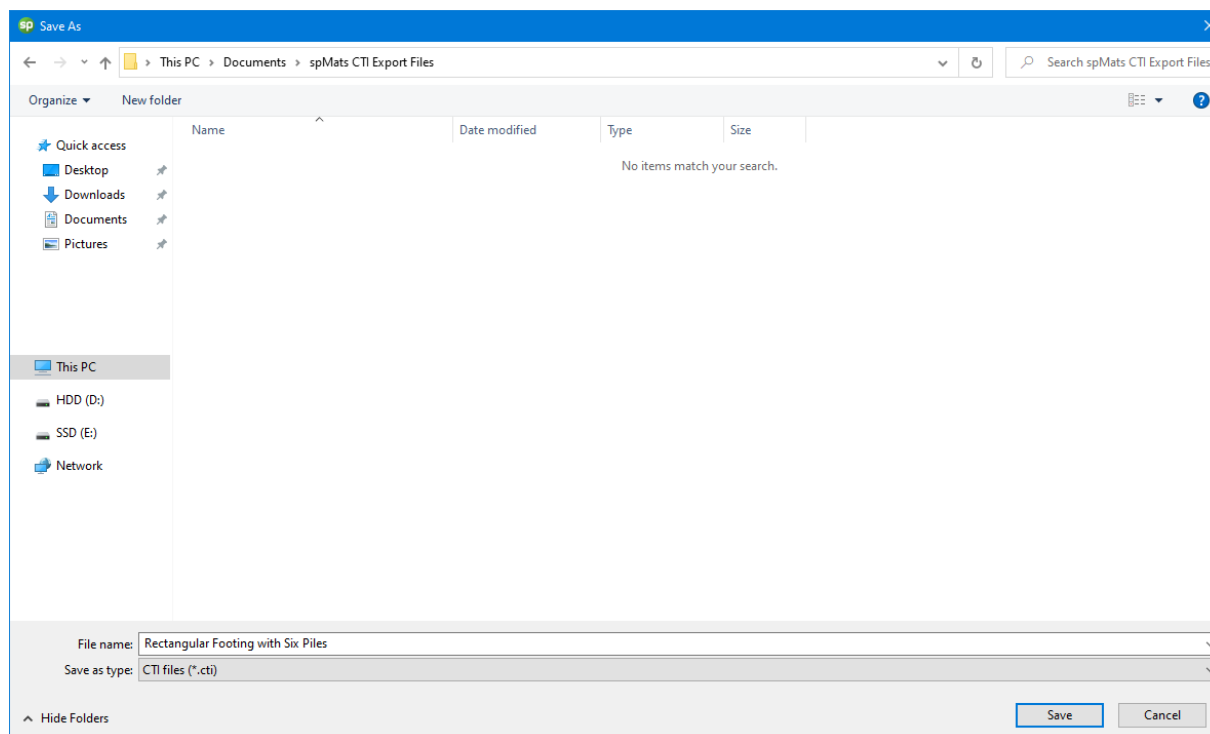
- Use the drop-down list and locate the folder where the file is to be saved. Once you locate the folder, the white listing area will display any other CTI files that have been saved in that folder. If you want to use a file name other than the default (same as input filename), double-click in the FILE NAME text box and type a filename. (You need not enter an extension since, by default, the program will affix the CTI extension to the filename.)

- Choose the SAVE button to finish exporting.

- Once spMats has finished exporting the files, you will be provided with a message box as shown below:
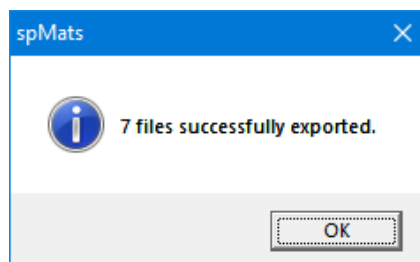
- Choose OK to return to spMats.

The exported files will be named in the following format:

Provided Filename-P (orC) – Object ID.cti

Where:

Provided Filename = the filename provided in the Save As dialog box.

P = is included in the name if the exported file contains a PILE section.

C = is included in the name if the exported file contains a COLUMN section.

Object ID = is the unique ID of the column/pile that is being exported.

Note: Steel piles will not be exported. When exporting for DESIGN, irregular piles (H-Type1 and H-Type2) will not be exported. Irregular piles (H-Type1 and H-Type2) will also not be exported for INVESTIGATION when the code used is CSA A23.3–14.

CHAPTER 6 — MODEL SOLUTION

Once the model creation and development are completed, the analysis can begin using the spMats FEM Solver by clicking on the **Solve** command. **Solve** Menu containing **Solver Options**, **Mesh Options** will appear on the **Left Panel**.

6.1   Solver Options

The SOLVER OPTIONS allow the user to enter input that is related to the uplift criteria for a specific foundation model and also select the required reinforcement calculation option. These options are important to consider carefully as they can be useful to manage and control the model behavior and corresponding analysis results.

Max. number of iterations

The MAX. NUMBER OF ITERATIONS **Solver Option** is a user input that is set to an initial default value of 10 by the Program and can be changed by the user based on the specific model needs. The initial default value of 10 that is set for number of iterations by the Program is to be considered as an upper-bound limit that the Program may not reach in most practical applications. In a given model, load combinations without an uplift will reach the solution at the first iteration. On the other hand, more than one iteration will be required for the load combinations with uplift. These load combinations will generally involve wind or earthquake load cases.

If uplift is detected at a given load combination during the run, the soil, and/or restraints such as nodal springs, and piles at the uplift nodes are eliminated and the program will iterate the modified

model until no further uplift nodes are encountered or until the set maximum number of iterations is reached, whichever comes first. During the run, if it is exceeded at any load combination, the solution will not be completed and an error message will appear after the run.
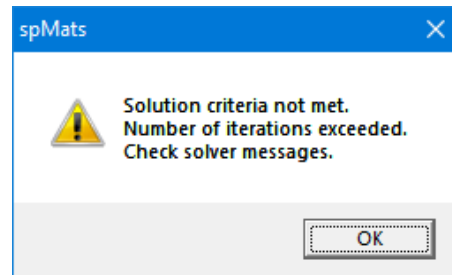
No output results and contour views will be generated. The Program will retain the **Solver Messages Dialog** which will display a "Solution error: Number of iterations exceeded" message for the first load combination that did not meet the criteria.

