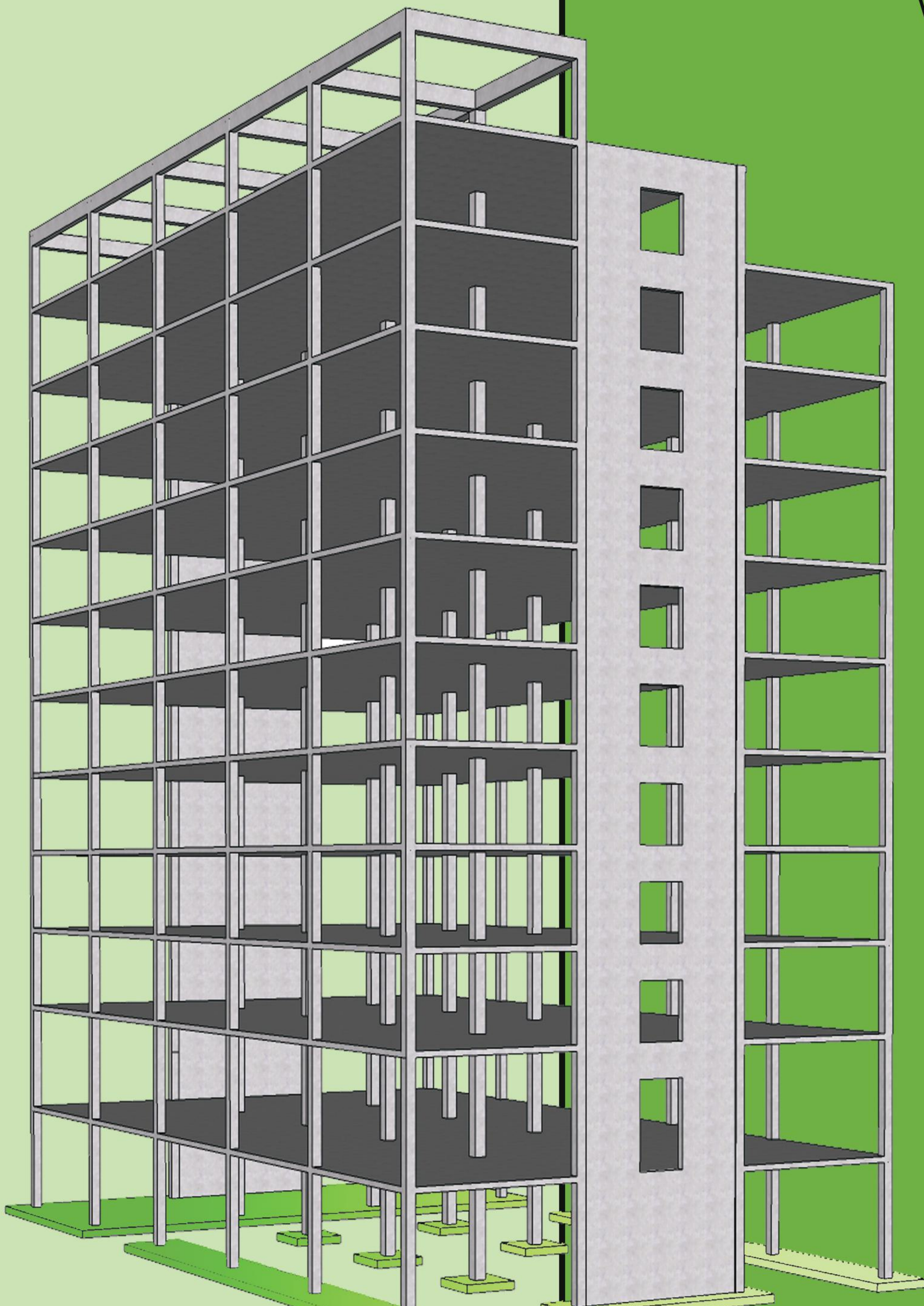
spMats

v10.50

Technical Manual



StructurePoint

Version 10.50

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: INTRODUCTION

Chapter 2: SOLUTION METHODS

Chapter 3: PROGRAM INTERFACE

Chapter 4: MODELING METHODS

Chapter 5: MODEL DEVELOPMENT

Chapter 6: MODEL SOLUTION

Chapter 7: MODEL OUTPUT

Chapter 8: EXAMPLES

Chapter: APPENDIX

CHAPTER

1

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 318-19/14/11/08/05/02 and CSA A23.3-19/14/04/94 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
- Templates of predefined and loaded models allowing the user to select and generate quick models for isolated and combined footings, mat foundations, pile supported foundations (pile caps), tank foundations, and equipment foundations.
- 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 (Pz, Mx, and My)
- 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, piles reactions, 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
- Smart snap assist features for improved modeling speed and accuracy
- 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
- 8, 12, 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

Any computer running Microsoft Windows 10 or Windows 11 operating system is sufficient to run the spMats program provided that .NET Framework v4.8 is installed. If it is not detected by the installation program, then it will be installed automatically.

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. It is recommended to run the model on the local computer hard drive for fastest response.

For instructions on how to purchase, download, install, license, and troubleshoot 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:

Italic	indicates a glossary item, or emphasizes a given word or phrase.
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	indicates a key combination. The plus sign indicates that you should press and hold the first key while pressing the second key, then release both keys. For example, "ALT + F" indicates that you 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.

CHAPTER

2

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:

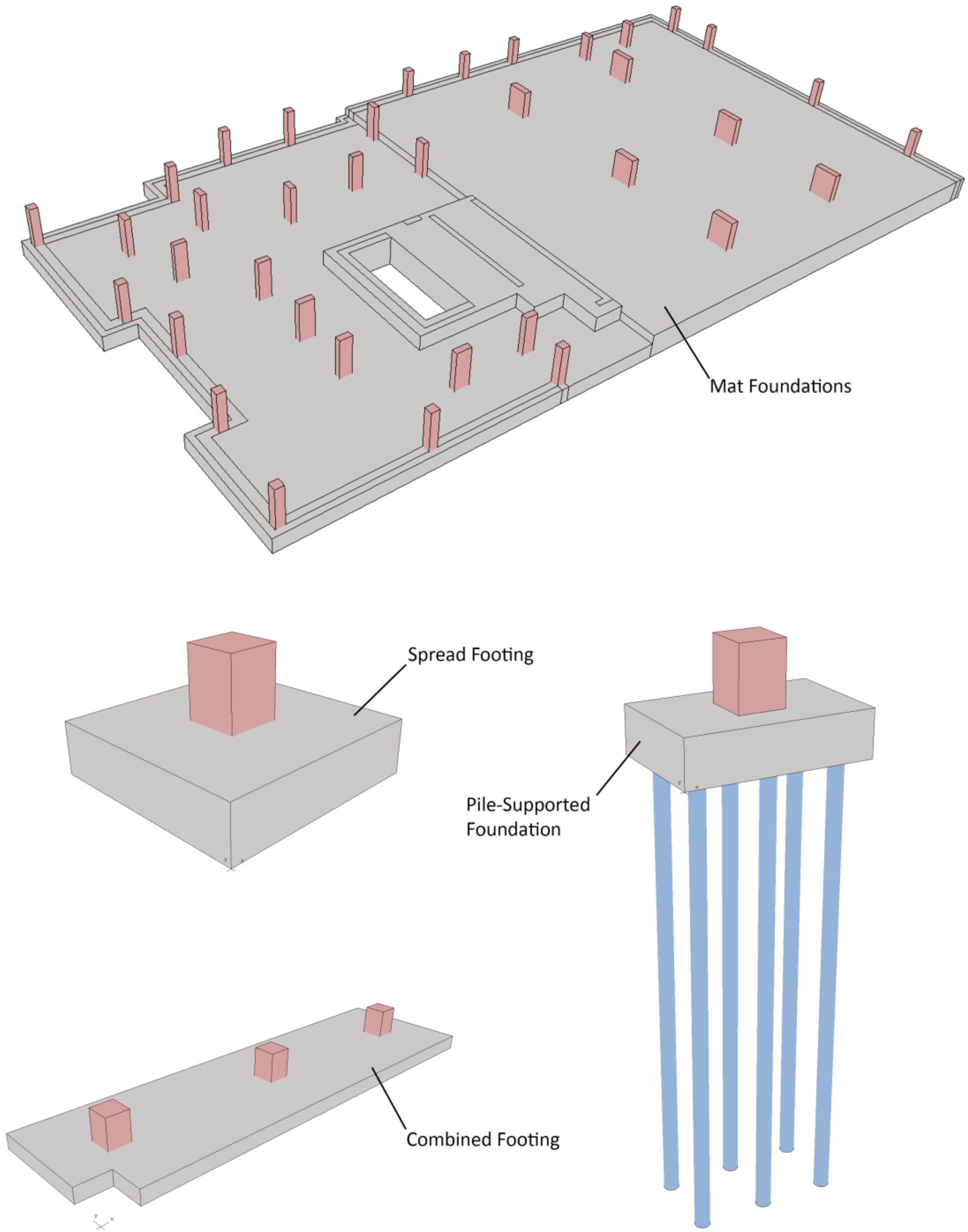


Figure 2.1 – Foundation Systems

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.

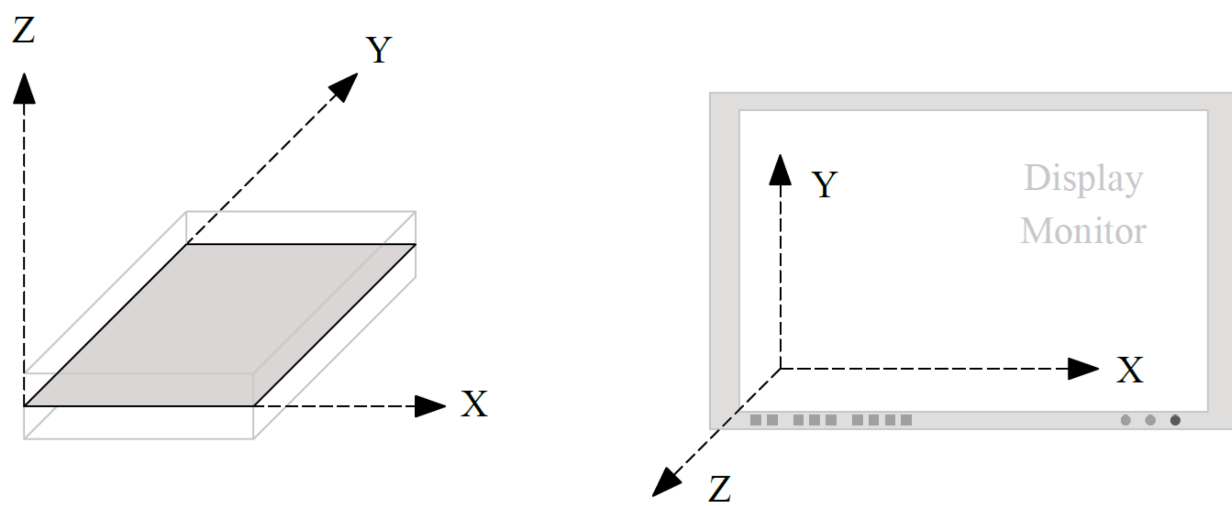


Figure 2.2 – Global Coordinate System

Local Coordinate System

There is no local coordinate system requirement in spMats.

2.2. Codes and Standards Provisions

2.2.1. Code Checks

2.2.1.1. Geometry Considerations

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

2.2.1.2. Material Considerations

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

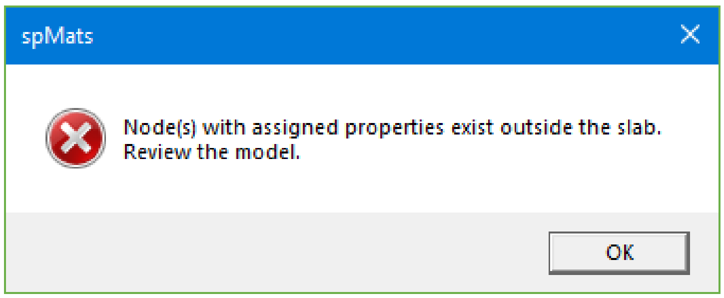
2.2.1.3. Loading Considerations

The program provides for load input in three basic categories: Foundation self weight, applied external point forces in the form of concentrated nodal loads, and applied surface loads in the form of area loads on elements. Modeling of loads and application convention is given in the Analysis Methods Section of this manual.

2.2.2. Geometry 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.

2.2.3. Definitions and Assumptions

The analysis of the reinforced concrete members performed by spMats conforms to the provisions of the Strength Design Method and Unified Design Provisions and is based on the following basic assumptions:

- All conditions of strength satisfy the applicable conditions of equilibrium and strain compatibility.
- Strain in the concrete and in the reinforcement is directly proportional to the distance from the neutral axis. In other words, plane sections normal to the axis of bending are assumed to remain plane after bending.
- An equivalent uniform rectangular concrete stress block is used with a maximum usable ultimate strain at the extreme concrete compression fiber equal to 0.003 for ACI codes and 0.0035 for CSA codes.
- Tensile strength of concrete in flexural calculations is neglected.
- For reinforcing steel, the elastic-plastic stress-strain distribution is used.
- The required reinforcement is determined based on bending for model elements.
- Assumptions related to the plate elements used in the Finite Element Method of Analysis method are outlined in Section 2.3.
- Detailed provisions and equations for the design codes supported by spMats are outlined in Section 2.4.

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.

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 2.3. 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.

[1] Reference [5]

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.

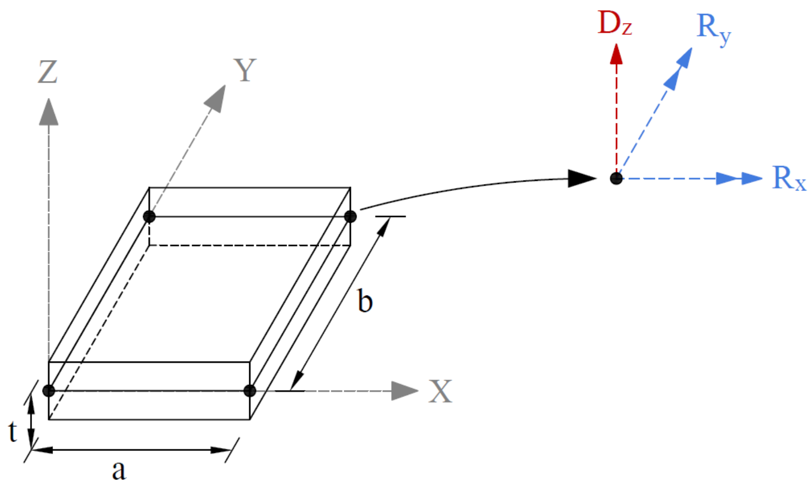


Figure 2.3 – Plate Finite Element

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.

2.3.2.1. 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.

$$P_z = K_s D_z \qquad \text{Eq. 2-1}$$

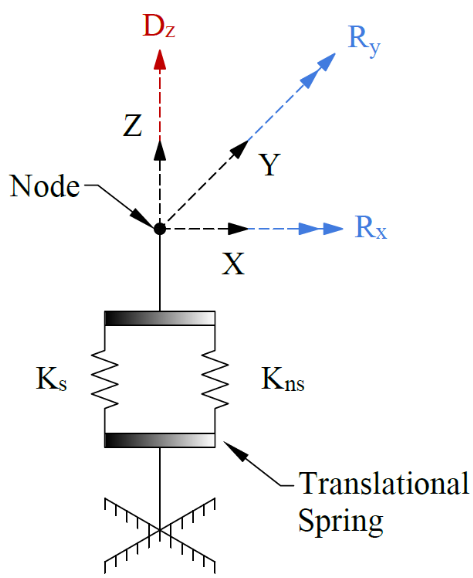


Figure 2.4 – Soil and Nodal Springs

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 2.4. 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.

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

2.3.2.2. 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:

$$K_p = \frac{Q_u}{S} \quad \text{Eq. 2-3}$$

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 $P_{all} \times A_p$, 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]:

$$S = \frac{D}{100} + \frac{Q_u L}{A_p E_p} \quad \text{Eq. 2-4}$$

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:

$$D = \sqrt{\frac{4A_p}{\pi}} \quad \text{Eq. 2-5}$$

2.3.2.3. 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.

2.3.2.4. Nodal Springs

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

2 Reference [8]

2.3.2.5. 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.

2.3.3. Modeling of Loads

An external load is applied as a point load or an area load. Positive forces are defined as forces in the positive direction of the global axes, and positive moments are defined in accordance with the right-hand rule. In other words, if the thumb of your right-hand points in the positive direction of an axis, curling the rest of your right-hand fingers defines the positive moment about that axis.

2.3.3.1. Point and Area Loads

External loads are applied as concentrated nodal loads and/or surface loads according to the sign convention shown in Figure 2.5.

A concentrated nodal load consists of a vertical load, P_z, and two concentrated moments about the X and Y axes, M_x and M_y. It should be noted that a positive vertical load is applied upward (in the positive Z-direction).

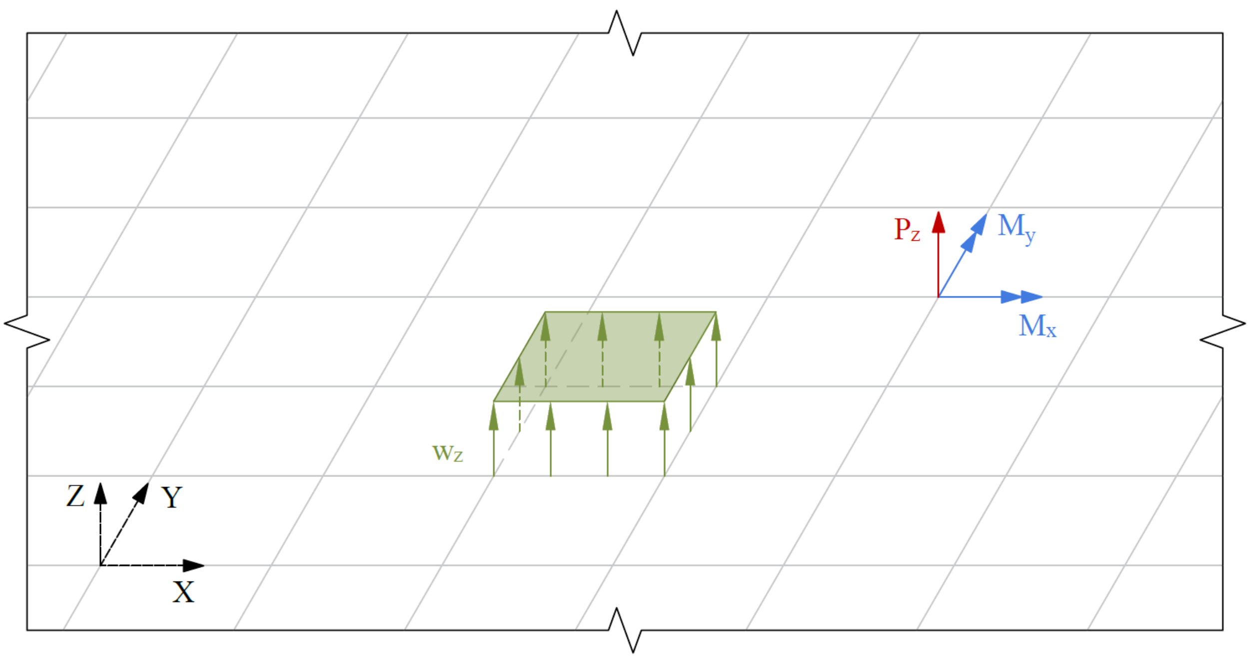


Figure 2.5 – Applied Loads

The uniform element surface load, w_z, applied over an element is internally discretized by the program into equivalent nodal loads as shown in Eq. 2-6:

$$\begin{Bmatrix} P_{iz} \\ M_{ix} \\ M_{iy} \\ P_{jz} \\ M_{jx} \\ M_{jy} \\ P_{kz} \\ M_{kx} \\ M_{ky} \\ P_{lz} \\ M_{lx} \\ M_{ly} \end{Bmatrix} = w_z \times \begin{Bmatrix} 1/4 \\ -b/24 \\ a/24 \\ 1/4 \\ b/24 \\ a/24 \\ 1/4 \\ -b/24 \\ -a/24 \\ 1/4 \\ b/24 \\ -a/24 \end{Bmatrix} \times a \times b \qquad \text{Eq. 2-6}$$

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.

2.3.3.2. Self Weight and Surcharge Loads

Self Weight can be applied automatically by selecting SELF WEIGHT for load CASE A in the spMats **Load Cases** dialog box, or it can be applied separately by the user as an external area load. In some cases, additional dead loads may be required beyond Self Weight, such as soil surcharge loads or stationary equipment loads. These additional dead loads can be defined as separate load cases, either including or excluding Self Weight.

2.3.3.3. Moving Loads

When modeling moving loads from lifting equipment such as forklifts, cranes, lifting rigs, or port container carriers, engineers must simulate realistic wheel loading scenarios on foundation slabs. Wheel loads can be modeled either as concentrated point loads applied to nodes or as distributed area loads over the tire contact patch. The latter approach is often preferred to reduce numerical stress concentrations, particularly in Finite Element Method of Analysis programs like spMats, by distributing forces across the actual tire footprint (e.g., 16.00–25 pneumatic wheels with approximate 360 in.^{2} contact area each).

To determine the worst-case effects on the slab, wheel placement scenarios should include loading at slab corners, edges, and center, where flexural and shear responses differ significantly. Because these lifting systems carry substantial loads and may have uneven wheel reactions, multiple load positions and configurations should be evaluated.

Due to limited standardized guidance, engineering judgment is essential, and developing multiple models with different wheel positions and load applications is advised to ensure conservative foundation design. Reference to ACI 360R and adaptation of static wheel data (e.g., from equipment datasheets) can aid in defining appropriate contact pressures and areas.

2.3.4. Determination of Internal Forces

The Program determines the element internal moments as shown below:

2.3.4.1. 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:

$$M_{r1} = M_{xx}\cos^2\theta + M_{yy}\sin^2\theta + M_{xy}\sin(2\theta) \quad \text{Eq. 2-7}$$

$$M_{r2} = M_{xx}\sin^2\theta + M_{yy}\cos^2\theta - M_{xy}\sin(2\theta) \quad \text{Eq. 2-8}$$

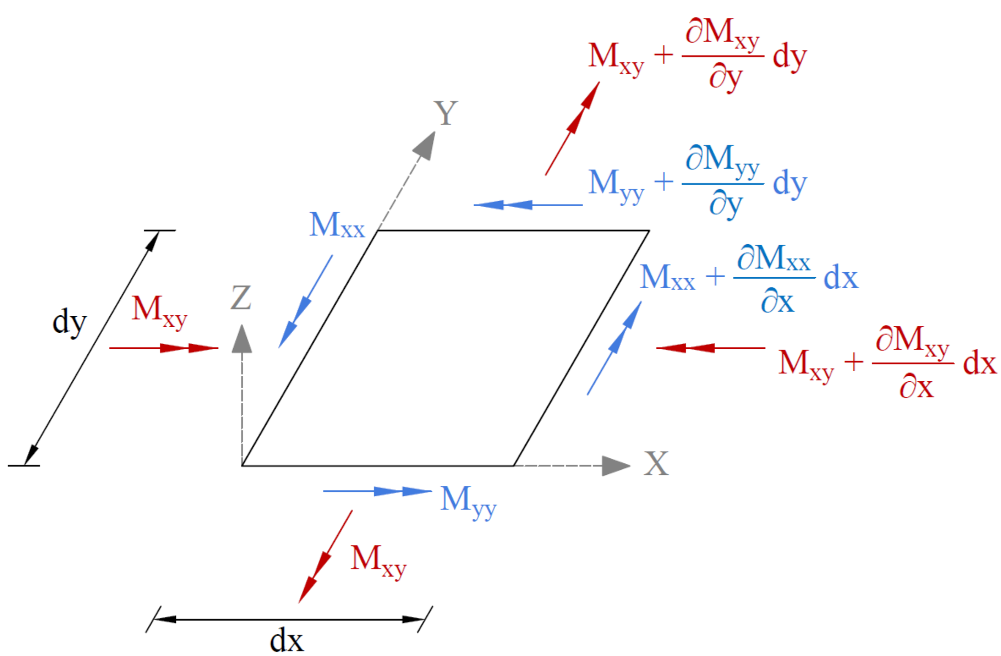


Figure 2.6 – Element Nodal Moments

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:

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

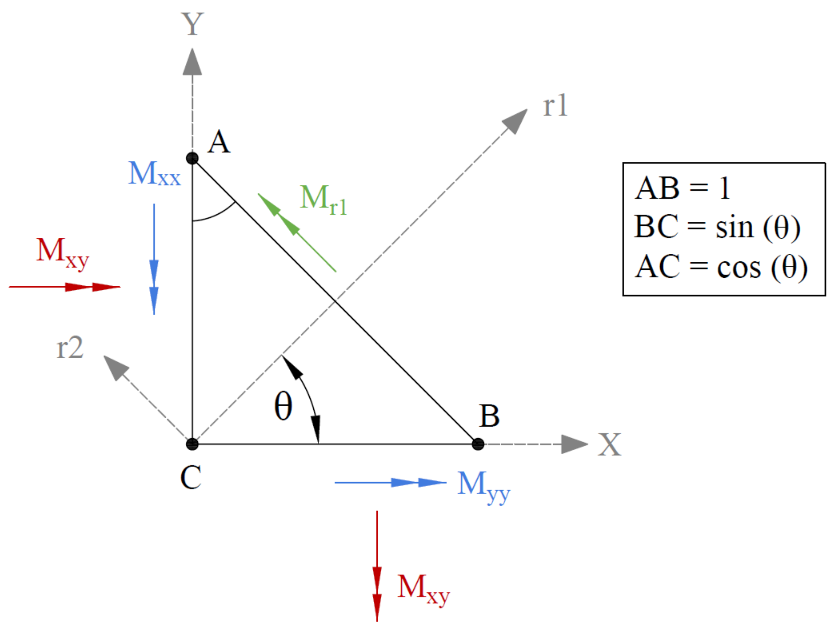


Figure 2.7 – Element Principal Moment

2.3.5. Displacements and Pressures

2.3.5.1. Displacements

The Program calculates the displacements, namely displacement, D_z, X-Rotation, R_x, and Y-Rotation, R_y, at all four nodes of an element for both service load and ultimate load combinations. Service-level nodal vertical displacement, 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 Solver Option and its default value is 10 in.

Under Solver 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.

2.3.5.2. 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.

2.4.1.1. 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:

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

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

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:

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

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

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

- For bottom reinforcement where negative moments produce tension:

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

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

However, if either M_{ux} or 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:

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

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

2.4.1.2. 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.

A_{sx} reinforcement is placed along X-direction and calculated based on the greater of the design moment, M_{ux}, code minimum reinforcement ratio, or the base reinforcement ratio (if selected) specified by the user under **Slab Design Criteria** in the DEFINITIONS Dialog Box.

Similarly, A_{sy} reinforcement is placed along Y-direction and calculated based on the greater of the design moment, M_{uy}, code minimum reinforcement ratio, or the base reinforcement ratio (if selected) specified by the user under **Slab Design Criteria** in the DEFINITIONS Dialog Box.

For computation of the required flexural reinforcement, the Program offers two options under **Slab Design 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:

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

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

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

   for ACI standards and

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

   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:

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

   which reinforcement ratio, ρ, equal to

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

   where factors m and g are calculated for ACI standards as

$$m = \frac{2M_u}{0.85\phi f_c' b d^2}$$ Eq. 2-24

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

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

$$m = \frac{2M_f}{\alpha_1 f_c' b d^2 \phi_c}$$ Eq. 2-26

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

where α_1 is defined as

$$0.67 \le \alpha_1 = 0.85 - 0.0015 f_c'$$ Eq. 2-28

and steel resistance factor[5] ϕ_s = 0.85 and concrete resistance factor[6], ϕ_c, that takes value of ϕ_c = 0.60 for CSA A23.3-94, ϕ_c = 0.65 for CSA A23.3-04, CSA A23.3-14 and CSA A23.3-19, and ϕ_c = 0.70 in case of precast concrete for CSA A23.3-04, CSA A23.3-14 and CSA A23.3-19 standards.

α_1 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 $0.85 - 0.0015 f_c'$ but not less than 0.67 for the CSA Standard[8].

4 ACI 318-19, 21.2; 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 A23.3-19, 8.4.3; CSA A23.3-14, 8.4.3; CSA A23.3-04, 8.4.3; CSA A23.3-94, 8.4.3

6 CSA A23.3-19, 8.4.2, 16.1.3; 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 318-19, 22.2.2.4.1; ACI 318-14, 22.2.2.4.1; ACI 318-11, 10.2.7.1; ACI 318-08, 10.2.7.1; ACI 318-05, 10.2.7.1; ACI 318-02, 10.2.7.1

8 CSA A23.3-19, 10.1.7; CSA A23.3-14, 10.1.7; CSA A23.3-04, 10.1.7; CSA A23.3-94, 10.1.7

2.4.1.3. 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 ε_{ty} + 0.003 for ACI 318-19 (not less than 0.004 for ACI 318-14 and prior codes).

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

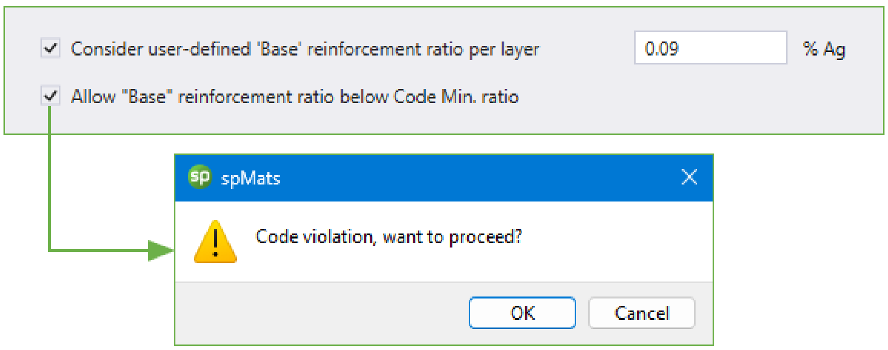
$$\frac{c}{d} \le \begin{cases} 0.8\left(\dfrac{700}{700+f_y}\right) & \text{for CSA A23.3-19} \\ \dfrac{700}{700+f_y} & \text{for CSA A23.3-14 and prior codes} \end{cases} \qquad \text{Eq. 2-29}$$

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.

[9] ACI 318-19, 7.3.3.1, Table 21.2.2; ACI 318-14, 7.3.3.1; 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 A23.3-19, 10.5.2; CSA A23.3-14, 10.5.2; CSA A23.3-04, 10.5.2; CSA A23.3-94, 10.5.2

2.4.1.4. Minimum Reinforcement

The Program automatically checks the minimum reinforcement ratio requirements in accordance with the selected design code. Additionally, the user may define a user-defined "Base" reinforcement ratio per layer under **Slab Design Criteria** in the DEFINITIONS dialog box. If the user permits the "Base" reinforcement ratio to fall below the code minimum, the Program will display a warning indicating a violation of code provisions. In such cases, the user is advised to proceed with caution.



For ACI, the minimum total reinforcement ratio required in foundations is equal to[10]:

$$\frac{A_{s,min}}{A_g} = 0.0018 \qquad \text{For ACI 318-19}$$

$$\frac{A_{s,min}}{A_g} = \begin{cases} 0.002 & f_y < 60 \text{ ksi} \\ \text{Greater of } \begin{cases} \dfrac{0.0018 \times 60{,}000}{f_y} \\ 0.0014 \end{cases} & f_y \geq 60 \text{ ksi} \end{cases} \qquad \text{For ACI 318-14}$$

$$\frac{A_{s,min}}{A_g} = \begin{cases} 0.002 & f_y = 40 \text{ ksi or } 50 \text{ ksi} \\ 0.0018 & f_y = 60 \text{ ksi} \\ \dfrac{0.0018 \times 60{,}000}{f_y} & f_y > 60 \text{ ksi} \end{cases} \geq 0.0014 \qquad \text{For ACI 318-11/08/05/02}$$

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

[10] ACI 318-19, 13.3.4.4, 8.6.1.1; 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 318-05, 10.5.4, 7.12.2.1; ACI 318-02, 10.5.4, 7.12.2.1

[11] CSA A23.3-19, 15.4.1, 10.5.1.2(a), 7.8.1; 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

2.4.2. One-Way (Beam) Shear

spMats does not perform in-plane or out-of-plane shear design. The program's finite element formulation is based on the Kirchhoff thin plate theory, which neglects transverse shear deformations and is primarily suitable for punching shear (two-way) design (see Section 2.3.1 for the complete list of assumptions). For very thick slabs or mat foundations where shear deformations may be significant, a more general finite element formulation, such as Mindlin's plate theory, may be required. In such cases, one-way shear design should be performed separately. As an alternative, for one-way shear calculation, the user may determine the location of critical sections and use the Soil Pressure Data and forces from spMats output to compute shear force and shear stress manually. An example of this alternative approach is provided in "Reinforced Concrete Shear Wall Foundation (Strip Footing) Analysis and Design" Design Example.

2.4.3. Two-Way (Punching) Shear

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

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

2.4.3.1. ACI Standard (318-14/11/08/05/02)

For ACI standards, the following condition is used:

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

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]:

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

[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

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

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

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_o = 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,

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

2.4.3.2. CSA Standard (A23.3-14/04/94)

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

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

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

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

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

[13] CSA A23.3-94, 13.4.4

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

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

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

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:

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

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).

[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

2.5. Detailing Provisions

2.5.1. Reinforcement Selection

The Program reports the area of flexural reinforcement per unit width based on four criteria, namely, the required reinforcement per analysis result (Design Moment), the minimum and maximum reinforcement ratio requirements as specified by the selected design code (Code Min. & Code Max.), and the user-defined “Base” reinforcement ratio input (“Base”). The final selection of reinforcement per layer is determined through the following process:

First, the Program determines whether the face (Top or Bottom) of the Element in consideration is in tension or in compression.

2.5.1.1. Element Face in Compression

If the face of the Element in consideration is in compression, this means the Design Moment equals to zero, and the reinforcement required per Design Moment shall subsequently be equal to zero. In this case, the Code Min. or Code Max requirements need not be checked.

- If “Base” is not checked, the governing reinforcement amount shall be equal to zero as it is per the Design Moment which is itself equal to zero.
- If “Base” is checked and user-defined “Base” reinforcement ratio input is greater than 0.00%, the governing reinforcement shall be based on “Base” reinforcement specified. If “Base” is checked and user-defined “Base” reinforcement ratio input is equal to 0.00%, the governing reinforcement amount shall be equal to zero as it is per the Design Moment which is itself equal to zero. Note that checking or unchecking the checkbox to Allow “Base” reinforcement ratio below Code Min. ratio shall not have any impact here as Code check for Min. and Max. is not required in compression face.

2.5.1.2. Element Face in Tension

If the face of the Element in consideration is in **tension**, the following steps are being considered in order to determine reinforcement selection.

- If "Base" is not checked, the governing reinforcement is per the Design Moment given that it is greater than or equal to Code Min. and less than or equal to Code Max. If the reinforcement required per the Design Moment is less than Code Min., then Code Min. governs the reinforcement selection. If the reinforcement required per the Design Moment is greater than Code Max., then, the reinforcement cannot be computed.

- If "Base" is checked, the following methodology is utilized for reinforcement selection:

  - If the user has not checked the checkbox to Allow "Base" reinforcement ratio below Code Min. ratio, the program compares "Base" reinforcement with required reinforcement per Design Moment:

    - If required reinforcement per Design Moment is greater than or equal to the "Base" reinforcement, the governing reinforcement selection shall be per the Design Moment given that it is greater than or equal to Code Min. and less than or equal to Code Max. If the reinforcement required per the Design Moment is less than Code Min., then Code Min. governs the reinforcement selection. If the reinforcement required per the Design Moment is greater than Code Max., then, the reinforcement cannot be computed.

    - If required reinforcement per Design Moment is less than the "Base" reinforcement, the governing reinforcement selection shall be per the "Base" reinforcement given that it is greater than or equal to Code Min. and less than or equal to Code Max. If the "Base" reinforcement is less than Code Min., then Code Min. governs the reinforcement selection. If the "Base" reinforcement is greater than Code Max., then, the "Base" reinforcement exceeds Code Max. Note is displayed in Results.

- If the user has checked the checkbox to Allow "Base" reinforcement ratio below Code Min. ratio, the program shall bypass the Code Min. check while displaying a warning message indicating a violation of Code Provision and ask the user whether to proceed or not. Then, the program compares "Base" reinforcement with required reinforcement per Design Moment:

  - If required reinforcement per Design Moment is greater than or equal to the "Base" reinforcement, the governing reinforcement selection shall be per the Design Moment given that it is less than or equal to Code Max. If the reinforcement required per the Design Moment is greater than Code Max., then, the reinforcement cannot be computed.

  - If required reinforcement per Design Moment is less than the "Base" reinforcement, the governing reinforcement selection shall be per the "Base" reinforcement given that it is less than or equal to Code Max. If the "Base" reinforcement is greater than Code Max., then, the "Base" reinforcement exceeds Code Max. Note is displayed in Results.

2.5.2. Slabs with Single Reinforcement Layer

Slabs on ground with a single layer of reinforcing or welded wire mesh are common foundation elements. In order to simulate single-layer reinforcing in a slab on grade, 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.

2.5.3. Slabs with Fiber Reinforcement

Ground-supported slabs in industrial and residential floors may be specified with fiber reinforcement instead 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.

2.5.4. Unreinforced Plain Concrete Slabs

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. Special Topics

2.6.1. Industrial Foundation Applications

spMats offers a robust and flexible solution for modeling and designing reinforced concrete foundations used in a wide range of industrial applications. These include pile-supported foundations for transmission towers, distribution and substations, wind turbines, and pipe racks, as well as soil-supported footings for solar panels, tank equipment, industrial equipment, and refrigerated container racks (reefer racks). Utilizing the Finite Element Method for analysis, spMats enables engineers to analyze complex loading scenarios, including gravity, wind, seismic, and overturning loads with high accuracy.

The following case studies highlight the capabilities of spMats in addressing diverse challenges within the industrial sector:

- Tall Wind Turbine Pile Supported Concrete Foundation
- Transmission Tower Concrete Pile Cap Foundation
- Industrial Plant Pipe Rack Concrete Foundation
- Telecommunication Tower Concrete Foundation
- Ground Mounted PV Solar Panel Concrete Foundation
- Short Wind Turbine Tower Concrete Foundation
- Tank Equipment Concrete Mat Foundation
- Refrigerated Container Racks (Reefer Racks) Concrete Foundation

2.6.2. Liquid Containing Tank Foundation

Reinforced concrete tanks are commonly used in water, wastewater treatment, agricultural, chemical, and process facilities to safely contain liquids, often under challenging chemical and environmental conditions. These structures are subject to unique and varied loading scenarios during their service life - including fill and empty cycles in outages between operating periods. This results in several loading scenarios to represent hydrostatic pressures, and backfill conditions that require careful analysis using multiple load combinations. spWall can be used to model and design the vertical tank walls. Then the base mat foundation can be analyzed in spMats using the wall reactions exported directly from the spWall model. This integration enables a seamless transfer of nodal forces and ensures consistency between wall and foundation design, while maintaining flexibility to evaluate construction stages and service phases accurately.

More information about this topic can be found in "Liquid Containing Rectangular Concrete Tank Design" Case Study.

2.6.3. Cantilever Retaining Wall Foundation

Cantilever retaining walls are commonly used in site development to retain soil and resist lateral earth pressures. These systems feature reinforced concrete vertical stem and foundation divided into heel and toe regions, designed to resist overturning, sliding, and bearing failure. spWall can be used to model and analyze the wall stem, accounting for soil loads, backfill pressures, and surcharge loads. The resulting stem reactions can then be transferred into spMats, where the foundation slab is modeled to evaluate bearing pressures, deflections, and reinforcement requirements under multiple load combinations.

More information about this topic can be found in "Reinforced Concrete Tapered Cantilever Retaining Wall Foundation Analysis and Design" Design Example and "Reinforced Concrete Cantilever Retaining Wall Foundation Analysis and Design" Case Study.

2.6.4. Combined Footings and Grade Beam Foundations

Combined footings, grade beams, and other similar foundations can be analyzed using different methods depending on the project needs, complexity, and design assumptions. spMats applies the Finite Element Method of Analysis by modeling the footing as a mesh of plate elements supported by linear springs, offering flexibility in representing varying soil conditions, two-way behavior, and foundation geometry. In contrast, spBeam uses a series of vertical support springs to simulate the Beam-on-Elastic-Foundation (BOEF) approach, ideal for evaluating one-way behavior along the footing length. While both tools can produce closely aligned results under typical loading conditions, their modeling capabilities differ in how they handle reinforcement distribution, soil pressures, and two-way behavior. For many practical scenarios, including those with irregular geometry or varying load placement, the choice of method may influence the level of detail captured in the design process. "Combined Footing - Finite Element Analysis vs Beam-on-Elastic-Foundation" Case Study highlights how each method performs under similar conditions, offering engineers insight into selecting the most appropriate approach for a given application.

2.6.5. Pile Cap Foundation Analysis and Design

Thick concrete foundation mats 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
- Pile (Caisson) Analysis and Design for Bridge Pier Foundations

2.6.6. Piles and Caissons Analysis and Design

The latest enhancements in spMats offer advanced tools for modeling and analyzing pile-supported foundations with greater clarity and control. Users can now define allowable service reactions for piles, enabling visual feedback in the graphical output: compression reactions are shown with upward arrows, tension with downward arrows, and reaction values exceeding the specified limits are flagged in red, while compliant values are shown in blue. A warning icon in the interface also highlights any piles that surpass allowable reactions. In the tabular output, spMats clearly lists each pile's label, Pile ID, assigned node, and Node ID alongside its service and ultimate reactions, providing a comprehensive and traceable summary of foundation behavior. Additionally, spMats supports direct integration with spColumn by exporting pile and column data - including geometry, materials, and applied loads - into spColumn Text Input (CTI) files. These files allow engineers to transition into detailed design and capacity checks for piles using spColumn's interaction diagrams and 3D failure surfaces. Whether piles are pinned or fixed to the cap, spMats and spColumn together enable full-cycle analysis and investigation, making them ideal tools for optimizing pile design.

Related Topics in the Manual and other resources:

- Modeling of Piles.
- Defining Piles.
- Assigning Piles.
- Editing Piles.
- Exporting Piles Data to spColumn.
- Pile Graphical Output: Envelope, Service, and Ultimate.
- "Pile (Caisson) Analysis and Design for Bridge Pier Foundations" Case Study.

2.6.7. Micropile Foundations Analysis and Design

Micropiles are small-diameter (typically less than 12 inches), high-capacity grouted piles used to support structural foundations, especially in constrained or difficult ground conditions. Constructed by drilling a borehole, placing reinforcement (usually a steel bar or casing), and injecting grout, micropiles are ideal where access is limited, soil conditions are poor, or traditional deep foundations are not feasible. They are commonly used for retrofitting existing foundations, supporting new structures with restricted clearances, and transferring loads through soft or unstable soils to more competent strata. Micropiles are often designed in clusters to share compressive, tensile, and lateral loads. spMats provides an efficient and accurate platform to model micropile-supported foundations by treating micropiles as vertical springs within a finite element model. This allows engineers to simulate load transfer, assess pile reactions, and verify performance under various load combinations, making spMats a practical tool for incorporating micropile systems into foundation analysis and design. Furthermore, spMats integrates with spColumn, enabling users to export pile reactions and cross-sectional properties for detailed strength design and capacity investigation of micropiles under the applied loads. More information about this topic can be found in "Pile (Caisson) Analysis and Design for Bridge Pier Foundations" Case Study.

2.7. References

[1] Wood, R.H., “The Reinforcement of Slabs in Accordance with a Predetermined Field of Moments,” Concrete, February 1968.

[2] Mills, H.B., Armer, G.S.T., and R.H. Wood, Correspondence regarding Article “The Reinforcement of Slabs in Accordance with Predetermined Field Moment,” Concrete, August 1986

[3] Gupta, A.K. and Sen Siddhartha, “Design of Flexural Reinforcement in Concrete Slabs,” Journal of the Structural Division, ASCE, Vol. 103, No. St4, April 1972.

[4] Park, R. and W.L. Gamble, Reinforced Concrete Slabs, John Wiley & Sons, 2000. (Chapter 5, Section 5-4) pp. 207-231.

[5] Zienkiewicz, O.C. and R.L. Taylor, The Finite Element Method, McGraw-Hill Book Company, Fourth Edition, 1988.

[6] Bowles, Joseph F., Foundation Analysis and Design, McGraw-Hill Book Company, Fifth Edition, 1996.

[7] ACI 336.2R-88, “Suggested Analysis and Design Procedures for Combined Footings and Mats.”

[8] Prakash, S., and H.D. Sharma, Pile Foundations in Engineering Practice, John Wiley & Sons, 1990, pp. 251.

[9] Building Code Requirements for Structural Concrete (ACI 318-19) and Commentary (ACI 318R-19), American Concrete Institute, 2019.

[10] Building Code Requirements for Structural Concrete (ACI 318-14) and Commentary (ACI 318R-14), American Concrete Institute, 2014.

[11] Building Code Requirements for Structural Concrete (ACI 318-11) and Commentary (ACI 318R-11), American Concrete Institute, 2011.

[12] Building Code Requirements for Structural Concrete (ACI 318-08) and Commentary (ACI 318R-08), American Concrete Institute, 2008.

[13] Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05), American Concrete Institute, 2005.

[14] Building Code Requirements for Structural Concrete (ACI 318-02) and Commentary (ACI 318R-02), American Concrete Institute, 2002.

[15] CSA A23.3-19, Design of Concrete Structures, Canadian Standards Association, 2019.

[16] CSA A23.3-14, Design of Concrete Structures, Canadian Standards Association, 2014.

[17] CSA A23.3-04, Design of Concrete Structures, Canadian Standards Association, 2004.

[18] CSA A23.3-94, Design of Concrete Structures, Canadian Standards Association, 1994 (Reaffirmed 2000).

[19] Minimum Design Loads for Buildings and Other Structures (ASCE 7-10), American Society of Civil Engineers, 2010.

[20] Minimum Design Loads for Buildings and Other Structures (ASCE 7-05), American Society of Civil Engineers, 2006.

[21] National Building Code of Canada (NBCC-15), Canadian Commission on Buildings and Fire Codes, National Research Council of Canada, 2015.

[22] National Building Code of Canada (NBCC-05), Canadian Commission on Buildings and Fire Codes, National Research Council of Canada, 2005.

[23] National Building Code of Canada (NBCC-95), Canadian Commission on Buildings and Fire Codes, National Research Council of Canada, 1995.

[24] International Building Code (2016 IBC), International Code Council, 2016.

[25] International Building Code (2012 IBC), International Code Council, 2012.

[26] International Building Code (2009 IBC), International Code Council, 2009.

[27] International Building Code (2006 IBC), International Code Council, 2006.

[28] International Building Code (2003 IBC), International Code Council, 2003.