The Program deems a node as an uplift node based on the user input "*Uplift occurs when displacement exceeds*" within the **Solver Options**. More information on this Solver Option can be found below.

Uplift occurs when nodal displacement exceeds

The UPLIFT OCCURS WHEN DISPLACEMENT EXCEEDS **Solver Option** is a user input that is set to an initial default value of 0 by the Program and can be changed by the user based on the specific model needs. The initial default value set by the Program deems a node as an uplift node for a specific load combination if the node undergoes a positive nodal displacement (i.e. > 0)

The initial default value of 0 that is set by the Program should be kept as 0 for foundation models supported solely by soil including the pile-supported foundations where soil supporting the foundation slab is considered in analysis. This is because, in these models, setting this value greater than 0 will cause the soil-supported nodes to undergo uplift and since the soil cannot take tension, the Program results will not be reliable.

This Solver Option can be utilized for the models that are solely supported on piles that can also resist tension. The user may input a positive value for the UPLIFT OCCURS WHEN DISPLACEMENT EXCEEDS **Solver Option** in these models. This will allow the nodes to undergo positive (upward) displacements without the Program eliminating them from them model by deeming them uplift nodes. The piles at the nodes with positive displacement not exceeding this input value can then be accounted as tension piles in the analysis. It is essential for the user to ensure separately that the computed pile reactions whether compression (positive sign) or tension (negative sign) meets the pile capacity for a given project.

Max. allowed service displacement

The MAX. ALLOWED SERVICE DISPLACEMENT **Solver Option** is a user input that is set to an initial default value of 11 in. (English unit) or 279 mm (Metric unit) by the Program and can be changed by the user based on the specific model needs. The initial default value set by the Program is to be considered as an upper-bound limit that the Program may not reach in most practical applications.

The displacement value (positive or negative) at each node under service load combinations is checked against this value. During the run, if it is exceeded at any service load combination, the solution will not be completed and an error message will appear after the run.
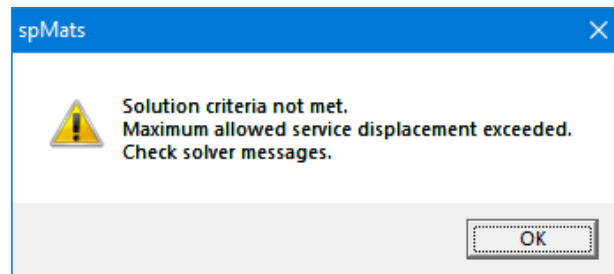
No output results and contour views will be generated. The Program will retain the Solver Messages Dialog which will display a "Solution error: Exceeded maximum allowed service displacement" message for the first service load combination that did not meet the criteria.

Min. allowed soil contact area

The MIN. ALLOWED SOIL CONTACT AREA **Solver Option** is a user input that is set to an initial default value of 50% by the Program and can be changed by the user based on the specific model needs. If uplift is detected during the run, the tributary area, A_{soil}, of all soil-supported nodes is computed. The program also computes the sum of the tributary area, $A_{uplift,soil}$, of all soil-supported nodes with uplift. The ratio of soil contact area is defined as $[A_{soil} - A_{uplift,soil}] / A_{soil}$. If during the solution, this ratio falls below the specified min. allowed soil contact area ratio for any load combinations, the solution will not be completed and an error message will appear during the run.
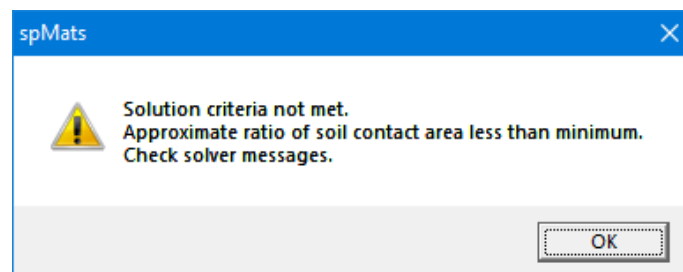
No output results and contour views will be generated. The Program will retain the **Solver Messages Dialog** which will display a "Solution error: Ratio of soil contact area is less than minimum = _ _ _%" message for the first load combination that did not meet the criteria.

Min. allowed active springs & piles

The MIN. ALLOWED ACTIVE SPRINGS & PILES **Solver Option** is a user input that is set to an initial default value of 0% by the Program and can be changed by the user based on the specific model needs. If uplift is detected during the run, the number of springs/piles in uplift NS/P, is computed. The program also computes the total number of springs/piles NS/P, total. The ratio of active springs/piles is defined as [NS/P, total - NS/P, uplift] / NS/P, total. If during the solution, this ratio falls below the specified min. allowed active springs & piles ratio for any load combinations, the solution will not be completed and an error message will appear during the run.

During the solution, if this ratio falls below the minimum ratio of active springs/piles for any load combination, the solution will not be completed and an error message will appear during the run.
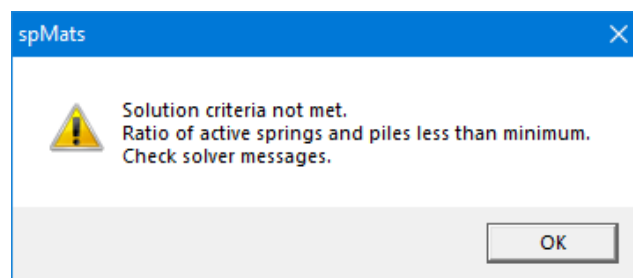
No results and contour views will be generated. The Program will retain the **Solver Messages Dialog** which will display a "Solution error: ratio of active springs and piles less than minimum = _ _ _%" message for the first load combination that did not meet the criteria.

Compute Required Reinforcement Based On

For each element, each of the design moments (M_{ux} and M_{uy}, top and bottom) is computed for a governing ultimate load combination. The governing load combination is the one that produces the maximum design moment M_{ux} and M_{uy} respectively in the X- and Y directions, and

separately for top and bottom reinforcement. The required area of reinforcement for an element may be computed based on:

- MAXIMUM MOMENT WITHIN AN ELEMENT (the maximum moment at any of the four element nodes.)

- AVERAGE MOMENT WITHIN AN ELEMENT (the average moment value at all element nodes.)

- The input is first verified for any inconsistencies or errors. If there is data missing or still required, a message box will be displayed.

- The program then switches control to the solver module. A message box reporting the progress and status of the solution is displayed.

6.2 Meshing Options

Unlike spMats v8.xx and earlier versions where the physical model contains the finite element mesh through the creation of grids at the first step of model creation, spMats v10.00 separates the physical model generation from the analytical model. It allows the user define parameters for the Finite Element mesh generation at the **Solve** Menu. The parameters that can be set by the user are discussed below:

Mesh

Max. allowed mesh size

The MAX. ALLOWED MESH SIZE **Analytical Model Option** is a user input that is set to an initial default value of 2.00 ft by the Program and can be changed by the user based on the specific model needs. As a rule of thumb, as the mesh size gets finer, the results become more accurate. The slab vertices, column, pile, node, restraint, and point load locations automatically get a finite element mesh grid. The factors that may be considered in max. allowed mesh size selection are the slab thickness, and the overall plan dimensions of the slab.

Max. allowed aspect ratio

The MAX. ALLOWED ASPECT RATIO **Analytical Model Option** is a user input that is set to an initial default value of 10 by the Program and can be changed by the user based on the specific model needs. As a rule of thumb, as the aspect ratio gets closer to 1, the results become more accurate.

Circle Segments

The CIRCLE SEGMENTS **Analytical Model Option** is a user input that is set to an initial default value of 36 by the Program and can be changed by the user to 24 or 48 based on specific model needs. This option is utilized for circular slabs.

Status
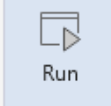
This section lists the number of elements, minimum and maximum element sizes in the model along with the maximum aspect ratio. If the maximum aspect ratio of an element in the model is greater than the defined maximum allowed aspect ratio then the maximum aspect ratio value is highlighted along with all the elements in the mesh whose aspect ratios exceed the maximum allowed aspect ratio. The user is warned before the program invokes the solver.
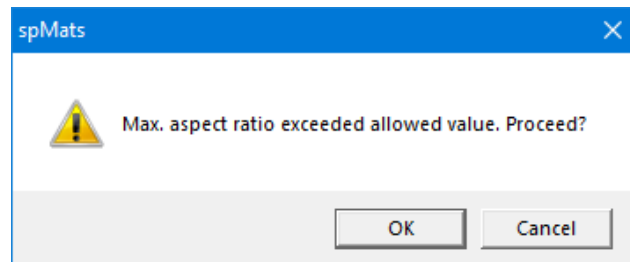
6.3   Running the Model

After inputting the model, the solver portion of the program can be executed using the **Run** button in the SOLVE panel.

SOLVE		
	Run	

∨ SOLVE OPTIONS

Max. number of iterations	10	
Max. allowed service displacement	11	in
Min. allowed soil contact area	50	%
Min. allowed active springs & piles	0	%
Uplift occurs when displacement exceeds	0	in

Compute required reinforcement based on
○ Maximum moment within an element
● Average moment within an element

∨ MESH OPTIONS

Max. allowed mesh size	2.00	ft
Max. allowed aspect ratio	10	
Circle segments	36	▼

∨ Status

Number of elements	1200	
Min. element size	2.00	ft
Max. element size	2.00	ft
Max. aspect ratio	1.00	

After you click the **Run** button, the program then switches control to the **Solver Module**. A dialog box reporting the progress and status of the solution is displayed.

```
sp  Please wait...                                                              ✕

Generating analysis model...

[████████████                                                    ]

09:41:17        Soil uplift nodes: none
09:41:17        Approximate ratio of soil contact area = 100.00 %
09:41:17 ....Load Combination 2 - Service
09:41:17        Maximum displacement = 0.000000000000E+000 in.
09:41:17        Minimum displacement = -4.553340422952E-001 in.
09:41:17        Soil uplift nodes: none
09:41:17        Approximate ratio of soil contact area = 100.00 %
09:41:17 ....Load Combination 3 - Service
09:41:17        Maximum displacement = 0.000000000000E+000 in.
09:41:17        Minimum displacement = -4.583042445879E-001 in.
09:41:18        Soil uplift nodes: none
09:41:18        Approximate ratio of soil contact area = 100.00 %
09:41:18 ....Load Combination 4 - Service. Nonlinear solution started
09:41:18 ......Iteration 0 (linear solution):
09:41:18        Maximum displacement = 1.151223070845E-003 in.
09:41:18        Minimum displacement = -4.617833552519E-001 in.
09:41:18        Soil uplift nodes: 2
                     31    1251
09:41:18        Approximate ratio of soil contact area = 99.92 %
09:41:18 ......Iteration 1:

                                                    Cancel        Close
```

When the solution is successfully completed, a contour map showing downward displacement envelope is displayed. If the solution procedure fails, a message box appears. Detailed information on the solution can be found in the **Solver Messages Dialog** under **Tables** Window.

CHAPTER

7

MODEL OUTPUT

The results of the finite element analysis and following design and code calculations are presented by spMats model output in two key categories.

1. **Text Results** tables including all relevant exact numerical results.
2. **Graphical Views** illustrating model behavior visually as an important and effective method to diagnose and verify expected results and critical parameters

A detailed description of all the features of both output types is given below.

7.1   Tabular Output

The Tabular output can be found both in the **Tables Module** and the **Reporter Module**. The **Tables Module** may be utilized to view and export the model output at any model development stage. The **Reporter Module** may be utilized to create, export and print customized reports when the design is finalized. Both modules have the same output sections. The differences being that of the **Reporter Module** contains the cover & contents, and screenshots sections. The **Tables Module** contains the **Solver Messages** section. The tables may be fully or partially output for all or for only selected nodes, and elements.