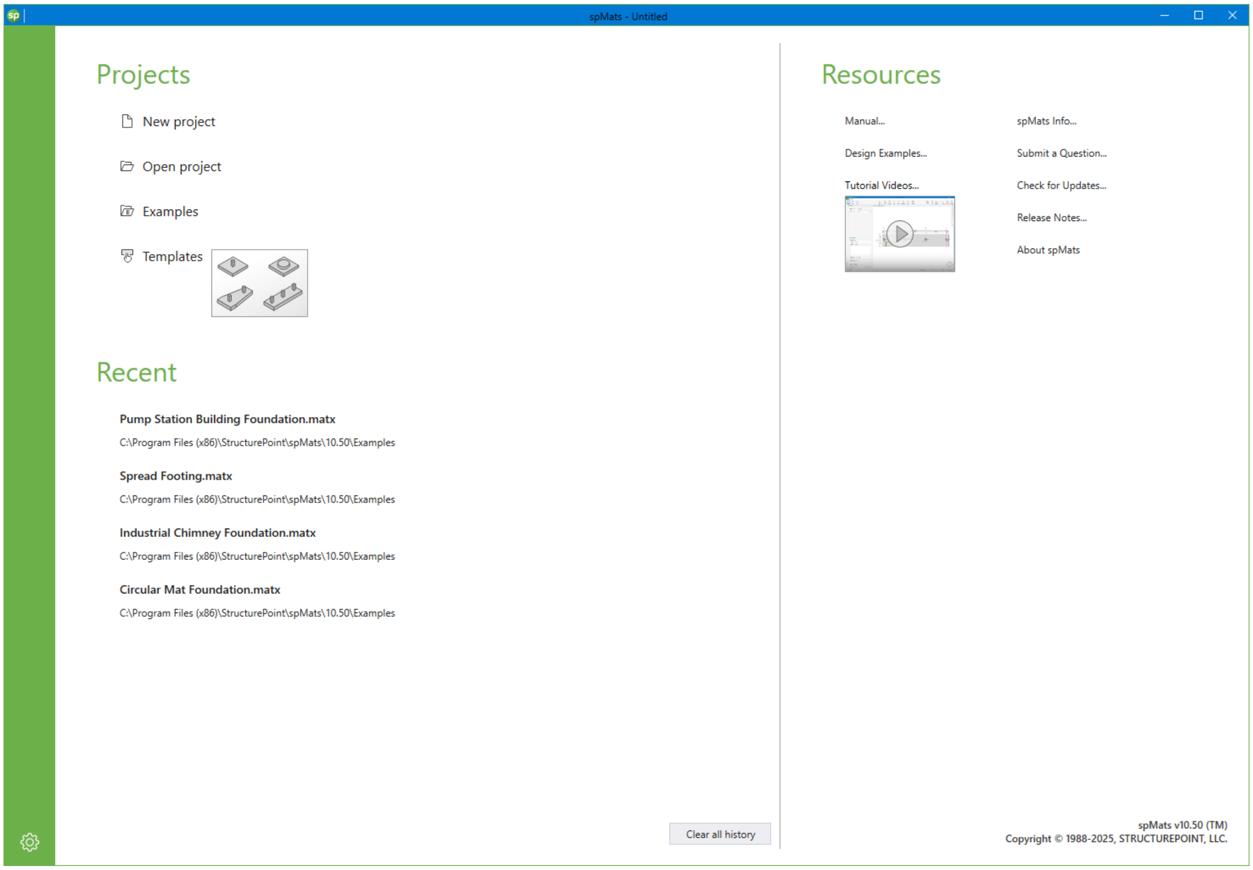
CHAPTER

3

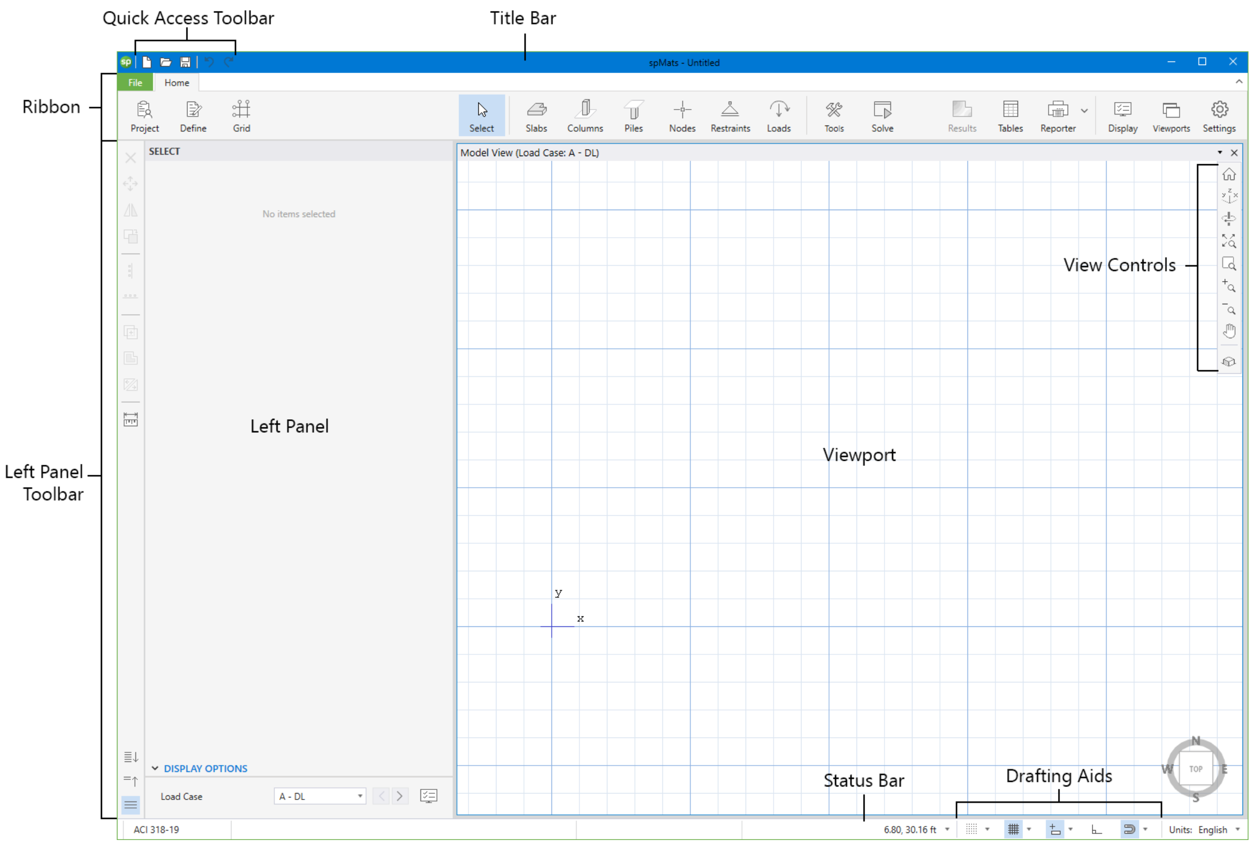
PROGRAM INTERFACE

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. Clicking the **Settings** icon in the bottom-left corner of the start screen opens the SETTINGS dialog, where you can adjust various program settings.


spMats - Untitled
Projects
New project
Open project
Examples
Templates
Recent
Pump Station Building Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Spread Footing.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Industrial Chimney Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Circular Mat Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Clear all history
Resources
Manual...
Design Examples...
Tutorial Videos...
spMats Info...
Submit a Question...
Check for Updates...
Release Notes...
About spMats
spMats v10.50 (TM)
Copyright © 1988-2025, STRUCTUREPOINT, LLC.

3.2. Main Program Window



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

3.2.1. Quick Access Toolbar

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

3.2.2. 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.

3.2.3. 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 **Slabs**, **Columns** and **Piles** objects; **Soil**, **Concrete**, **Reinforcement**, and **Slab Design Criteria** 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 $\boldsymbol{D_z}$, $\boldsymbol{R_x}$, and $\boldsymbol{R_y}$ restraints to the model.

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

Tools: enables to open the **MTX Editor** and **MTX Editor - Internal**.

Solve: enables to specify solver options, mesh generation options, slab design options, and solve the model.

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

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.

3.2.4. Left Panel

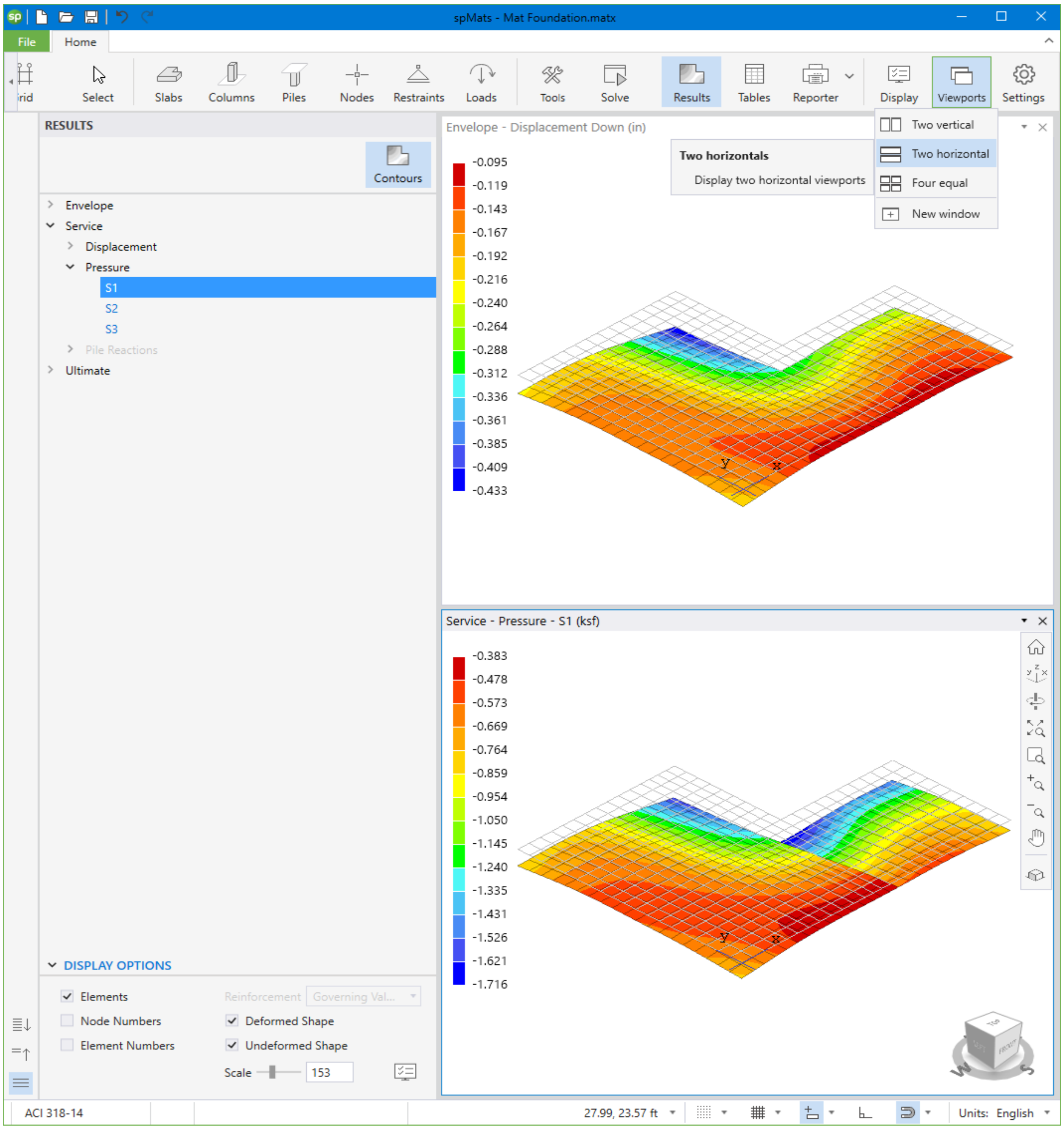
The properties of active commands under **Home** Tab or the properties of items selected in the **Viewport** are displayed in the **Left Panel** which can then be used to execute the commands or edit the selected items. After execution **Left Panel** also displays various solutions which can be explored using the **Viewport**.

3.2.5. Left Panel Toolbar

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

3.2.6. 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. Viewports can be moved and docked in a number of predefined locations using the docking tool. A viewport may be split out to a separate screen entirely for added flexibility and to enlarge the model view work area providing more accurate drafting controls.



3.2.7. 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.

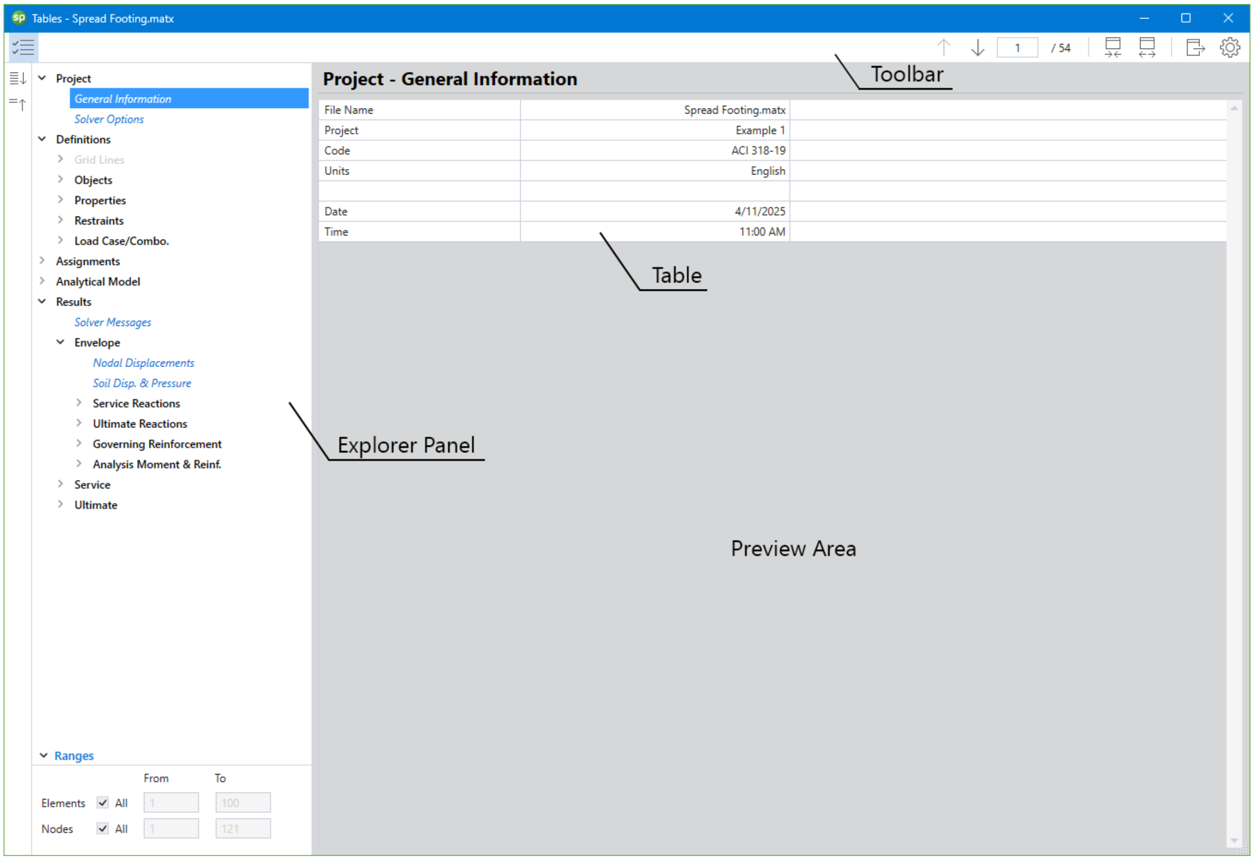
3.2.8. Drafting Aids

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

3.2.9. 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.

3.3.1. 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

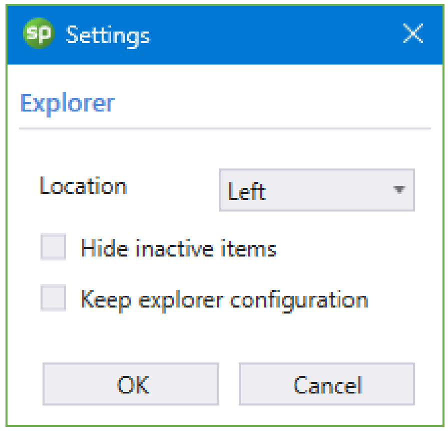
Restores all table columns to their default maximum width.

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**.

3.3.2. Explorer Panel

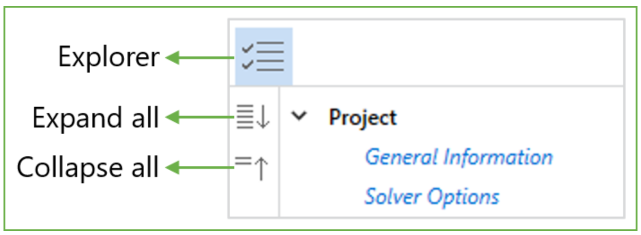
The **Explorer Panel** consists of all the available items of the inputs and results classified into sections and arranged hierarchically. Any item in the **Explorer Panel** can be clicked on to display the corresponding table in the **Preview Area**.

Expand all

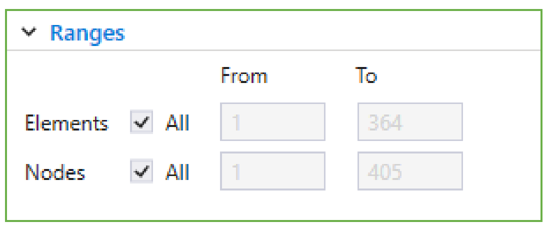
Expands item list.

Collapse all

Collapses item list.

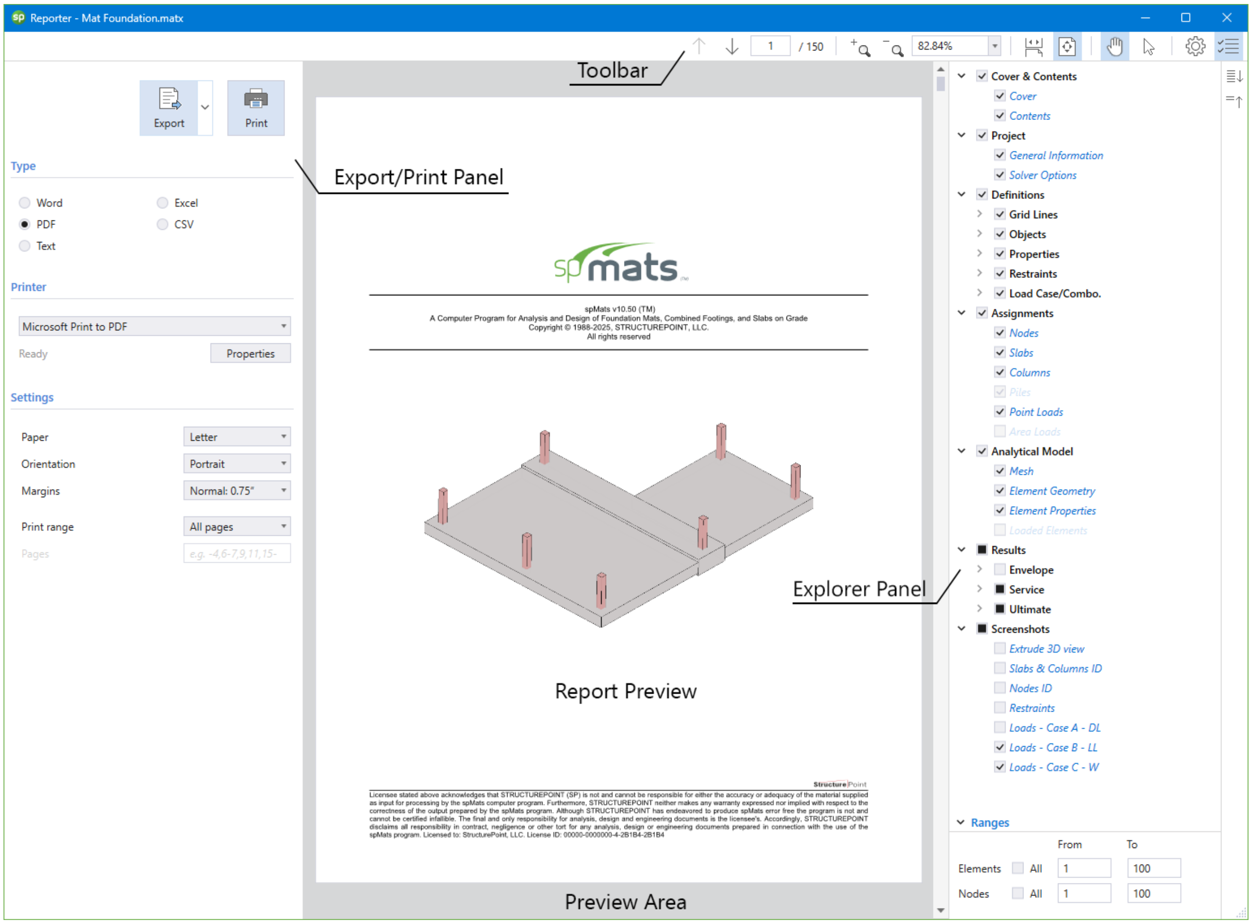


Ranges



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 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 **Reporter** window.

3.4.1. Toolbar

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

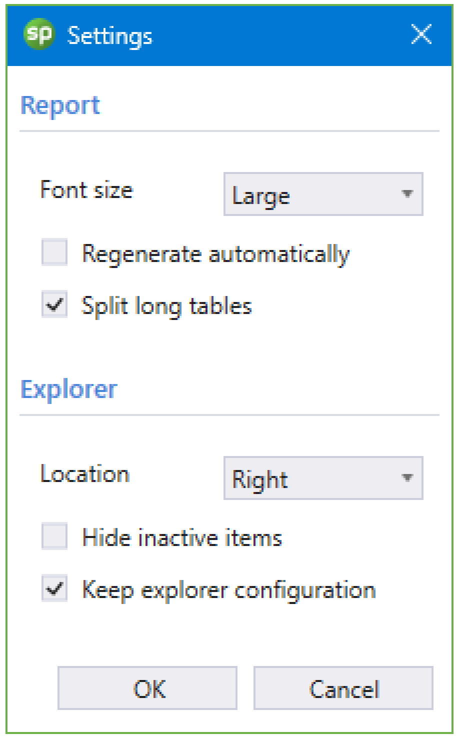
When toggled on and report is bigger than preview window, enables panning the report.

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**.

3.4.2. Export / Print panel

Export

Exports the report in the selected format, with an option to automatically open the report or its file location.

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.

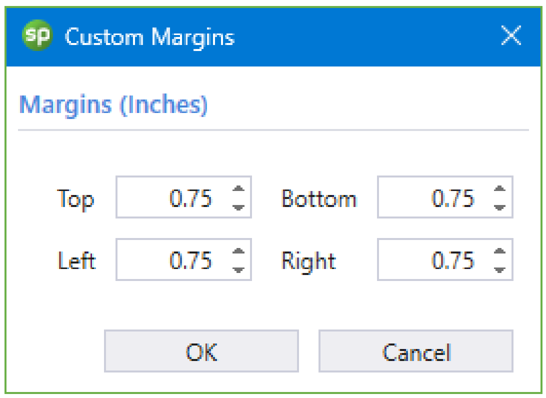
Printer

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.

3.4.3. 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.

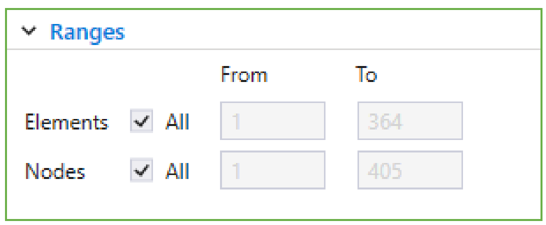
Expand all

Expands item list.

Collapse all

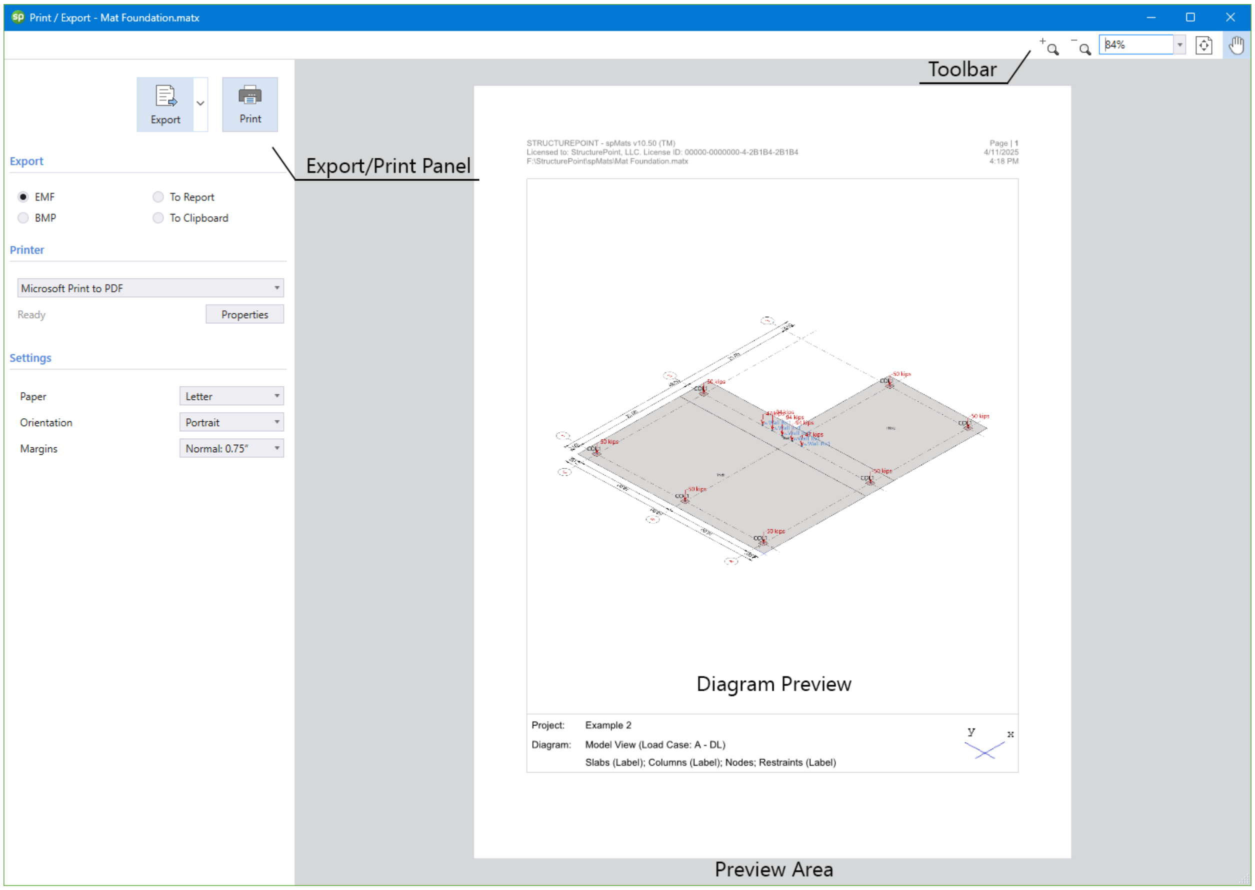
Collapses item list

Ranges



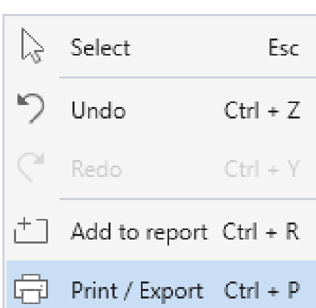
The RANGES feature allows you to explore the results for a selected range of ELEMENTS, and/or NODES. To view element and nodal results for the entire model simply check the ALL checkbox. Alternatively, to view element and nodal results within a range, make sure that the ALL checkbox is unchecked and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

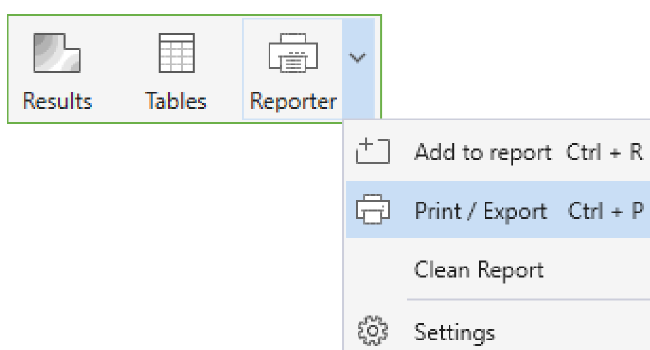
3.5. Print/Export Window



Print/Export Module interface shown above enables the user to view, customize, print and export contours and 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.

3.5.1. 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 is bigger than preview window, enables panning the report.

3.5.2. Export / Print panel

Export

Exports the report in the selected format, with an option to automatically open the report or its file location.

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

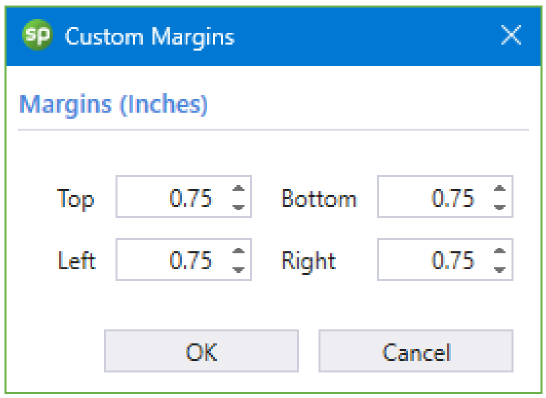
Printer

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

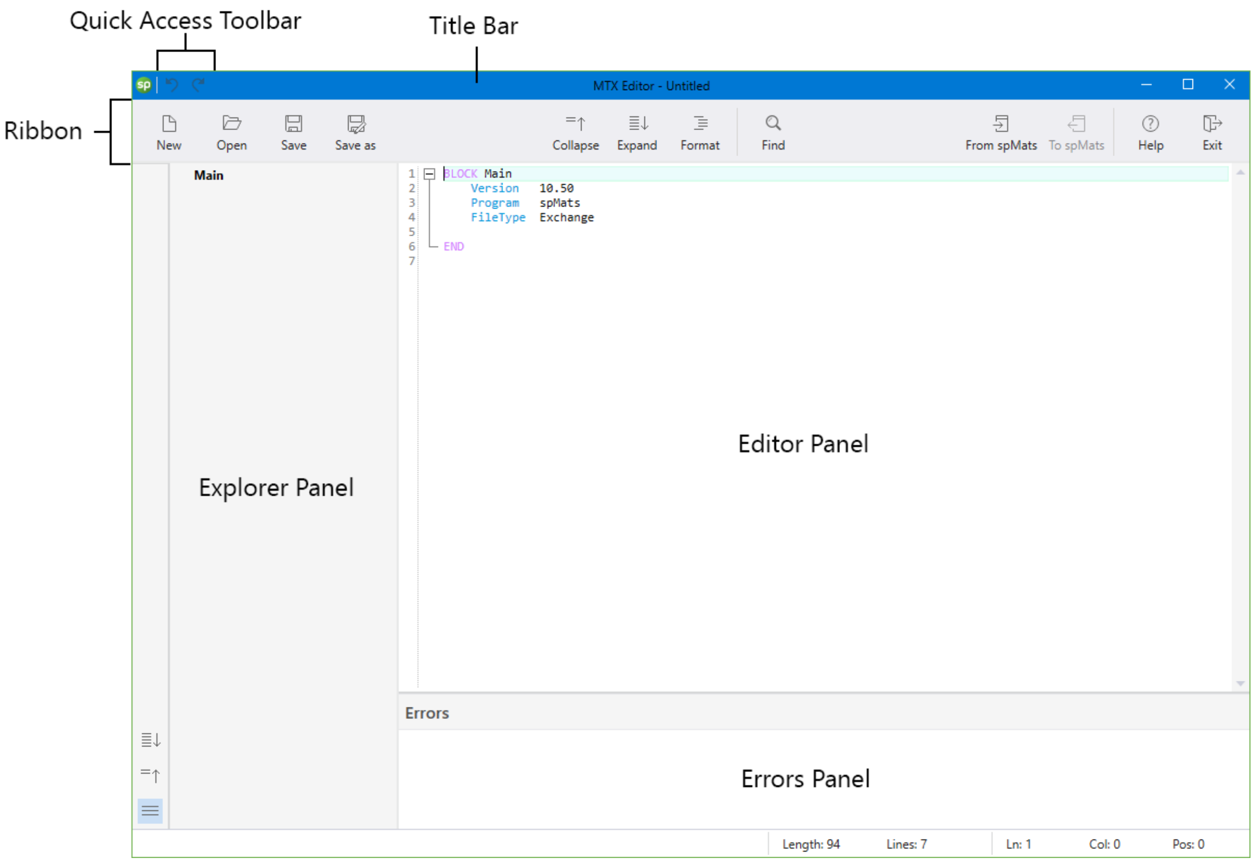
Settings

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.



3.6. MTX Editor



The **Main MTX Editor** shown above consists of the following:

3.6.1. Quick Access Toolbar

The **Quick Access Toolbar** includes the **Undo** and **Redo** commands.

3.6.2. Title Bar

The Title Bar displays the editor's name, 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.

3.6.3. Ribbon

The **Ribbon** gives quick access to the editor commands.

New:	create a New file.
Open:	open an existing file.
Save:	save changes.
Save as:	save as a new file.
Collapse:	collapse all blocks except the "Main" block.
Expand:	expand all blocks.
Format:	reformat (realign) the MTX file.
Find:	find & replace.
From spMats:	transfer the model in spMats to the MTX Editor.
To spMats:	transfer MTX data to spMats.
Help:	open MTX help.
Exit:	exit MTX editor.

3.6.4. Explorer Panel

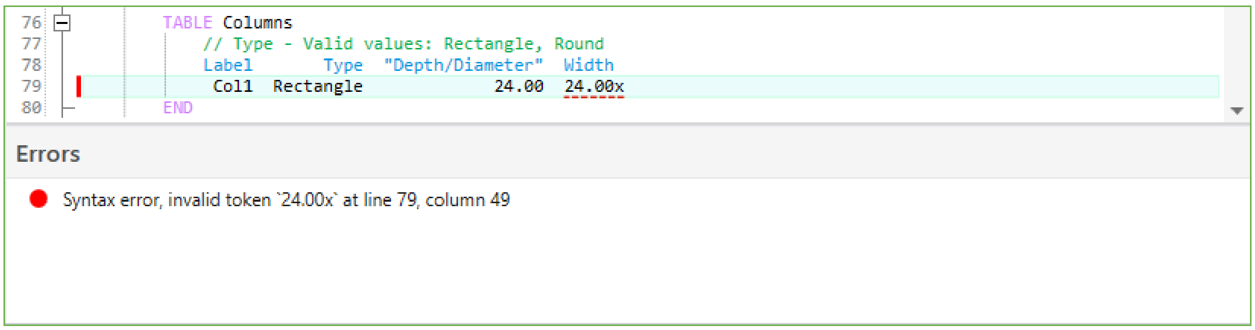
The **Explorer Panel** lists all the available Blocks and their Tables arranged hierarchically. Any item in the **Explorer Panel** can be clicked on to navigate to it in the **Editor Panel**.

3.6.5. Editor Panel

The **Editor Panel** lists the contents of the MTX file. Different parameters like blocks, keys, values and comments are color coded to facilitate quick and efficient working with the format. User can make changes to the MTX file in the **Editor Panel**.

3.6.6. Errors Panel

The MTX editor uses spMats' internal error handling mechanism to identify and show errors in MTX files. Errors if any, in the MTX file are highlighted in the **Editor Panel** with the description listed in the **Errors Panel**.



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.

4.1.1. 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.

4.1.2. 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.

4.1.3. 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 slab design criteria properties, are named entities that must be specified before assigning them to objects. If a property is assigned to an object, for example a slab design criteria 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.

4.1.4. 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.

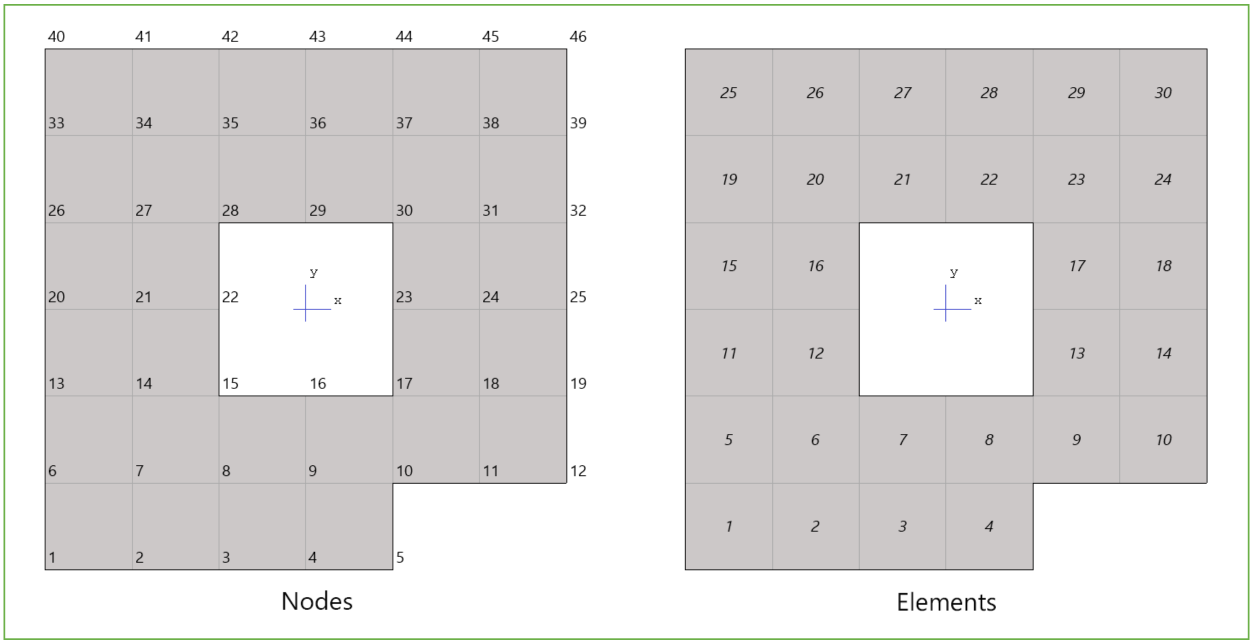


Figure 4.1 – Node and Element Numbering in Analytical Model

4.1.5. Modeling Considerations

Modeling and analyzing foundation systems using the finite element method requires careful attention to factors that influence accuracy, stability, and interpretation of results. Key considerations include mesh size and aspect ratio, modeling of construction joints, variable soil support conditions, pile location and pacing, and the proper representation of pedestals and piers. Each of these aspects directly affects the behavior of the foundation model and should be addressed thoughtfully to ensure reliable and structurally meaningful outcomes in spMats.

4.1.5.1. Finite Element Mesh Size

The selection of finite element mesh size significantly influences the accuracy and convergence of analysis results in foundation modeling. Finer mesh densities generally yield more accurate deflection profiles, improved moment distribution, and optimized reinforcement design. Deflections under surface loads converge faster than those under concentrated loads, and finer meshes help capture the peaking behavior of moments and reinforcement near concentrated load applications more accurately. However, overly fine meshes may increase computational demands and result interpretation complexity. While mesh sizes around 10% of the smallest slab dimension are often sufficient for convergence within 1%, users should balance accuracy with practicality. Using spMats, engineers can refine mesh density, distribute concentrated loads to the surface they physically occupy, or average element moments to reduce artificial stress concentrations and obtain structurally meaningful reinforcement patterns - especially under column or pedestal reactions. Thoughtful mesh size selection can lead to more economical and constructible designs without compromising analysis results quality.

More information about this topic can be found in Section 6.2.1, "Finite Element Mesh Sizing Influence on Mat Foundation Reinforcement", and "Finite Element Mesh Density Influence on spMats Model Results" Technical Articles from StructurePoint.

4.1.5.2. Finite Element Aspect Ratio

The aspect ratio of a finite element - defined as the ratio of its longer side to shorter side - can significantly influence the accuracy and reliability of Finite Element Method of Analysis results in foundation modeling. Ideally, square elements with an aspect ratio of 1 are preferred to minimize interpolation errors and ensure uniform solution accuracy. Aspect ratios up to 10 have negligible impact on deflection results. However, ratios exceeding 100 lead to noticeable deviations and should be avoided. If elements with high aspect ratios are unavoidable due to geometric or modeling constraints, increasing mesh density in the elongated direction can help mitigate their impact. In spMats, users can directly control the maximum allowable aspect ratio through the Mesh Options available under the Solve command. Maintaining low aspect ratios, especially near areas of high stress gradients or concentrated loads, enhances solution stability and reflects more accurate reinforcement demands and deflection profiles in the model.

More information about this topic can be found in Section 6.2.2 and "Finite Element Aspect Ratio Influence in Concrete Foundation Models" Technical Article from StructurePoint.

4.1.5.3. Modeling of Construction Joints

Construction joints are intentional discontinuities in concrete mat foundations or slabs-on-grade, typically introduced to manage staged construction and control cracking due to drying shrinkage, curling, and thermal effects. Structurally, these joints often transfer vertical shear between adjacent concrete placements, while moment transfer may or may not be required. In spMats, when modeling construction joints that transfer shear but do not transfer moment, two methods may be used. The first involves inserting weak elements at the joint - these are narrow elements with a low modulus of elasticity, allowing vertical shear to pass while minimizing moment continuity due to their flexibility. The second method simulates dowel action by slaving node pairs in the vertical (D_z) direction only. Each node pair should be assigned separate slaved node criteria to avoid unintended constraints across the joint.

More information about this topic can be found in the "Modeling Construction Joints Structural Foundations in spMats" Technical Article from StructurePoint.

4.1.5.4. Modeling of Foundation Slab Supported by Dissimilar Soils

When modeling foundation slabs supported by dissimilar soils in spMats, users can define varying soil stiffness across different regions of the slab to reflect realistic subgrade conditions. This is particularly important in cases involving expansions or partial replacements, where matching the existing soil subgrade modulus may not be feasible. In spMats, each slab object can be assigned a unique subgrade modulus, allowing for accurate representation of dissimilar support conditions within a single model. The resulting soil pressure contours will reflect abrupt changes along soil boundaries, with higher pressures occurring over stiffer soils and lower pressures over softer ones. It is essential to interpret these results in conjunction with both the graphical output (graphical contour views) and the tabular output for soil pressure and displacement. Since each finite element's pressure is governed by its most critical nodal value, engineers should pay special attention to pressure transitions at the boundary. Comparing individual service-level pressure results and the corresponding contour displays ensures a reliable assessment of soil performance under varying stiffness conditions

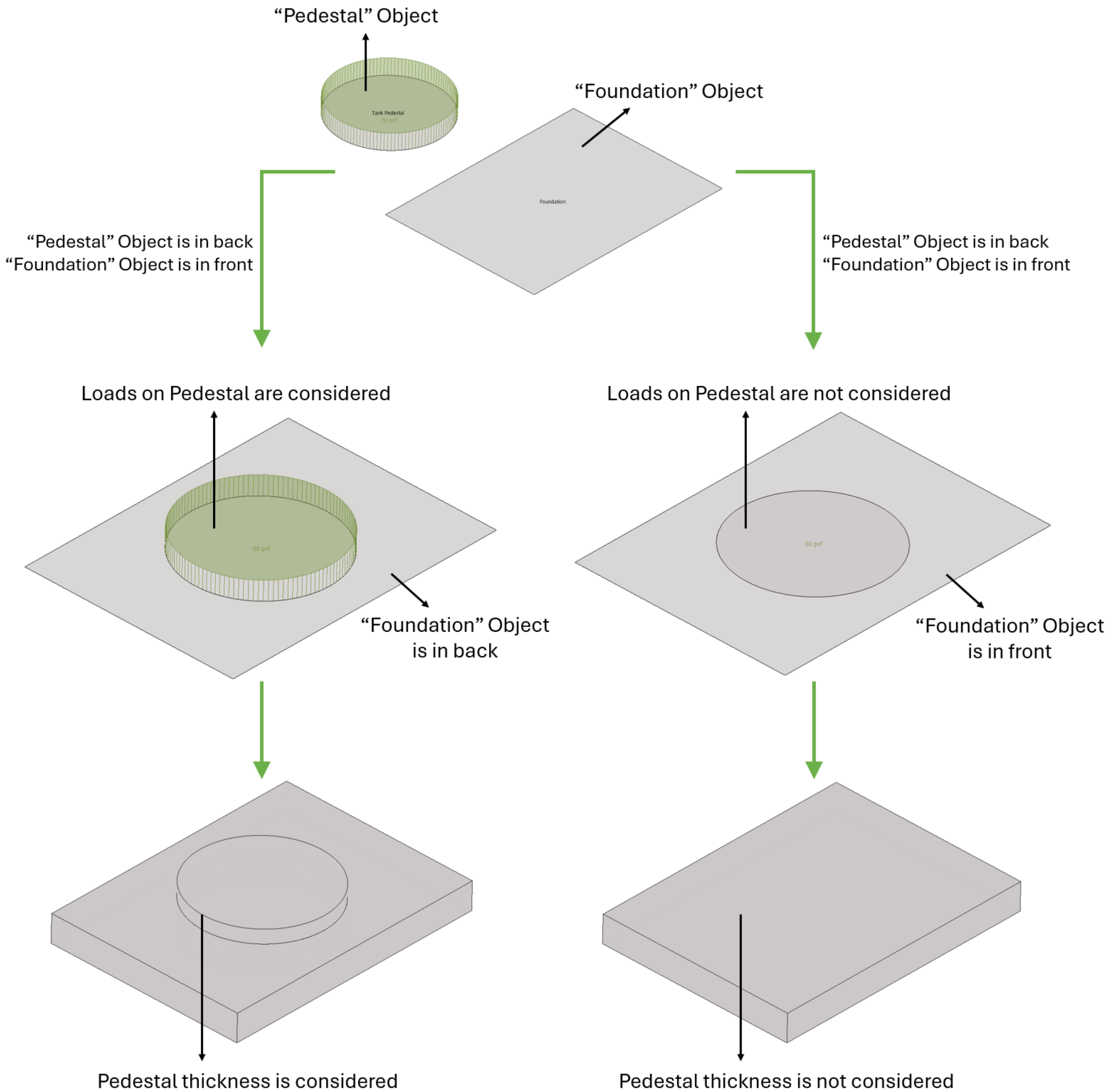
More information about this topic can be found in the "Foundations Slabs Supported by Dissimilar Soils" Technical Article from StructurePoint.

4.1.5.5. Modeling of Pedestals and Piers

Pedestals and piers are elevated structural elements used to transfer concentrated loads from equipment, tanks, or columns to the supporting foundation. Pedestals are generally short, wide elements cast integrally with the mat or slab, while piers are taller vertical elements. Frequently found in industrial facilities, this combination of elements is widely used to support tanks, vessels, equipment, machinery, cranes, hoists, and myriad other non-building structures. Proper modeling of these elements in spMats is essential to capture their effects on load distribution, stiffness discontinuities, and potential stress concentrations in the foundation.

In spMats, the inclusion of pedestal loads and thickness in the analysis depends on the order the objects are drawn in the model. If the pedestal object is set forward and the foundation object is backward, loads applied to the pedestal are properly considered, and its thickness is included in the model analysis. However, if the pedestal is drawn behind the foundation, the loads and thickness will be ignored in the analysis. The following figure illustrates this modeling behavior and highlights the importance of ensuring the correct object order to achieve the intended model behavior and correct analysis results.