The program distinguishes between individual (service or ultimate) combination results and envelope results (which include the maximum values from all load combinations).

Contour views are also provided for selected output results to facilitate the graphical examination of results by the user. However, the tabular results reports should be used to make and finalize modeling decisions.

The Tabular output contains the following common input and results sections:

Project

This section contains the following input data blocks:

General Information

This block contains the information regarding to the **Project** such as FILE NAME, PROJECT NAME, DESIGN CODE, UNITS, DATE and TIME.

Solver Options

This block contains the information regarding to the **Solver Options** input entered by the user.

Definitions

This section contains the following **Definitions** input data subsections:

Grid Lines

This subsection contains the information regarding to the definitions input data for **Grids** utilized in the model. This subsection has data blocks for VERTICAL and HORIZONTAL grid input data.

Objects

This subsection contains the information regarding to the definitions input data for **Objects** utilized in the model. This subsection has data blocks for SLABS, COLUMNS, PILE – PROPERTIES, and PILE – GEOMETRY input data.

Properties

This subsection contains the information regarding to the definitions input data for **Properties** utilized in the model. This subsection has data blocks for SOIL, CONCRETE, REINFORCEMENT, and DESIGN PARAMETERS input data.

Restraints

This subsection contains the information regarding to the definitions input data for **Restraints** utilized in the model. This subsection has data blocks for NODAL SPRINGS, and SLAVED NODES input data.

Load Case/Combo.

This subsection contains the information regarding to the definitions input data for **Load Case/ Combo.** utilized in the model. This subsection has data blocks for LOAD CASES, SERVICE LOAD COMBINATIONS, and ULTIMATE LOAD COMBINATIONS input data.

Assignments

This section contains the following **Assignments** input data blocks:

Nodes

This block contains the **Nodes** input data such as NODE ID, COORDINATES, ASSIGNMENTS FOR COLUMN, PILE, and RESTRAINTS.

Slabs

This block contains the **Slabs** input data such as ID, LABEL, SHAPE and GEOMETRIC INFORMATION.

Columns

This block contains the **Columns** input data such as ID, LABEL, TYPE and COORDINATES.

Piles

This block contains the **Piles** input data such as ID, LABEL, TYPE and COORDINATES.

Point Loads

This block contains the **Point Loads** input data such as NODES ID, LOAD CASE, and LOAD VALUES.

Area Loads

This block contains the **Area Loads** input data such as SLABS ID, LOAD CASE, and LOAD VALUE.

Analytical Model

This section contains the following **Analytical Model** input data blocks:

Mesh

This block contains **Mesh** input data such as MAX. ALLOWED MESH SIZE, CIRCLE SEGMENTS and MESH STATUS.

Element Geometry

This block contains **Element Geometry** data such as ELEMENT NUMBER, NODE NUMBERS OF AN ELEMENT, PLAN DIMENSIONS, and THICKNESS OF AN ELEMENT.

Element Properties

This block contains **Element Properties** data labels such as SLAB, CONCRETE, SOIL, STEEL, DESIGN PARAMETER and whether the element LOADED or not.

Loaded Elements

This block contains **Loaded Elements** data showing the AREA LOAD VALUES per load case for all loaded elements.

Results

This section contains the **Envelope**, the **Service**, and the **Ultimate** level results.

Envelope

This section contains the **Envelope** results information on Nodal Displacements, Service Reactions, Ultimate Reactions, Soil Displacement & Pressure, Element Top Moment, Element Bottom Moment, Element Top Design Moment & Reinforcement, Element Top Design Moment & Reinforcement.

Nodal Displacements

This block contains the upward and downward service displacements, D_z, envelopes along with the governing combinations labels. Positive displacements are upward in the positive Z-direction.

Service Reactions

This subsection contains the information regarding to the envelope (minimum and maximum) **Service Reactions** for the nodes with soil, spring, pile, restraints, and slaved nodes from all service load combinations along with the governing load combinations labels.

Ultimate Reactions

This subsection contains the information regarding to the envelope (minimum and maximum) **Ultimate Reactions** for the nodes with soil, spring, pile, restraints, and slaved nodes from all ultimate load combinations along with the governing load combinations labels.

Soil Displacement and Pressure

For the elements with specified soil, this block contains the **Soil Displacement and Pressure** envelopes resulting from all service load combinations and all four element nodes. The governing load combination and the governing element node are also listed.

Element Top Moment

At each node of each element, this block reports envelope positive values of Wood-Armer design bending moments, M_{ux} and M_{uy}, in X and Y direction respectively, together with the moments M_{xx}, M_{yy}, M_{xy}, M_{r1}, angle of the major principal direction, and the ultimate load combination that produces the envelope design moments.

Element Bottom Moment

At each node of each element, this block reports envelope negative values of Wood-Armer design bending moments, M_{ux} and M_{uy}, in X- and Y-directions, respectively, together with the moments M_{xx}, M_{yy}, M_{xy}, M_{r1}, angle of the major principal direction, and the ultimate load combination that produces the envelope design moments.

Element Top Design Moment and Reinforcement

When steel design is based on average moment within an element, the table reports the ultimate load combination that produces the extreme average value of positive Wood-Armer design bending moments, M_{ux} and M_{uy}, together with the values of extreme design moments and the corresponding steel area requirement.

When steel design is based on maximum moment within an element, the table reports the node and the ultimate load combination for which the values of positive Wood-Armer design bending moments, M_{ux} and M_{uy}, are extreme together with the values of extreme design moments and the corresponding steel area requirement.

Element Bottom Design Moment and Reinforcement

When steel design is based on average moment within an element, the table reports the ultimate load combination that produces the extreme average value of negative Wood-Armer design bending moments, M_{ux} and M_{uy}, together with the values of extreme design moments and the corresponding steel area requirement.

When steel design is based on maximum moment within an element, the table reports the node and the ultimate load combination for which the values of negative Wood-Armer design bending moments, M_{ux} and M_{uy}, are extreme together with the values of extreme design moments and the corresponding steel area requirement.

Service

This section contains the **Service** level results information on **Force Vector**, **Displacement Vector**, **Reactions**, **Sum of Reactions**, and **Soil Displacement & Pressure**.

Force Vector

This block is output for individual **Service Load Combinations**. It lists the **Nodal Load Vector** that is actually used by the program for each load combination. The force, F_z, moment about X-axis, M_x, and moment about Y-axis, M_y, at each node includes the effects of loads concentrated at the node and the discretized effects of uniform surface loads. Positive forces are applied in the direction of the positive Z-axis (upward) and positive moments are determined using the right-hand rule.

Displacement Vector

This block is output for individual **Service Load Combinations**. It lists the **Displacement Vector** for each load combination. The table lists the displacement, D_z, and the two rotations about the X- and Y-axis, R_x and R_y, respectively. Positive displacement is in the positive Z-direction, and the right-hand rule is used to determine the direction of the rotations.

Reactions

This block is output for individual **Service Load Combinations**. It lists **Reactions** for the nodes with soil, spring, pile, restraints, and slaved nodes. Positive translational reactions (forces) are in the direction of the positive axes and positive rotational reactions (moments) are determined using the right-hand rule. The table also reports the sum of forces and moments (with respect to center of gravity) for applied loads and reactions due to restraints, slaved nodes, soil springs, nodal springs, and piles.

Sum of Reactions

This block is output for individual **Service Load Combinations**. It lists the **Sum of Forces and Moments** (with respect to center of gravity) for applied loads and reactions due to restraints, slaved nodes, soil springs, nodal springs, and piles.

Soil Displacement and Pressures

This block is output for individual **Service Load Combinations**. For the elements with specified soil, the displacement and pressure at all four nodes are listed. Since the soil is assumed tensionless, the pressure is set to zero for positive (upward) displacements.

Ultimate

This section contains the **Ultimate** level results information on **Force Vector**, **Displacement Vector**, **Reactions**, **Sum of Reactions**, and **Element Nodal Moments**.

Force Vector

This block is output for individual **Ultimate Load Combinations**. It lists the **Nodal Load Vector** that is actually used by the program for each load combination. The force, F_z, moment about X-axis, M_x, and moment about Y-axis, M_y, at each node includes the effects of loads concentrated at the node and the discretized effects of uniform surface loads. Positive forces are applied in the direction of the positive Z-axis (upward) and positive moments are determined using the right-hand rule.

Displacement Vector

This block is output for individual **Ultimate Load Combinations**. It lists the **Displacement Vector** for each load combination. The table lists the displacement, D_z, and the two rotations about the X- and Y-axis, R_x and R_y, respectively. Positive displacement is in the positive Z-direction, and the right-hand rule is used to determine the direction of the rotations.

Reactions

This block is output for individual **Ultimate Load Combinations**. It lists **Reactions** for the nodes with soil, spring, pile, restraints, and slaved nodes. Positive translational reactions (forces) are in

the direction of the positive axes and positive rotational reactions (moments) are determined using the right-hand rule. The table also reports the sum of forces and moments (with respect to center of gravity) for applied loads and reactions due to restraints, slaved nodes, soil springs, nodal springs, and piles.

Sum of Reactions

This block is output for individual **Ultimate Load Combinations**. It lists the sum of forces and moments (with respect to center of gravity) for applied loads and reactions due to restraints, slaved nodes, soil springs, nodal springs, and piles.

Element Nodal Moments

This block is output for individual **Ultimate Load Combinations**. At each of the four nodes of the element (i, j, k and l), listed are the bending moments (M_{xx} and M_{yy}), the twisting moment (M_{xy}) the equivalent principal moments (M_{r1} and M_{r2}), along with the principal angle, and equivalent design bending moments (M_{ux} and M_{uy}) at the top and bottom. Note that M_{xx} and M_{yy} are positive when they produce tension at the top and are referred to as moments along the X and Y-axes, respectively. For more information about these moments and the sign convention, refer to Fig 2-6 Element Nodal Moments.

7.2 Graphical Output

The Graphical output is organized into contour views that may be viewed, printed, exported or added to Report. Contour views are to facilitate the graphical examination of the results by the user. Contour views show the results in three distinct sections, namely, **Envelope**, **Service**, and **Ultimate**.

Envelope

This section contains the **Envelope** graphical results information on **Element Design Moment along X-direction**, **Element Design Moment along Y-direction**, **Element Design Reinforcement along X-direction**, **Element Design Reinforcement along Y-direction**, **Pressure Down**, **Displacement Up**, and **Displacement Down**.

Element Design Moment along X-direction, M_{ux}

This graphical view displays the contour envelopes for M_{ux} **Top** and M_{ux} **Bottom**. These design moments are along the X-direction and are utilized to compute the required top and bottom reinforcement along X-direction respectively.

Element Design Moment along Y-direction, M_{uy}

This graphical view displays the contour envelopes for M_{uy} **Top** and M_{uy} **Bottom**. These design moments are along the Y-direction and are utilized to compute the required top and bottom reinforcement along Y-direction respectively.

Element Reinforcement along X-direction, A_{sx}

This graphical view displays the contour envelopes for A_{sx} **Top** and A_{sx} **Bottom**. These reinforcements are along the X-direction and are the maximum of the required reinforcement due to design moment and minimum reinforcement ratio input by the user.

Element Reinforcement along Y-direction, A_{sy}

This graphical view displays the contour envelopes for A_{sy} **Top** and A_{sy} **Bottom**. These reinforcements are along the Y-direction and are the maximum of the required reinforcement due to design moment and minimum reinforcement ratio input by the user.

Pressure Down

For the elements with specified soil, this graphical view displays **Soil Pressure Down** envelopes resulting from all service load combinations and all four element nodes.

Displacement Up

This graphical view displays the upward service displacements, D_z, envelope.

Displacement Down

This graphical view displays the downward service displacements, D_z, envelope.

Service

This section contains the **Service** level graphical results information on **Displacement**, and **Soil Pressure**.

Displacement

This graphical view displays the displacement, D_z, contours for individual service load combinations.