More information about this topic can be found in Section 5.2.5.4. Understanding Slab Layers.

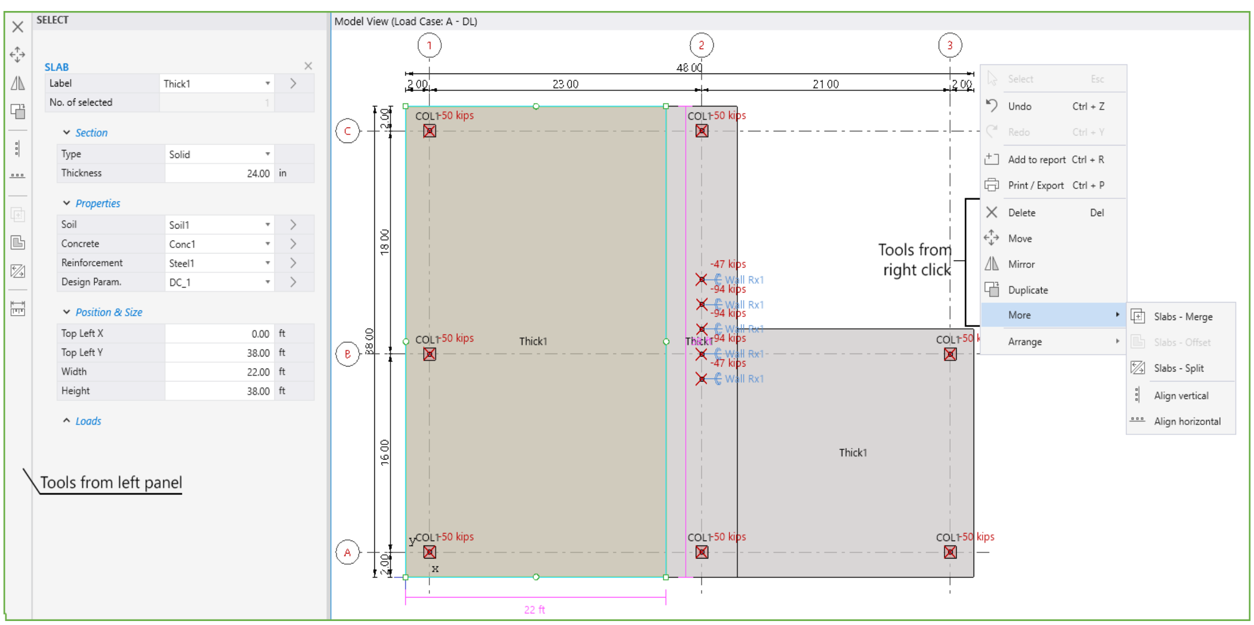


4.2. Model Editing Concepts

4.2.1. Editing Objects

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:

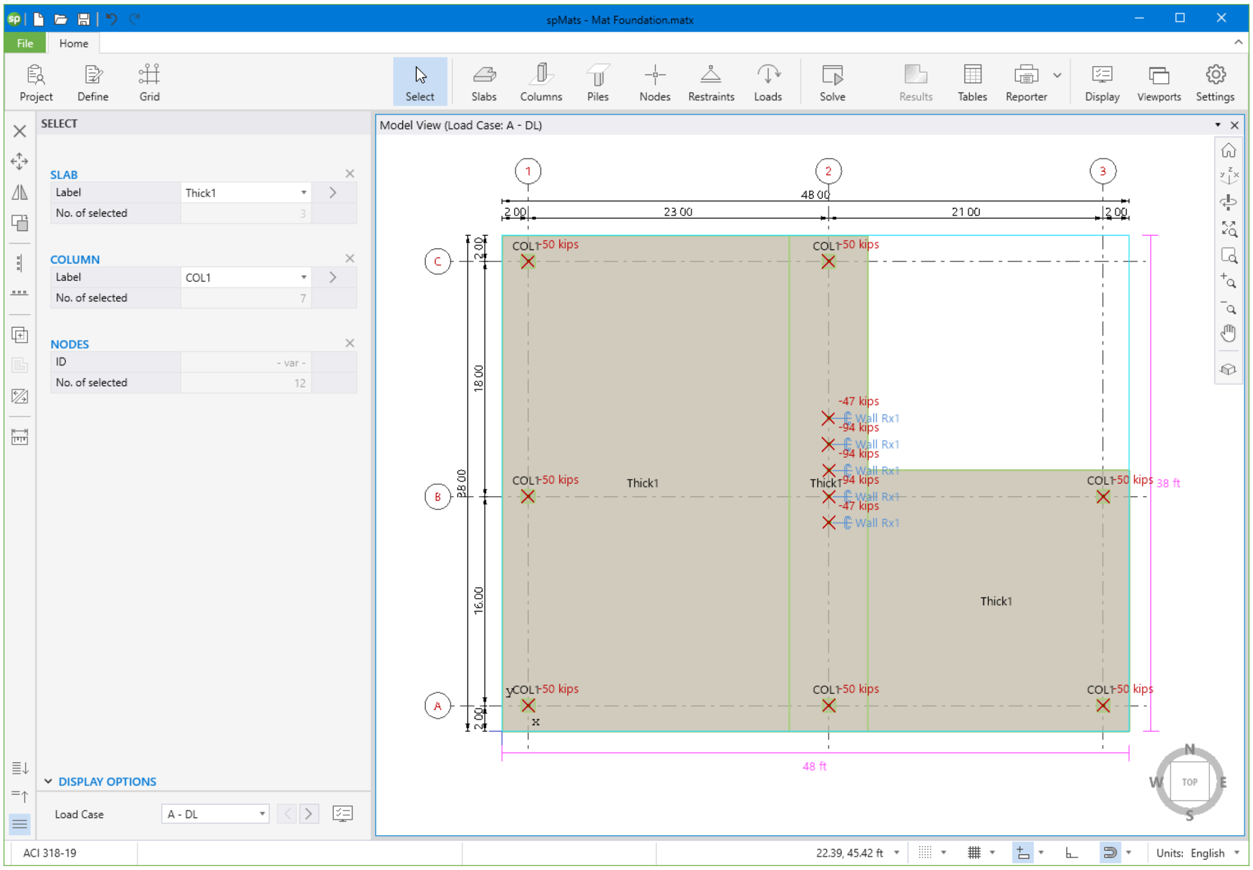
- **Slab Object**: Slab objects can be deleted, moved, duplicated, merged, offset or split.
- **Column Object**: Column objects can be deleted, moved or duplicated.
- **Pile Object**: Pile objects can be deleted, moved or duplicated.
- **Node Object**: Node objects can be deleted, moved, duplicated, 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.

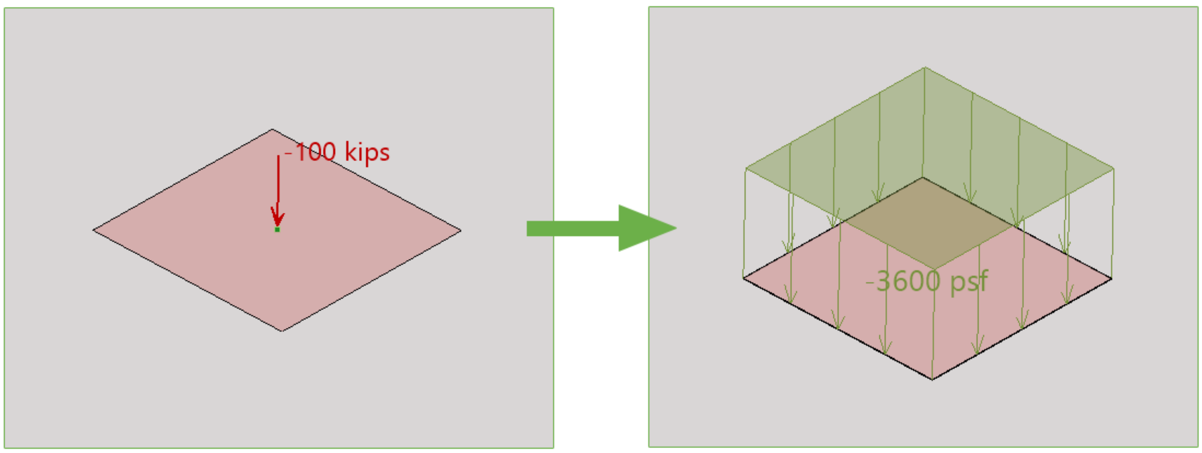
When multiple different object types are selected, all applicable editing tools will be populated at the **Left Panel Toolbar**. The object types can be unselected by clicking the x button next to the Object Type at the **Left Panel**.



Press any white space in the **Viewport** in order to undo the selection.

4.2.2. Modeling Point Loads as Distributed Area Loads

In spMats, point loads applied to a mat foundation can be redistributed as area loads to more accurately represent the load distribution. This approach is particularly useful when modeling heavy equipment, column reactions, or other concentrated loads that, in practice, are dispersed over a larger area due to the stiffness of the mat. To achieve this, users can define a load distribution area around the point load, allowing the program to evenly distribute the force across the specified area, resulting in more realistic stress and deflection results.



4.2.3. Embedment in Mat Foundations

In mat foundations supporting heavy equipment or vertical structural systems, accurately simulating anchor bolt behavior is critical. As highlighted in heavy industrial applications such as the construction of coal-fired electric generation power plants, large diameter anchor bolts are often embedded deeply into the mat foundation to secure heavy machinery and resist tension from uplift and shear forces. In spMats, while anchor bolts themselves are not explicitly modeled as structural elements, their influence on the mat behavior - particularly the localized tensile stresses around anchorage zones - can be captured using distributed area loads (see the previous section). Users can simulate the uplift forces caused by tension plates or equipment attachments by redistributing these concentrated anchor forces as upward distributed area loads over the anchored base plate region. This technique helps also to evaluate the required development of reinforcement to transfer forces from the anchor bolts.

More information about this topic can be found in "Constructing a Mat Slab" Technical Article.

4.2.4. Optimizing Mesh Conditions

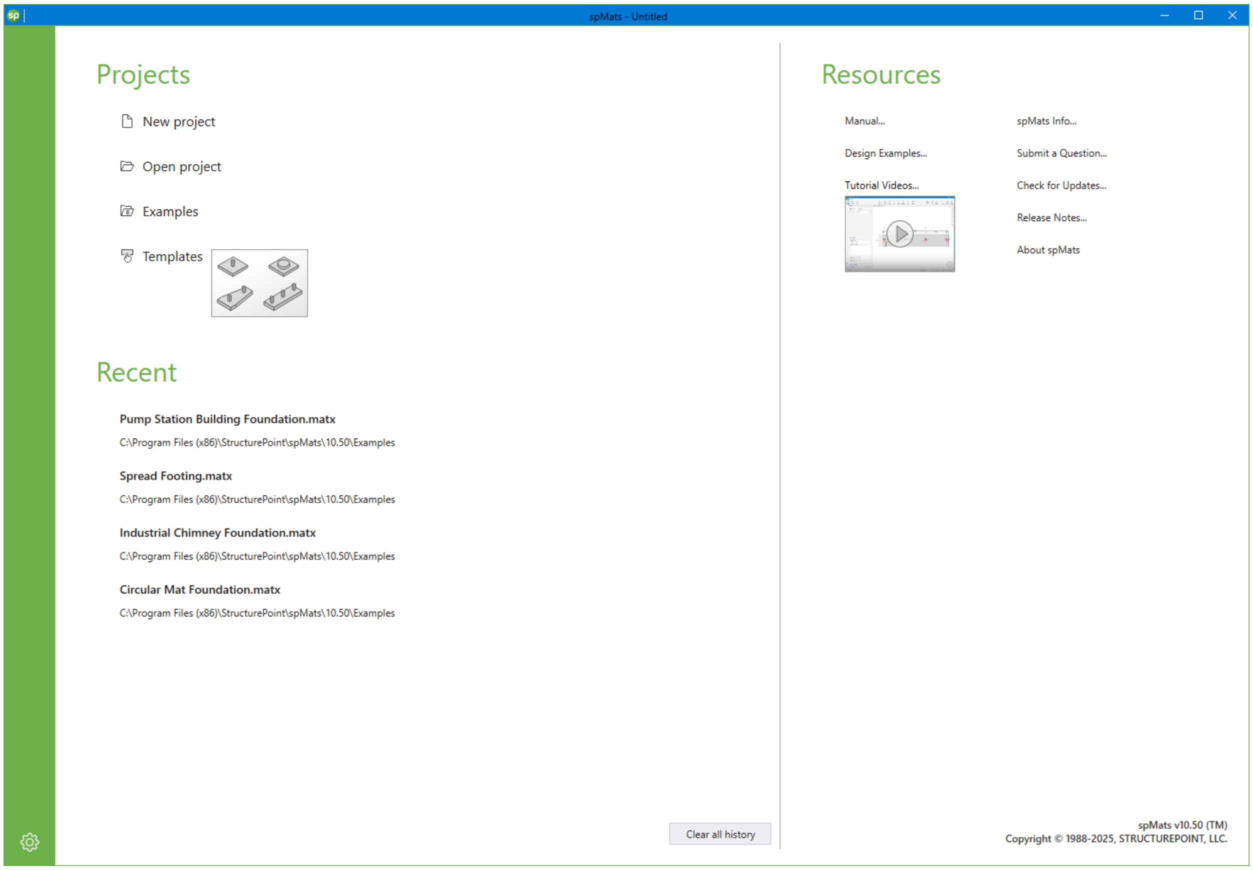
spMats provides several features to help users optimize mesh quality and improve analysis stability, particularly in geometrically complex regions such as circular slabs or where diagonal edges are present. The CIRCLE SEGMENTS analytical model option, located under **Solve** command, allows users to control the level of discretization for circular geometries by specifying the number of circular segments - selectable from 8, 12, 24, 36 (default), or 48 segments. Increasing this value results in a smoother approximation of curved boundaries, which is essential for accurate modeling of circular foundations. Additionally, spMats addresses potential mesh distortions through the AUTO ALIGNMENT feature, located under **Solve** command. This tool identifies misaligned nodes that cause high aspect ratio elements and potential numerical inaccuracies. By setting a NODES AUTO ALIGN MARGIN and activating the AUTO ALIGNMENT function, users can automatically adjust nodes to the nearest common axis, reducing mesh distortion and improving element quality. Elements exceeding the maximum aspect ratio threshold are highlighted in red, guiding users to areas that require correction.

CHAPTER

5

MODEL DEVELOPMENT

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**. Each of these methods can be used to create a new model from scratch, edit a model developed previously for an earlier project, start with pre-defined template, or use an existing example file from the provided library. Each of the methods is described in detail in this chapter.

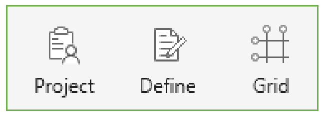

spMats - Untitled
Projects
New project
Open project
Examples
Templates
Recent
Pump Station Building Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Spread Footing.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Industrial Chimney Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Circular Mat Foundation.matx
C:\Program Files (x86)\StructurePoint\spMats\10.50\Examples
Clear all history
Resources
Manual...
Design Examples...
Tutorial Videos...
spMats Info...
Submit a Question...
Check for Updates...
Release Notes...
About spMats
spMats v10.50 (TM)
Copyright © 1988-2025, STRUCTUREPOINT, LLC.

5.1. 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.

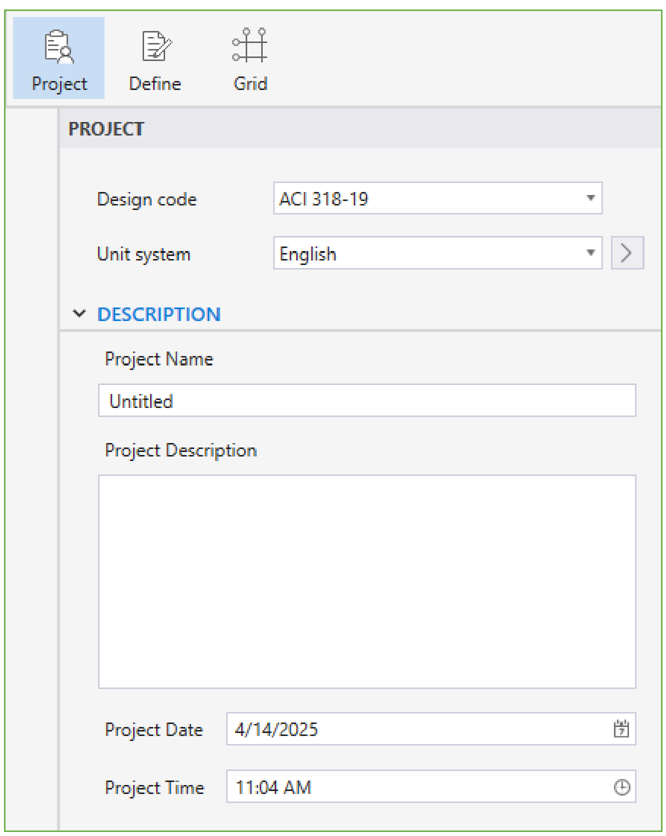
5.2. Creating New Models

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.



5.2.1. 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.

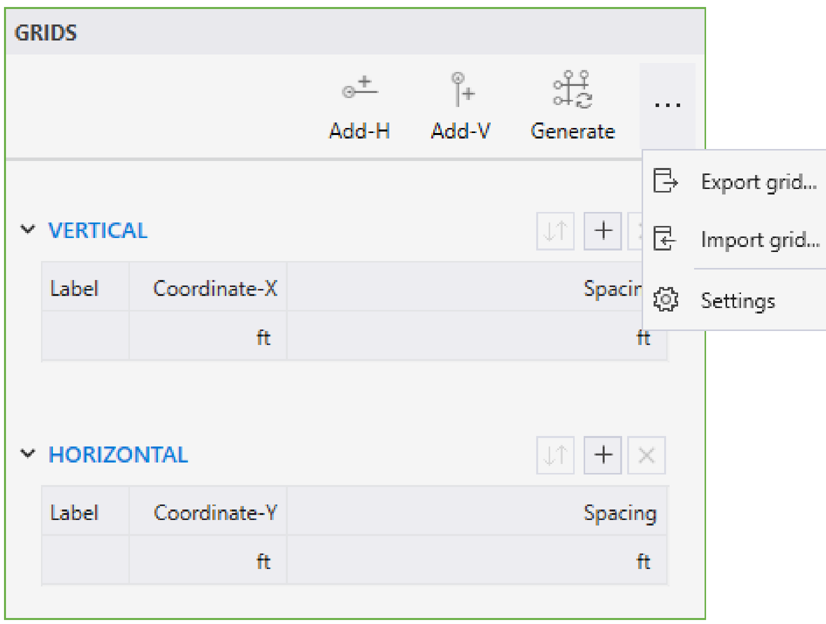


5.2.2. Structural Grids

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.

5.2.2.1. Working with Grids

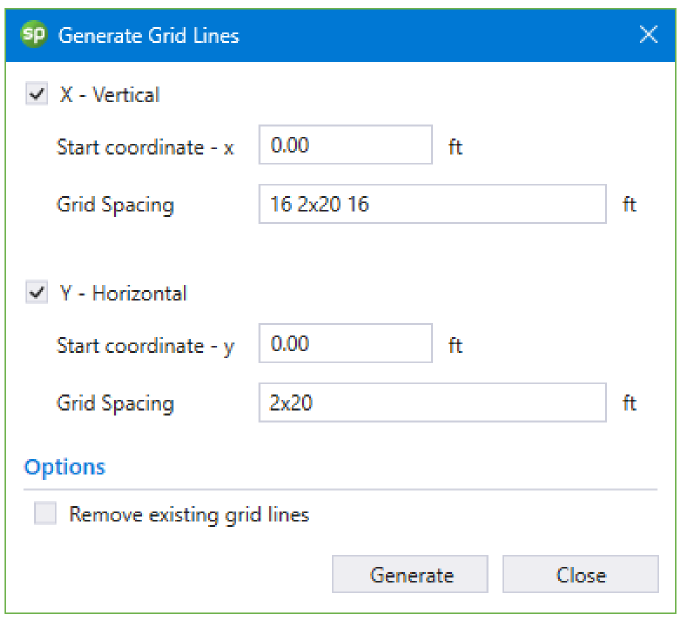
You can select the **Grid** command button from the **Ribbon**. The corresponding **Grids Left Panel** provides various tools and options for effectively working with grids.



5.2.2.2. 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 x 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



5.2.2.3. Adding Grids

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

5.2.2.4. 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

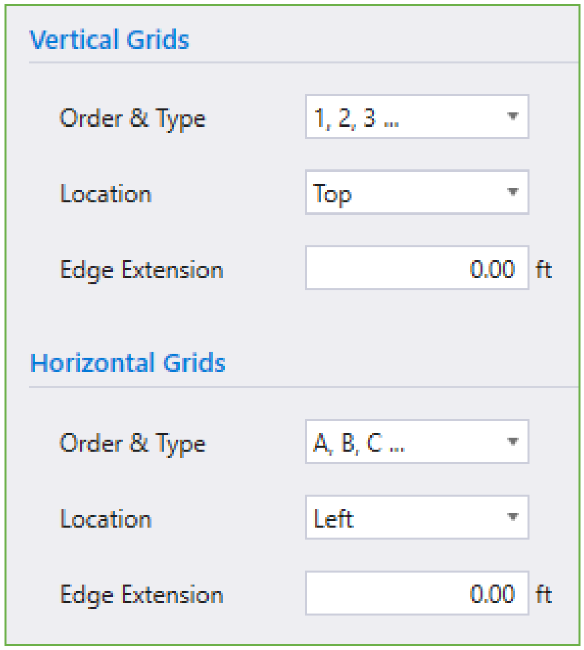
Label	Coordinate-X	Spacing
	ft	ft
2	3.00	3.00
3	33.00	30.00
4	63.00	30.00
5	93.00	30.00
6	103.00	10.00

- 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.

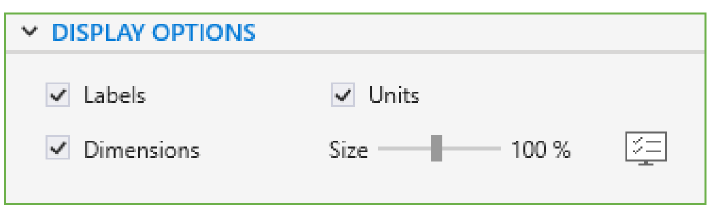
5.2.2.5. Working with labels

You can use the **Settings** / **General** / **Grids** to change the ORDER & TYPE and LOCATION of the grid labels. You can also change how far the grid labels are located from the grid lines by changing the EDGE EXTENSION value.



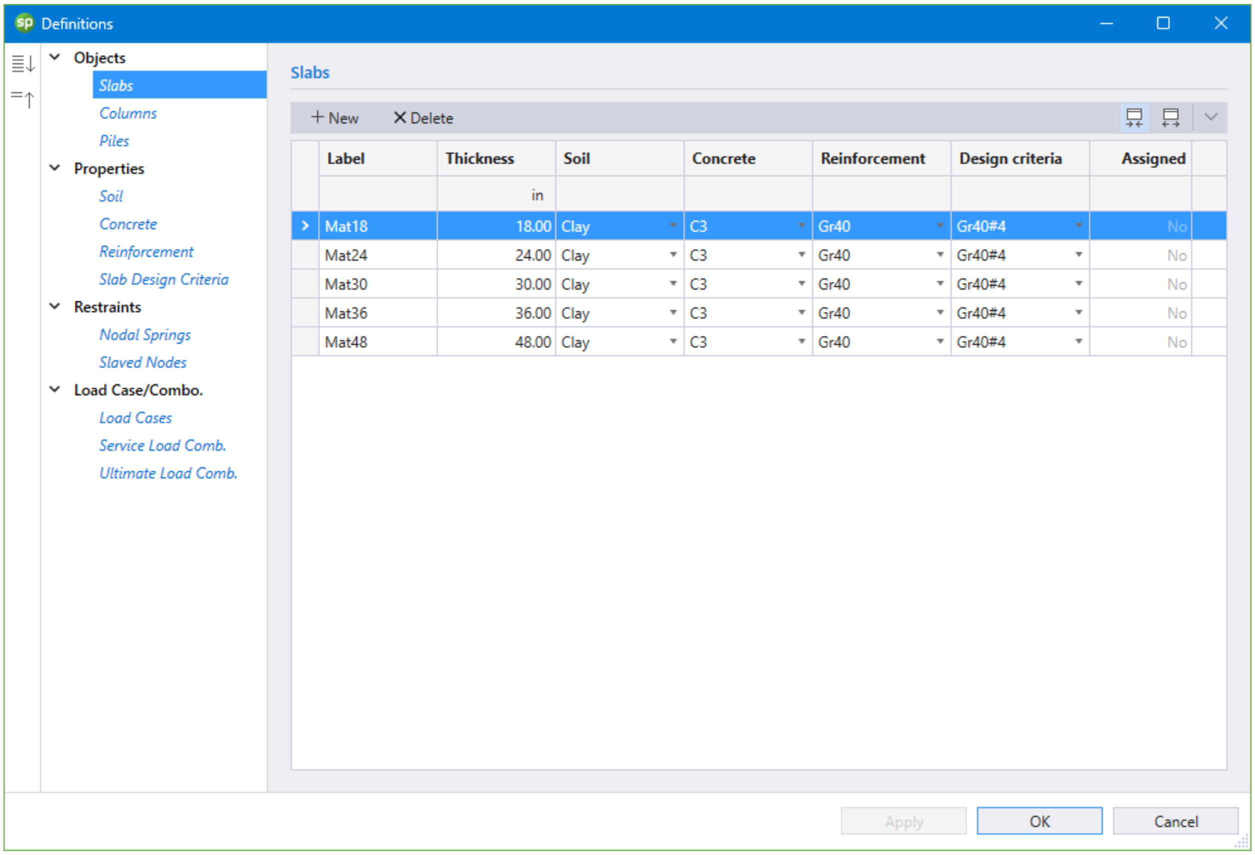
5.2.2.6. Grid Display Options

- 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.



5.2.3. Generating Definitions

Once the project information has been completed, editing and adding new definitions can be done in the DEFINITIONS Dialog Box by selecting the **Define** command button from the **Ribbon**.



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.

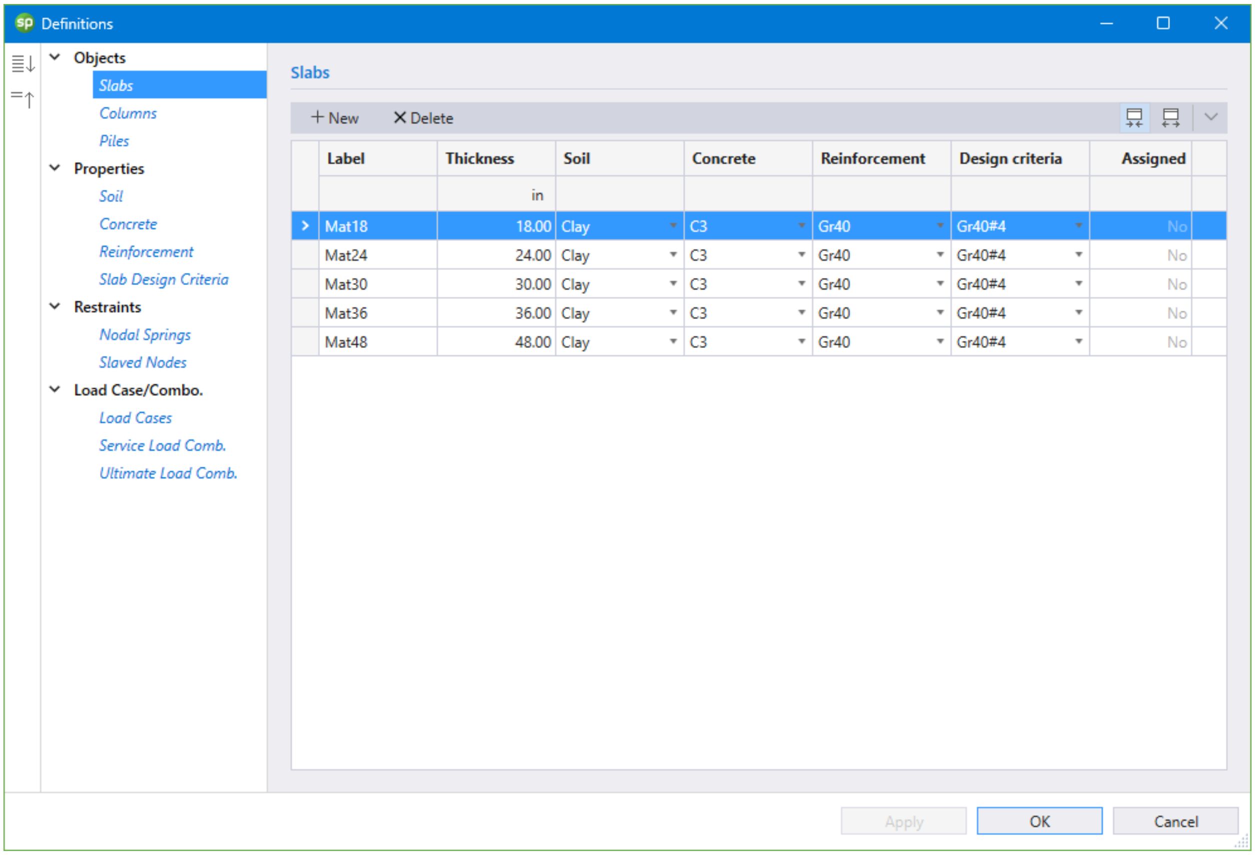
Clicking the dialog launcher  next to an item in the **Left Panel** will open the DEFINITIONS dialog box.

5.2.3.1. Objects

The **Objects** that can be defined are: **Slabs**, **Columns** and **Piles**.

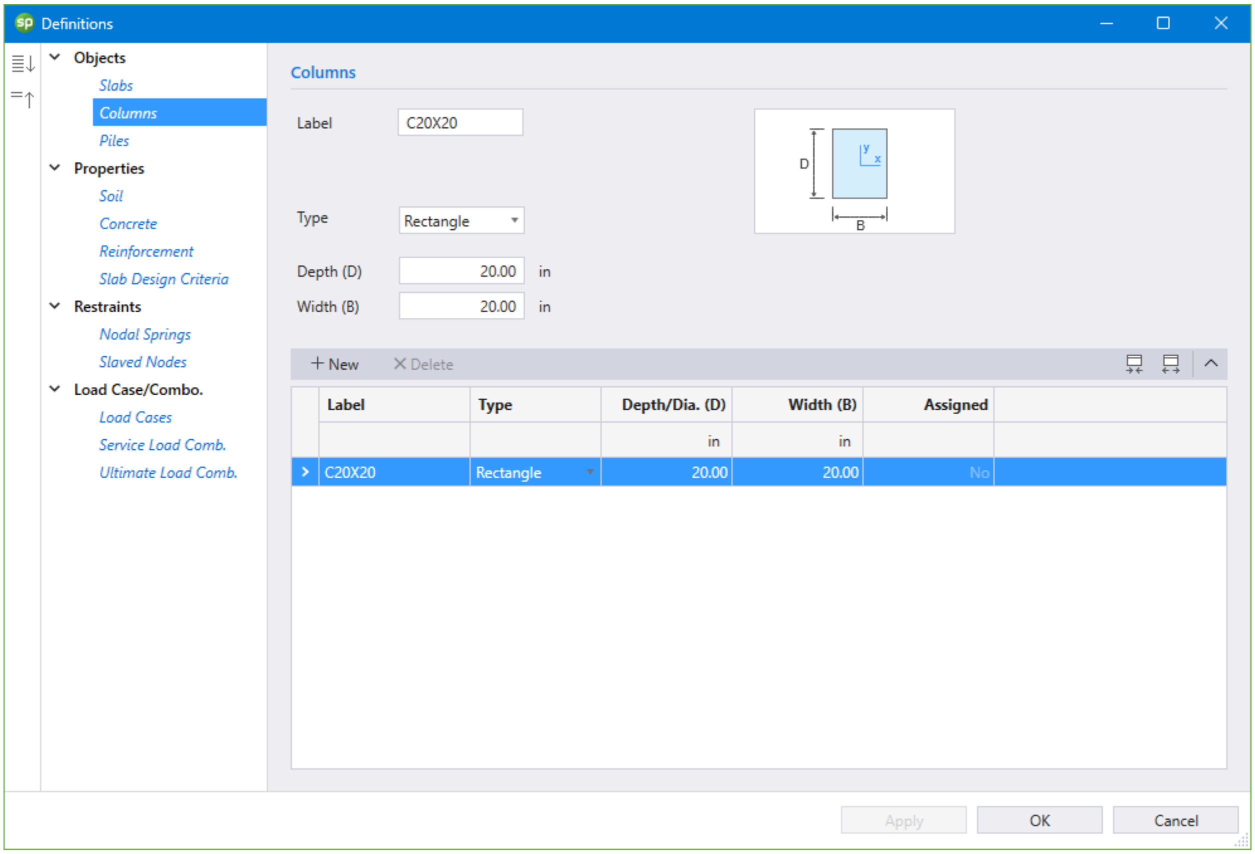
Slabs

The **Slabs** Object definition consists of the LABEL, THICKNESS, PROPERTIES, and whether the **Slabs** label is ASSIGNED in the model or not.



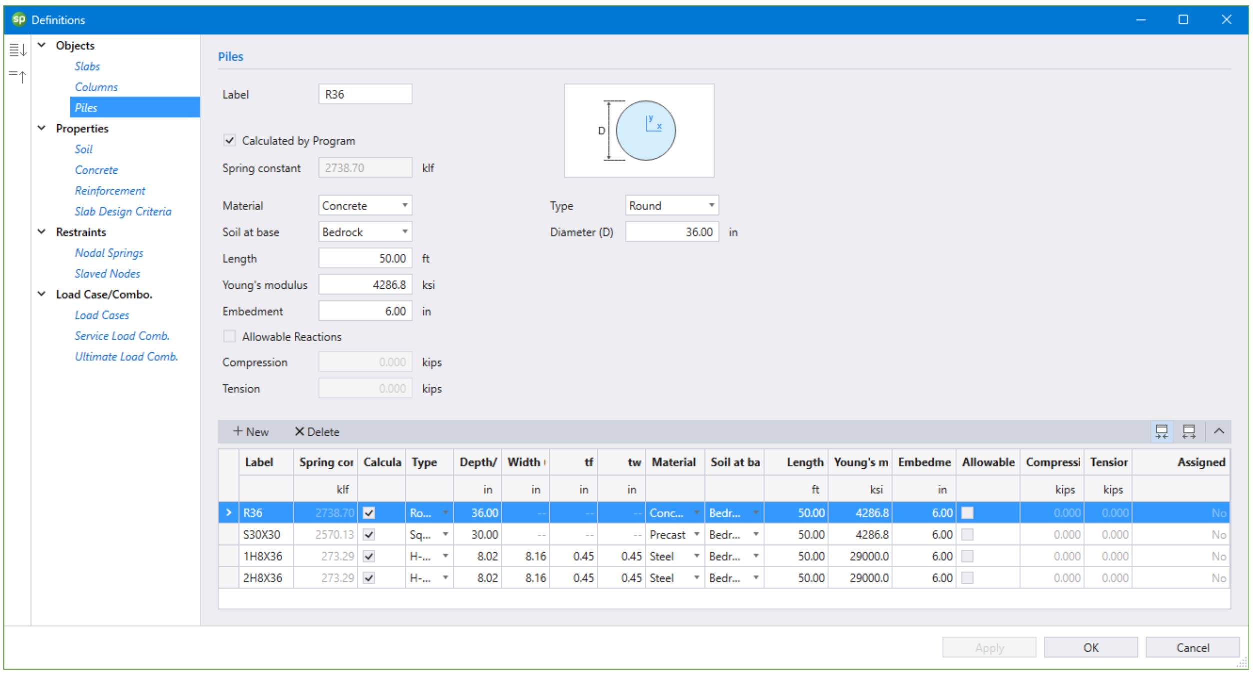
Columns

The **Columns** Object definition consists of the LABEL, TYPE, DIMENSIONS, and whether the **Column** label is ASSIGNED in the model or not.



Piles

The **Piles** Object definition consists of the LABEL, TYPE, DIMENSIONS, and CHARACTERISTICS OF PILE, SUPPORTING SOIL, ALLOWABLE REACTIONS, and whether the **Pile** label is ASSIGNED in the model or not.

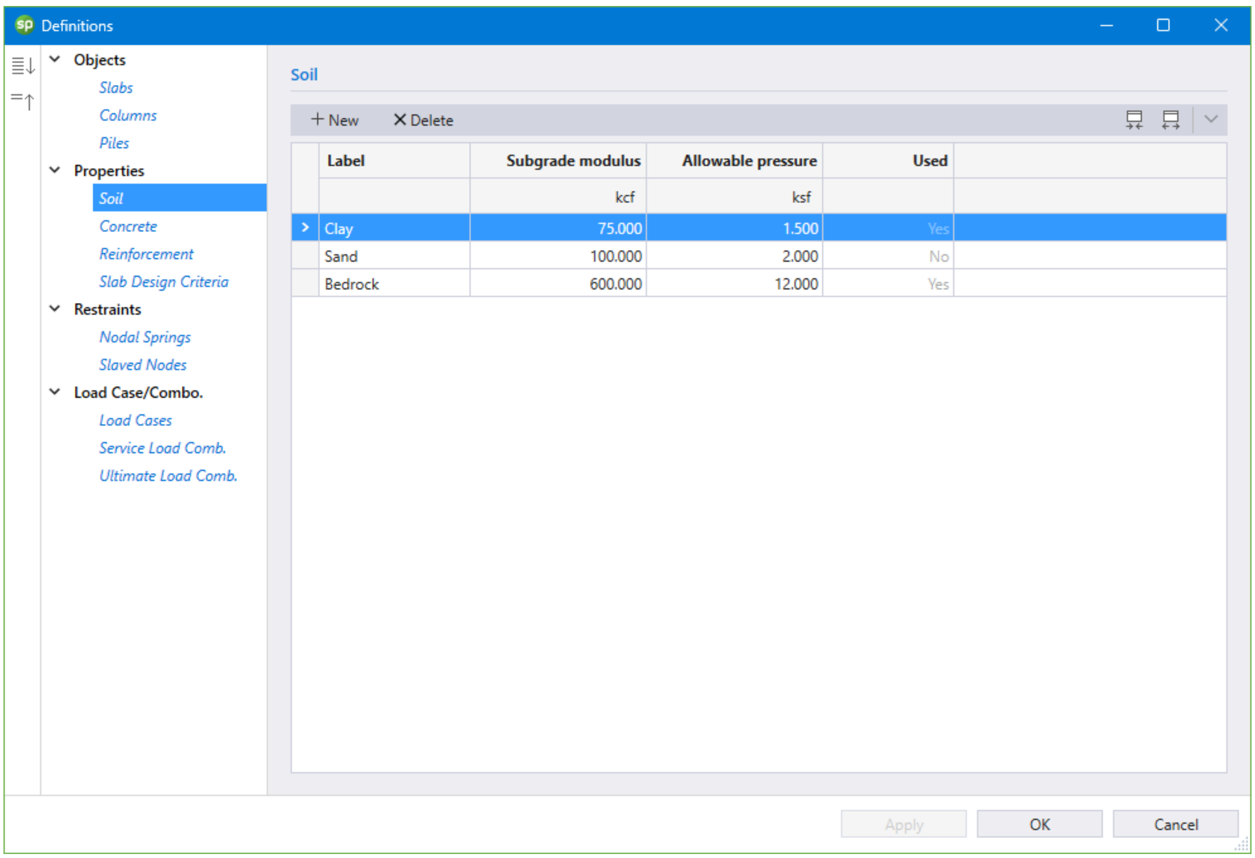


5.2.3.2. Properties

The **Properties** that can be defined are: **Soil**, **Concrete**, **Reinforcement**, and **Slab Design Criteria**.

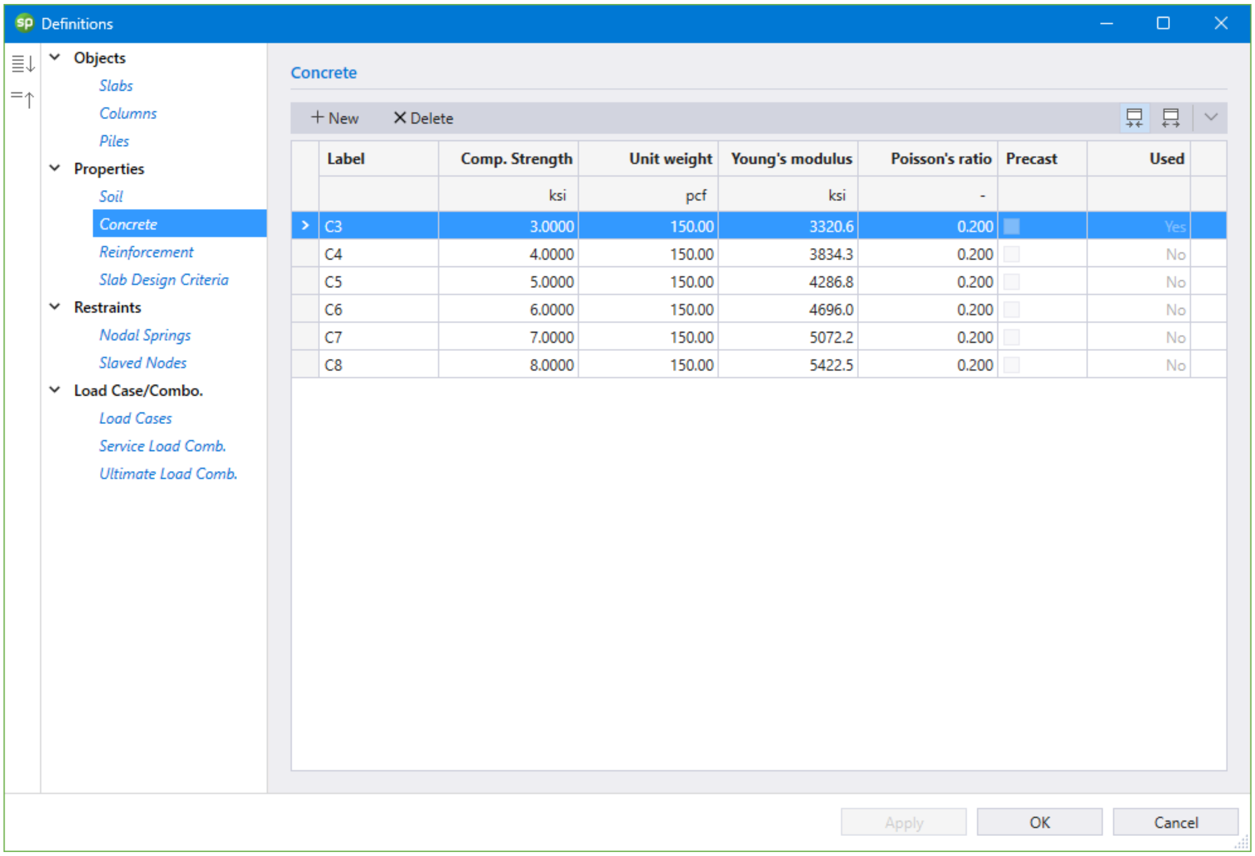
Soil

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



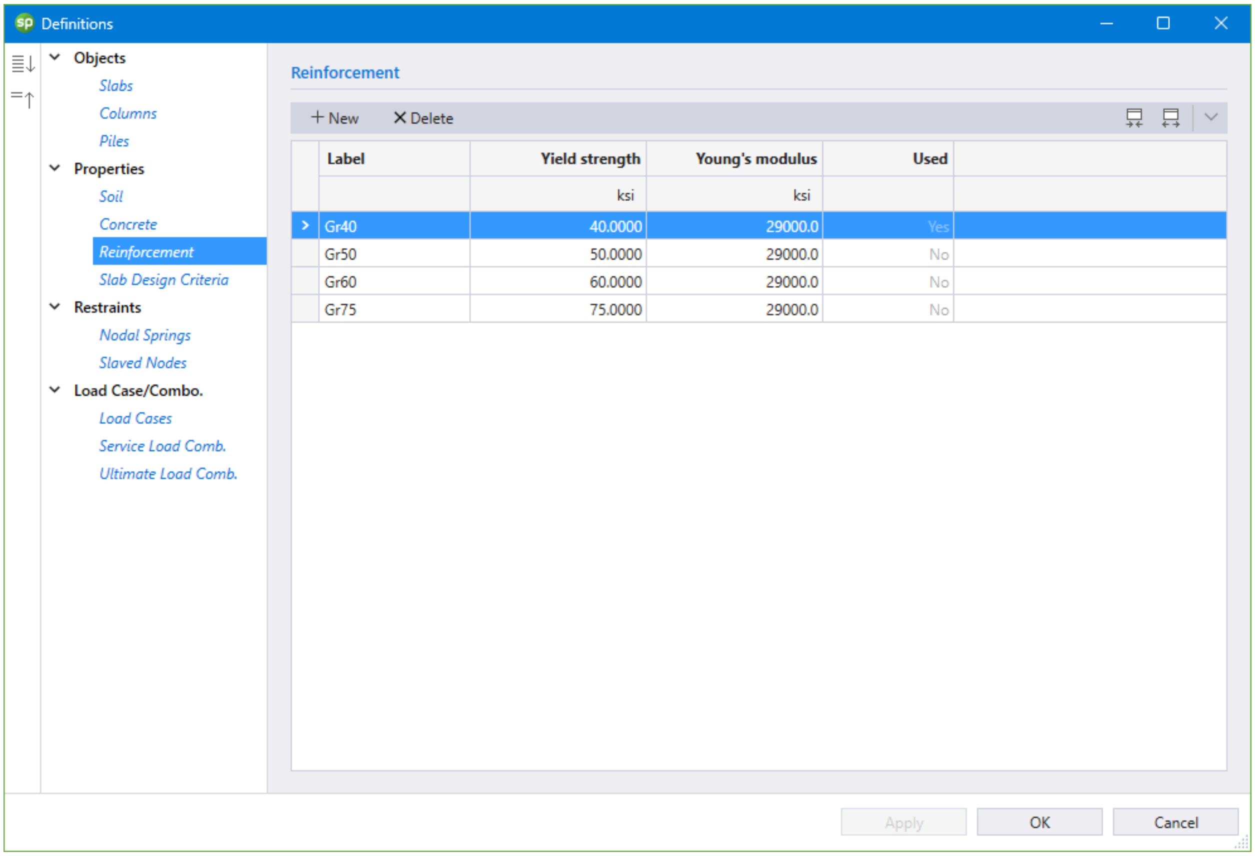
Concrete

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



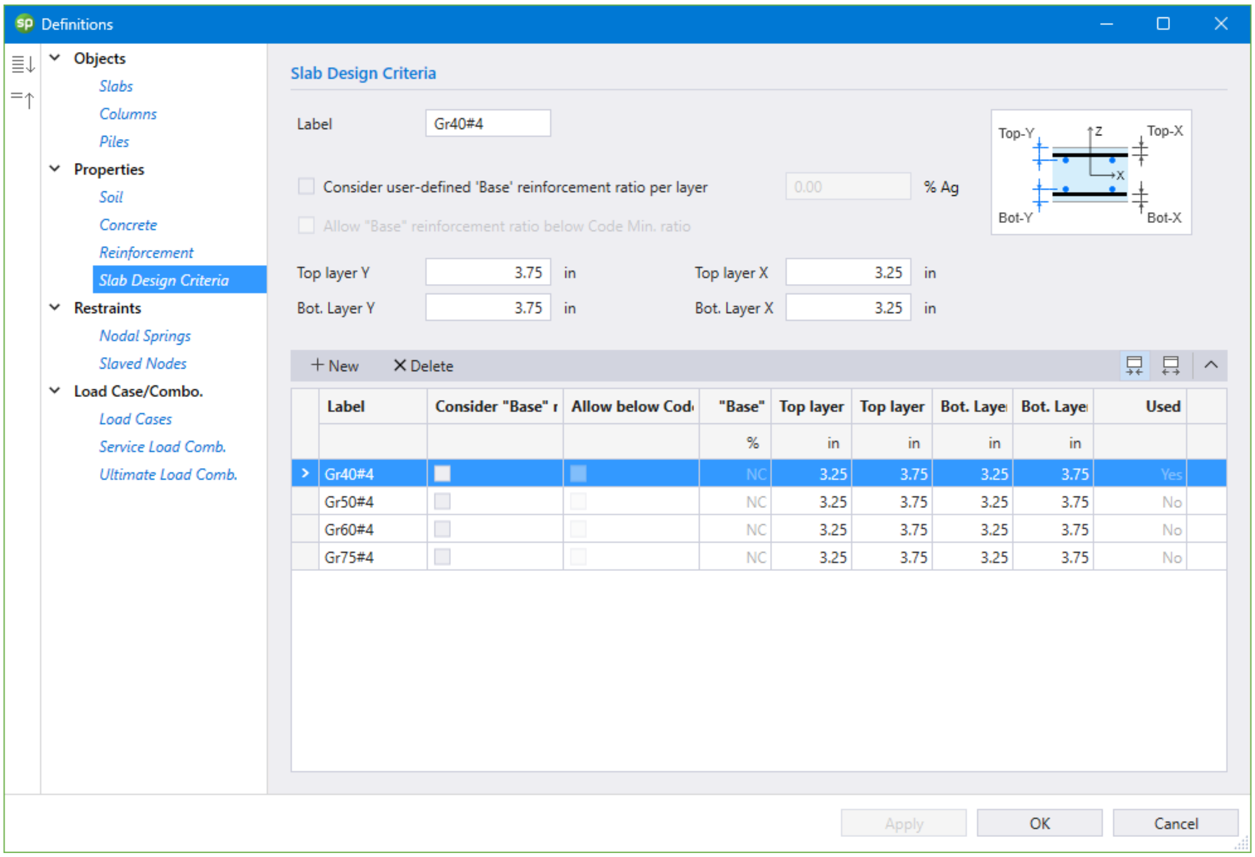
Reinforcement

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



Slab Design Criteria

The **Slab Design Criteria** definition consists of the LABEL, CONSIDER USER-DEFINED BASE REINFORCEMENT RATIO PER LAYER, ALLOW BASE REINFORCEMENT RATIO BELOW CODE MIN. RATIO, REINFORCEMENT CENTERLINE LOCATIONS, and whether the **Slab Design Criteria** label is USED in the model or not.