Pressure

This graphical view displays soil pressure contours for individual service load combinations.

Ultimate

This section contains the **Ultimate** level graphical results information on Displacement, M_{xx}, M_{yy}, M_{xy}, M_{r1}, and M_{r2}.

Displacement

This graphical view displays the displacement, D_z, contours for individual ultimate load combinations.

M_{xx}

This graphical view displays the bending moment, M_{xx} contours for individual ultimate load combinations.

M_{yy}

This graphical view displays the bending moment, M_{yy} contours for individual ultimate load combinations.

M_{xy}

This graphical view displays the twisting moment, M_{xy} contours for individual ultimate load combinations.

M_{r1}

This graphical view displays the equivalent principal moment, M_{r1} contours for individual ultimate load combinations.

M_{r2}

This graphical view displays the equivalent principal moment, M_{r2} contours for individual ultimate load combinations.

A | APPENDIX

A.1   Default Load Case and Combination Factors

Each load is applied to the slab under one of the 26 (A through Z) load cases. The slab is analyzed and designed under load combinations. A load combination is the algebraic sum of each of the load cases multiplied by a load factor.

Load combinations are categorized into service-level and ultimate-level. For service-level load combinations, force vector, displacement vector, reactions, and soil displacements and pressures are output. For ultimate-level load combinations, force vector, displacement vector, reactions, and element nodal moments are output. The output is available only when solution criteria are met for all load combinations.

Basic load cases and the corresponding load factors for service and ultimate load combinations are provided as defaults in the input file templates (activated in SETTINGS / STARTUP DEFAULTS) to facilitate user's input. The default load cases and load combination factors should be modified as necessary at the discretion of the user.

The following default load cases are suggested: A – Dead (D), B – Live (L), C – Snow (S), D – Wind (W), and E – Earthquake (E). Service-level load values are considered for all load cases except case E, in all design codes, and case W, in ACI 318-14 and ACI 318-11 only, which are taken at ultimate-level. The suggested default load combination factors for service and ultimate load levels are shown below. Under service load level, allowable stress design load factors are

considered for calculating foundation pressure1. The user should introduce additional service-level load combinations as required, e.g., for checking displacement limits2.

Any load combination in the Tables below is VALID only if all Principal Loads exist for a given load combination. The Companion Load has no impact in the validity of a given load combination.

For ACI 318-14/11

- Service load combinations:

SERVICE LOAD COMBINATIONS - ACI 318 - 14 / 11						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
S1	1.00					D	-
S2	1.00	1.00				D, L	-
S3	1.00		1.00			D, S	-
S4	1.00	0.75	0.75			D, L, W	-
S5	1.00			0.60		D, W	-
S6	1.00			-0.60		D, W	-
S7	1.00				0.70	D, E	-
S8	1.00				-0.70	D, E	-
S9	1.00	0.75	0.75	0.45		D, L, W	S
S10	1.00	0.75	0.75	-0.45		D, L, W	S
S11	1.00	0.75	0.75	0.45		D, S, W	L
S12	1.00	0.75	0.75	-0.45		D, S, W	L
S13	1.00	0.75	0.75		0.525	D, L, E	S
S14	1.00	0.75	0.75		-0.525	D, L, E	S
S15	1.00	0.75	0.75		0.525	D, S, E	L
S16	1.00	0.75	0.75		-0.525	D, S, E	L
S17	0.60			0.60		D, W	-
S18	0.60			-0.60		D, W	-
S19	0.60				0.70	D, E	-
S20	0.60				-0.70	D, E	-

- Ultimate load Combinations:

					ULTIMATE LOAD COMBINATIONS - ACI 318 - 14 / 11		
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
U1	1.40					D	-
U2	1.20	1.60	0.50			D, L	S
U3	1.20	1.00	1.60			D, S	L
U4	1.20		1.60	0.50		D, S	W
U5	1.20		1.60	-0.50		D, S	W
U6	1.20	1.00	0.50	1.00		D, W	L, S
U7	1.20	1.00	0.50	-1.00		D, W	L, S
U8	1.20	1.00	0.20		1.00	D, E	L, S
U9	1.20	1.00	0.20		-1.00	D, E	L, S
U10	0.90			1.00		D, W	-
U11	0.90			-1.00		D, W	-
U12	0.90				1.00	D, E	-
U13	0.90				-1.00	D, E	-

For ACI 318-08/05

- Service load combinations:

SERVICE LOAD COMBINATIONS - ACI 318 - 08 / 05						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
S1	1.00					D	-
S2	1.00	1.00				D, L	-
S3	1.00		1.00			D, S	-
S4	1.00	0.75	0.75			D, L, S	-
S5	1.00			1.00		D, W	-
S6	1.00			-1.00		D, W	-
S7	1.00				0.70	D, E	-
S8	1.00				-0.70	D, E	-
S9	1.00	0.75	0.75	0.75		D, L, W	S
S10	1.00	0.75	0.75	-0.75		D, L, W	S
S11	1.00	0.75	0.75	0.75		D, S, W	L
S12	1.00	0.75	0.75	-0.75		D, S, W	L
S13	1.00	0.75	0.75		0.525	D, L, E	S
S14	1.00	0.75	0.75		-0.525	D, L, E	S
S15	1.00	0.75	0.75		0.525	D, S, E	L
S16	1.00	0.75	0.75		-0.525	D, S, E	L
S17	0.60			1.00		D, W	-
S18	0.60			-1.00		D, W	-
S19	0.60				0.70	D, E	-
S20	0.60				-0.70	D, E	-

- Ultimate load combinations:

ULTIMATE LOAD COMBINATIONS - ACI 318 - 08 / 05						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
U1	1.40					D	-
U2	1.20	1.60	0.50			D, L	S
U3	1.20	1.00	1.60			D, S	L
U4	1.20		1.60	0.80		D, S	W
U5	1.20		1.60	-0.80		D, S	W
U6	1.20	1.00	0.50	1.60		D, W	L, S
U7	1.20	1.00	0.50	-1.60		D, W	L, S
U8	1.20	1.00	0.20		1.00	D, E	L, S
U9	1.20	1.00	0.20		-1.00	D, E	L, S
U10	0.90			1.60		D, W	-
U11	0.90			-1.60		D, W	-
U12	0.90				1.00	D, E	-
U13	0.90				-1.00	D, E	-