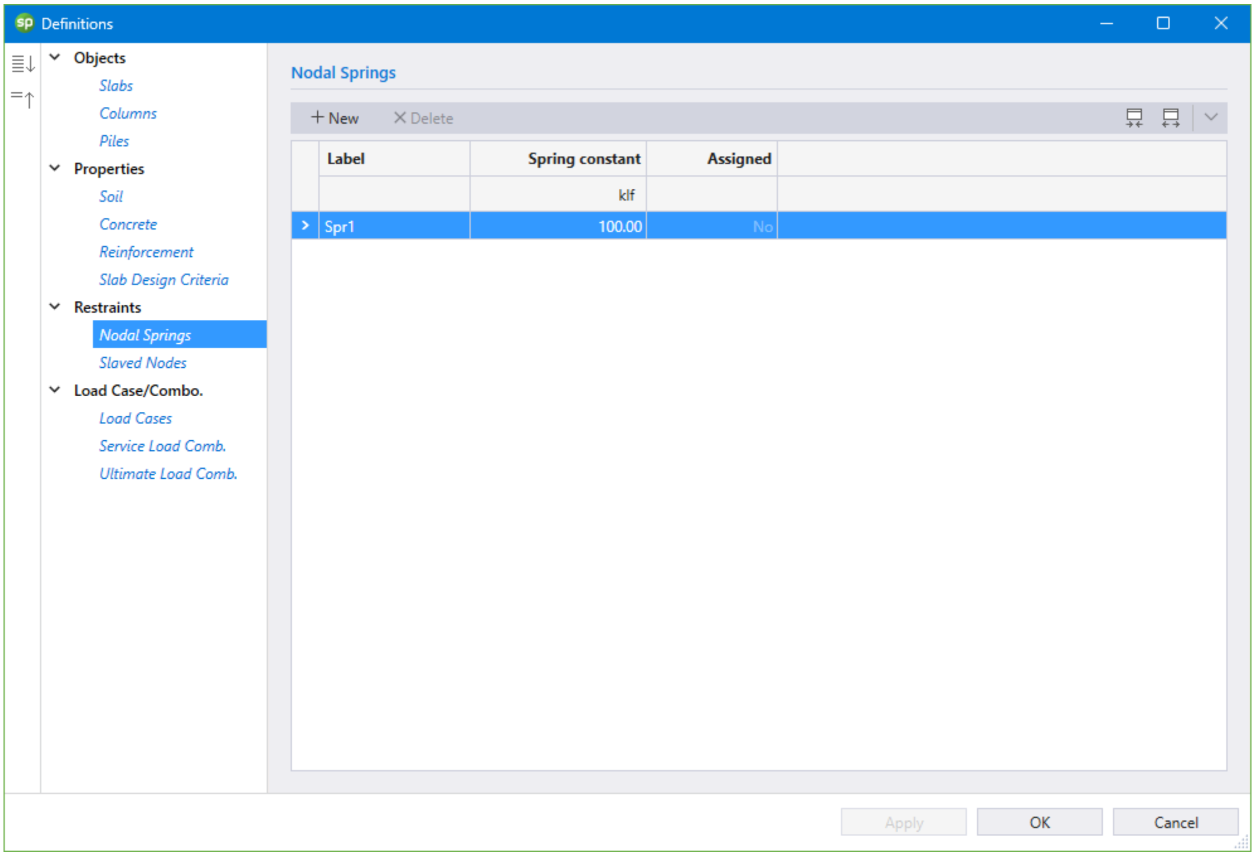


5.2.3.3. Restraints

The **Restraints** that can be defined are: **Nodal Springs,** and **Slaved Nodes**.

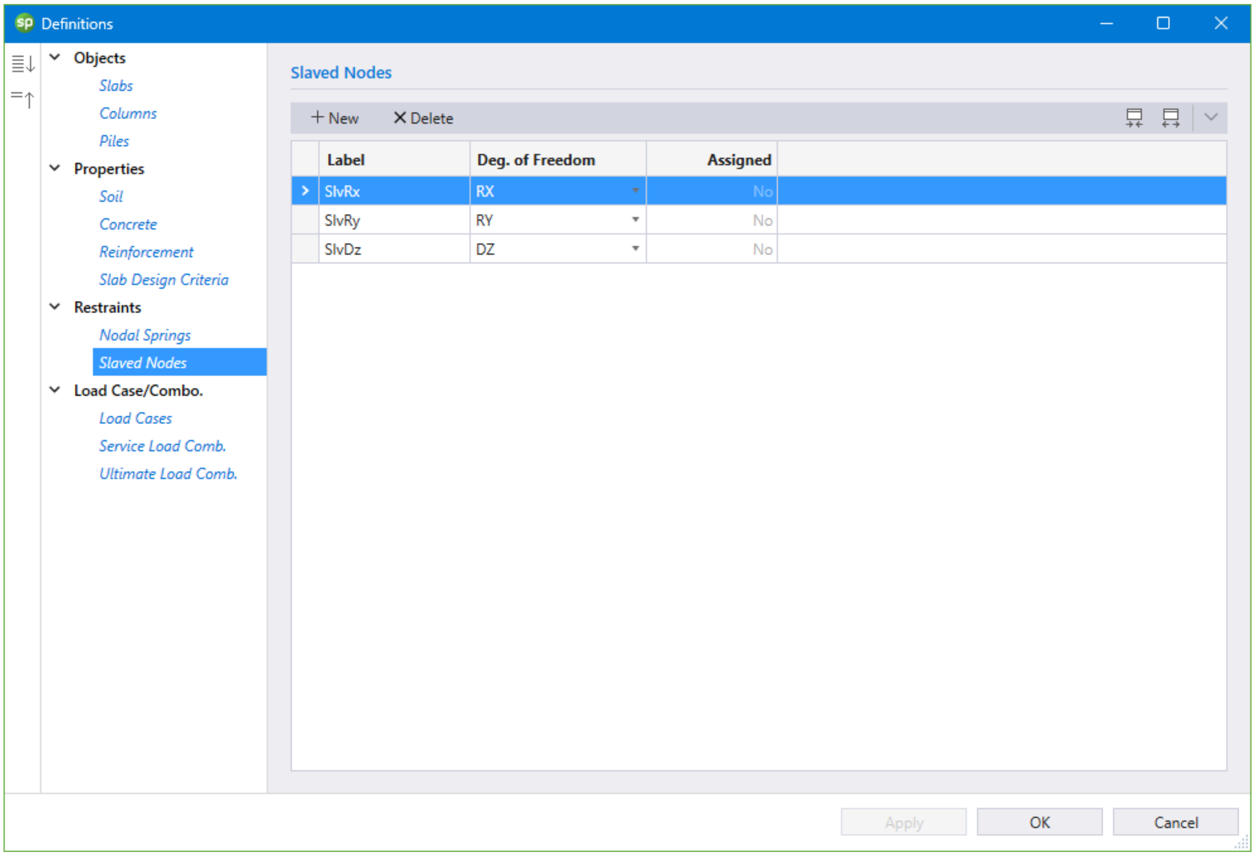
Nodal Springs

The **Nodal Springs** definition consists of the LABEL, SPRING CONSTANT, and whether the **Nodal Spring** label is ASSIGNED in the model or not.



Slaved Nodes

The **Slaved Nodes** definition consists of the LABEL, DEGREE OF FREEDOM, and whether the **Slaved Node** label is ASSIGNED in the model or not.

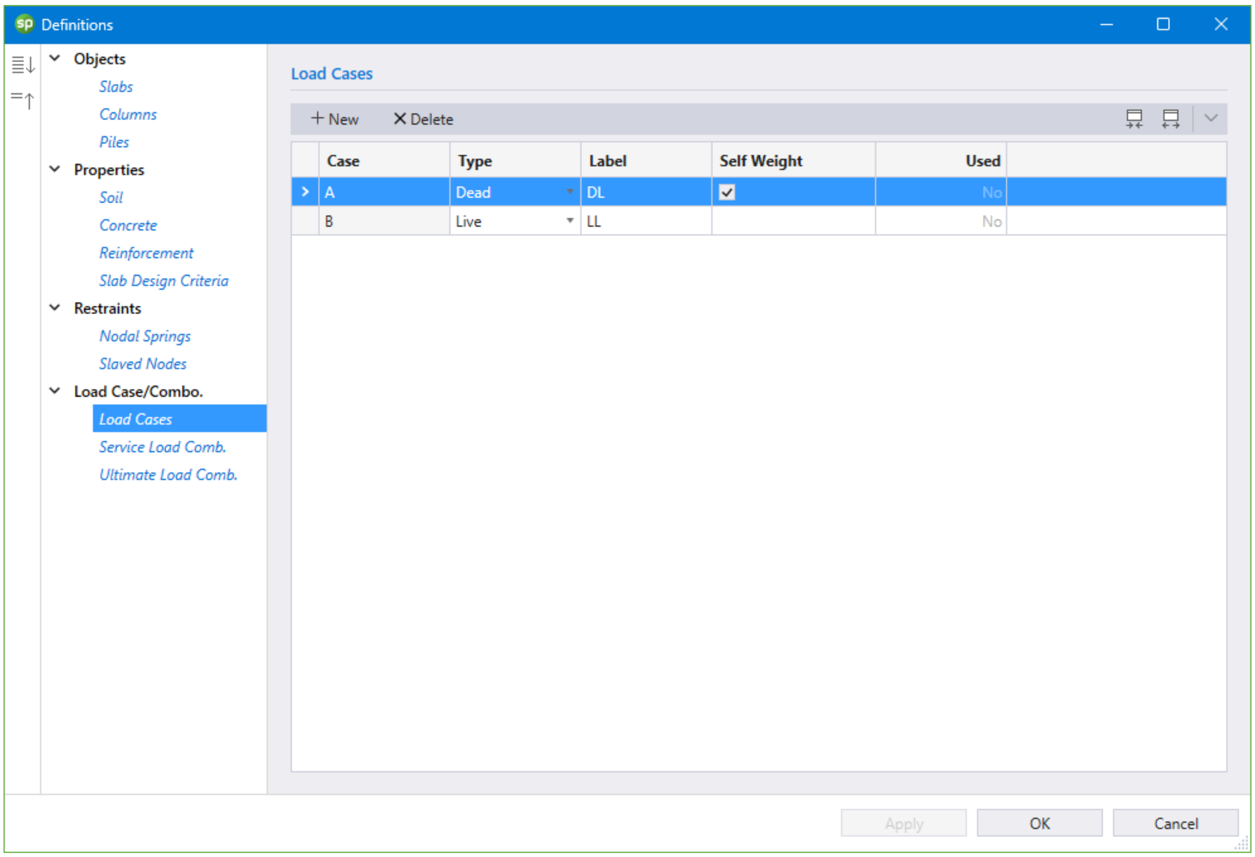


5.2.3.4. Load Case / Combo.

The **Load Case / Combo.** that can be defined are: **Load Cases**, **Service Load Combinations**, and **Ultimate Load Combinations**.

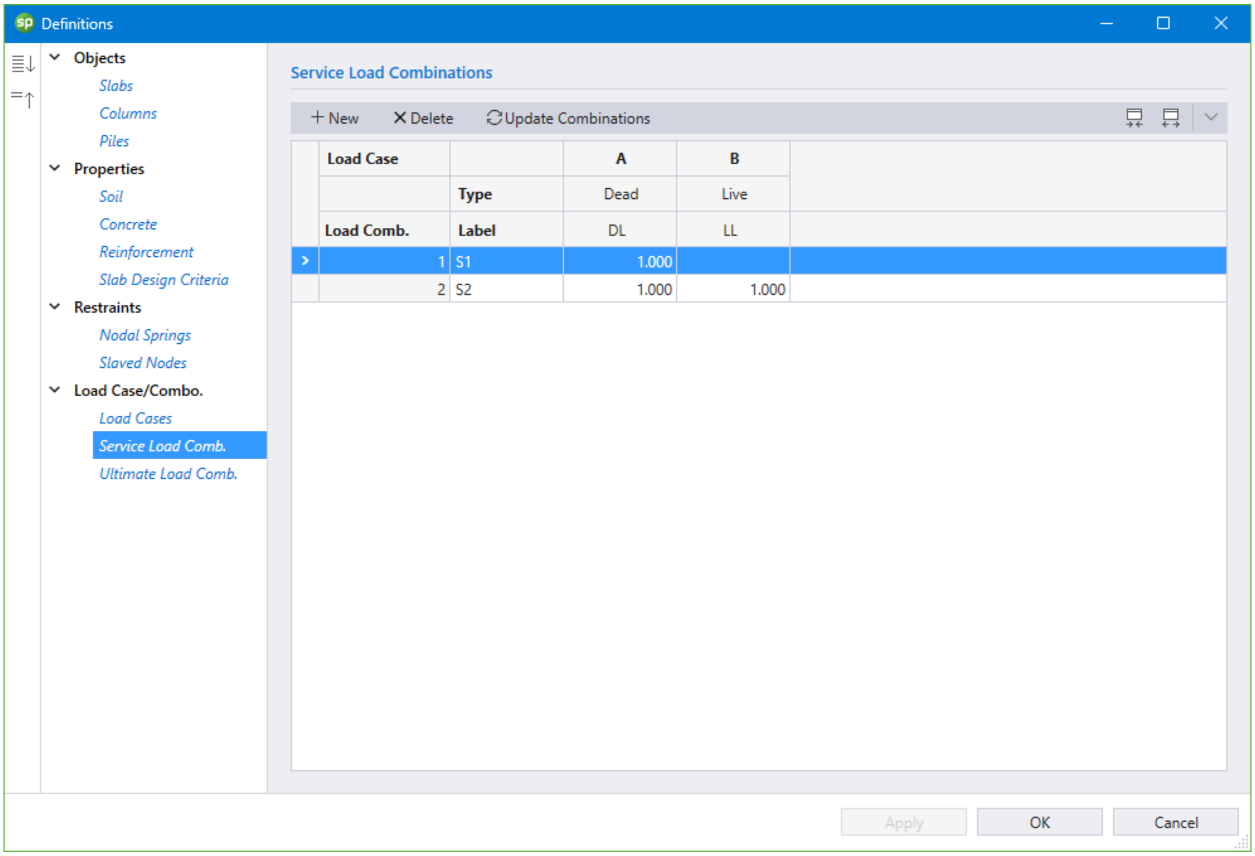
Load Cases

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.



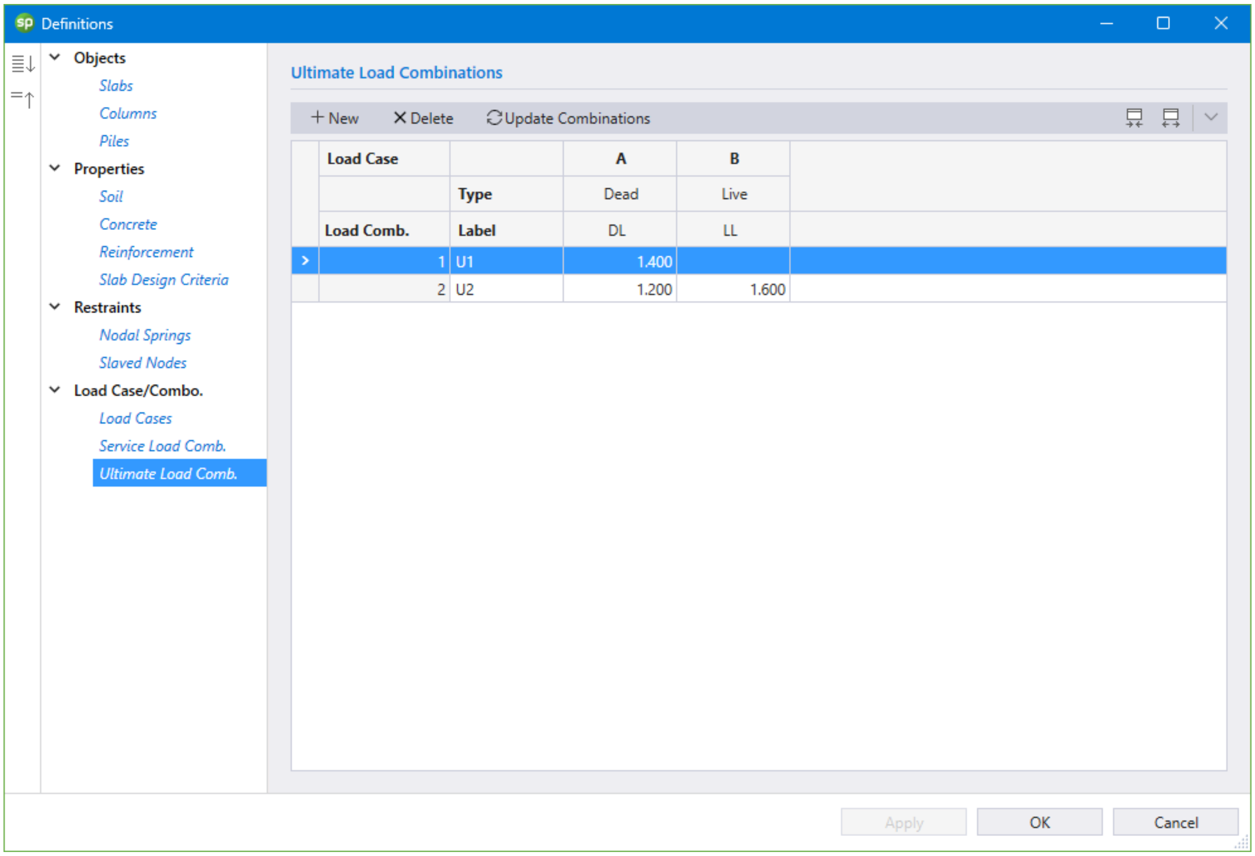
Service Load Combinations

The **Service Load Combinations** definition consists of the LOAD CASES, LOAD CASE 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, LOAD CASE 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.

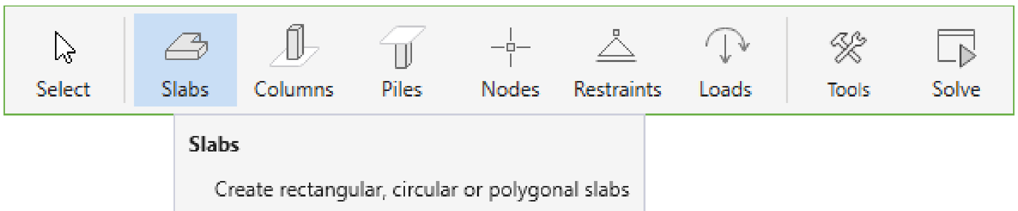


5.2.4. Creating Model Objects

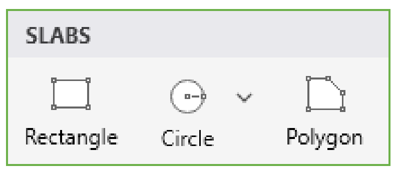
The Objects that can be created are: **Slabs**, **Columns**, and **Piles**.

5.2.4.1. Slabs

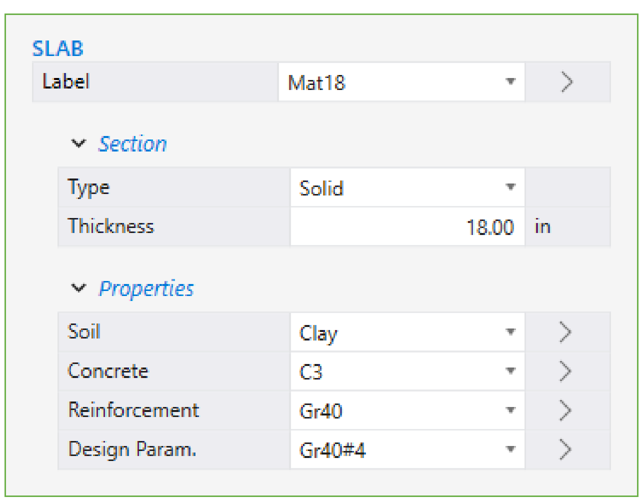
You can create **Slabs** using the **Slabs** command from the **Ribbon**.



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.

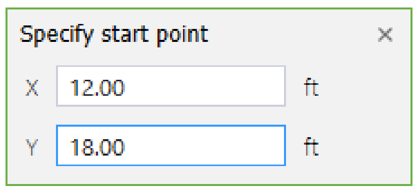


To draw a slab shape using the mouse, first select the desired tool.

- 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.

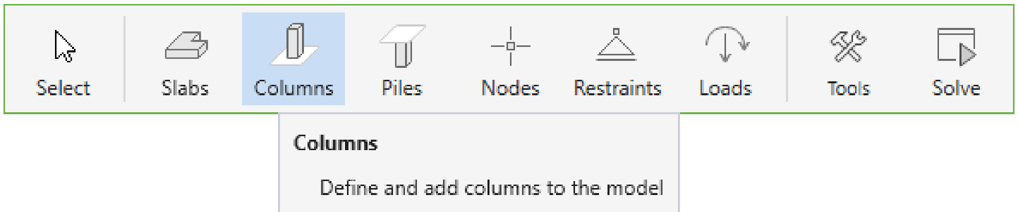
- To display a dynamic input box, press ENTER after you have selected the desired slab tool.



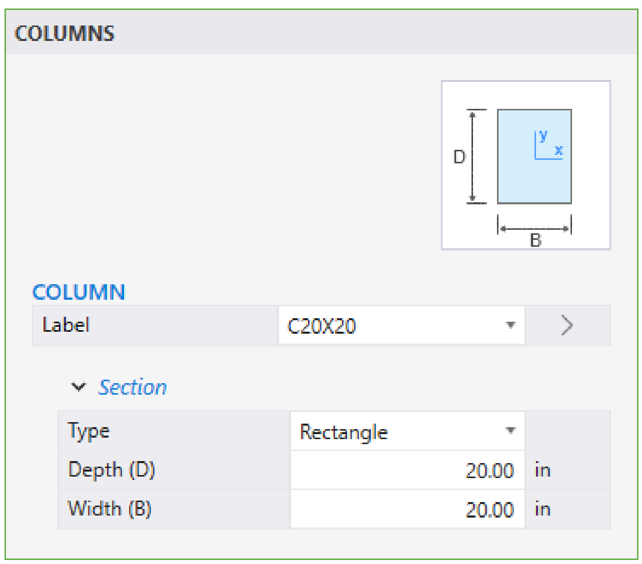
- You will be required to provide the start point (center point for circular slabs).
- Type in the required coordinates and press ENTER.
- Press ENTER once more to display the dynamic input box again. Type in the required quantities i.e. width and height for a rectangular slab or radius for a circular 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.

5.2.4.2. Columns

You can create **Columns** using the **Columns** 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 DEFINITIONS dialog.

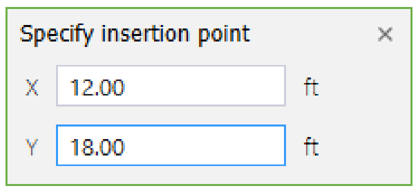


To assign a column using the mouse, first decide on the parameters of the column to be assigned. The cursor shape changes to the column selected to be assigned.

- 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.

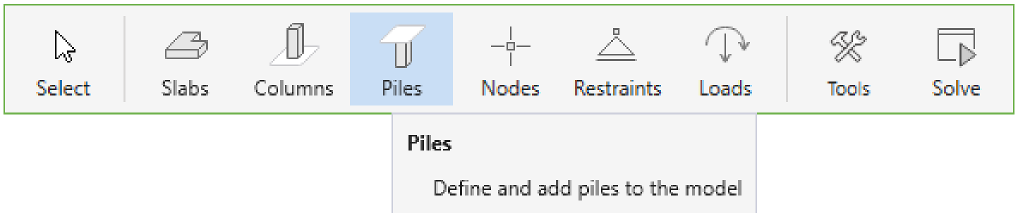
- To display a dynamic input box, press ENTER after you have selected the column to be assigned.



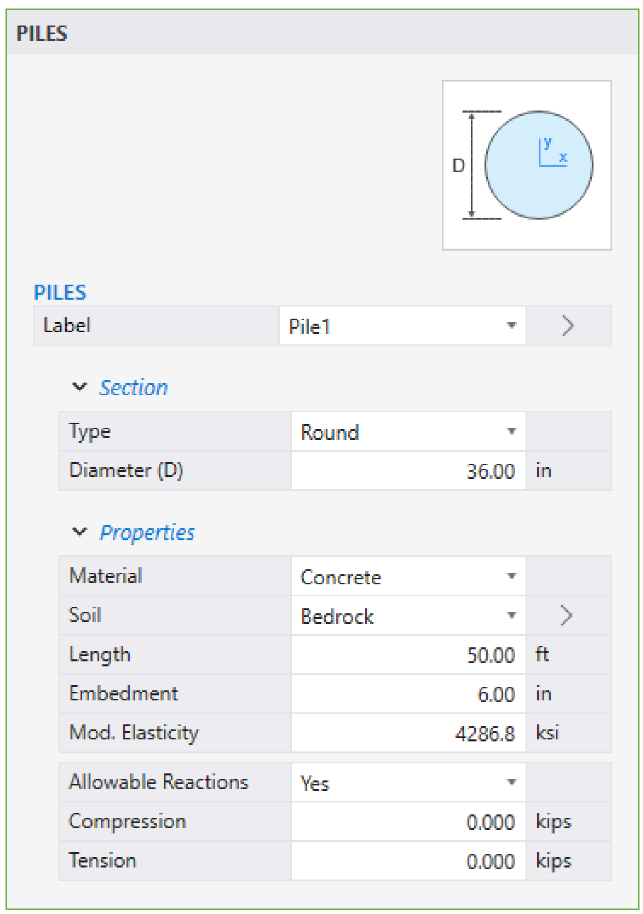
- You will be required to provide the insertion point.
- Type in the required coordinates and press ENTER.

5.2.4.3. Piles

You can create **Piles** using the **Piles** 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 allowable reactions (if applicable), and start assigning, the corresponding pile definition created will automatically be added to the DEFINITIONS dialog.

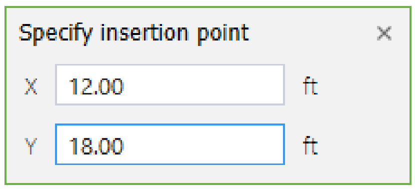


To assign a pile using the mouse, first decide on the parameters of the pile to be assigned. You can also define whether ALLOWABLE REACTIONS are considered for the pile in the **Properties** section. The cursor shape changes to the pile selected to be assigned.

- 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.

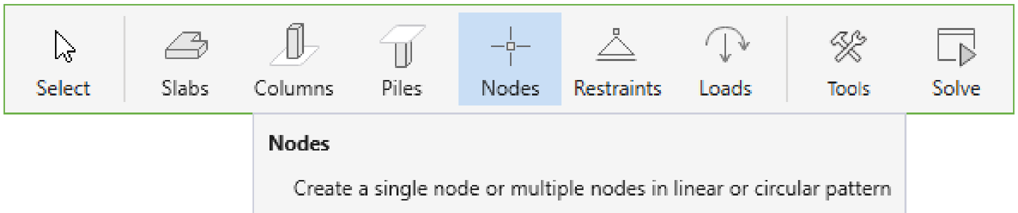
- To display a dynamic input box, press ENTER after you have selected the pile to be assigned.



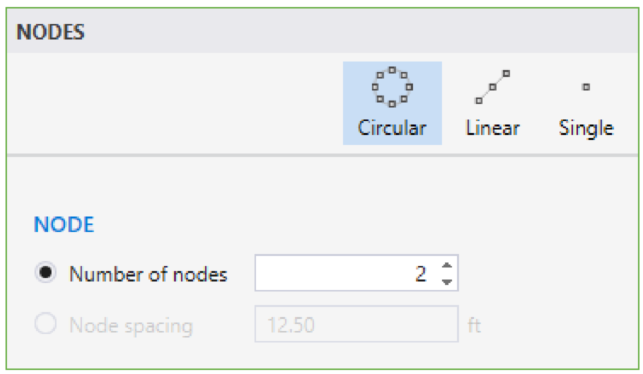
- You will be required to provide the insertion point.
- Type in the required coordinates and press ENTER.

5.2.4.4. Nodes

You can create **Nodes** scope using the **Nodes** 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**.

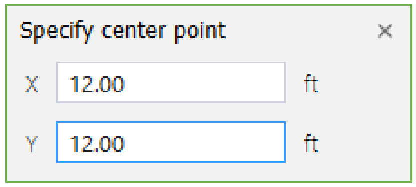


To draw nodes, first select the desired **Nodes** Tool.

- 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.

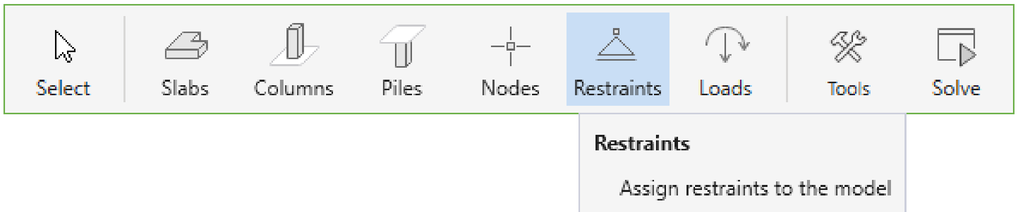
- To display a dynamic input box, press ENTER after you have selected the node tool and decided on the number or spacing of the nodes to be created.



- 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.

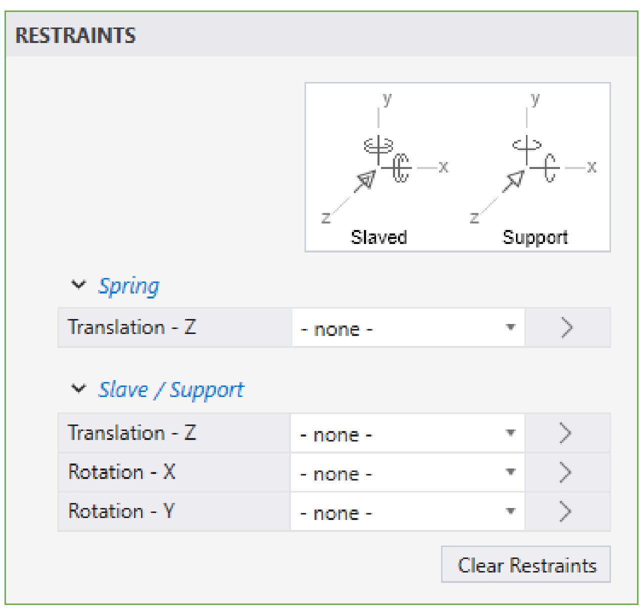
5.2.4.5. Restraints

The **Restraints** that can be assigned are: **Nodal Springs**, **Slaved Nodes**, and **Supports**. You can create **Restraints** using the **Restraints** command from the **Ribbon**.



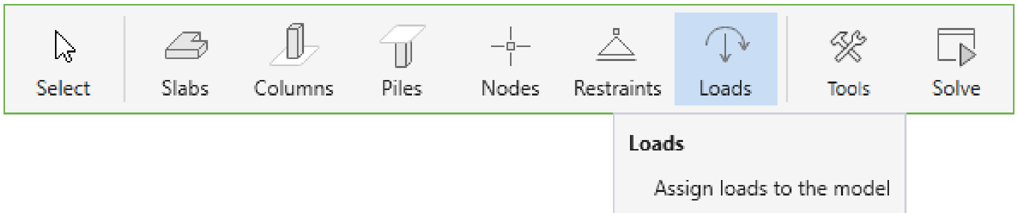
spMats considers restraints as nodal properties. It is possible to assign more than one kind of restraint to a node at the same time.

- 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.

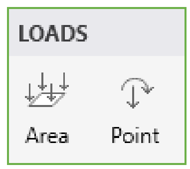


5.2.4.6. Loads

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

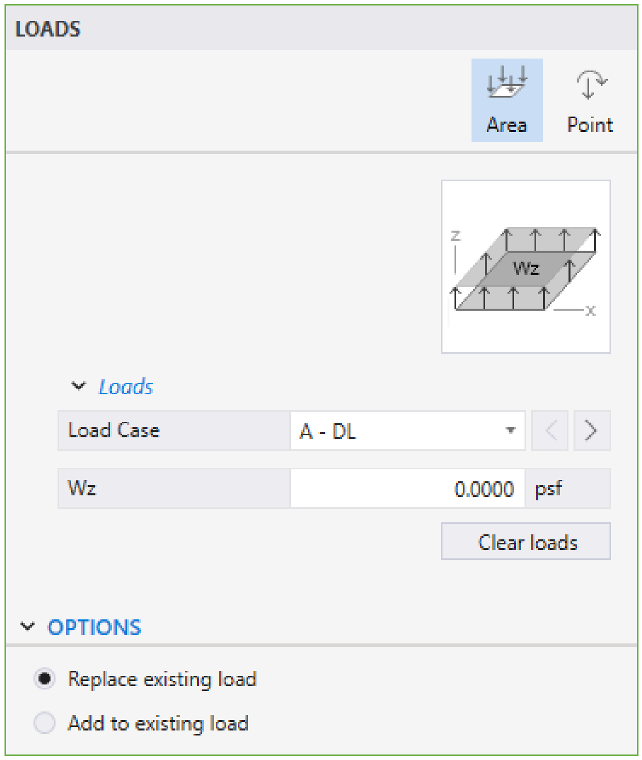


You can assign **Point** loads or **Area** loads by using one of the two options that are presented in the **Loads Left Panel**.



Assigning Area Loads

Area loads can only be assigned to **Slabs**.

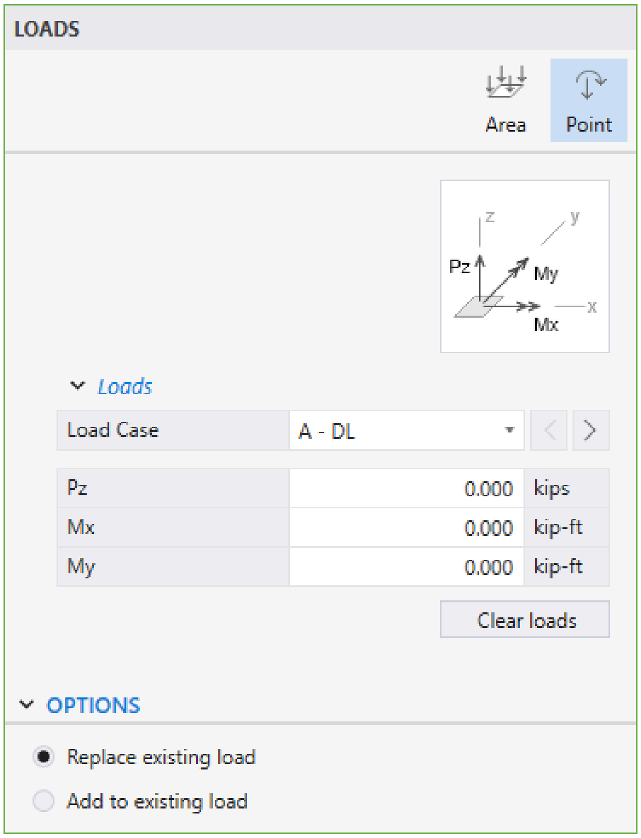


To assign an **Area** load, make sure that the **Area** command in the **Left Panel** is selected. The **Left Panel** should be displaying various **Area Loads** options.

- Select the LOAD CASE you want the **Area** load to belong to. You can always define LOAD CASES in the DEFINITIONS dialog.
- In the W_z box, type in the required load value. Note that the 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.

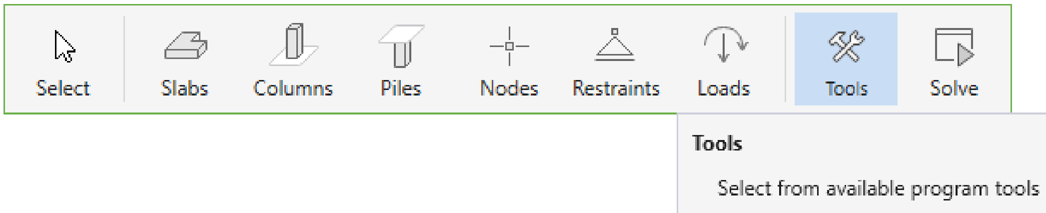


To assign a **Point** load, make sure that the **Point** command in the **Left Panel** is selected. The **Left Panel** should be displaying various **Point Loads** options.

- Select the LOAD case you want the **Point** load to belong to. You can always define LOAD CASES in the DEFINITIONS dialog.
- In the W_z, M_x, and M_y boxes, type in the required load and moment values. Note that the downward forces have negative values. To determine the direction of the moments M_x and M_y, 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.

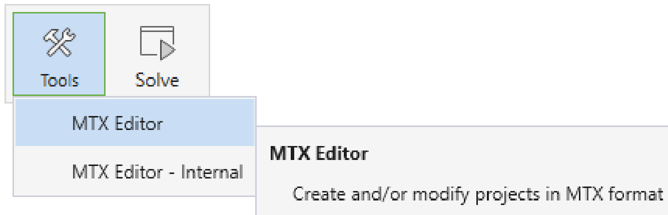
5.2.4.7. Tools

You can open **Tools** scope using the **Tools** command from the **Ribbon**.

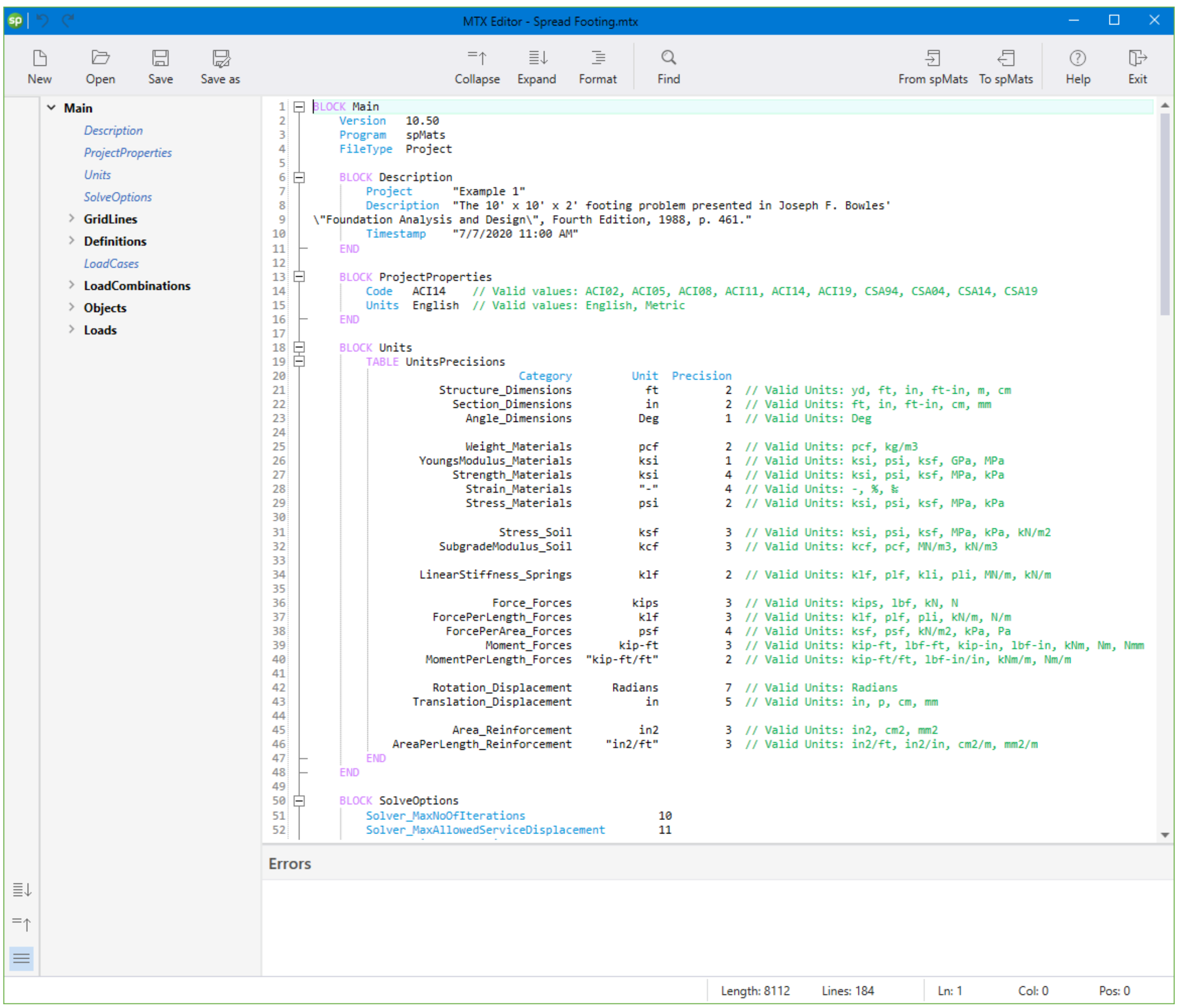


MTX Editor

You can open the **MTX Editor** using the **Tools** command from the **Ribbon**.



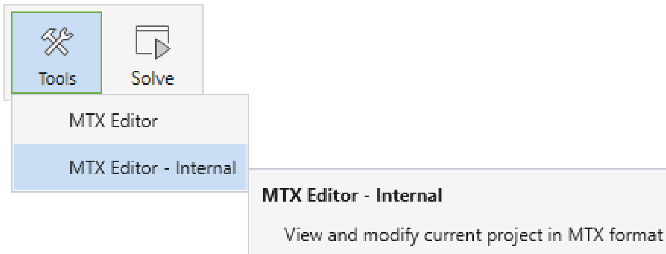
The **Main MTX Editor** works like a normal text editor. It can be used for opening, editing and saving both project and import/export (exchange) MTX files.



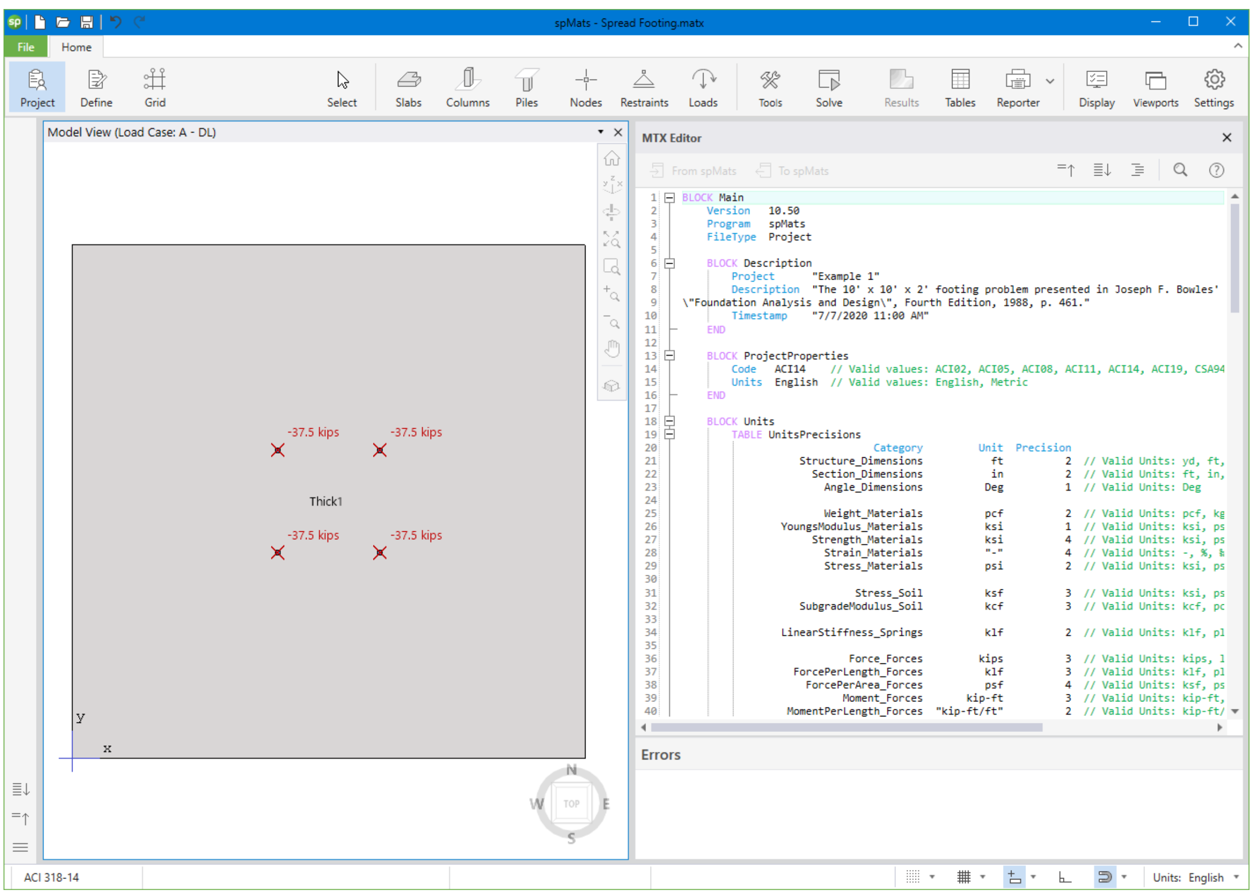
More information about this topic can be found in Section 3.6 and A.3.