For ACI 318-02

- Service load combinations:

SERVICE LOAD COMBINATIONS - ACI 318 - 02						
Load Combo	**DEAD D**	**LIVE L**	**SNOW S**	**WIND W**	**EQ E**	**Principle Loads**	**Companion Loads**
S1	**1.00**					**D**	-
S2	**1.00**	**1.00**				**D, L**	-
S3	**1.00**	1.00	**1.00**			**D, S**	L
S4	**1.00**	**1.00**	1.00	**1.00**		**D, L, W**	S
S5	**1.00**	**1.00**	1.00	**-1.00**		**D, L, W**	S
S6	**1.00**	1.00	**1.00**	**1.00**		**D, S, W**	L
S7	**1.00**	1.00	**1.00**	**-1.00**		**D, S, W**	L
S8	**1.00**	**1.00**	1.00		**0.70**	**D, L, E**	S
S9	**1.00**	**1.00**	1.00		**-0.70**	**D, L, E**	S
S10	**1.00**	1.00	**1.00**		**0.70**	**D, S, E**	L
S11	**1.00**	1.00	**1.00**		**-0.70**	**D, S, E**	L
S12	**0.60**			**1.00**		**D, W**	-
S13	**0.60**			**-1.00**		**D, W**	-
S14	**0.60**				**0.70**	**D, E**	-
S15	**0.60**				**-0.70**	**D, E**	-

* Ultimate load combinations:

Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
ULTIMATE LOAD COMBINATIONS - ACI 318 - 02							
U1	1.40					D	-
U2	1.20	1.60	0.50			D, L	S
U3	1.20	1.00	1.60			D, S	L
U4	1.20		1.60	0.80		D, S	W
U5	1.20		1.60	-0.80		D, S	W
U6	1.20	1.00	0.50	1.60		D, W	L, S
U7	1.20	1.00	0.50	-1.60		D, W	L, S
U8	1.20	1.00	0.20		1.00	D, E	L, S
U9	1.20	1.00	0.20		-1.00	D, E	L, S
U10	0.90			1.60		D, W	-
U11	0.90			-1.60		D, W	-
U12	0.90				1.00	D, E	-
U13	0.90				-1.00	D, E	-

For CSA A23.3-14/04/94

spMats reports soil pressure for service combinations only. The suggested service load combinations are based on CSA A23.3-94. To comply with clause N15.2.2 in Explanatory Notes on CSA A23.3-04 (Ref. [22]), the user should use appropriate load factors in conjunction with service level combinations to determine soil pressure for factored loads1.

- Service load combinations:

SERVICE LOAD COMBINATIONS - CSA A23.3-19 / 14 / 04 / 94						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
S1	1.00					D	-
S2	1.00	1.00				D, L	-
S3	1.00	1.00	1.00			D, S	L
S4	0.75	0.75	1.00	1.00		D, L, W	S
S5	0.75	0.75	1.00	-1.00		D, L, W	S
S6	0.75	0.75	1.00	1.00		D, S, W	L
S7	0.75	0.75	1.00	-1.00		D, S, W	L
S8	0.75	0.75	1.00		0.70	D, L, E	S
S9	0.75	0.75	1.00		-0.70	D, L, E	S
S10	0.75	0.75	1.00		0.70	D, S, E	L
S11	0.75	0.75	1.00		-0.70	D, S, E	L
S12	1.00			1.00		D, W	-
S13	1.00			-1.00		D, W	-
S14	1.00				0.70	D, E	-
S15	1.00				-0.70	D, E	-

- Ultimate load combinations for CSA A23.3-14:

ULTIMATE LOAD COMBINATIONS - CSA A23.3 - 14						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
U1	1.40					D	-
U2	1.25	1.50	1.00			D, L	S
U3	0.90	1.50	1.00			D, L	S
U4	1.25	1.50		0.40		D, L	W
U5	1.25	1.50		-0.40		D, L	W
U6	0.90	1.50		0.40		D, L	W
U7	0.90	1.50		-0.40		D, L	W
U8	1.25	1.00	1.50			D, S	L
U9	0.90	1.00	1.50			D, S	L
U10	1.25		1.50	0.40		D, S	W
U11	1.25		1.50	-0.40		D, S	W
U12	0.90		1.50	0.40		D, S	W
U13	0.90		1.50	-0.40		D, S	W
U14	1.25	0.50		1.40		D, W	L
U15	1.25	0.50		-1.40		D, W	L
U16	1.25		0.50	1.40		D, W	S
U17	1.25		0.50	-1.40		D, W	S
U18	0.90	0.50		1.40		D, W	L
U19	0.90	0.50		-1.40		D, W	L
U20	0.90		0.50	1.40		D, W	S
U21	0.90		0.50	-1.40		D, W	S
U22	1.00	0.50	0.25		1.00	D, E	L, S
U23	1.00	0.50	0.25		-1.00	D, E	L, S

- Ultimate load combinations for CSA A23.3-04:

ULTIMATE LOAD COMBINATIONS - CSA A23.3 - 04							
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
U1	**1.40**					**D**	-
U2	**1.25**	**1.50**	**0.50**			**D, L**	S
U3	**0.90**	**1.50**	**0.50**			**D, L**	S
U4	**1.25**	**1.50**		**0.40**		**D, L**	W
U5	**1.25**	**1.50**		**-0.40**		**D, L**	W
U6	**0.90**	**1.50**		**0.40**		**D, L**	W
U7	**0.90**	**1.50**		**-0.40**		**D, L**	W
U8	**1.25**	**0.50**	**1.50**			**D, S**	L
U9	**0.90**	**0.50**	**1.50**			**D, S**	L
U10	**1.25**		**1.50**	**0.40**		**D, S**	W
U11	**1.25**		**1.50**	**-0.40**		**D, S**	W
U12	**0.90**		**1.50**	**0.40**		**D, S**	W
U13	**0.90**		**1.50**	**-0.40**		**D, S**	W
U14	**1.25**	**0.50**		**1.40**		**D, W**	L
U15	**1.25**	**0.50**		**-1.40**		**D, W**	L
U16	**1.25**		**0.50**	**1.40**		**D, W**	S
U17	**1.25**		**0.50**	**-1.40**		**D, W**	S
U18	**0.90**	**0.50**		**1.40**		**D, W**	L
U19	**0.90**	**0.50**		**-1.40**		**D, W**	L
U20	**0.90**		**0.50**	**1.40**		**D, W**	S
U21	**0.90**		**0.50**	**-1.40**		**D, W**	S
U22	**1.00**	**0.50**	**0.25**		**1.00**	**D, E**	L, S
U23	**1.00**	**0.50**	**0.25**		**-1.00**	**D, E**	L, S

- Ultimate load combinations for CSA A23.3-94:

ULTIMATE LOAD COMBINATIONS - CSA A23.3- 94						
Load Combo	DEAD D	LIVE L	SNOW S	WIND W	EQ E	Principle Loads	Companion Loads
U1	1.25					D	-
U2	1.25	1.50	1.00			D, L	S
U3	0.85	1.50	1.00			D, L	S
U4	1.25	1.50		0.40		D, L	W
U5	1.25	1.50		-0.40		D, L	W
U6	0.85	1.50		0.40		D, L	W
U7	0.85	1.50		-0.40		D, L	W
U8	1.25	1.00	1.50			D, S	L
U9	0.85	1.00	1.50			D, S	L
U10	1.25		1.50	0.40		D, S	W
U11	1.25		1.50	-0.40		D, S	W
U12	0.85		1.50	0.40		D, S	W
U13	0.85		1.50	-0.40		D, S	W
U14	1.25			1.40		D, W	L
U15	1.25			-1.40		D, W	L
U16	0.85		0.50	1.40		D, W	S
U17	0.85		0.50	-1.40		D, W	S
U18	1.00			1.40		D, W	L
U19	1.00			-1.40		D, W	L
U20	1.00	0.50	0.50	1.40		D, W	S
U21	1.00	0.50	0.50	-1.40		D, W	S
U22	1.00	0.50	0.25		1.00	D, E	L, S
U23	1.00	0.50	0.25		-1.00	D, E	L, S

A.2   Import File Formats

Grid, load, and load combination data may be imported from a text file. The import file must be saved in pure ASCII (text) format. Data fields on each line should be separated by spaces or TABs. Comments or blank lines should not be placed within the import file. The first line of each text file must be the unique keyword associated with the input.

A.2.1   Grid Data:

```
GRID                   //Keyword
3                      //Number of vertical grid lines
1 2.0                  //Label of vertical grid line   X coordinate of vertical grid line
2 25.0
3 46.0
3                      //Number of horizontal grid lines
A 2.0                  //Label of horizontal grid line   Y coordinate of horizontal grid
line
B 18.0
C 36.0
```

A.2.2   Load Data:

```
LOADCASES              //Keyword
Dead DL                //Load case typeLoad case label
Live LL
Wind WL

                       //Space
POINTS                 //Keyword
1 2.0 2.0              //Point numberX CoordinateY coordinate
2 25.0 2.0
3 32.0 6.0
LOADS                  //Keyword
1 DL-50.0 12.0 7.0     //Point numberLoad case labelPzMxMy
1 LL-35.0 0.0 0.0
1 WL0.0 0.0 0.0
2 DL-50.0 0.0 0.0
2 LL-35.0 0.0 0.0
2 WL-32.0 0.0 0.0
3 DL0.0 0.0 0.0
3 LL0.0 0.0 0.0
3 WL52.0 21.0 8.0
```

A.2.3   Load Combination Data:

```
COMBINATIONS           //Keyword
0                      //Parameter indicating self-weight
3                      //Number of service load combinations
S1 1.000 1.000 0.000   //Label coefficient for each load case type
S2 1.000 1.000 1.000
S3 1.000 0.000 1.000
9                      //Number of ultimate load combinations
U1 1.400 0.000 0.000   //Label coefficient for each load case type
U2 1.200 1.600 0.000
U3 1.200 1.000 0.000
U4 1.200 0.000 0.800
```

```
U5 1.200 1.000  1.600
U6 0.900 0.000  1.600
U7 1.200 0.000 -0.800
U8 1.200 1.000 -1.600
U9 0.900 0.000 -1.600
```

A.3   Conversion Factors

A.3.1   English to SI

To convert from	To	Multiply by
in.	m (1000 mm)	0.025400
ft	m	0.304800
lb	N (0.001 kN)	4.448222
kip (1000 lbs)	kN	4.448222
plf (lb/ft)	kN/m	14.593904
psi (lb/in.^{2})	MPa	6.894757
ksi (kips/in.^{2})	MPa	6.894757
psf (lb/ft^2)	kN/m^2 (kPa)	47.88026
pcf (lb/ft^3)	kg/m^3	16.018460
ft-kips	kN • m	1.355818

A.3.2   SI to English

To convert from	To	Multiply by
m (1000 mm)	in.	39.37008
m	ft	3.28084
N (0.001 kN)	lb	0.224809
kN	kip (1000 lbs)	0.224809
kN/m	plf (lb/ft)	68.52601
MPa	psi (lb/in.^{2})	145.0377
MPa	ksi (kips/in.^{2})	0.145038
kN/m^2 (kPa)	psf (lb/ft^2)	20.88555
kg/m^3	pcf (lb/ft^3)	0.062428
kN • m	ft-kips	0.737562

A.4   Contact Information

Mail: StructurePoint
                     Customer Service
                     5420 Old Orchard Road
                     Skokie, IL 60077
                     USA

Phone: +1-847-966-4357

Fax: +1-847-966-1542

Web: http://www.StructurePoint.org
Sales: info@StructurePoint.org
Support: support@StructurePoint.org
Licensing: licensing@StructurePoint.org