MTX Editor - Internal

You can open the **MTX Editor - Internal** using the **Tools** command from the **Ribbon**.



MTX Editor Internal has the exact same interface as the **Main MTX Editor**. The difference between the two is that while **Main MTX Editor** can be used to load, edit and save files independent of what is loaded in the GUI, **MTX Editor Internal** is synced with the spMats project loaded in the main program. It can therefore be easily used to investigate spMats models, compare MTX data with spMats Models and also modify the active spMats models.



More information about this topic can be found in Section 3.6 and A.3.

5.2.5. Editing Model Objects

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

5.2.5.1. 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**.

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 section, properties, allowable reactions, 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.

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**.

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:

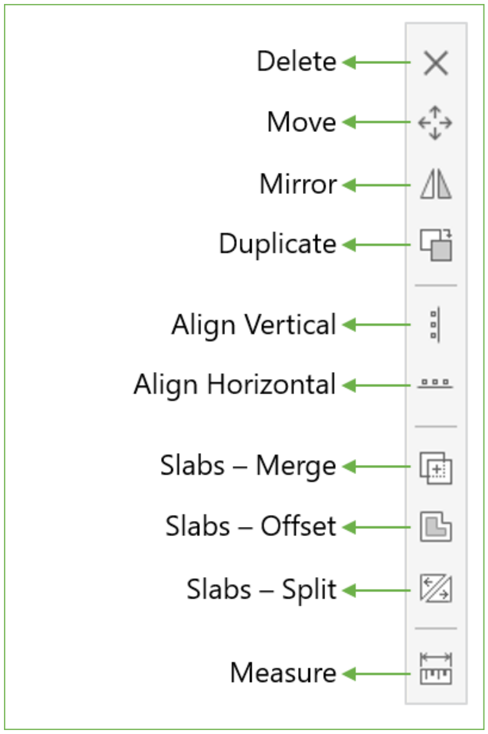
- Click on the node containing the load you want to edit to display its properties and position in the **Left Panel**.
- Change the desired load parameter.

To edit multiple point loads at once:

- Use marquee select to select multiple nodes with desired loads.
- Change the desired load parameter or parameters.

5.2.5.2. 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.

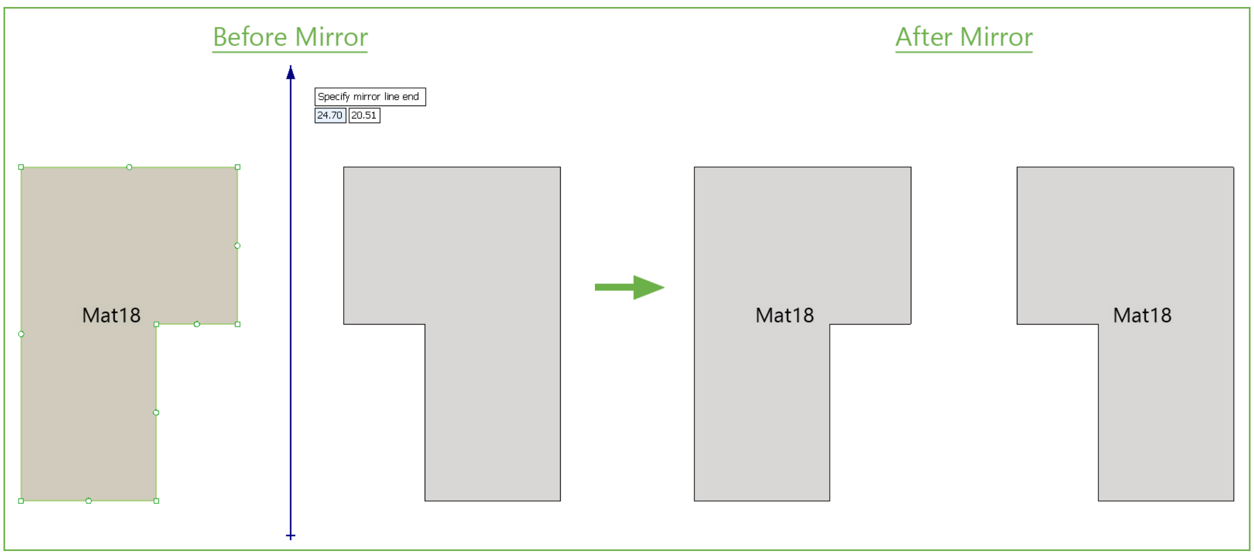
Mirror

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

- Select the item or items you want to mirror and click the **Mirror** command.
- Click on the screen to specify the start point of the mirror line.

  Alternatively, you can also press ENTER and manually enter the coordinates of the start of the mirror line.
- Next click on the screen to specify the end point of the mirror line.

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



Duplicate

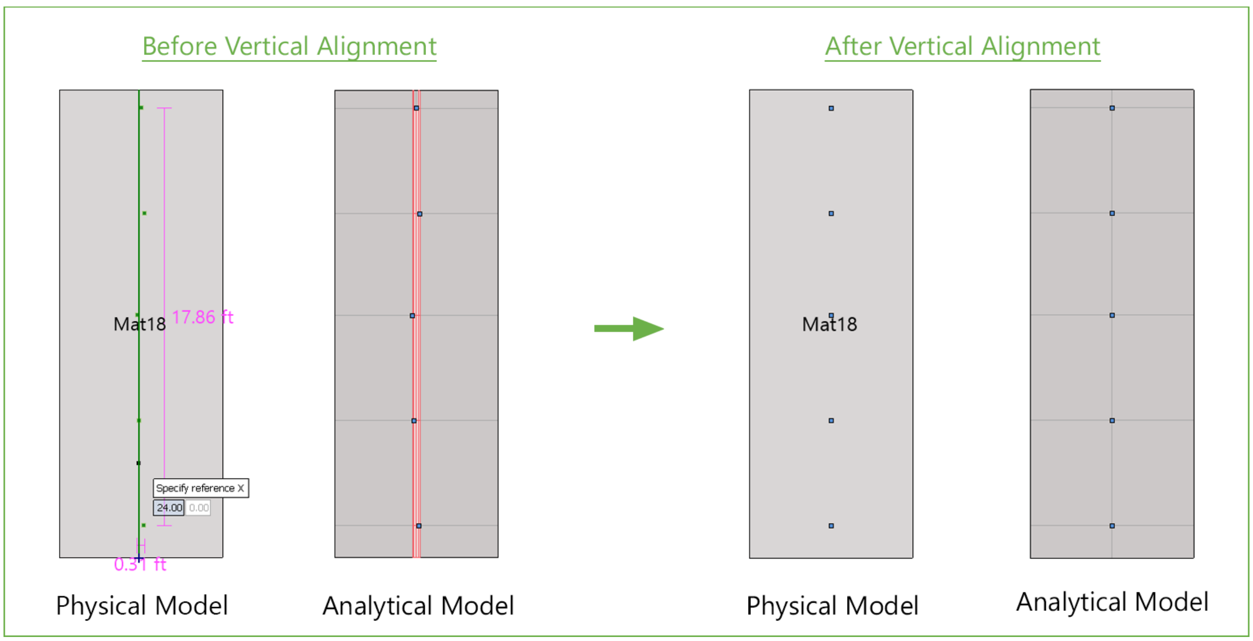
The **Duplicate** command is active only when one or more items are selected.

- Select the item or items you want to duplicate of and click the **Duplicate** command.
- Click on the screen to specify the base point from which to start the duplicate process. Alternatively, you can also press ENTER and manually enter the coordinates of the base point.
- Drag the duplicated 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 duplicated instances of items to their new location.

Align Vertical

The **Align Vertical** command is active only when one or more model items are selected. This command is used to eliminate very small difference in location coordinates that could result in unwanted elements with an irregular or high aspect ratio.

- Select the item or items you want to align vertically in a straight line and click the **Align Vertical** command.
- Click on the screen to specify the reference X point at which to vertically align all the selected items.
- Alternatively, you can also press ENTER and manually enter the coordinate of the X reference point and press ENTER to complete alignment.



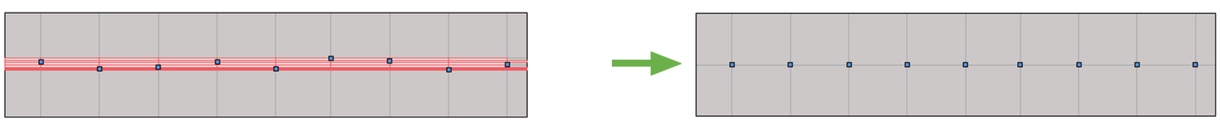
Align Horizontal

The **Align Horizontal** command is active only when one or more items are selected. This command is used to eliminate very small difference in location coordinates that could result in unwanted elements with an irregular or high aspect ratio.

- Select the item or items you want to align horizontally in a straight line and click the **Align Horizontal** command.
- Click on the screen to specify the reference Y point at which to horizontally align all the selected items.
- Alternatively, you can also press ENTER and manually enter the coordinate of the Y reference point and press ENTER to complete alignment.

Before Horizontal Alignment   After Horizontal Alignment

Physical Model   Physical Model

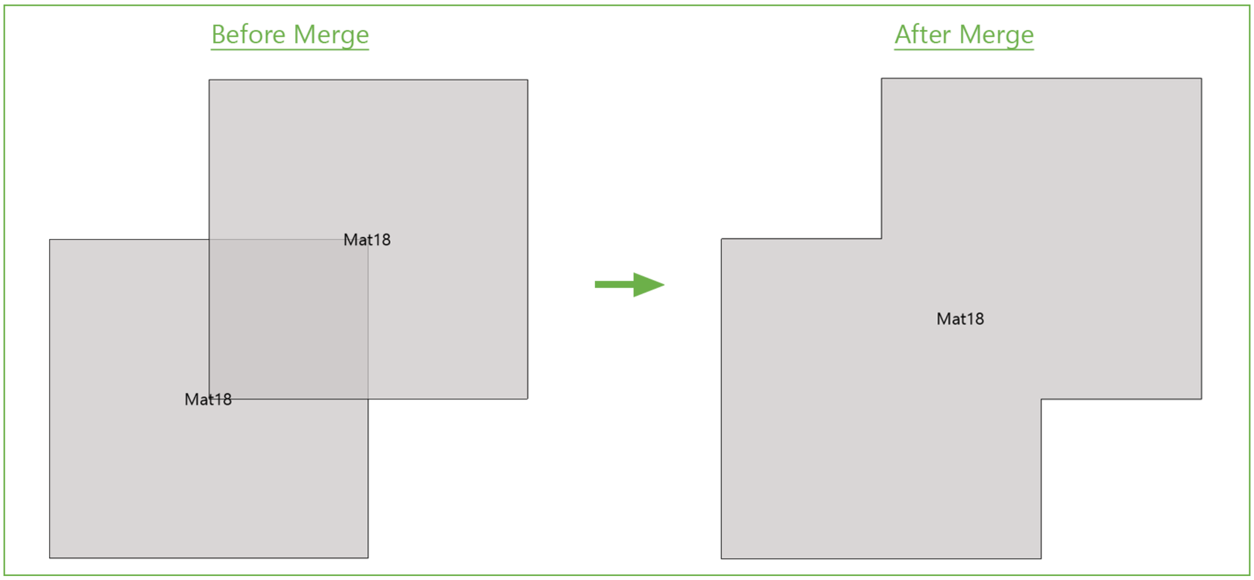


Analytical Model   Analytical Model

Slabs - Merge

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

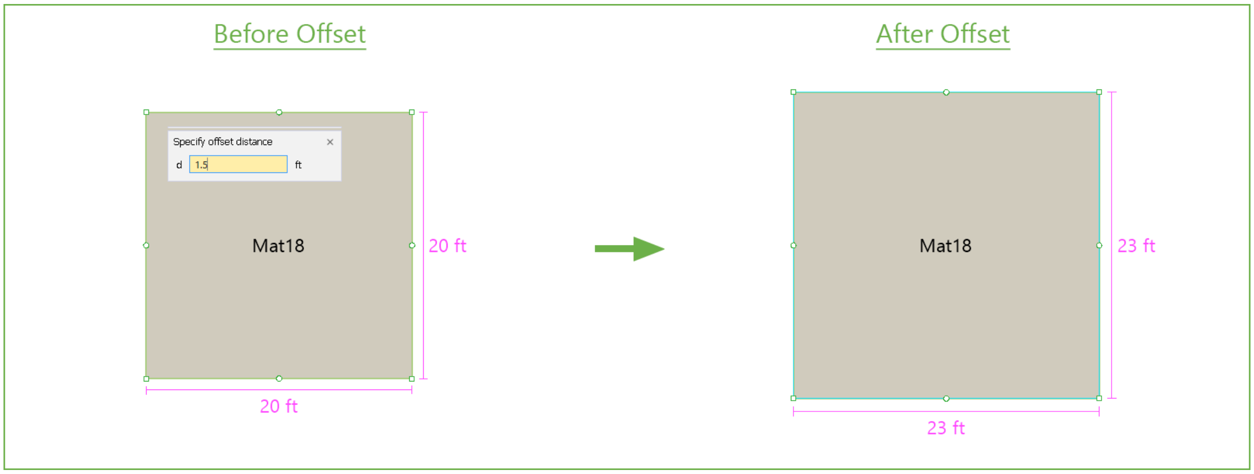
- Select the slabs that you want to merge and click the **Slabs - Merge** command.
  While merging two or more slabs, the program requires you to select a reference slab, the properties of which will be applied to the final merged slab. These include section type and thickness along with properties like 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. It is used to enlarge or reduce a slab overall size proportionally all around the perimeter to create accommodation for architectural or other features in a slab element.

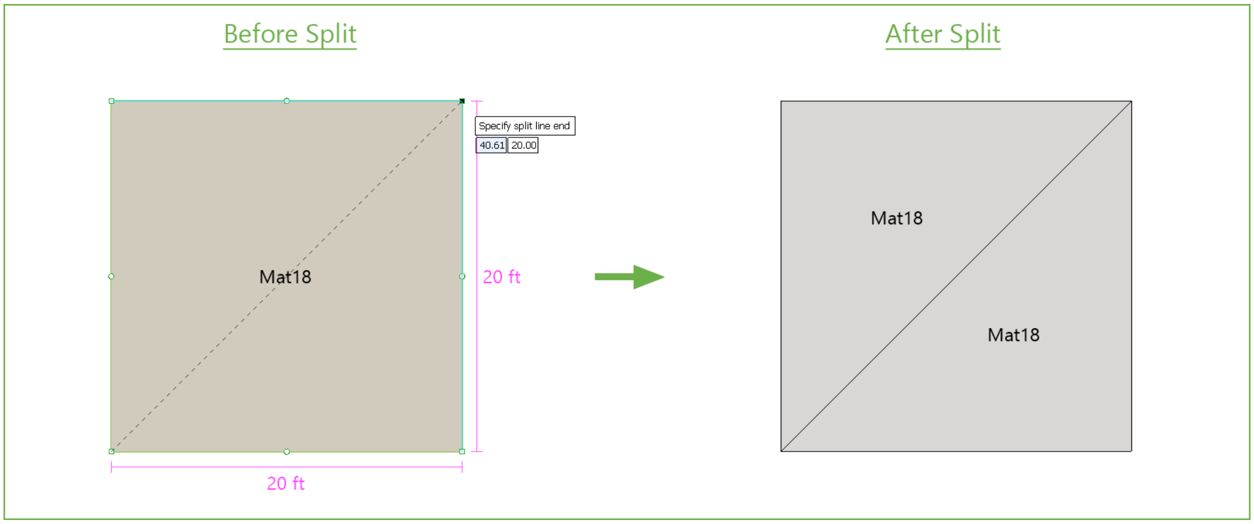
- Select the slabs that you want to offset and click the **Slabs - Offset** command.
  It is possible to offset the slab inwards or outwards.
- Click the side of the slab you want to offset.
- In the input box that appears, specify the offset distance and press ENTER.



Slabs - Split

The **Slabs - Split** command is active only when one or more slabs are selected. Splitting a slab is useful in situations where different load or support options need to be assigned to two separate parts of a slab created previously.

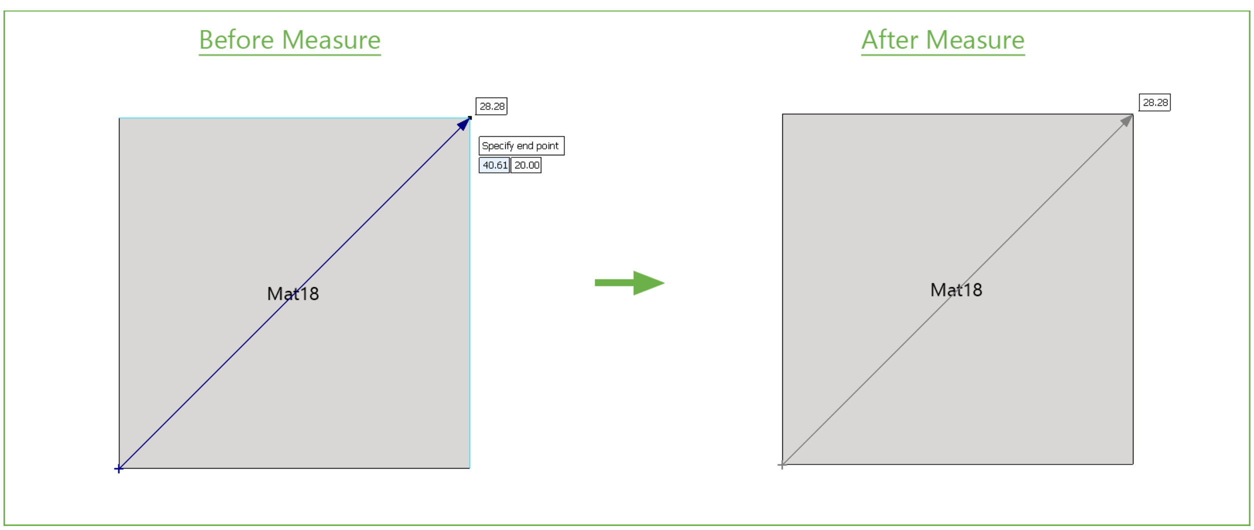
- Select the slabs that you want to split and click the **Slabs - Split** command.
- Click on the screen to specify the start point of the cutting line.
- Alternatively, you can also press ENTER and manually enter the coordinates of the start of the cutting line.
  Note that a slab can be split only 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.



Measure

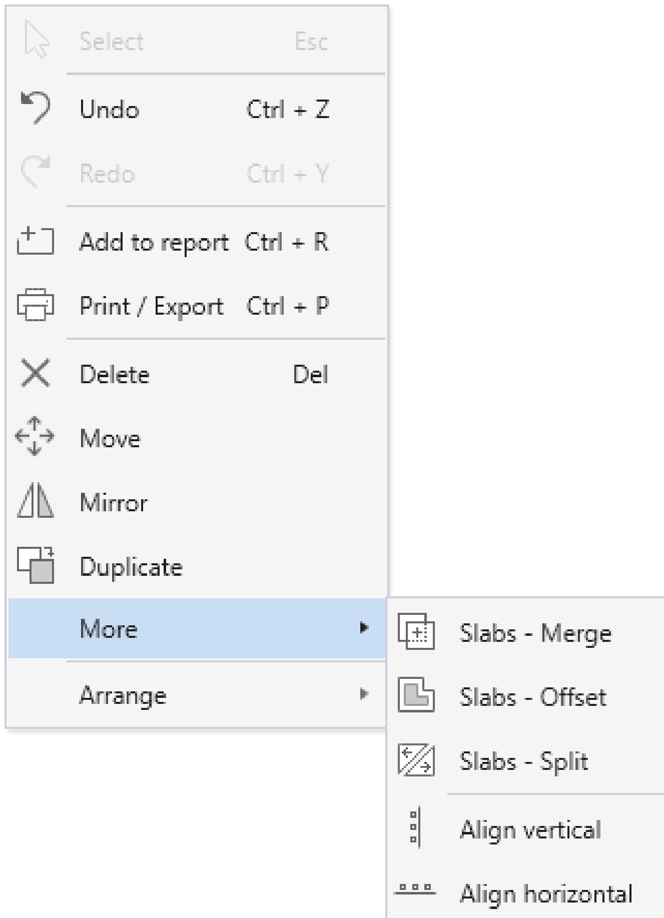
The **Measure** command is always active and can be used to measure distance between any two points. This command is particularly useful for obtaining the distance between two points along a diagonal.

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



5.2.5.3. 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.

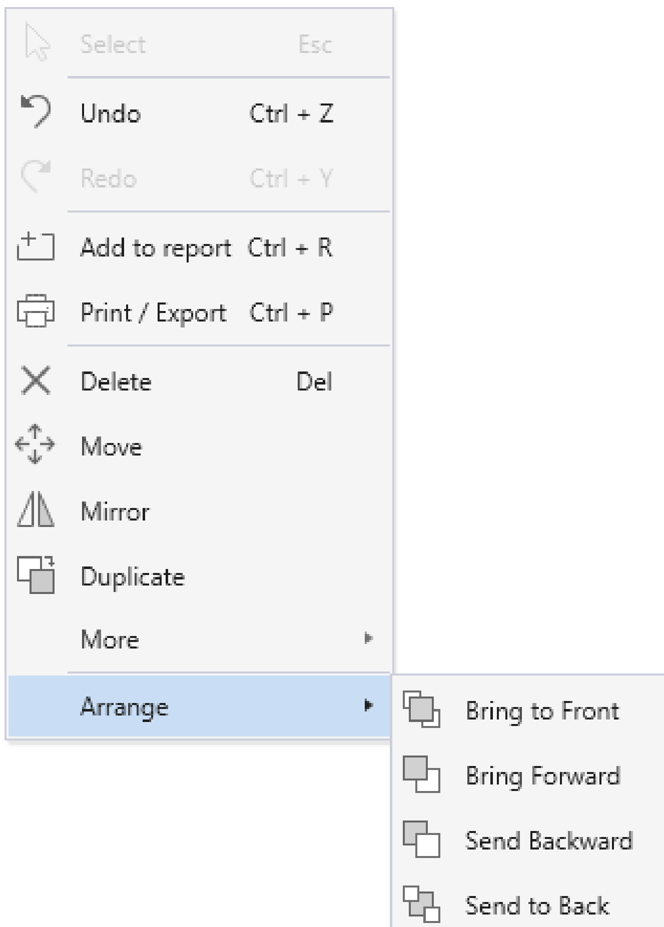


5.2.5.4. Understanding Slab Layers

While creating models, the need may arise 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.

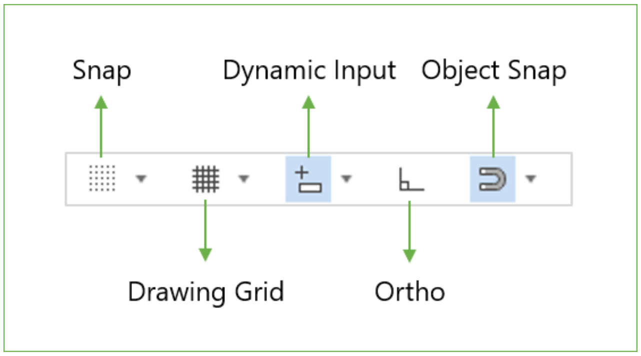


- From the **Arrange** sub menu, select the desired action.

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

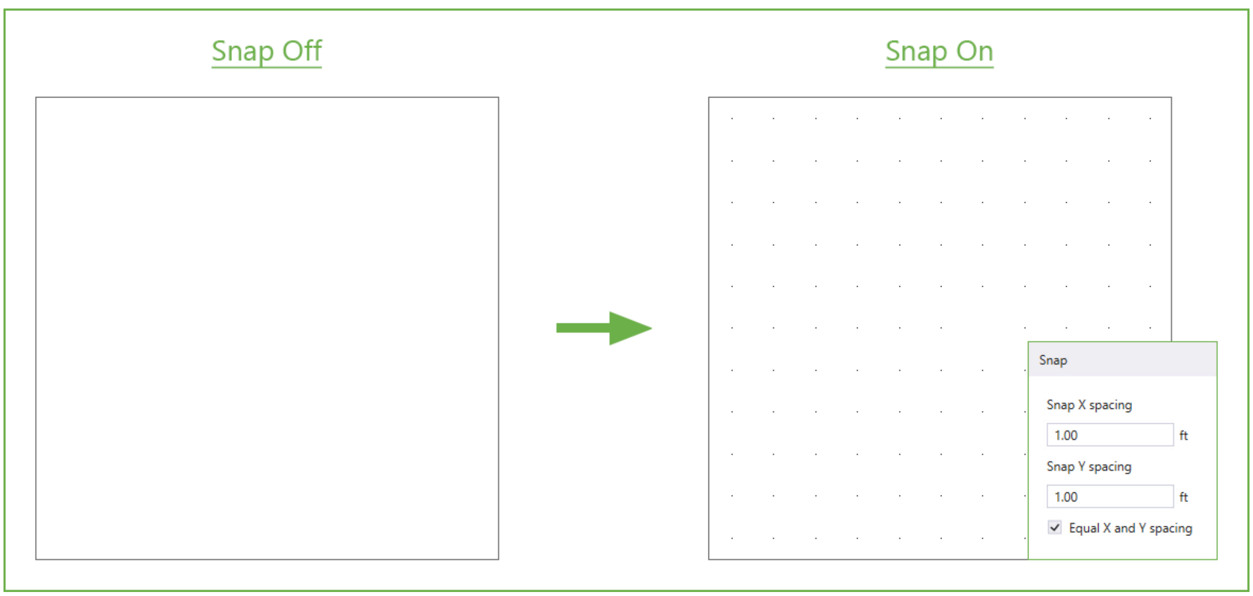
5.2.6. Using Drafting Aids

The following figure shows essential **Drafting Aids** that enhance modeling precision and workflow during the modeling or editing process.



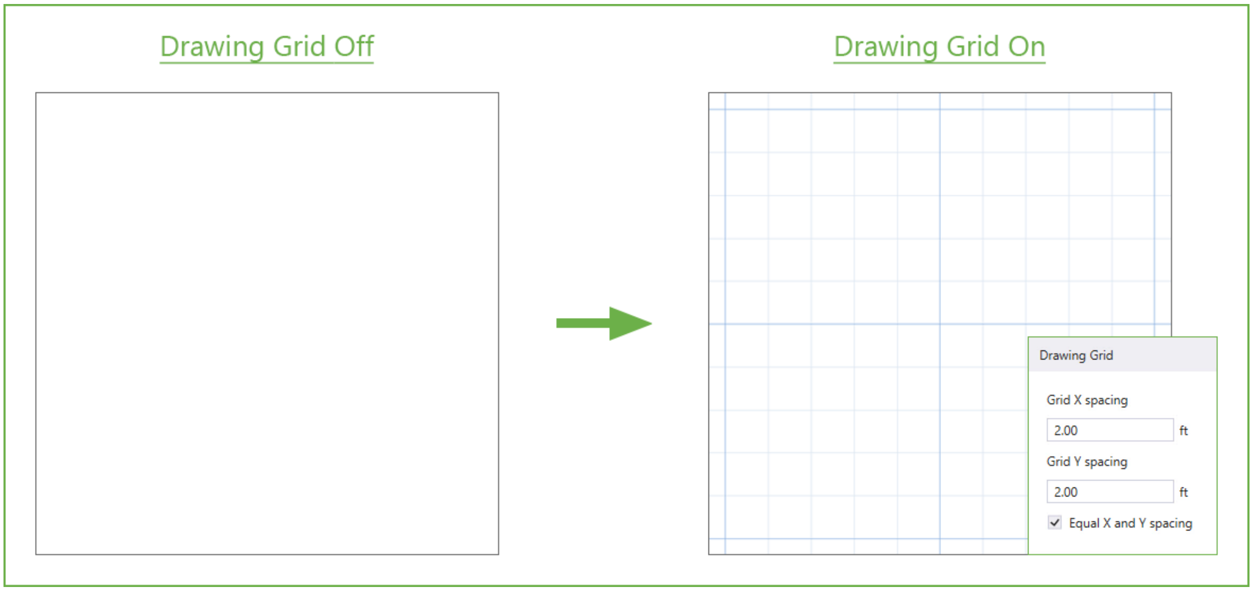
5.2.6.1. Snap

The **Snap** command enables the cursor to automatically align with a predefined snap points, allowing for precise placement of elements during the modeling or editing process.



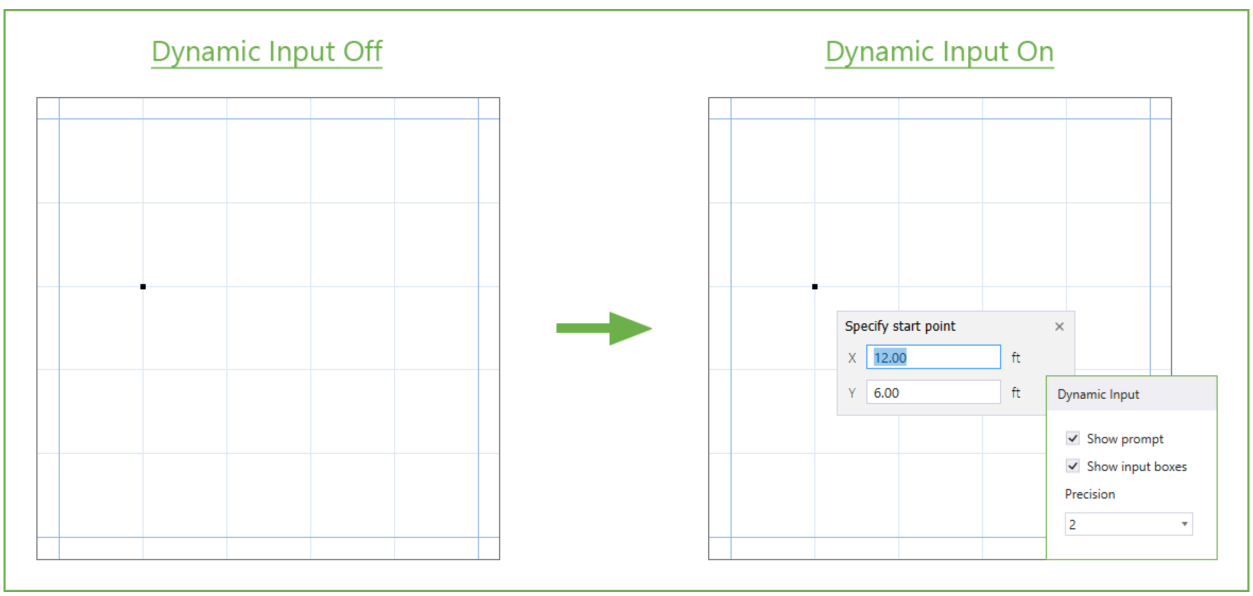
5.2.6.2. Drawing Grid

The **Drawing Grid** command enables the cursor to automatically align with a predefined grids, allowing for precise placement of elements during the modeling or editing process.



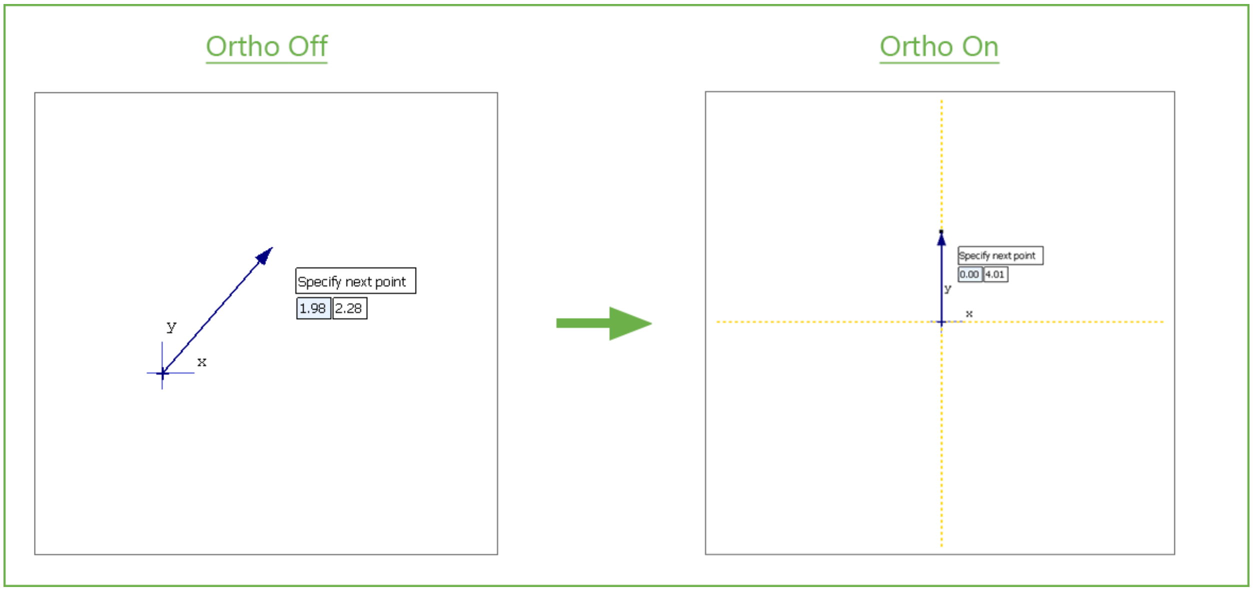
5.2.6.3. Dynamic Input

The **Dynamic Input** command allows to enter coordinates and dimensions directly on-screen near the cursor. This feature can be toggled on or off while creating or editing objects in the viewport.



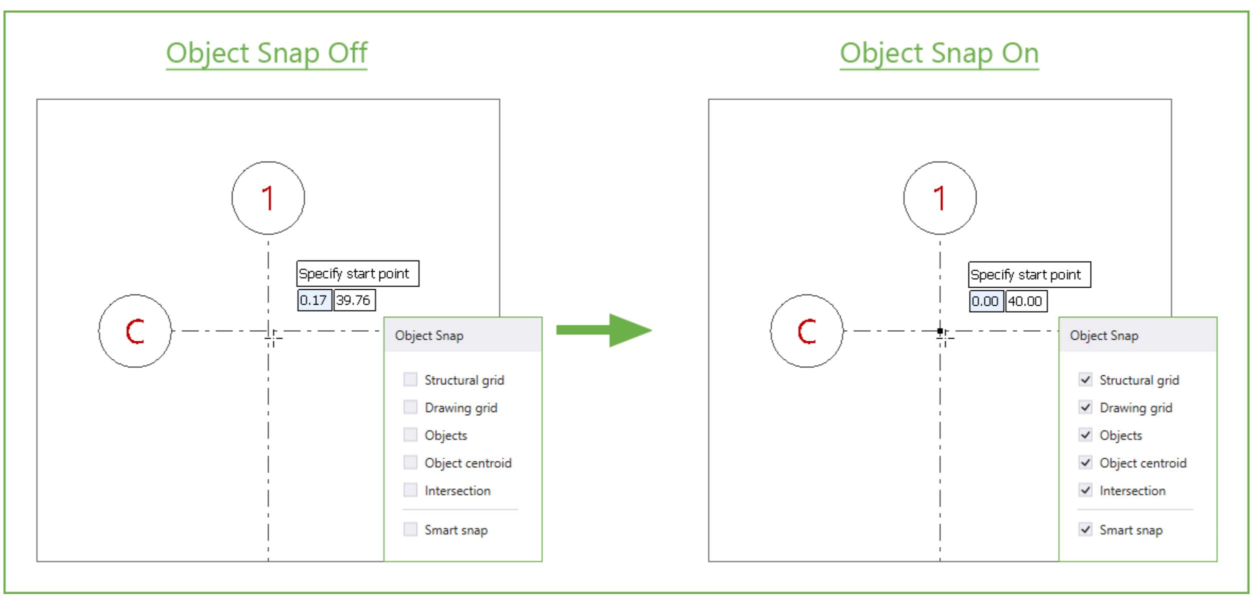
5.2.6.4. Ortho

The **Ortho** command restricts cursor movement to horizontal or vertical directions, when creating or modifying objects.

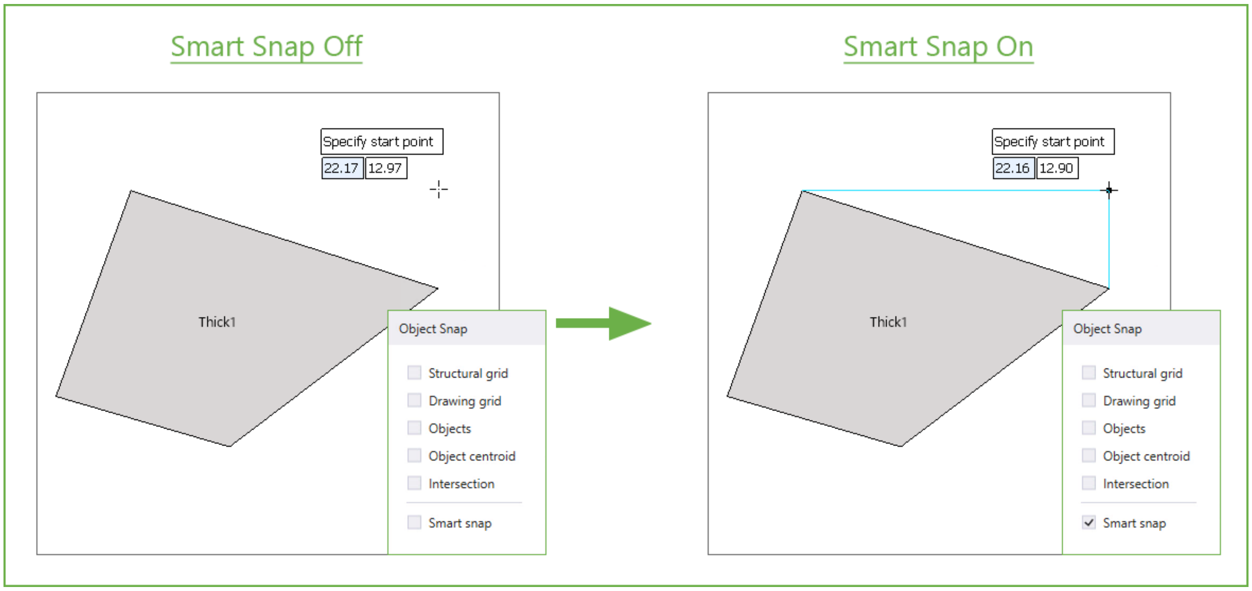


5.2.6.5. Object Snap

The **Object Snap** command allows the cursor to automatically detect and lock onto specific reference points such as structural grids, drawing grids, objects, object centroids, and intersections.



Additionally, the **Smart Snap** command shows Horizontal and Vertical tracking guides to object vertices, midpoints, geometric center and apparent intersections even when the cursor is at a significant distance from the points. The cursor snaps to these guides and any location on the guide can be used as a reference to add to or edit the model.

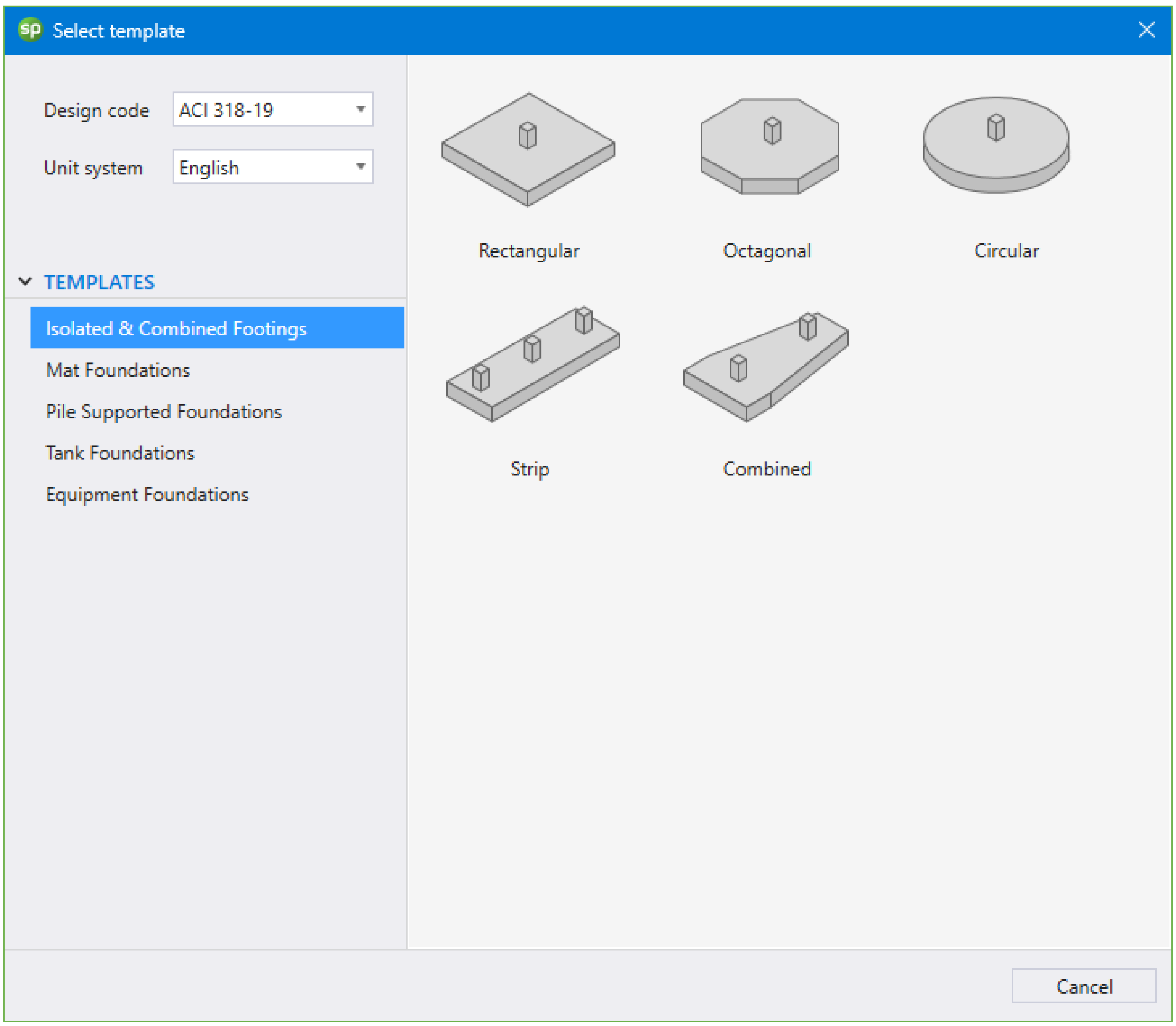


5.3. Modeling with Templates

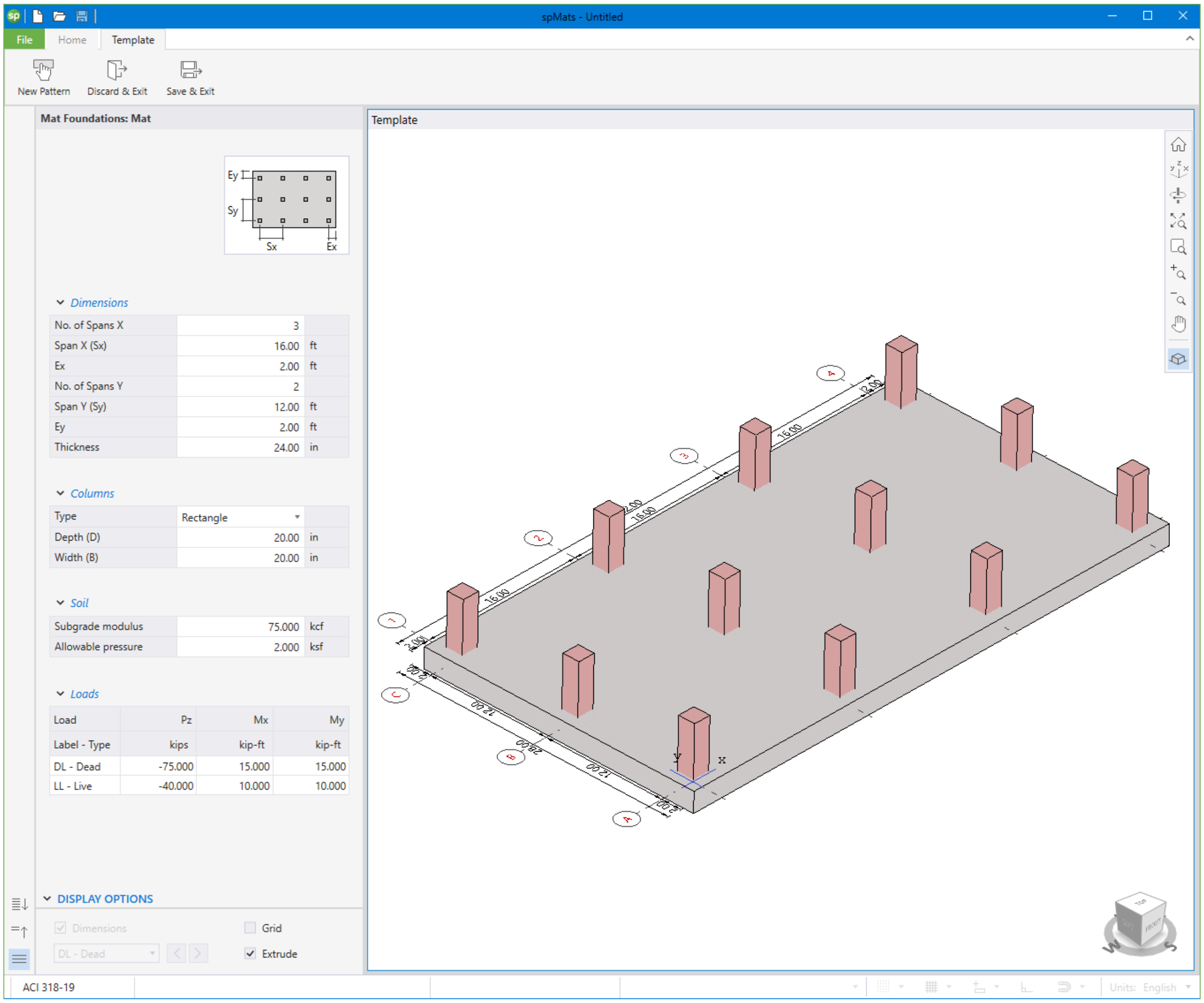
5.3.1. Utilizing Templates

Templates are 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. Clicking on a template image 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.

5.3.2. Template Ribbon

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

5.3.2.1. New Pattern

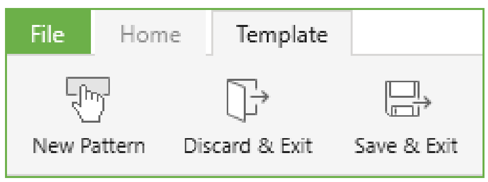
Opens the TEMPLATE Selection Dialog Box. Selecting a New template will discard the old template and load the new one.

5.3.2.2. Discard & Exit

Discards the current template and exits to spMats home screen.

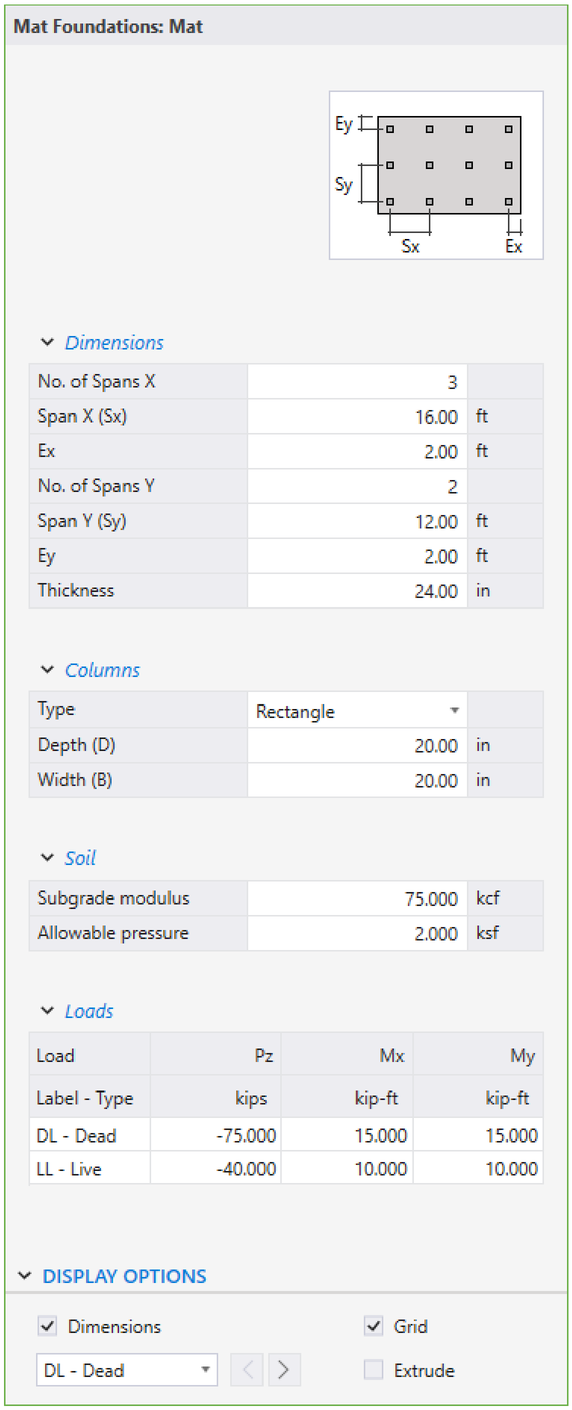
5.3.2.3. Save & Exit

Imports the current template to spMats.



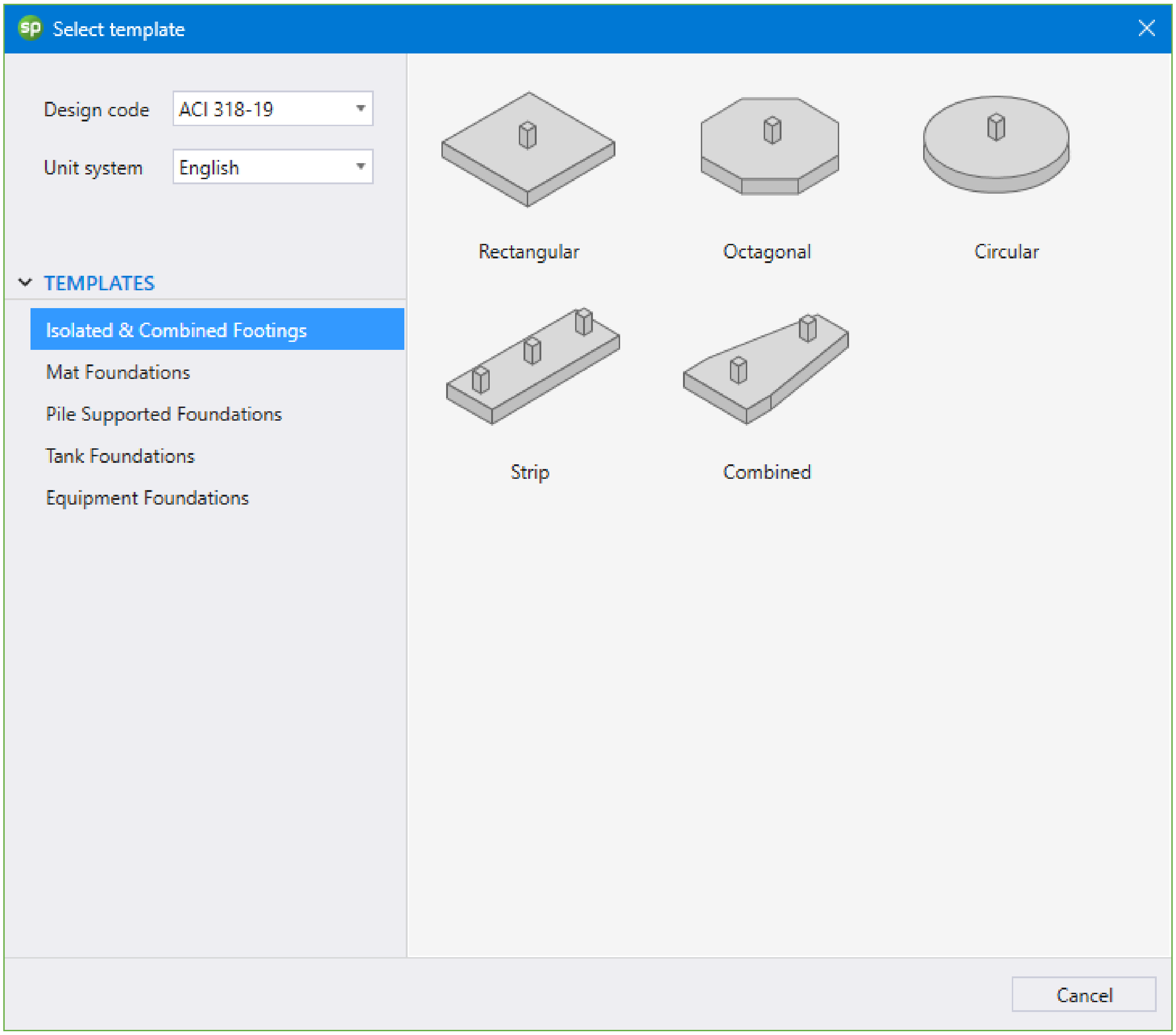
5.3.3. Template Left Panel

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.

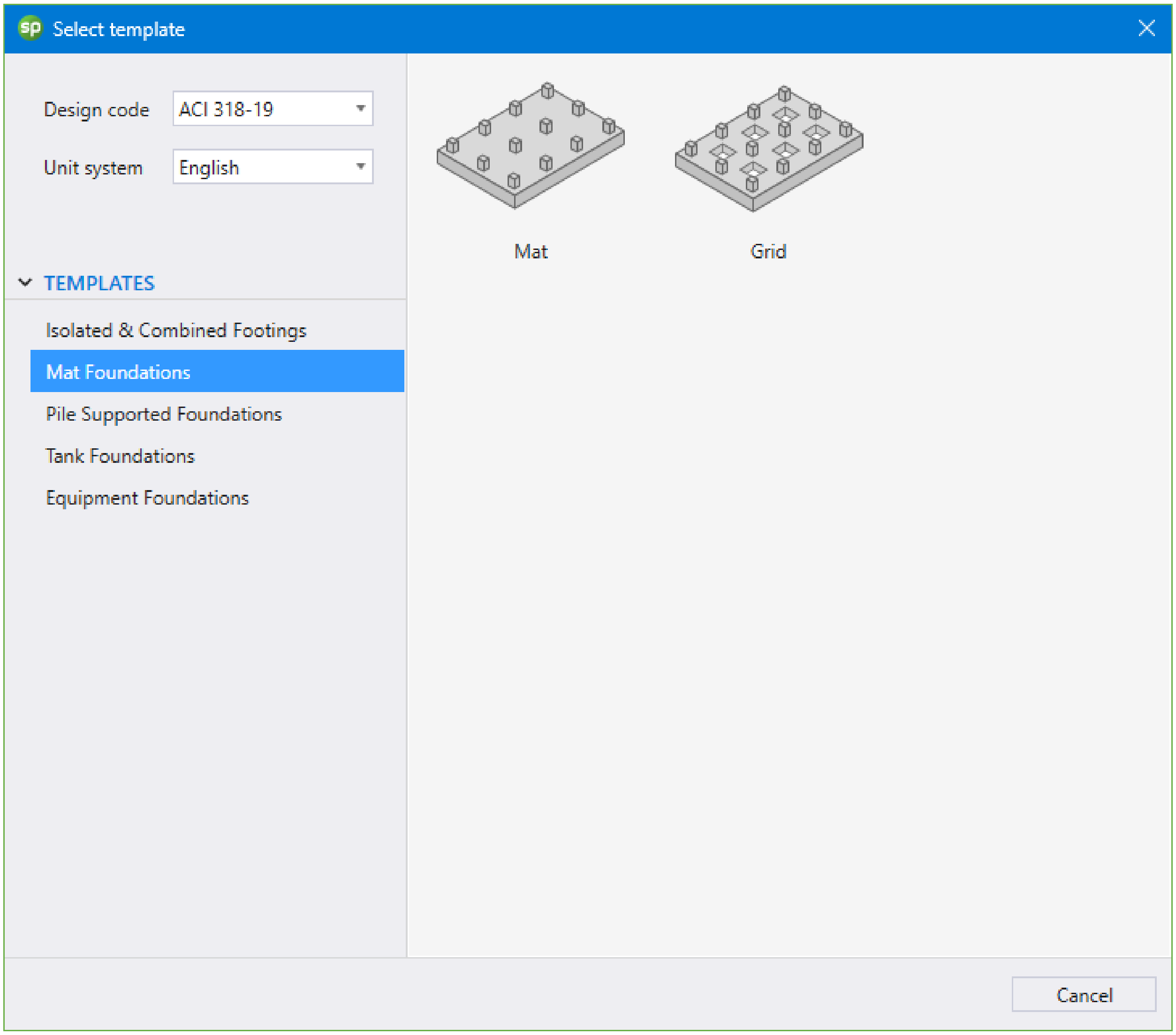


5.3.4. Types of Templates

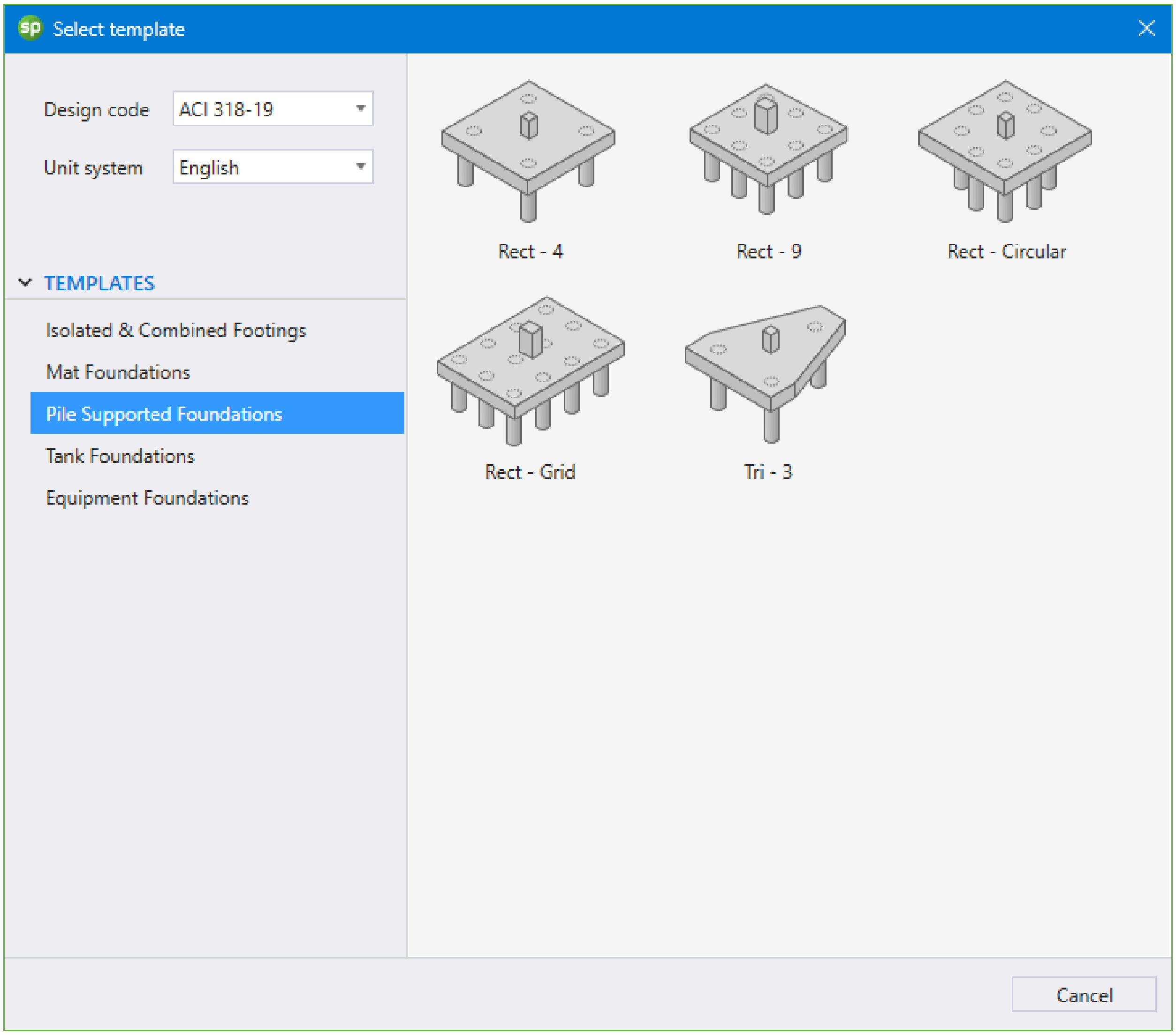
Isolated and Combined Footings



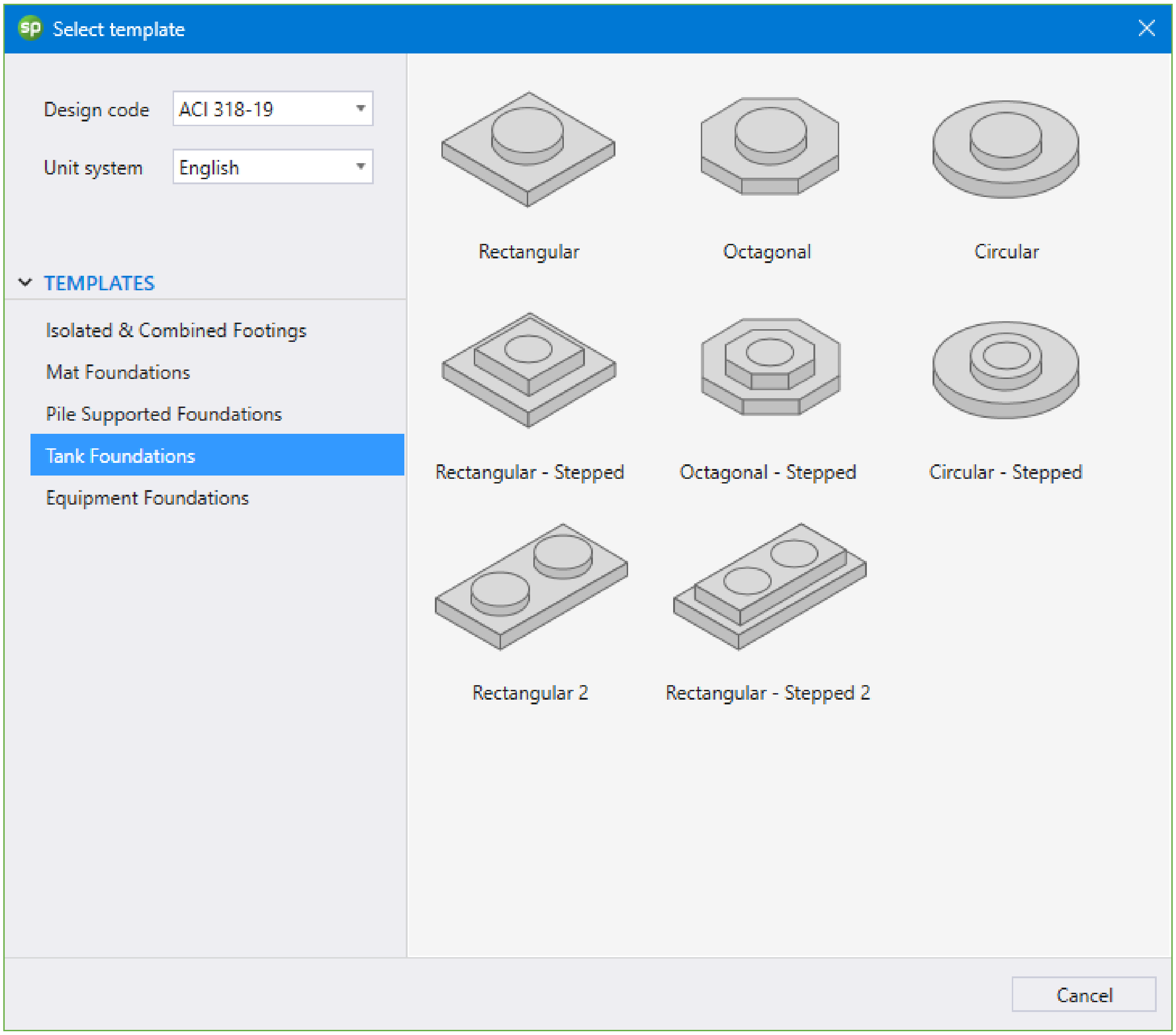
Mat Foundations



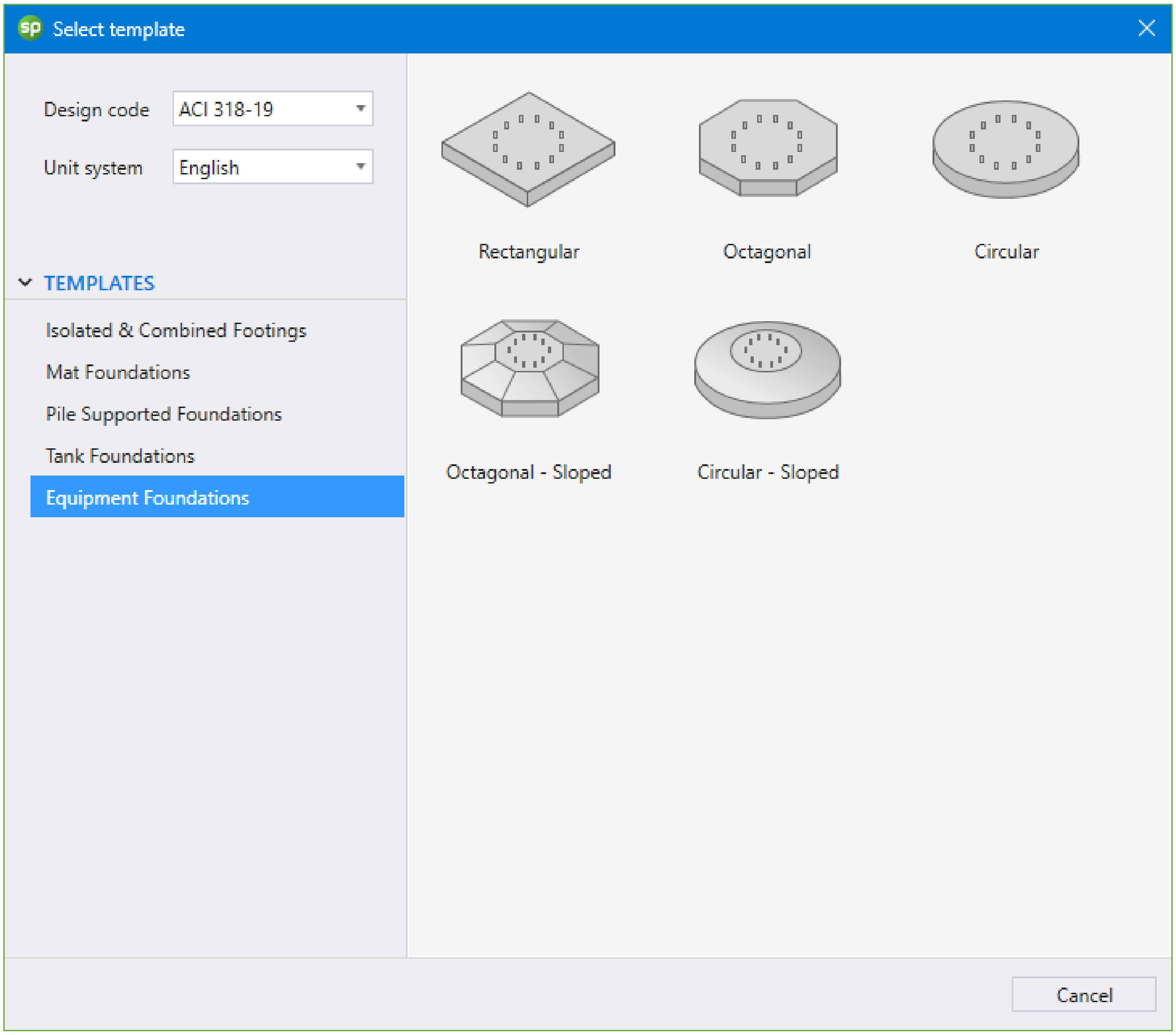
Pile Supported Foundations



Tank Foundations

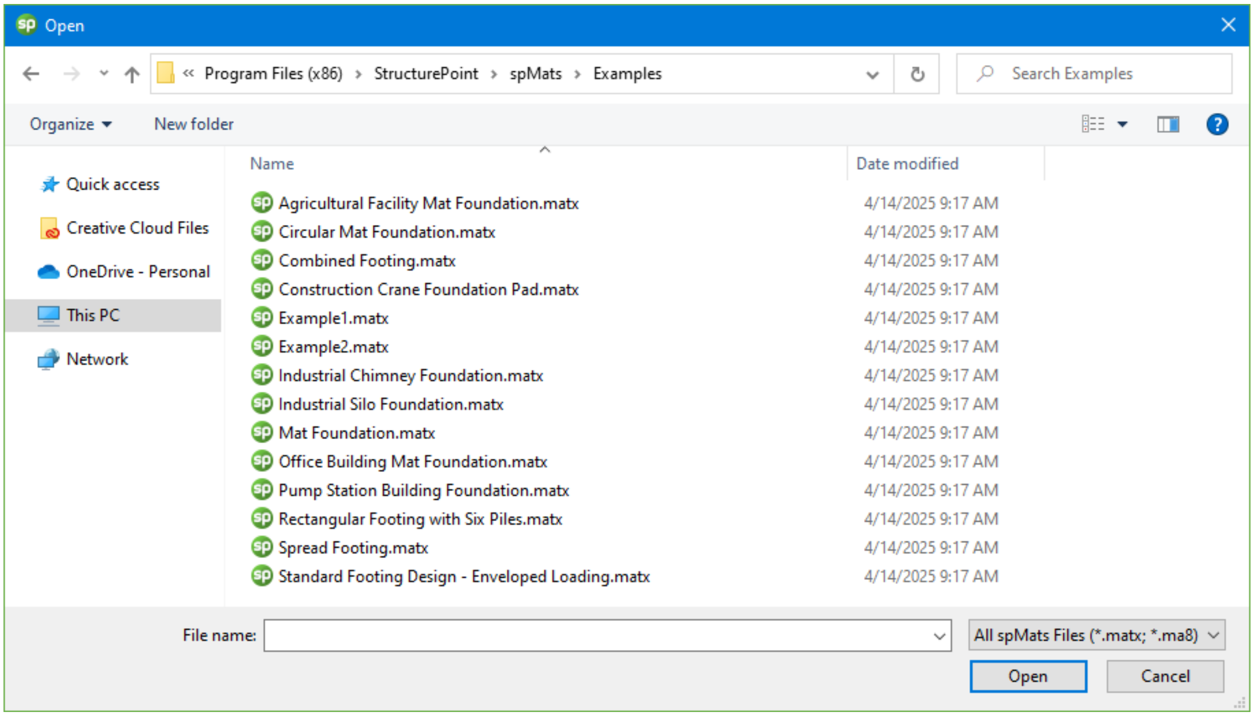


Equipment Foundations



5.4. 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.

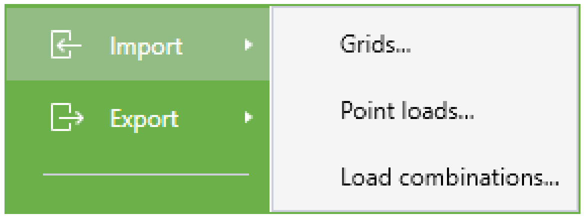


The **Examples** folder contains predefined foundation models that can be further utilized by the user.

5.5. Importing Model Data

Importing Data

spMats provides the options to import GRIDS, POINT LOADS and LOAD CASES & COMBINATIONS data from a text file. The data import options can be obtained from **File | Import**.

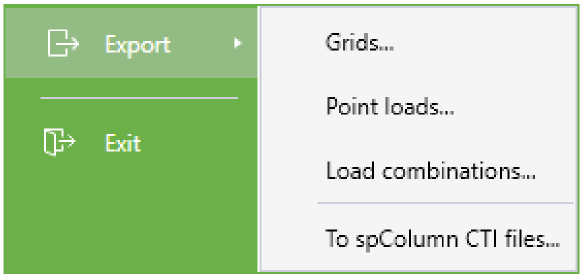


It should be noted that importing a data set will completely replace existing data of that particular set in the program. For details on the import file formats please refer to section A.3. spMats Text Exchange (MTX) File Format in the **Appendix**.

5.6. Exporting Model Data

Exporting Data

spMats provides the options to export GRID, POINT LOADS and LOAD CASES & COMBINATIONS data to a text file. The data export options can be obtained from **File | Export**.

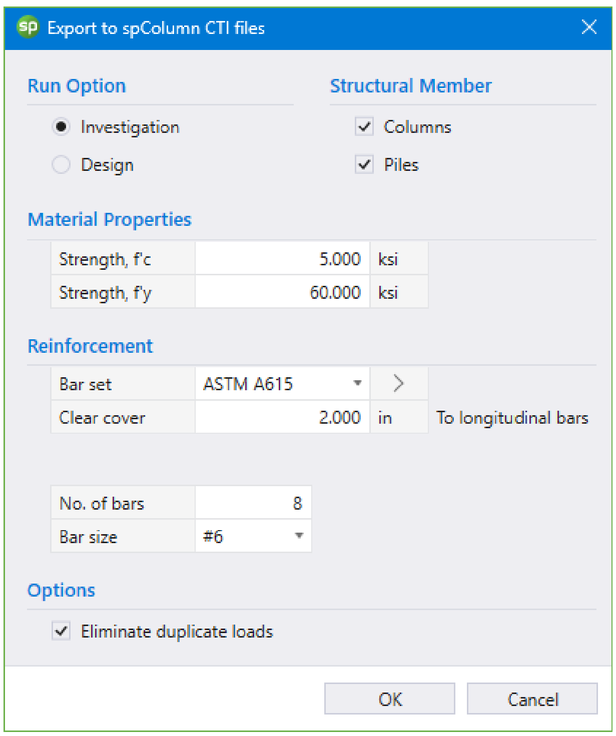


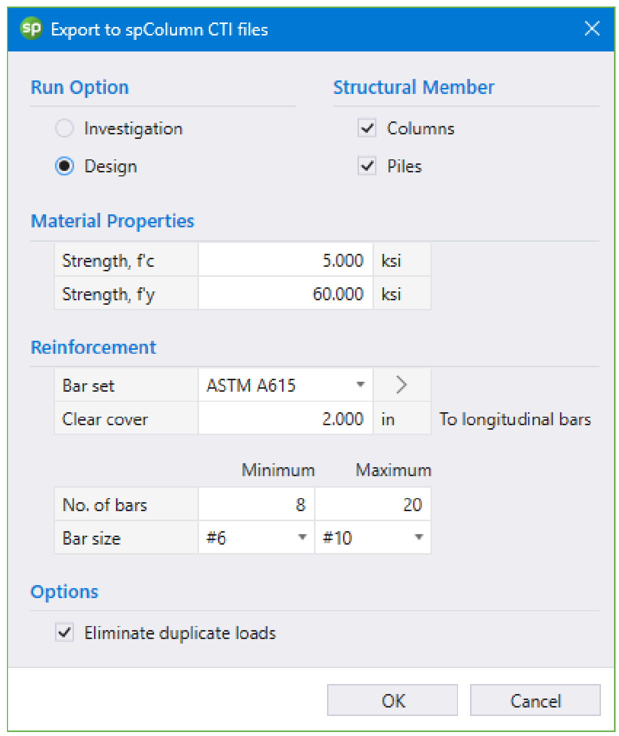
The exported data format will be the same as the import data format provided in section A.3. spMats Text Exchange (MTX) File Format in the **Appendix**.

5.7. 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.

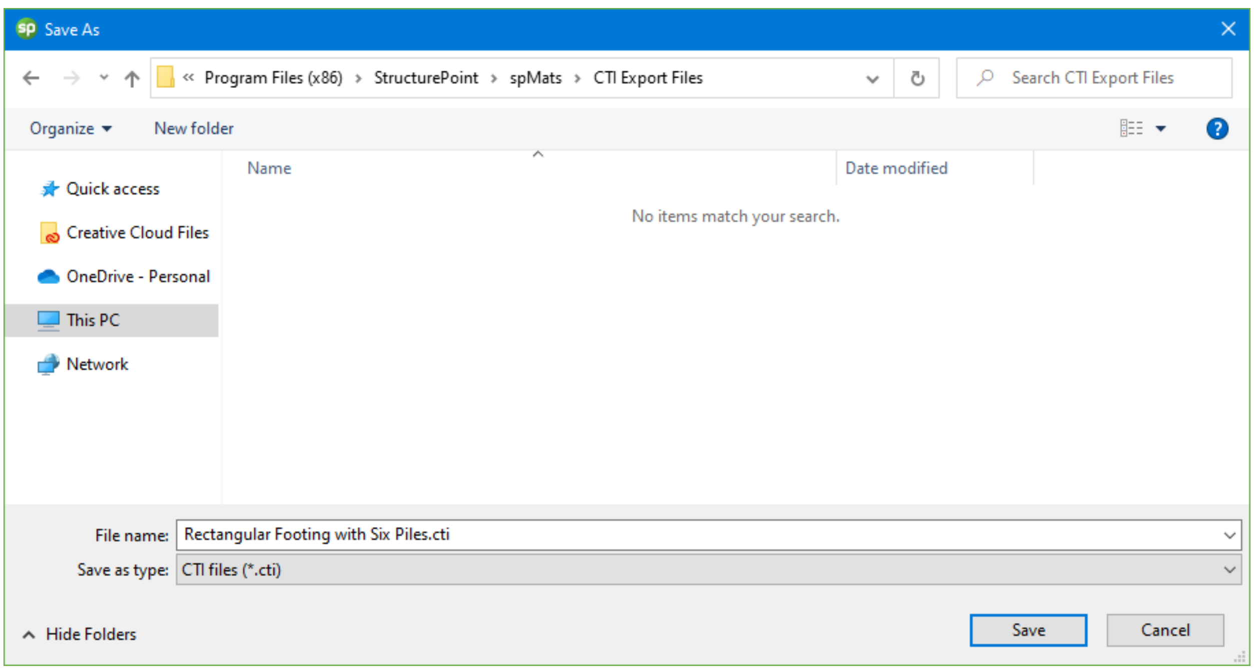
- From the **File** menu, select **Export | To spColumn CTI files**. The first dialog box among the following two appears.





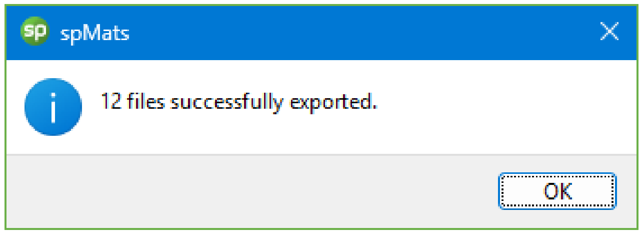
- In the **Run Option** group, select if you want to export the selected sections for INVESTIGATION or DESIGN by spColumn. By default INVESTIGATION is selected. If you select DESIGN the **Reinforcement** group changes to that as shown in the second dialog box image.
- In the **Structural Member** group, 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

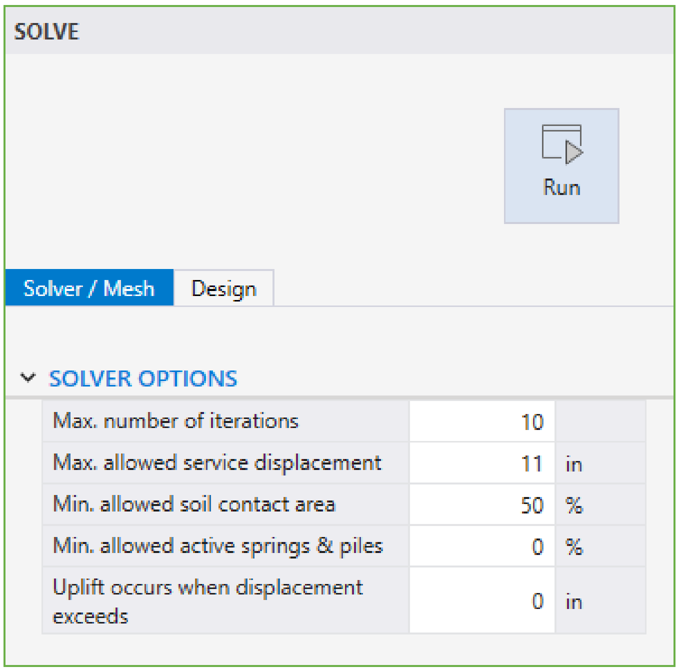
6

MODEL SOLUTION

Once the model creation and development are completed, the analysis can begin using the spMats Finite Element Method of Analysis Solver by clicking on the **Solve** command. Solve Menu containing **Solver Options**, **Mesh Options**, and **Slab Design Options** will appear on the **Left Panel**.

6.1. Solver Options

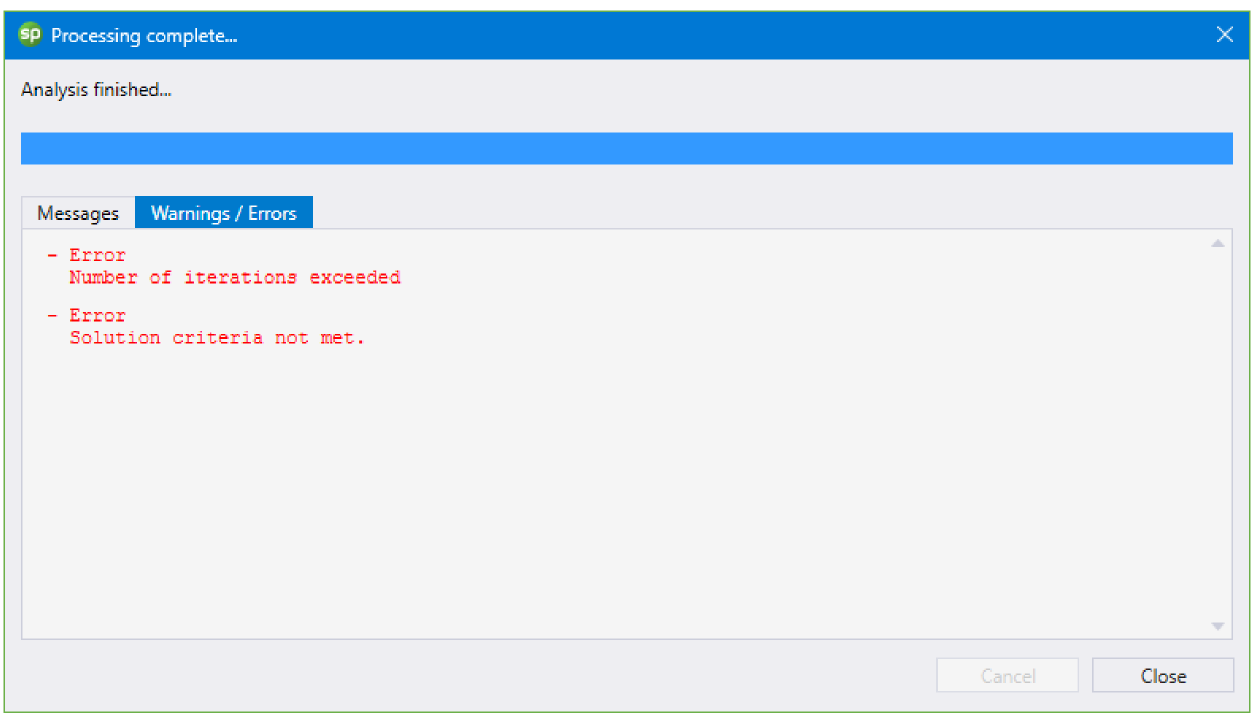
The **Solver Options** allow the user to enter input that is related to the uplift criteria for a specific foundation model and also select the required reinforcement calculation option. These options are important to consider carefully as they can be useful to manage and control the model behavior and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



6.1.1. Maximum 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 the maximum number of iterations is exceeded for any load combination, the solution will not be completed, and the program will display a message in the **Solver Warnings/Errors Dialog** - as shown in the next figure. The message is triggered for the first load combination that did not meet the criteria. In such cases, no output results or contour views will be generated.

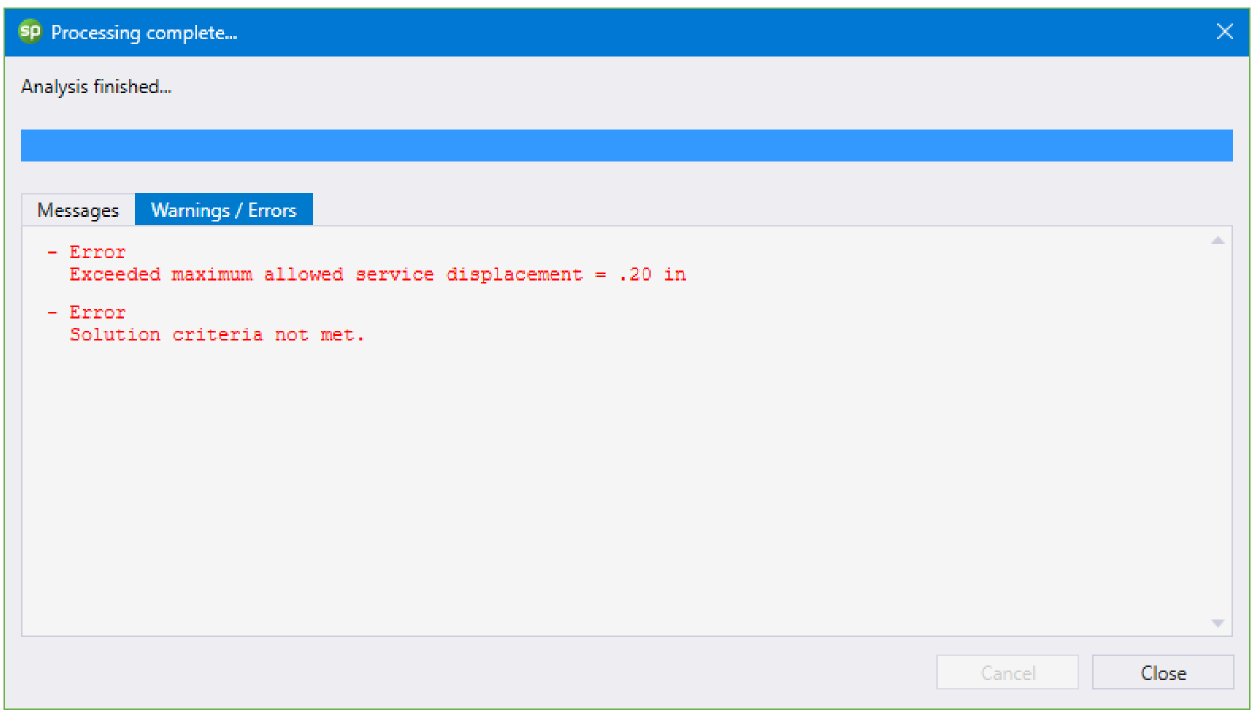


The Program deems a node as an uplift node based on the user input “Uplift occurs when displacement exceeds” within the **Solver Options**. More information on this **Solver Option** can be found below.

6.1.2. Maximum Allowed Service Displacement

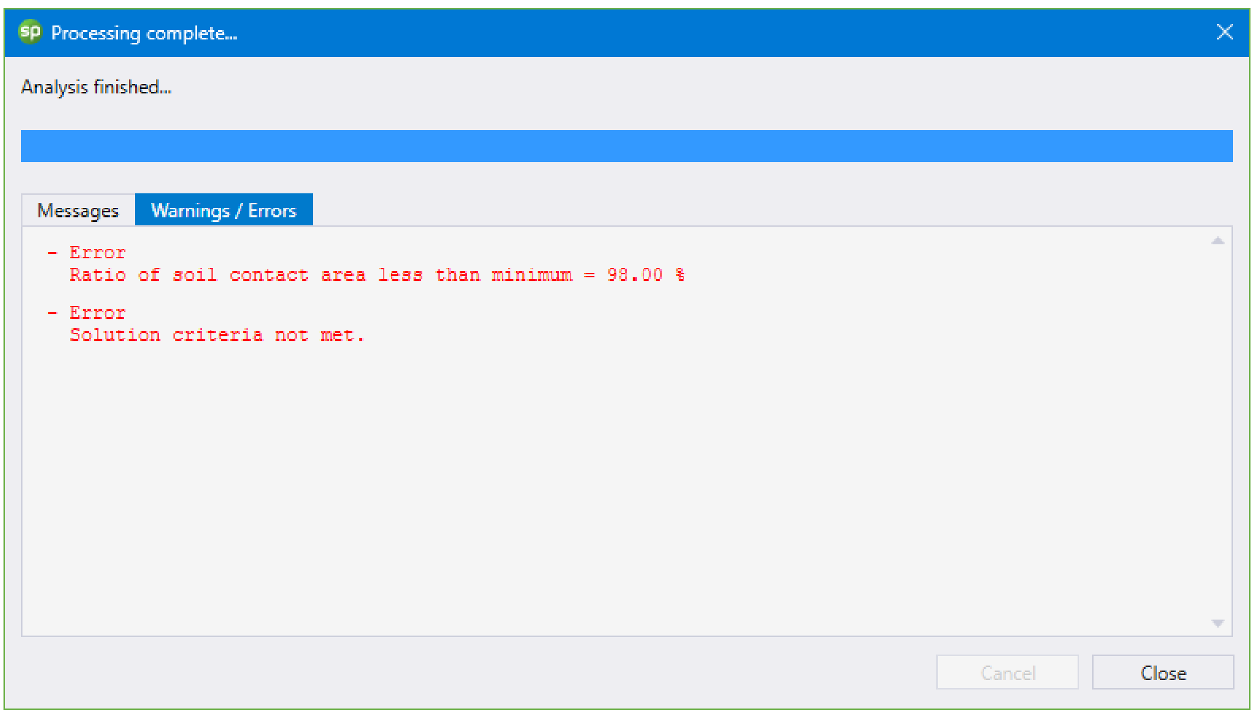
The MAX. ALLOWED SERVICE DISPLACEMENT **Solver Option** is a user input that is set to an initial default value of 11 in. (English unit) or 279 mm (Metric unit) by the Program and can be changed by the user based on the specific model needs. The initial default value set by the Program is to be considered 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 the maximum allowed service displacement is exceeded for any service load combination, the solution will not be completed, and the program will display a message in the **Solver Warnings/Errors Dialog** - as shown in the next figure. The message is triggered for the first service load combination that did not meet the criteria. In such cases, no output results or contour views will be generated.



6.1.3. Minimum Allowed Soil Contact Area

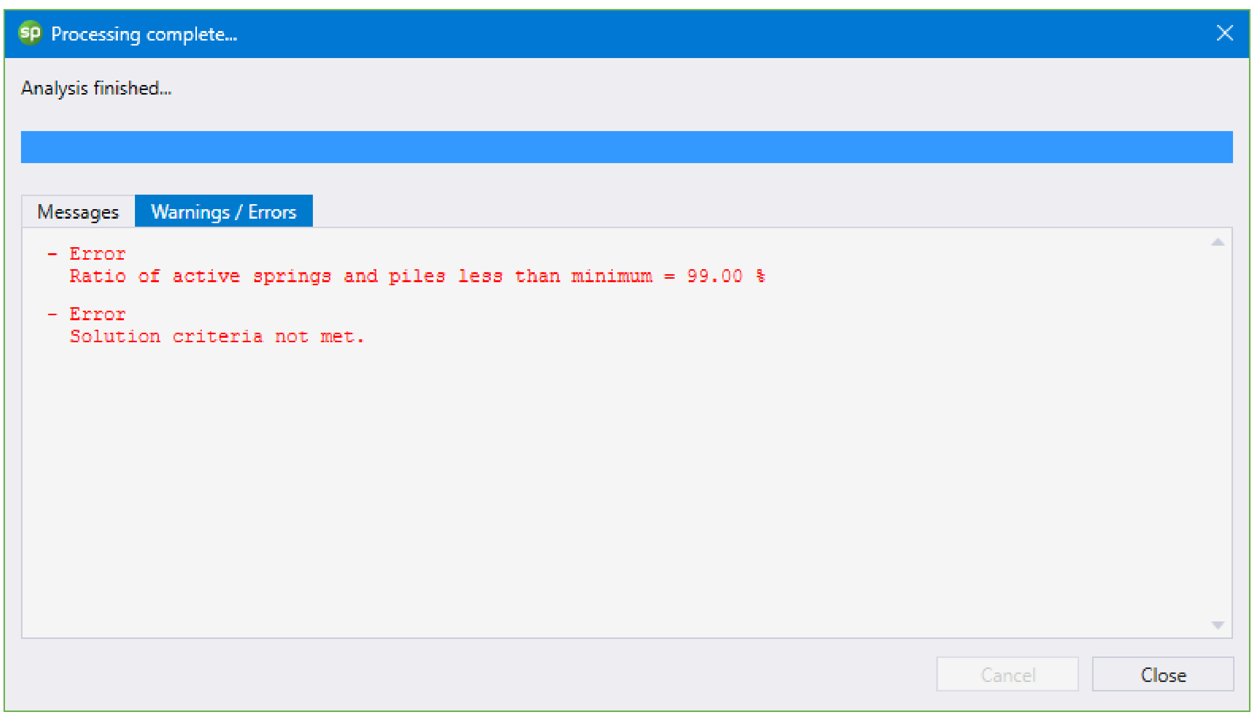
The MIN. ALLOWED SOIL CONTACT AREA **Solver Option** is a user input that is set to an initial default value of 50% by the Program and can be changed by the user based on the specific model needs. If uplift is detected during the run, the tributary area, A_{soil}, of all soil-supported nodes is computed. 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 $[A_{soil} - A_{uplift,soil}] / A_{soil}$. During the run, if this ratio falls below the specified minimum allowed soil contact area ratio for any load combination, the solution will not be completed, and the program will display a message in the **Solver Warnings/Errors Dialog** - as shown in the next figure. The message is triggered for the first load combination that did not meet the criteria. In such cases, no output results or contour views will be generated.



6.1.4. Minimum 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 $N_{S/P}$, is computed. The program also computes the total number of springs/piles $N_{S/P,\ total}$. The ratio of active springs/piles is defined as $[N_{S/P,\ total} - N_{S/P,\ uplift}] / N_{S/P,\ total}$. 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 run, if this ratio falls below the specified minimum ratio of active springs/piles for any load combination, the solution will not be completed, and the program will display a message in the **Solver Warnings/Errors Dialog** - as shown in the next figure. The message is triggered for the first load combination that did not meet the criteria. In such cases, no output results or contour views will be generated.



6.1.5. Uplift occurs when nodal displacement exceeds

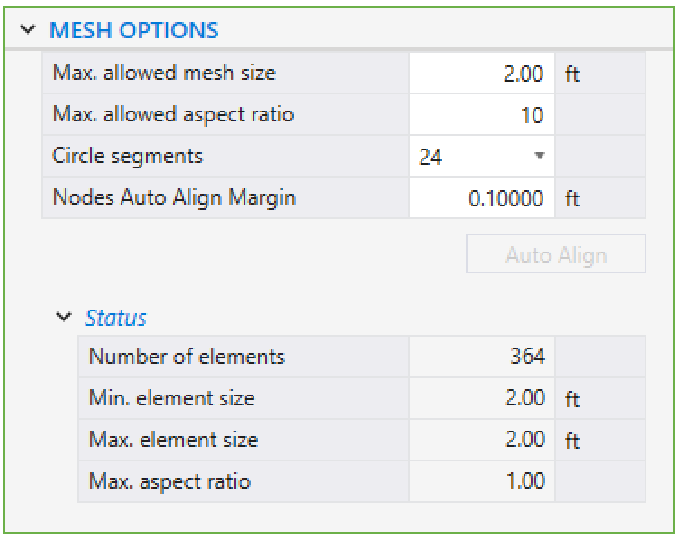
The UPLIFT OCCURS WHEN DISPLACEMENT EXCEEDS **Solver Option** is a user input that is set to an initial default value of 0 by the Program and can be changed by the user based on the specific model needs. The initial default value set by the Program deems a node as an uplift node for a specific load combination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

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 v10.50 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:



6.2.1. Maximum 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.

6.2.2. Maximum 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.

6.2.3. 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 8, 12, 24 or 48 based on specific model needs. This option is utilized for circular slabs.

6.2.4. Auto Alignment

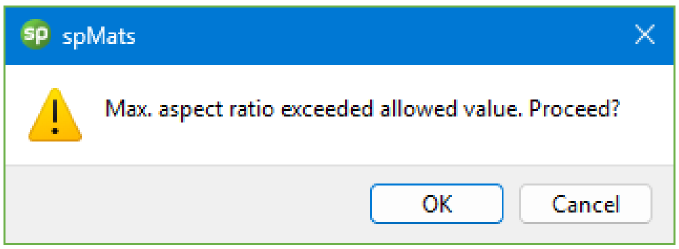
The AUTO ALIGNMENT feature allows users to improve mesh quality by eliminating slender elements that exceed the defined maximum aspect ratio.

After the mesh is generated, some elements may appear distorted or elongated due to minor misalignments between adjacent nodes - especially in regions where nodes are intended to lie on a straight line. Such elements, highlighted in red, typically indicate an aspect ratio higher than the allowable threshold (MAX. ASPECT RATIO).

To address this, the user can specify a tolerance value for the NODES AUTO ALIGN MARGIN and activate the AUTO ALIGN command by clicking the AUTO ALIGN button. The program then identifies and aligns all nodes located within this margin to a common vertical or horizontal axis. This correction results in a cleaner mesh configuration and helps eliminate high aspect ratio elements, which can otherwise compromise the accuracy and stability of the analysis.

6.2.5. Status

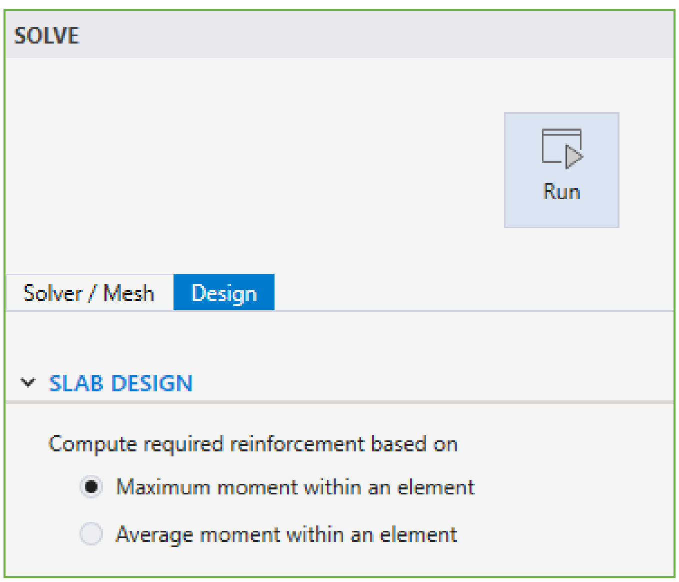
This section lists the number of elements, minimum and maximum element sizes in the model along with the maximum aspect ratio. If the maximum aspect ratio of an element in the model is greater than the defined maximum allowed aspect ratio then the maximum aspect ratio value is highlighted along with 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. Slab Design Options

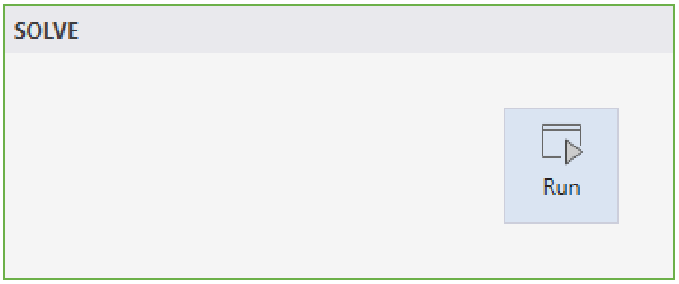
For each element, each of the design moments (M_{ux} and M_{uy}, top and bottom) is computed for a governing ultimate load combination. The governing load combination is the one that produces the maximum design moment M_{ux} and M_{uy} respectively in the X- and Y directions, and separately for top and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

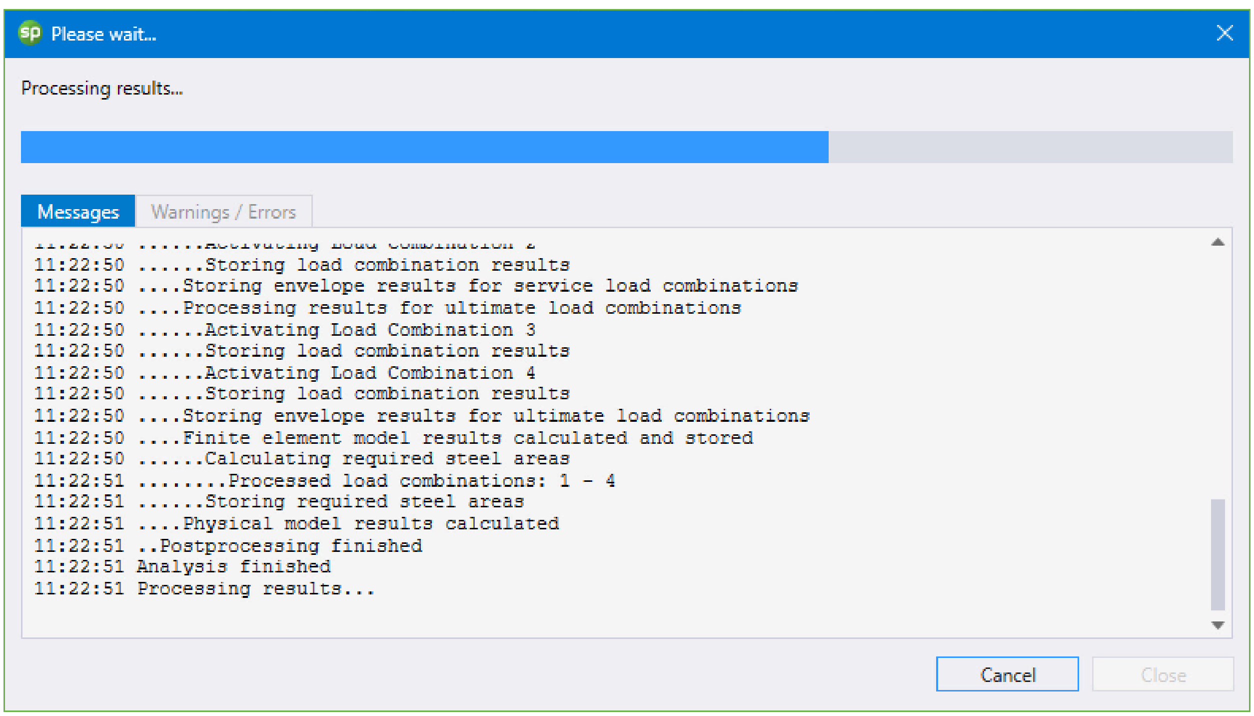


6.4. Running the Model

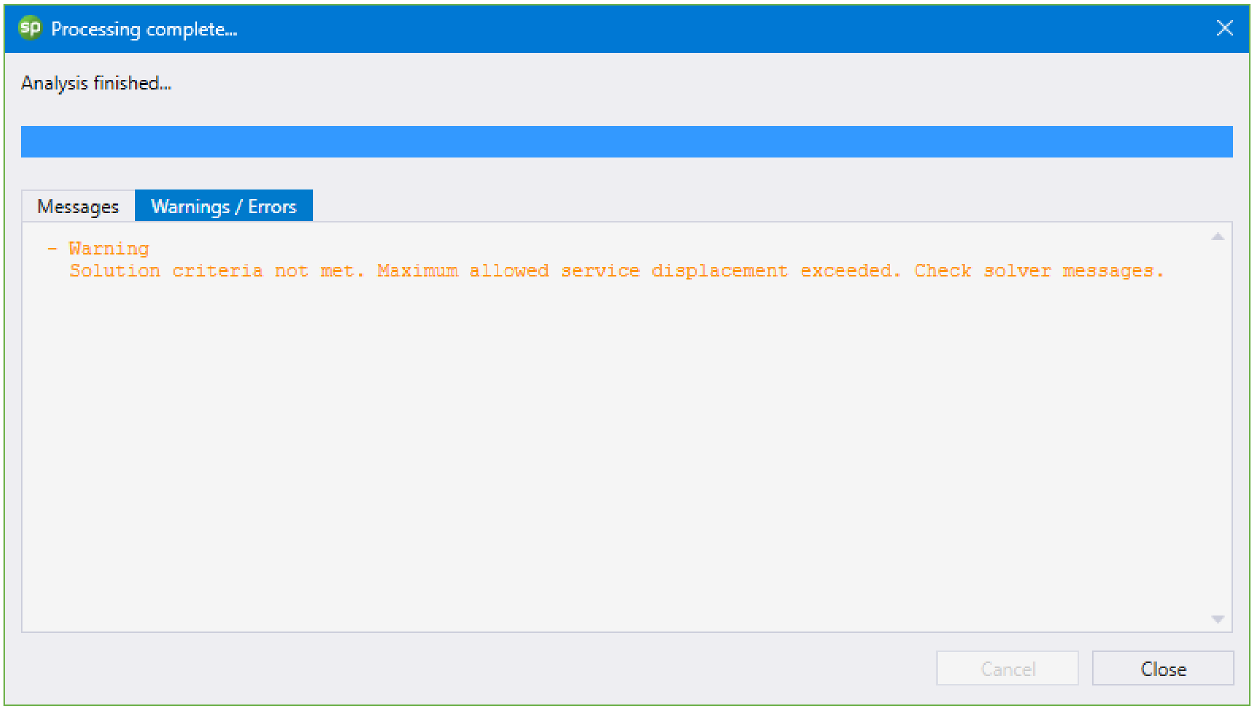
After inputting the model, the solver portion of the program can be executed using the **Run** button in the SOLVE panel.



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.



When the solution is successfully completed, a contour map showing downward displacement envelope is displayed. If the solution fails, the program will display the related warnings and/or errors in the **Solver Warnings/Errors Dialog**. Detailed information on the solution can be found in the **Solver Messages** table under the **Tables** window.



Processing complete...
Analysis finished...
Messages
Warnings / Errors
- Warning
Solution criteria not met. Maximum allowed service displacement exceeded. Check solver messages.
Cancel
Close

6.5. Running from Command Prompt

spMats solver can be run in batch mode from the command line prompt. Command line runs can be done by invoking **spMats.CLI.exe** (spMats Command Line Interface module) after navigating to the directory in which spMats is installed. Input data file, output data file, and run options can be passed to the program via parameters. For instance, when in the command prompt, navigating to the directory containing spMats.CLI.exe and typing

```
spMats.CLI /i:Examples\Example1.matx
```

will run spMats and solve the model defined in input file Example1.matx. Each step of the solve process and its timestamp along with color coded Errors and/or Warnings, if any, will also be listed.

```
C:\WINDOWS\system32\cmd.

C:\Program Files (x86)\StructurePoint\spMats v10.50>spMats.CLI.exe /i:"Examples\Example1.matx"
spMats v10.50 (TM)
Copyright c 1988-2025, STRUCTUREPOINT, LLC.
04/17/2025 17:18:52 - Processing arguments...
04/17/2025 17:18:52 - Reading input file 'C:\Program Files (x86)\StructurePoint\spMats v10.50\Examples\Example1
.matx'...
04/17/2025 17:18:52 - Preparing solver model...
04/17/2025 17:18:52 - Preparing analytical model...
04/17/2025 17:18:52 - Preparing solver input file...
04/17/2025 17:18:52 - Executing solver...
04/17/2025 17:18:53 - Solver execution finished. Processing results...
04/17/2025 17:18:53 - Solver results processed successfully.
04/17/2025 17:18:53 - Calculating punching shear...
04/17/2025 17:18:55 - Punch shear calculation completed.
04/17/2025 17:18:55 - Preparing data for reporting...
04/17/2025 17:18:55 - Generating reports...
04/17/2025 17:18:55 - Generating TXT report...
04/17/2025 17:18:56 - Done

C:\Program Files (x86)\StructurePoint\spMats v10.50
Press any key to continue . . .
```

Multiple models can be analyzed by running spMats with multiple input files using batch (BAT) files (see Examples.bat in the spMats program folder). Combined with spMats Text Input files (MTX), this feature can be used for automating spMats runs for large number of models.

Help on how to use command line parameters can be obtained by typing `spMats.CLI -h` or `spMats.CLI --help` at the command prompt.

```
C:\WINDOWS\system32\cmd.

C:\Program Files (x86)\StructurePoint\spMats v10.50>spMats.CLI.exe -h
spMats v10.50 (TM)
Copyright c 1988-2025, STRUCTUREPOINT, LLC.

USAGE:
Basic usage:
    spMats.CLI -i "C:\project.matx" -o "C:\project.txt"

Generate TXT, PDF, Word Excel and CSV reports in project directory:
    spMats.CLI -i "C:\project.matx" --rpdf --rdoc --rxls --rcsv

Generate report in custom location
    spMats.CLI -i "C:\project.matx" --rpdf "C:\myReport.pdf"

PARAMETERS:

    -i, --input                 Required. The input path.
    -o, --output                Output TXT file path.
    -e, --useExistingResults    (Default: false) Use existing solver results if present
    -h, --help                  (Default: false) Prints help info.
    --mtx                       Export input data to spMats Text Input file format
    --rpdf                      Output PDF report path.
    --rdoc                      Output Word report path.
    --rxls                      Output Excel report path.
    --rcsv                      Output CSV report path.

C:\Program Files (x86)\StructurePoint\spMats v10.50
Press any key to continue . . .
```

CHAPTER

7

MODEL OUTPUT

The results of the Finite Element Method of Analysis, along with subsequent design and code calculations, are presented in the spMats model output under two key categories.

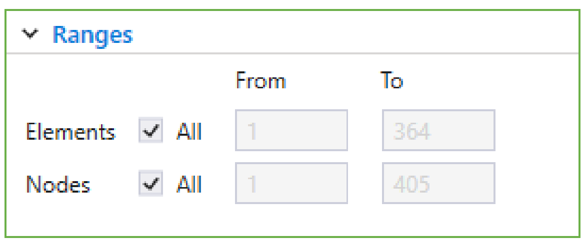
1. **Tabular Output:** Text results organized in tables including all relevant exact numerical results formatted in columnar tables.

2. **Graphical Output:** Illustrating model behavior visually as an important and effective method to diagnose and verify expected results and critical parameters. Graphical output is presented as contours or other two-dimensional graphics.

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 using the Ranges section of the results explorer panel. Selecting "All" nodes or elements provide the complete output for each category in the results table.



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:

7.1.1. Project

This section contains the following input data blocks:

7.1.1.1. General Information

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

7.1.1.2. Solver Options

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

7.1.2. Definitions

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

7.1.2.1. 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.

7.1.2.2. 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.

7.1.2.3. 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 SLAB DESIGN CRITERIA input data.

7.1.2.4. 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.

7.1.2.5. Load Cases / 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.

7.1.3. Assignments

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

7.1.3.1. Nodes

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

7.1.3.2. Slabs

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

7.1.3.3. Columns

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

7.1.3.4. Piles

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

7.1.3.5. Point Loads

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

7.1.3.6. Area Loads

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

7.1.4. Analytical Model

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

7.1.4.1. Mesh

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

7.1.4.2. Element Geometry

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

7.1.4.3. Element Properties

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

7.1.4.4. Loaded Elements

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

7.1.5. Results

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

7.1.5.1. Solver Messages

This block displays the progress and status of the solution. It also displays warnings or error messages generated during model execution.

7.1.5.2. Envelope

This section contains the **Envelope** results information on **Nodal Displacements**, **Soil Displacement & Pressure**, **Service Reactions**, **Ultimate Reactions**, **Governing Reinforcement**, and **Analysis Moment and Reinforcement**.

Nodal Displacements

This block contains the upward and downward service displacements, $\mathbf{D_Z}$, envelopes along with the governing combinations labels. Positive displacements are upward in the positive Z-direction.

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.

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.

Governing Reinforcement

This subsection contains the **Governing Reinforcement** results information on **Element Top - X**, **Element Top - Y**, **Element Bottom - X**, and **Element Bottom - Y**.

Element Top - X

This block reports the **Governing A_s** for the top reinforcement in the X-direction for each element. It reports the **Governing A_s** which is selected from among the four types of reinforcement: **Design Moment A_s**, user-defined **"Base" A_s**, **Code Min. A_s**, and **Code Max. A_s**. The table also provides the corresponding **Design Moment A_s** values, **"Base" A_s** values, and the applicable **Code Min. A_s** and **Code Max. A_s** limits.

Element Top - Y

This block reports the **Governing A_s** for the top reinforcement in the Y-direction for each element. It reports the **Governing A_s** which is selected from among the four types of reinforcement: **Design Moment A_s**, user-defined **"Base" A_s**, **Code Min. A_s**, and **Code Max. A_s**. The table also provides the corresponding **Design Moment A_s** values, **"Base" A_s** values, and the applicable **Code Min. A_s** and **Code Max. A_s** limits.

Element Bottom - X

This block reports the **Governing A_s** for the bottom reinforcement in the X-direction for each element. It reports the **Governing A_s** which is selected from among the four types of reinforcement: **Design Moment A_s**, user-defined **"Base" A_s**, **Code Min. A_s**, and **Code Max. A_s**. The table also provides the corresponding **Design Moment A_s** values, **"Base" A_s** values, and the applicable **Code Min. A_s** and **Code Max. A_s** limits.

Element Bottom - Y

This block reports the **Governing A_s** for the bottom reinforcement in the Y-direction for each element. It reports the **Governing A_s** which is selected from among the four types of reinforcement: **Design Moment A_s**, user-defined **"Base" A_s**, **Code Min. A_s**, and **Code Max. A_s**. The table also provides the corresponding **Design Moment A_s** values, **"Base" A_s** values, and the applicable **Code Min. A_s** and **Code Max. A_s** limits.

Analysis Moment and Reinforcement

This subsection contains the **Analysis Moment and Reinforcement** results information on **Element Top Moment**, **Element Bottom Moment**, **Element Top Design Moment and Reinforcement**, and **Element Bottom Design Moment and Reinforcement**.

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, $\mathbf{M_{ux}}$ and $\mathbf{M_{uy}}$, 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, $\mathbf{M_{ux}}$ and $\mathbf{M_{uy}}$, are extreme together with the values of extreme design moments and the corresponding steel area requirement.

7.1.5.3. 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, $\mathbf{F_z}$, moment about X-axis, $\mathbf{M_x}$, and moment about Y-axis, $\mathbf{M_y}$, 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, $\mathbf{D_z}$, and the two rotations about the X and Y-axis, $\mathbf{R_x}$ and $\mathbf{R_y}$, 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.

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.

7.1.5.4. 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, $\mathbf{F_z}$, moment about X-axis, $\mathbf{M_x}$, and moment about Y-axis, $\mathbf{M_y}$, 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, $\mathbf{D_z}$, and the two rotations about the X and Y-axis, $\mathbf{R_x}$ and $\mathbf{R_y}$, 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.

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

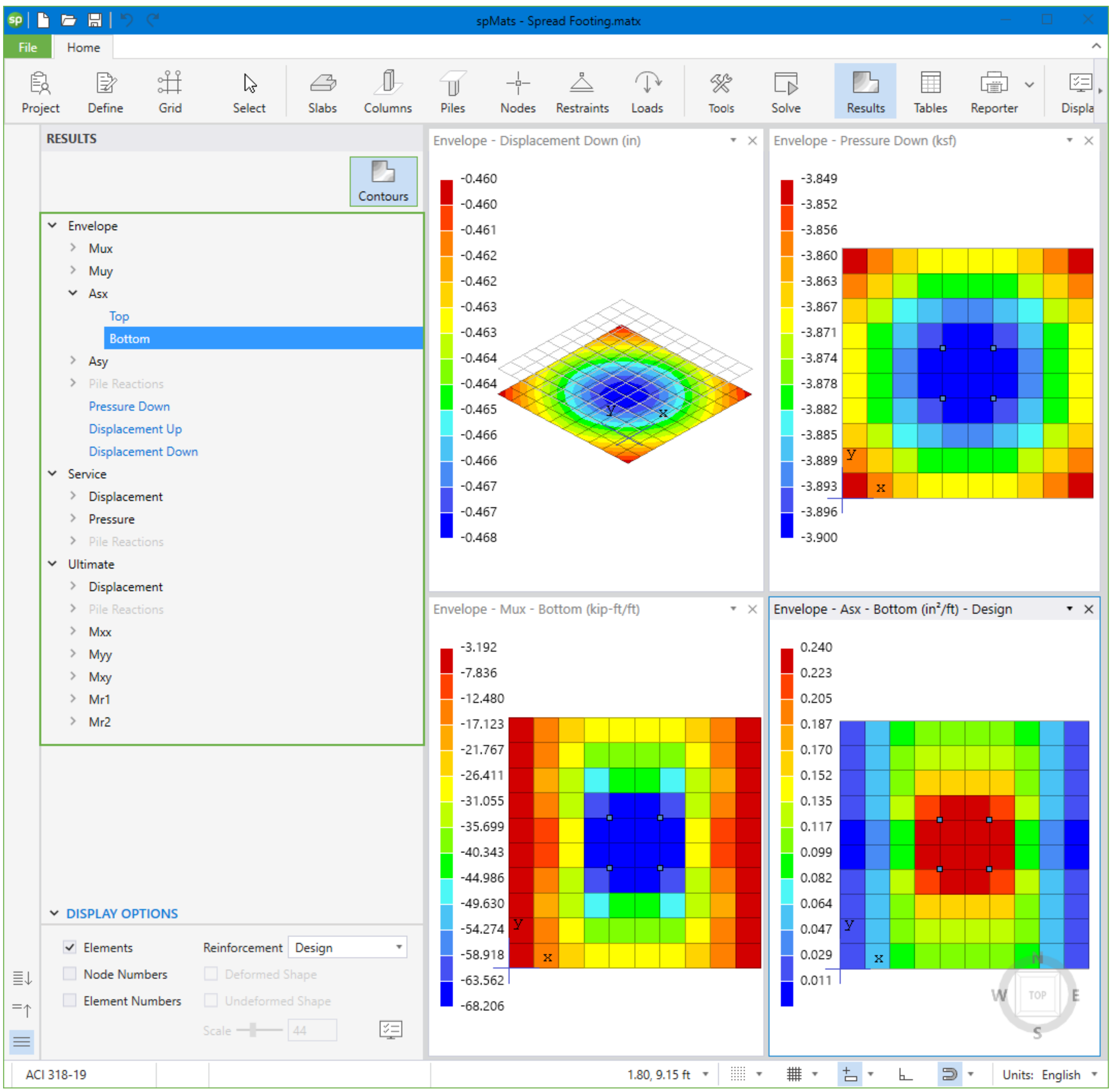
This block is output for individual **Ultimate Load Combinations**. At each of the four nodes of the element (i, j, k and l), listed are the bending moments ($\mathbf{M_{xx}}$ and $\mathbf{M_{yy}}$), the twisting moment ($\mathbf{M_{xy}}$) the equivalent principal moments ($\mathbf{M_{r1}}$ and $\mathbf{M_{r2}}$), along with the principal angle, and equivalent design bending moments ($\mathbf{M_{ux}}$ and $\mathbf{M_{uy}}$) at the top and bottom. Note that $\mathbf{M_{xx}}$ and $\mathbf{M_{yy}}$ 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 Figure 2.6.

7.2. Graphical Output

The Graphical output is organized into **Contour** views that may be viewed, printed, exported or added to Report.

7.2.1. Contours

Contour views are used 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**.



7.2.1.1. 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**, **Pile Reactions**, **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 aligned along the X-direction. The default contour shown is for the **Governing Value** of A_{sx}. User can select and view **A_{sx} Top** and **A_{sx} Bottom** contours for **Governing Type**, **Design**, **Base**, **Code Min.** and **Code Max.** reinforcements.

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 aligned along the Y-direction. The default contour shown is for the **Governing Value** of A_{sy}. User can select and view **A_{sy} Top** and **A_{sy} Bottom** contours for **Governing Type**, **Design**, **Base**, **Code Min.** and **Code Max.** reinforcements.

Pile Reactions

This graphical view displays the envelope results for reactions of **Piles in Compression** and **Piles in Tension** under **Service** and **Ultimate** load combinations. Pile reactions are represented graphically with directional arrows - upward arrows for compression and downward arrows for tension. When allowable pile reactions are defined by the user, the program will automatically evaluate each pile's reaction against these limits. Pile reactions that do not exceed the allowable values are displayed in blue, while those that exceed are highlighted in red to alert the user. A warning icon will also appear in the Contours tree to indicate that one or more pile reactions exceed the allowable limit. Note that allowable pile reactions, when specified, are only applicable to and compared with the **Service** pile reactions.

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.

7.2.1.2. Service

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

Displacement

This graphical view displays the displacement, $\mathbf{D_z}$, contours for individual service load combinations.

Pressure

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

Pile Reactions

This graphical view displays pile reactions for individual service load combinations. Pile reactions are represented graphically with directional arrows - upward arrows for compression and downward arrows for tension. When allowable pile reactions are defined by the user, the program will automatically evaluate each pile's reaction against these limits. Pile reactions that do not exceed the allowable values are displayed in blue, while those that exceed are highlighted in red to alert the user.

7.2.1.3. Ultimate

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

Displacement

This graphical view displays the displacement, $\mathbf{D_z}$, contours for individual ultimate load combinations.

Pile Reactions

This graphical view displays pile reactions for individual ultimate load combinations. Pile reactions are represented graphically with directional arrows - upward arrows for compression and downward arrows for tension.

$\mathbf{M_{xx}}$

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

$\mathbf{M_{yy}}$

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

$\mathbf{M_{xy}}$

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

$\mathbf{M_{r1}}$

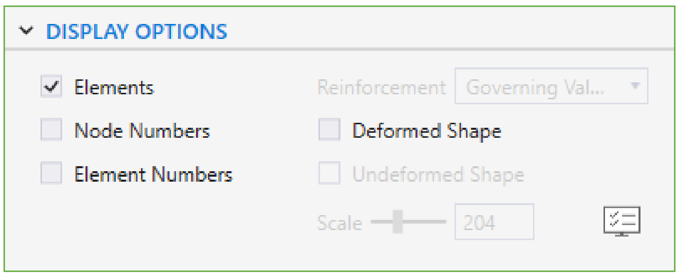
This graphical view displays the equivalent principal moment, $\mathbf{M_{r1}}$ contours for individual ultimate load combinations.

M_{r2}

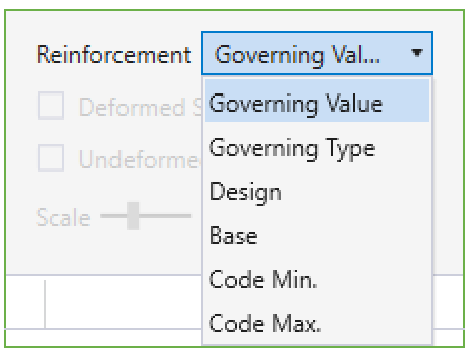
This graphical view displays the equivalent principal moment, **M_{r2}** contours for individual ultimate load combinations.

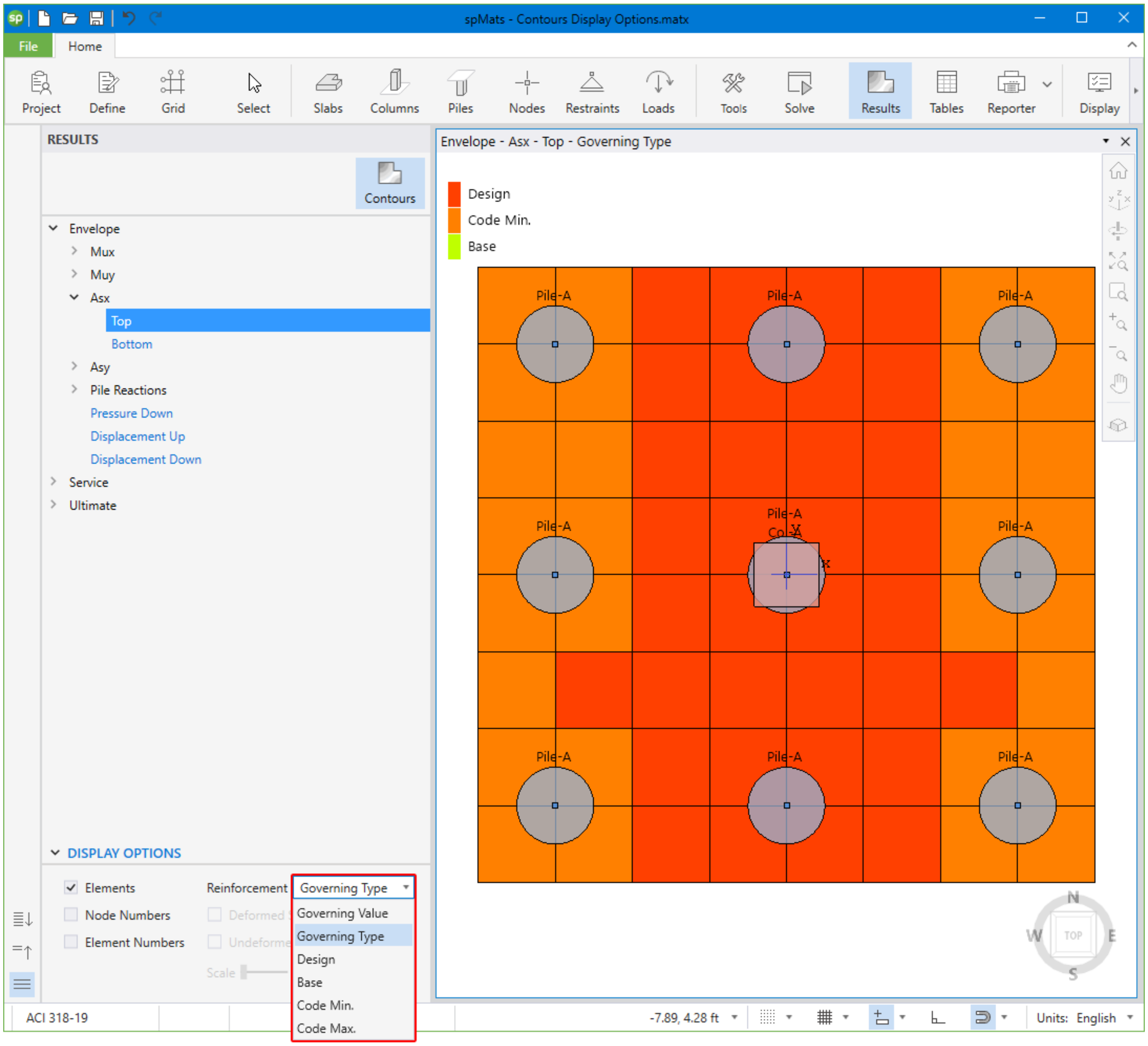
7.2.1.4. Contours Display Options

The contours **Display Options** allow the user to control the view of slab elements, slab element numbers, node numbers, reinforcement options, deformed shape, undeformed shape, and deformed shape scale. It also contains a shortcut button for **Display** command.



The REINFORCEMENT dropdown menu in the **Display Options** panel allows users to visually examine different reinforcement contour types. Users can select from GOVERNING VALUE, GOVERNING TYPE, DESIGN, BASE, CODE MIN., and CODE MAX. options.



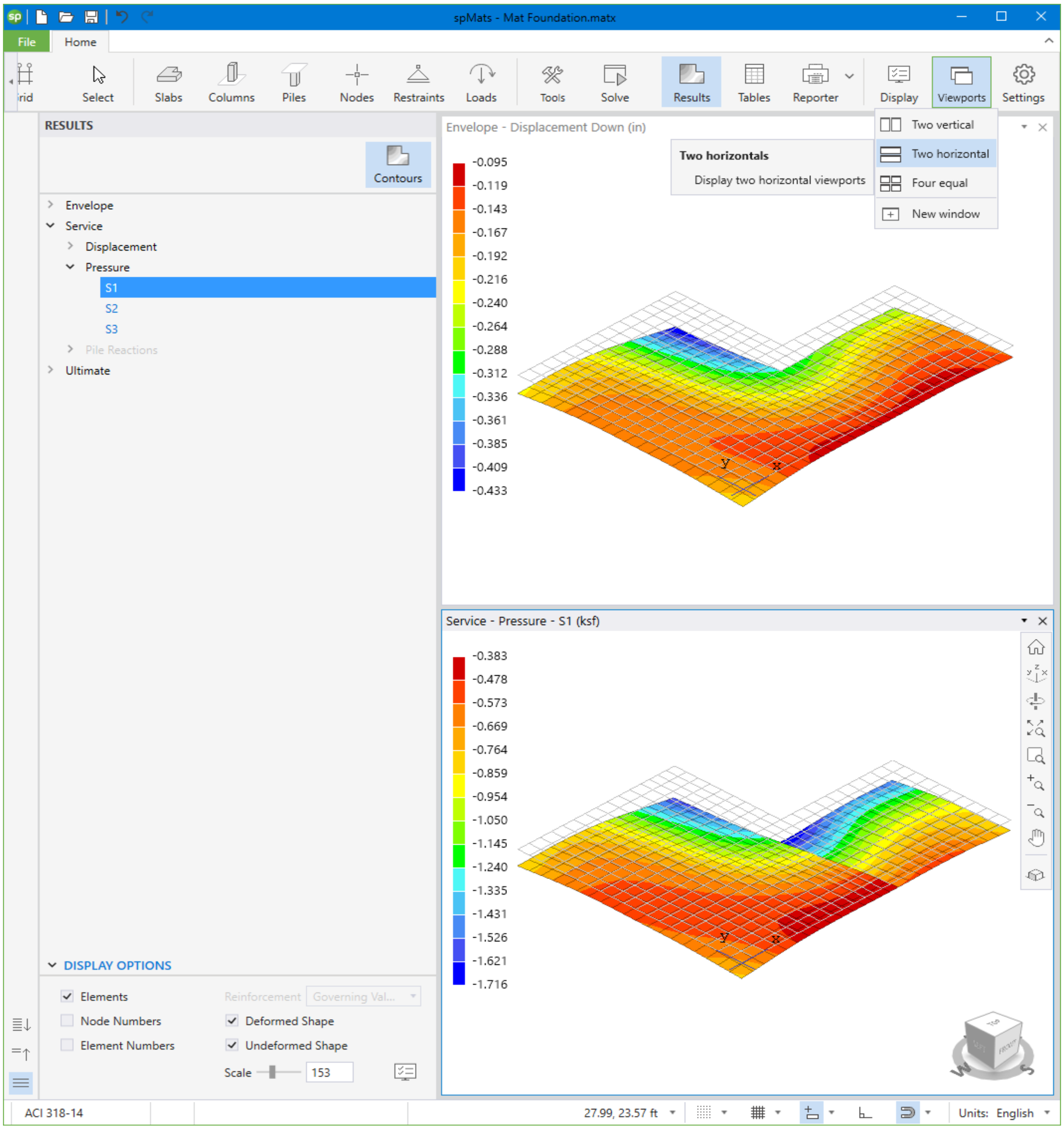


7.2.2. Viewing Aids

Viewing aids are those features in the program that facilitate viewing the graphical output results produced by the program.

7.2.2.1. Multiple Viewports

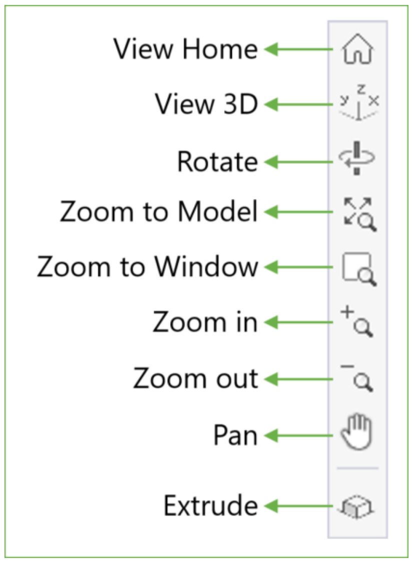
Multiple viewports can be used to view different contours, and model views at the same time. The **Viewports** Command in the **Ribbon** can be used to select from a set of pre-defined viewport configurations or create a new viewport window. A maximum of 6 viewports can be used at one time.



7.2.2.2. View Controls

When a viewport is active it has a set of **View Controls** located in the top right corner. These commands can be used to aid in viewing the model and contours.

Commands in the **View Controls** can be used to VIEW HOME, VIEW 3D, ROTATE, ZOOM TO MODEL, ZOOM TO WINDOW, ZOOM IN, ZOOM OUT, PAN and EXTRUDE.



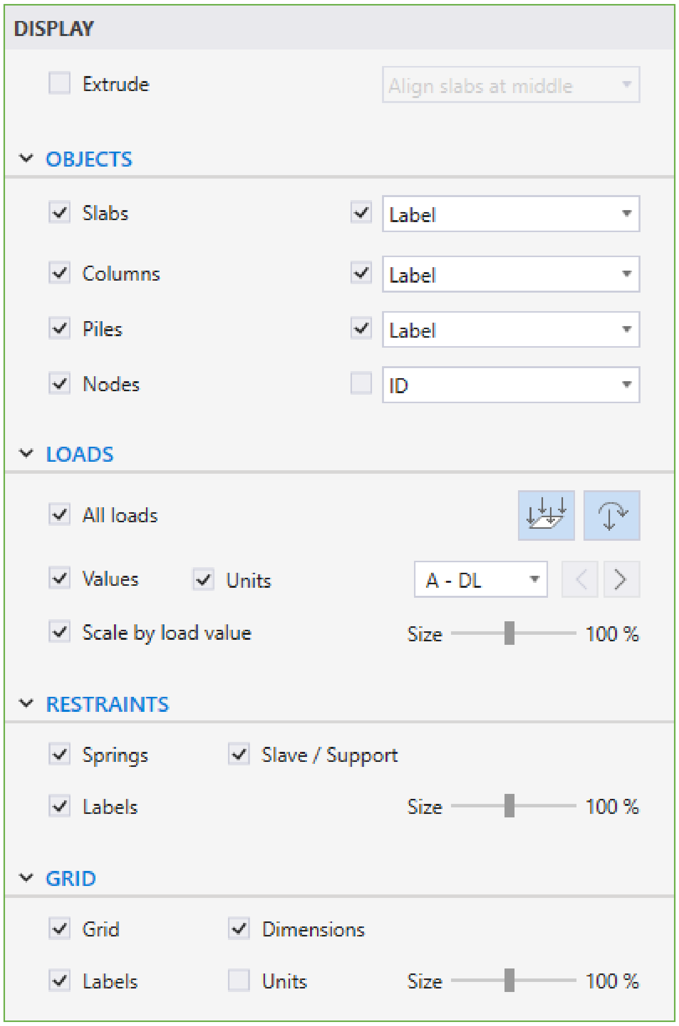
Users can also:

- Rotate Section in 3D: Enables rotating the model, or contour in three dimensions (shift + middle mouse button)

- Zoom in and zoom out using the mouse wheel and panned by holding the middle mouse button and moving the mouse around.

7.2.2.3. Display Options

The **Display** Command in the **Ribbon** can be used to open the DISPLAY OPTIONS dialog. This dialog facilitates toggling on/off the different Objects, Loads, Restraints, and Grids.

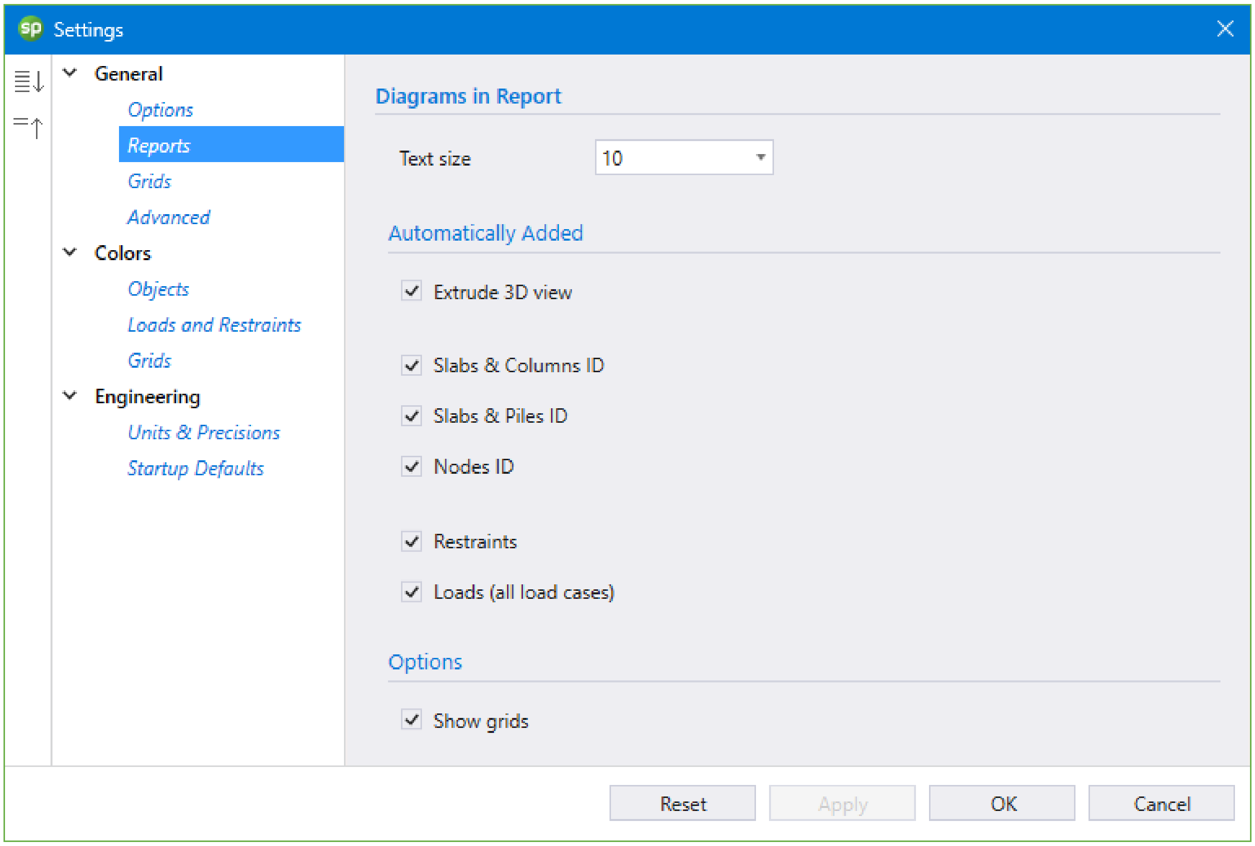


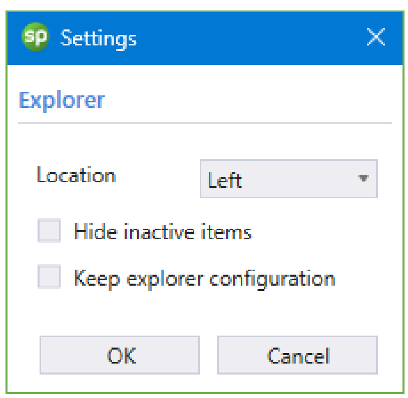
7.3. Output Settings

The **Settings** Command in the **Ribbon** can be used to open the SETTINGS dialog which can be used to change various program settings. The settings dialog can also be accessed from the settings button at the bottom left of the start screen. Beyond general options and program startup default values the settings provide the user with numerous ways to personalize results reports, including colors for objects, grids, loads and restraints, as well as engineering units and precisions.

7.3.1. Settings – Tabular Results

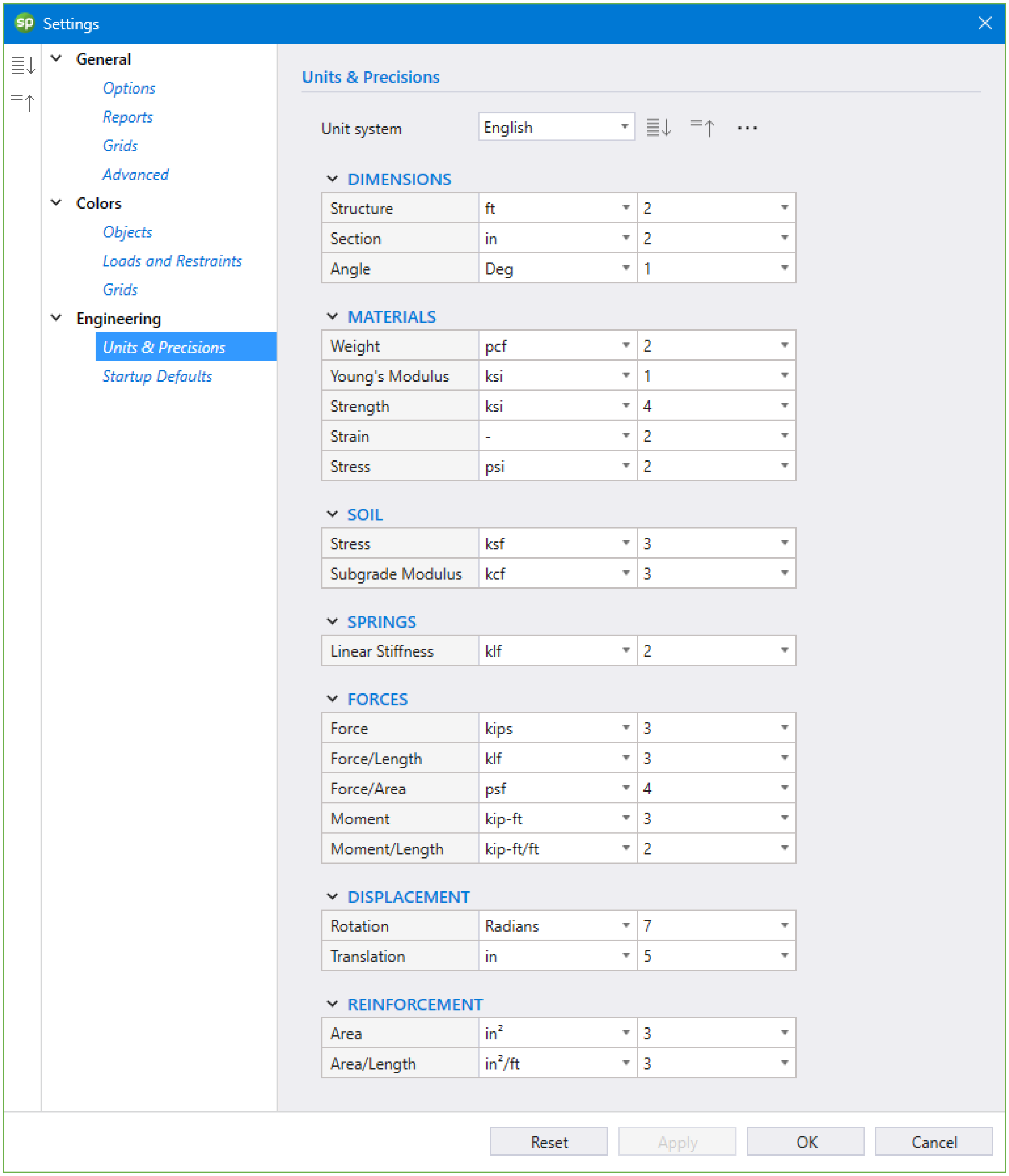
Tabular results settings are simply obtained from the ribbon to provide user options for result tables and reports as follows:





7.3.2. Settings – Engineering

Provides user options to set Units & Precisions for Dimensions, Materials, Soil, Springs, Forces, Displacement, and Reinforcement for input data and results.



CHAPTER

8

EXAMPLES

8.1. Example 1 – Spread Footing

8.1.1. Problem Formulation

A square footing with dimensions 10 ft × 10 ft and a thickness of 2 ft will be analyzed and designed to support a 500 kip axial load applied at its center.[1]

Design data

Concrete: $f_c' = 3.00$ ksi

$w_c = 148$ pcf

$E_c = 3{,}245$ ksi

ν (Poisson's ratio) = 0.15

Soil: Subgrade modulus = 100 kcf

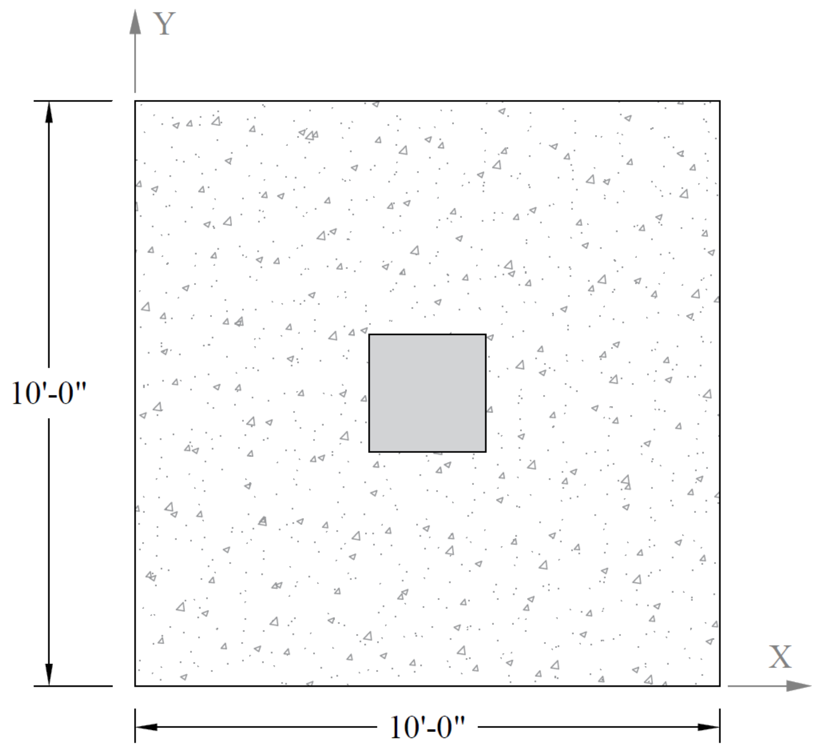
Allowable pressure = 6 kcf

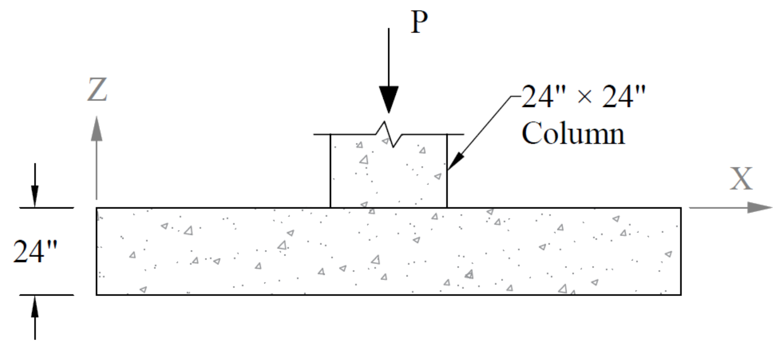
Steel: $f_y = 60$ ksi

$E_s = 29{,}000$ ksi

The origin of the XY plane will be located at the lower left-hand corner of the footing. Two-foot square elements will be used.

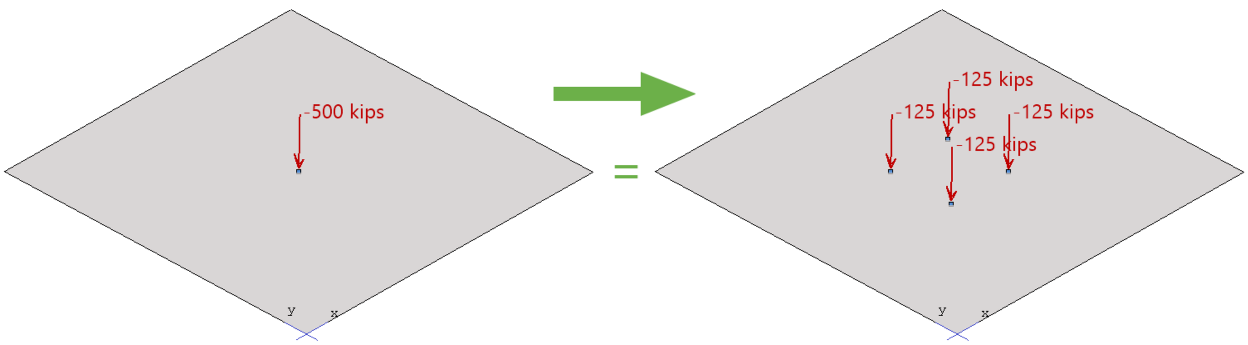
1 *Foundation Analysis and Design*, Fourth Edition, 1988, p. 461.





<u>Column Loading Conditions</u>

The concentrated load will be applied as four nodal loads (500 kips / 4 = 125 kips per node).

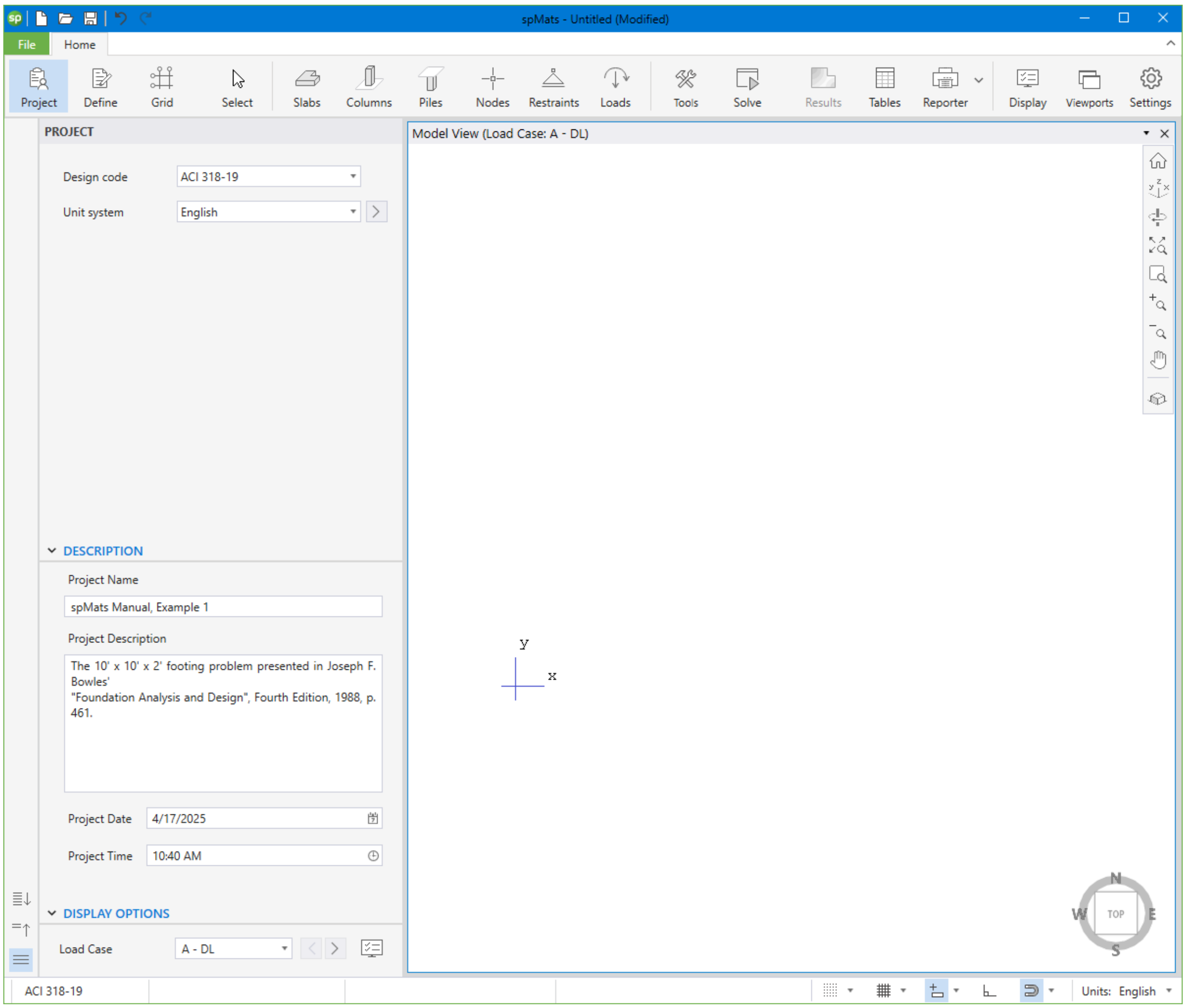


8.1.2. Preparing the Input[2]

1. From the Start screen, select **New Project**.

2. In the **Main Program Window**, select **Project** from the **Ribbon**.

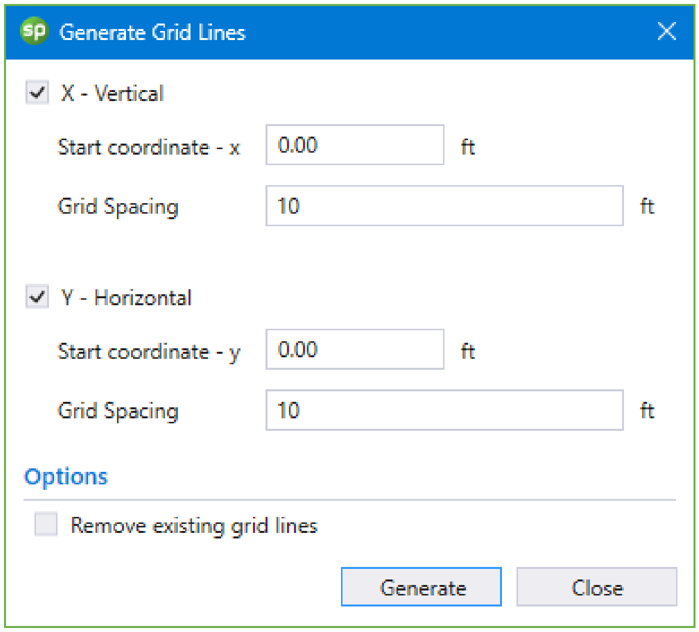
   - Select the DESIGN CODE, UNIT SYSTEM, and enter the PROJECT NAME and PROJECT DESCRIPTION.



2 All input data entered manually assuming that LOAD DEFAULT DEFINITIONS option is not selected in Startup Defaults.

3. From the **Ribbon**, select **Grid**.

- Click on the **Generate** in the left panel to have the program surface the following:



- Place a check mark in the X - VERTICAL box and enter the following values in the corresponding text boxes:

START COORDINATE - X: `0.0`

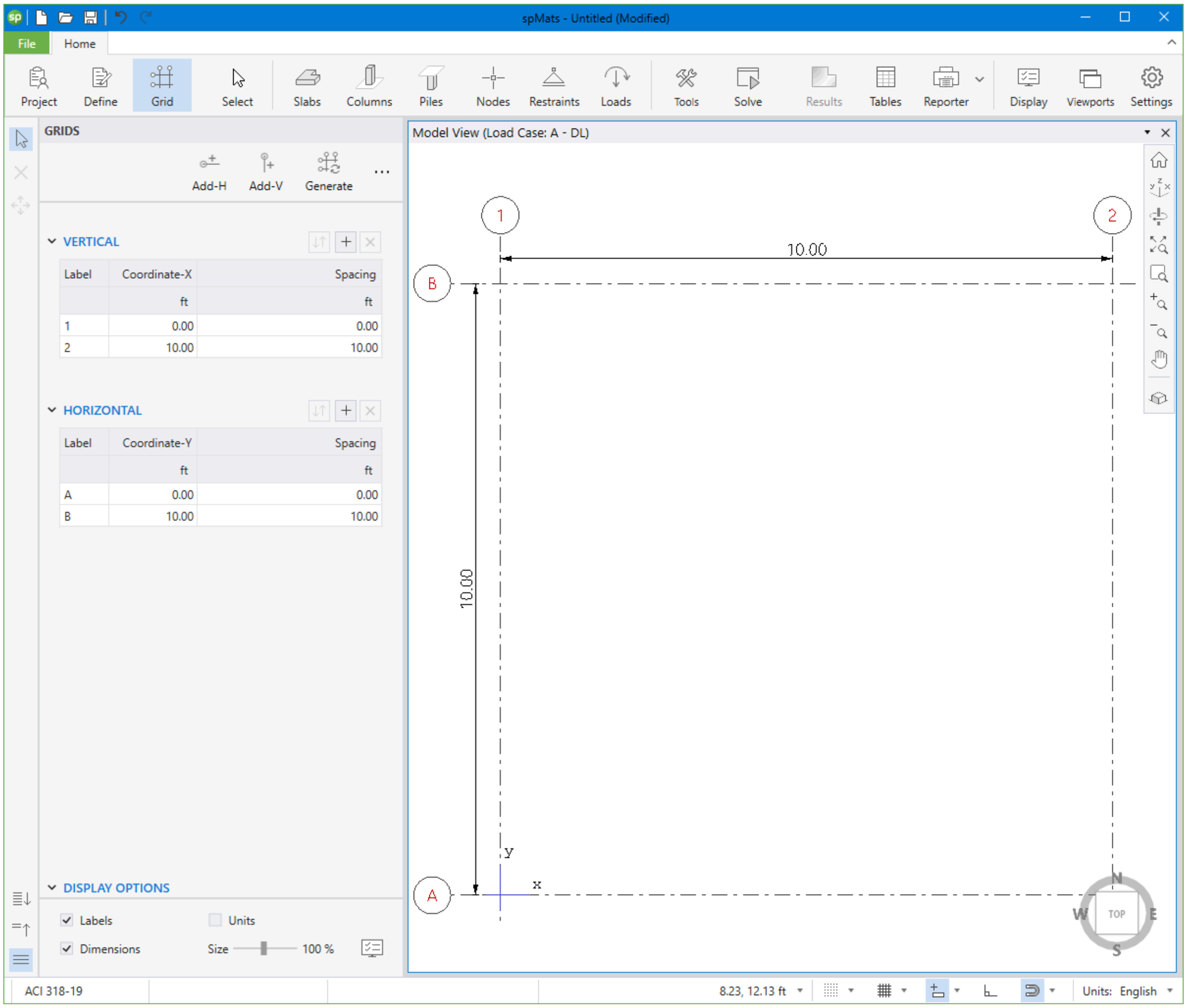
GRID SPACING: `10.0`

- Place a check mark in the Y - VERTICAL box and enter the following values in the corresponding text boxes:

START COORDINATE - Y: `0.0`

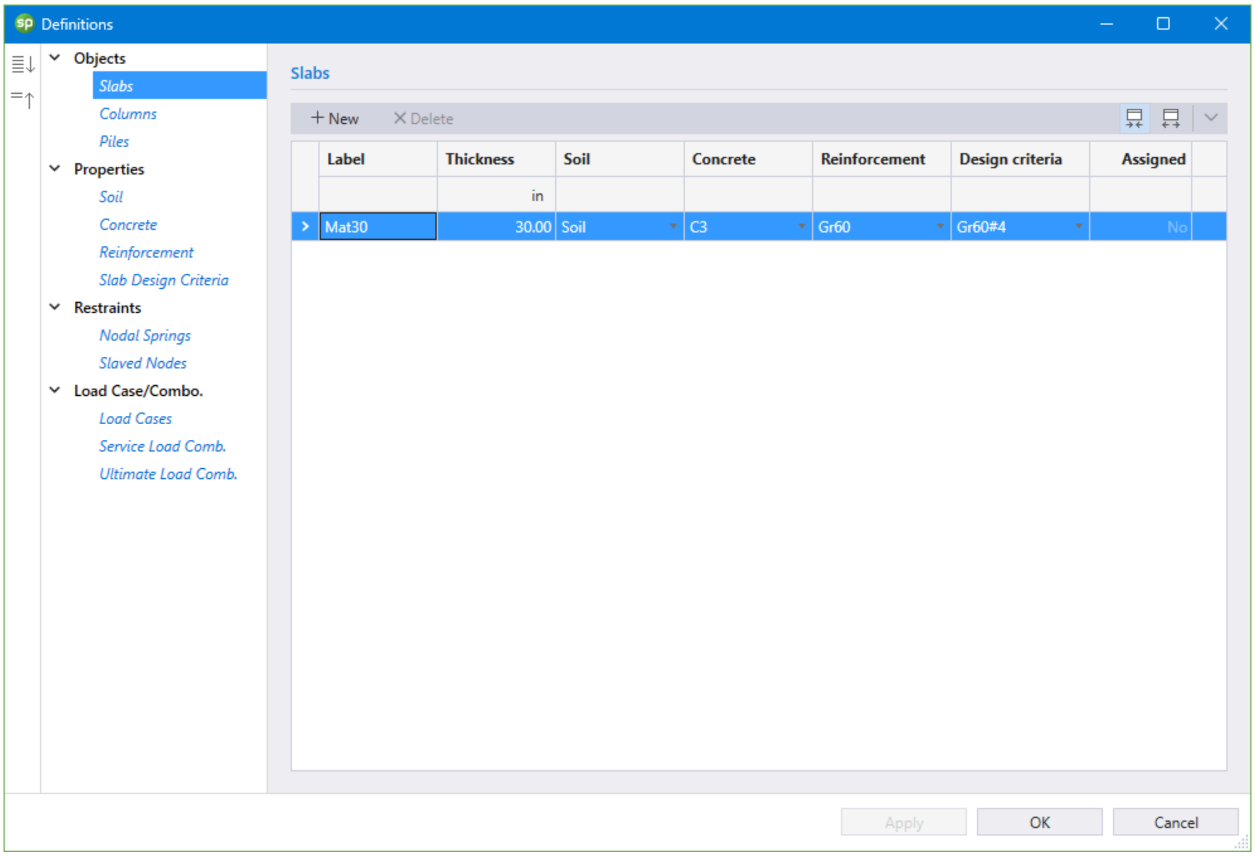
GRID SPACING: `10.0`

- Click on the GENERATE button to return to the main window. Notice how the VERTICAL and HORIZONTAL grid lines now appear in the VIEWPORT.


spMats - Untitled (Modified)
File
Home
Project
Define
Grid
Select
Slabs
Columns
Piles
Nodes
Restraints
Loads
Tools
Solve
Results
Tables
Reporter
Display
Viewports
Settings
GRIDS
Add-H
Add-V
Generate
VERTICAL
Label Coordinate-X Spacing
ft ft
1 0.00 0.00
2 10.00 10.00
HORIZONTAL
Label Coordinate-Y Spacing
ft ft
A 0.00 0.00
B 10.00 10.00
DISPLAY OPTIONS
Labels
Units
Dimensions
Size
100 %
Model View (Load Case: A - DL)
1
2
10.00
B
10.00
A
y
x
N
W
TOP
E
S
ACI 318-19
8.23, 12.13 ft
Units: English

4. From the **Ribbon**, select **Define,** then choose **Slabs** from **Objects** to display the **Slabs** dialog box.

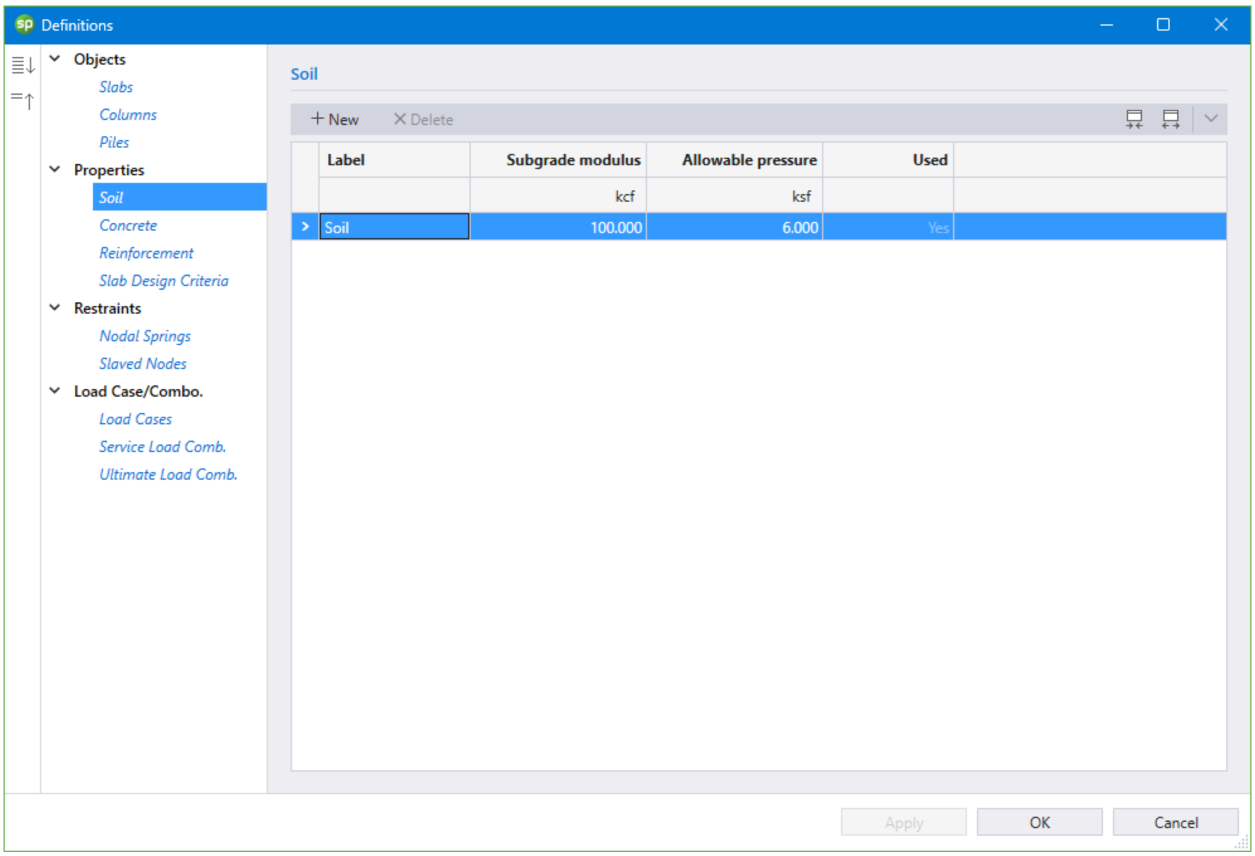
- Input `Mat30` for LABEL and `30.00` in. for THICKNESS.



5. Click on **Soil** from **Properties** to display the **Soil** dialog box.

- Enter the following:

LABEL:	`SOIL`
SUBGRADE MODULUS:	`100.00` kcf
ALLOWABLE PRESSURE:	`6.00` ksf



6. Click on **Concrete** from **Properties** to display the **Concrete** dialog box.

- Enter the following:

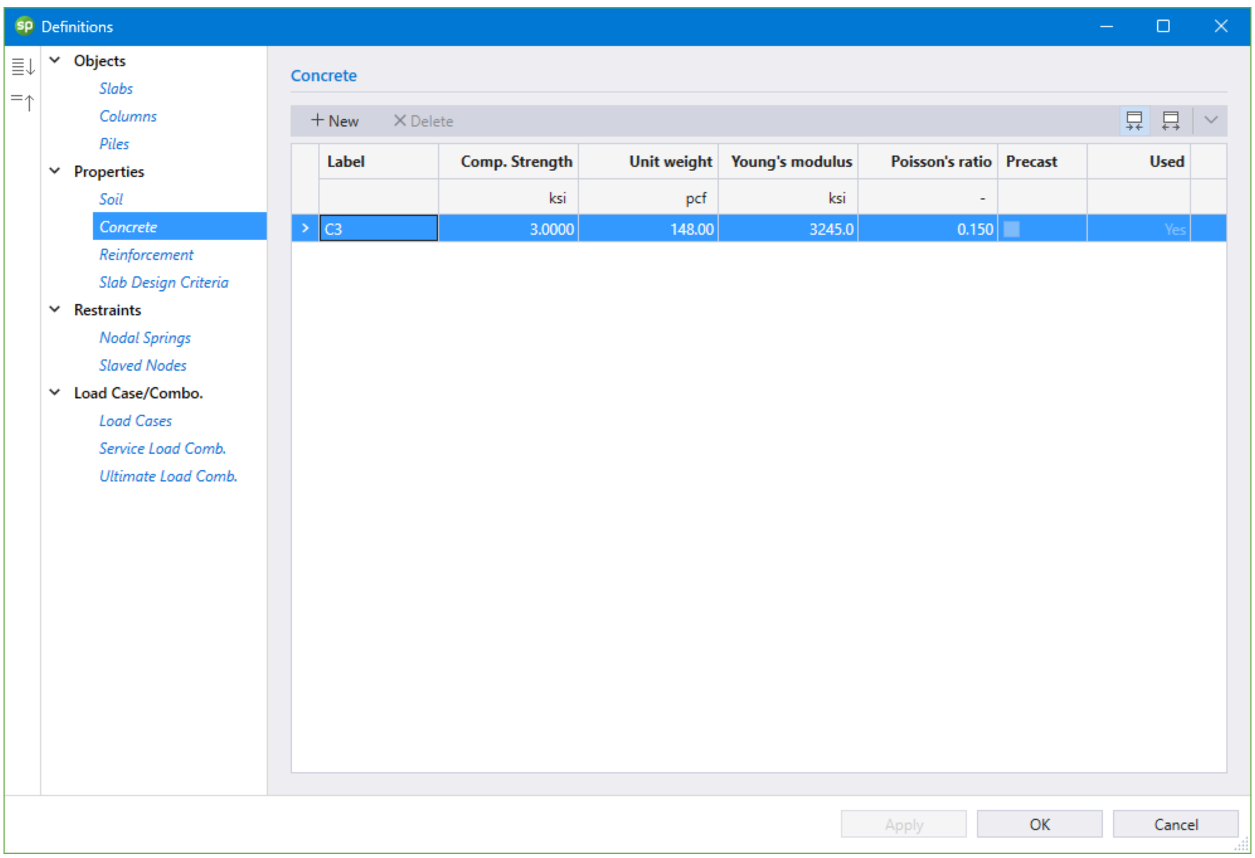
LABEL: C3

COMPRESSIVE STRENGTH: 3.00 ksi

UNIT WEIGHT: 148.00 pcf

YOUNG'S MODULUS: 3245.00 ksi

POISSON'S RATIO: 0.15



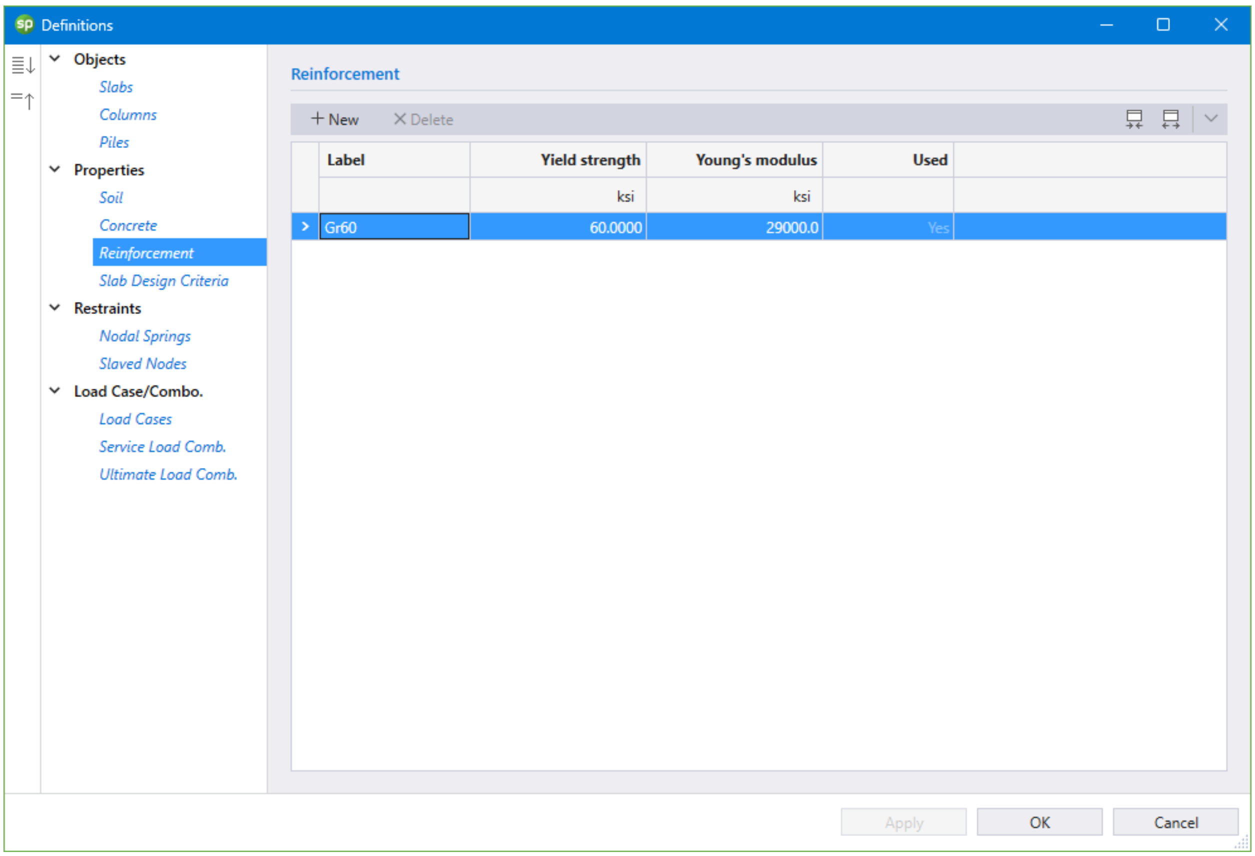
7. Click on **Reinforcement** from **Properties** to display the **Reinforcement** dialog box.

- Enter the following:

LABEL: `Gr60`

YIELD STRENGTH: `60.00` ksi

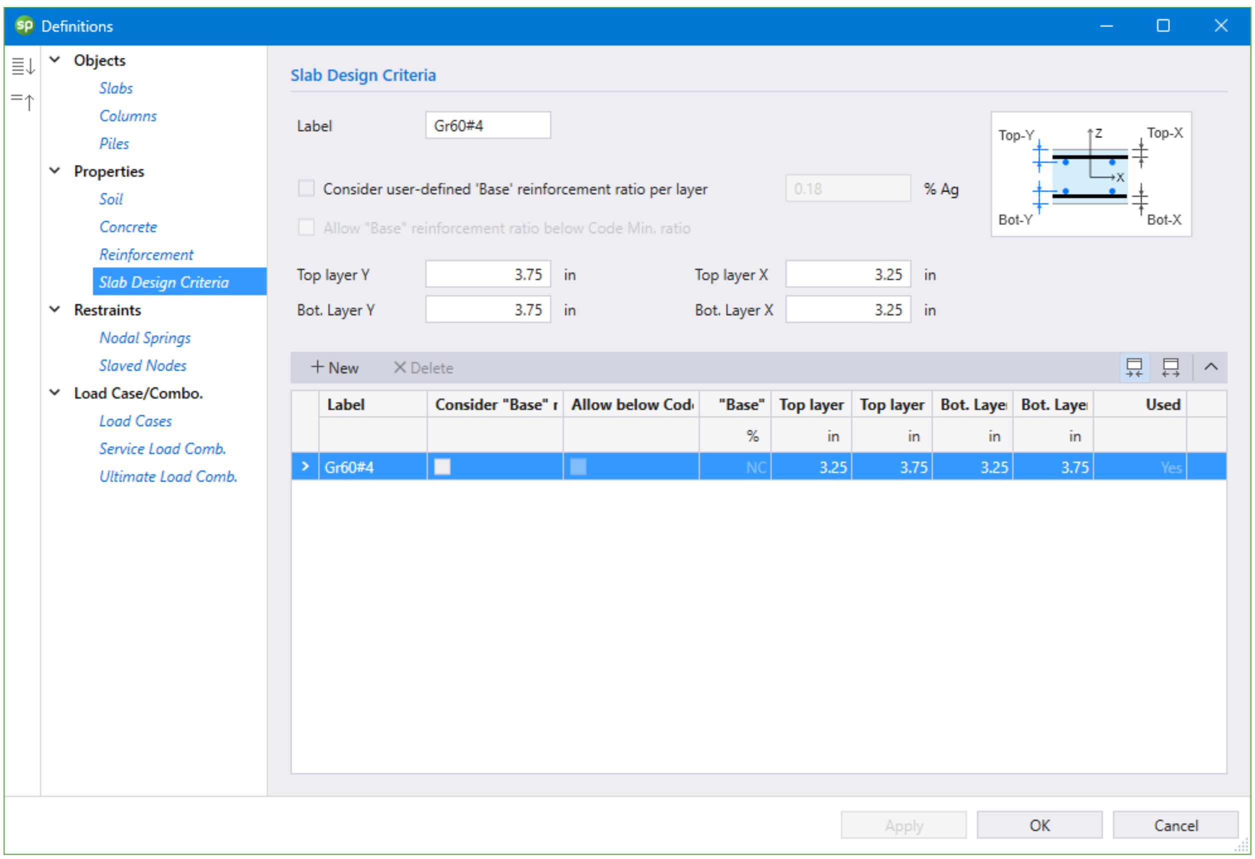
YOUNG'S MODULUS: `29000.00` ksi



8. Click on **Slab Design Criteria** from **Properties** to display the **Slab Design Criteria** dialog box.

- Enter the following:

LABEL:	Gr60#4
TOP LAYER Y:	3.75 in.
BOTTOM LAYER Y:	3.75 in.
TOP LAYER X:	3.25 in.
BOTTOM LAYER X:	3.25 in.

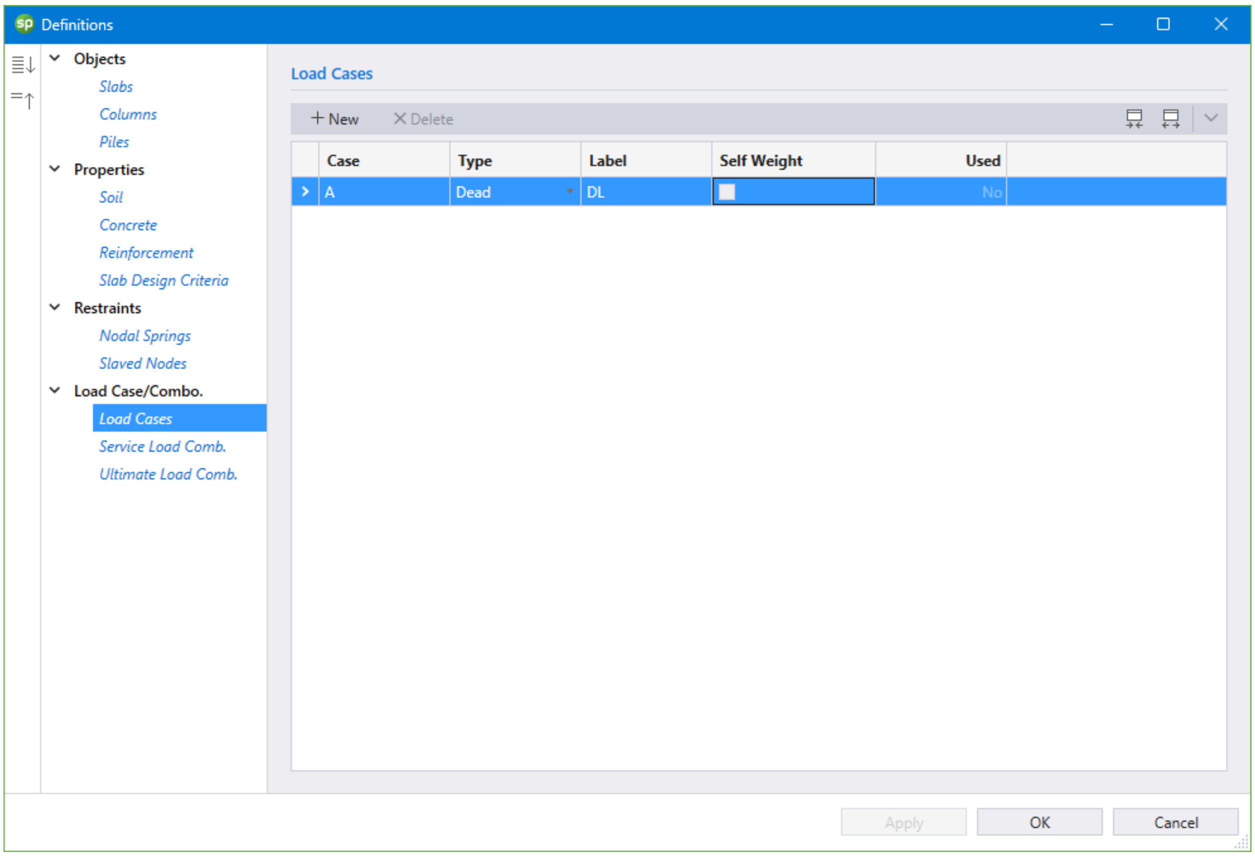


9. Click on **Load Cases** from **Load Case/Combo.** to display the **Load Cases** dialog box.

- Enter the following:

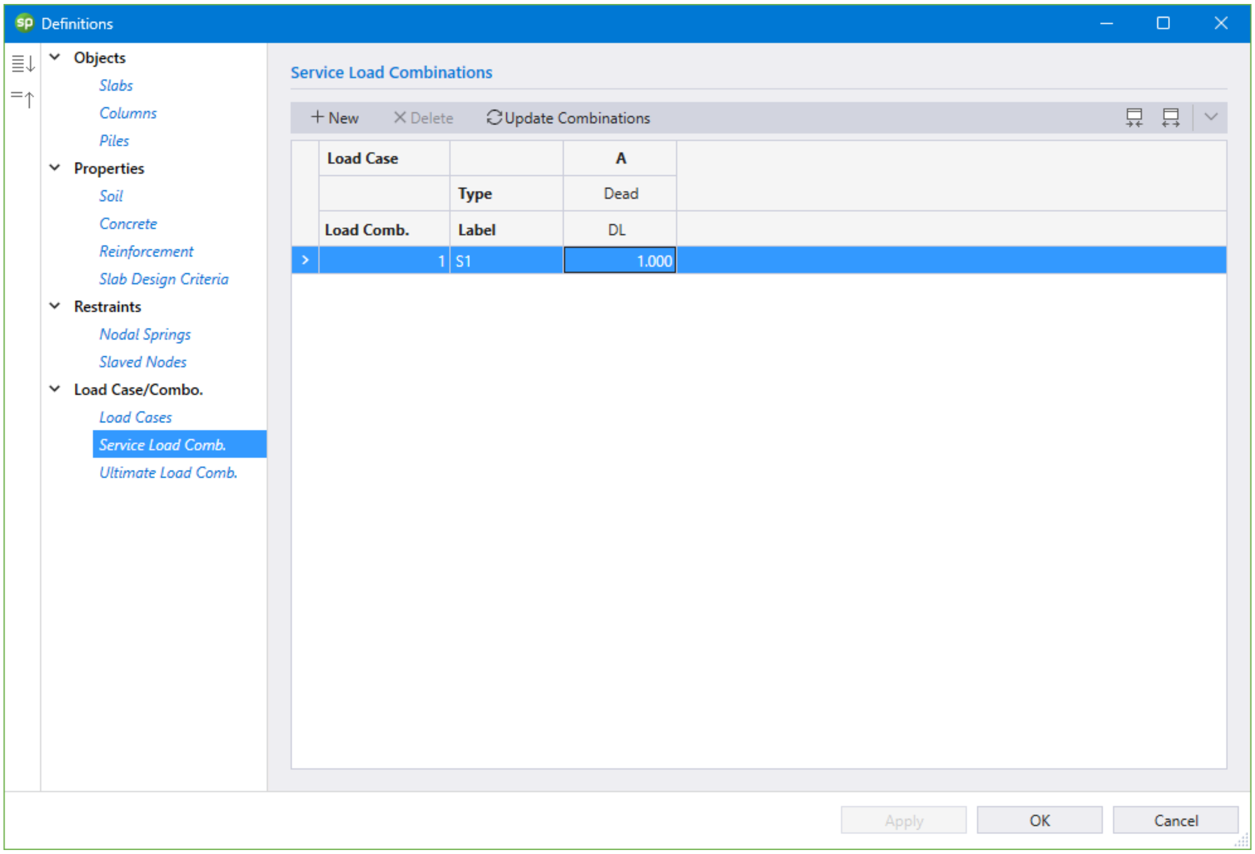
  CASE A: DL

- Uncheck SELF WEIGHT for CASE A.



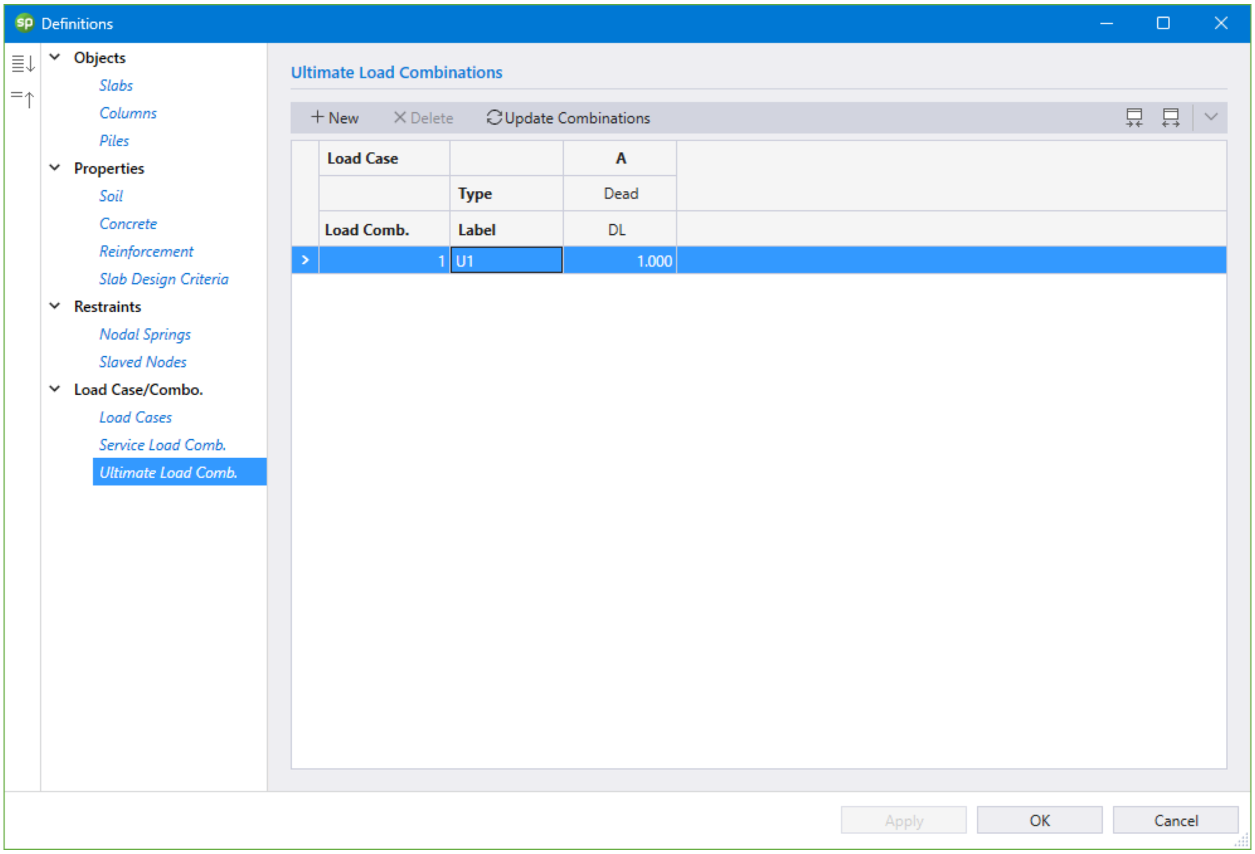
10. Click on **Service Load Combinations** from **Load Case/Combo.** to display the **Service Load Combinations** dialog box.

   - Enter the following service load combinations shown in the figure below:



11. Click on **Ultimate Load Combinations** from **Load Case/Combo.** to display the **Ultimate Load Combinations** dialog box.

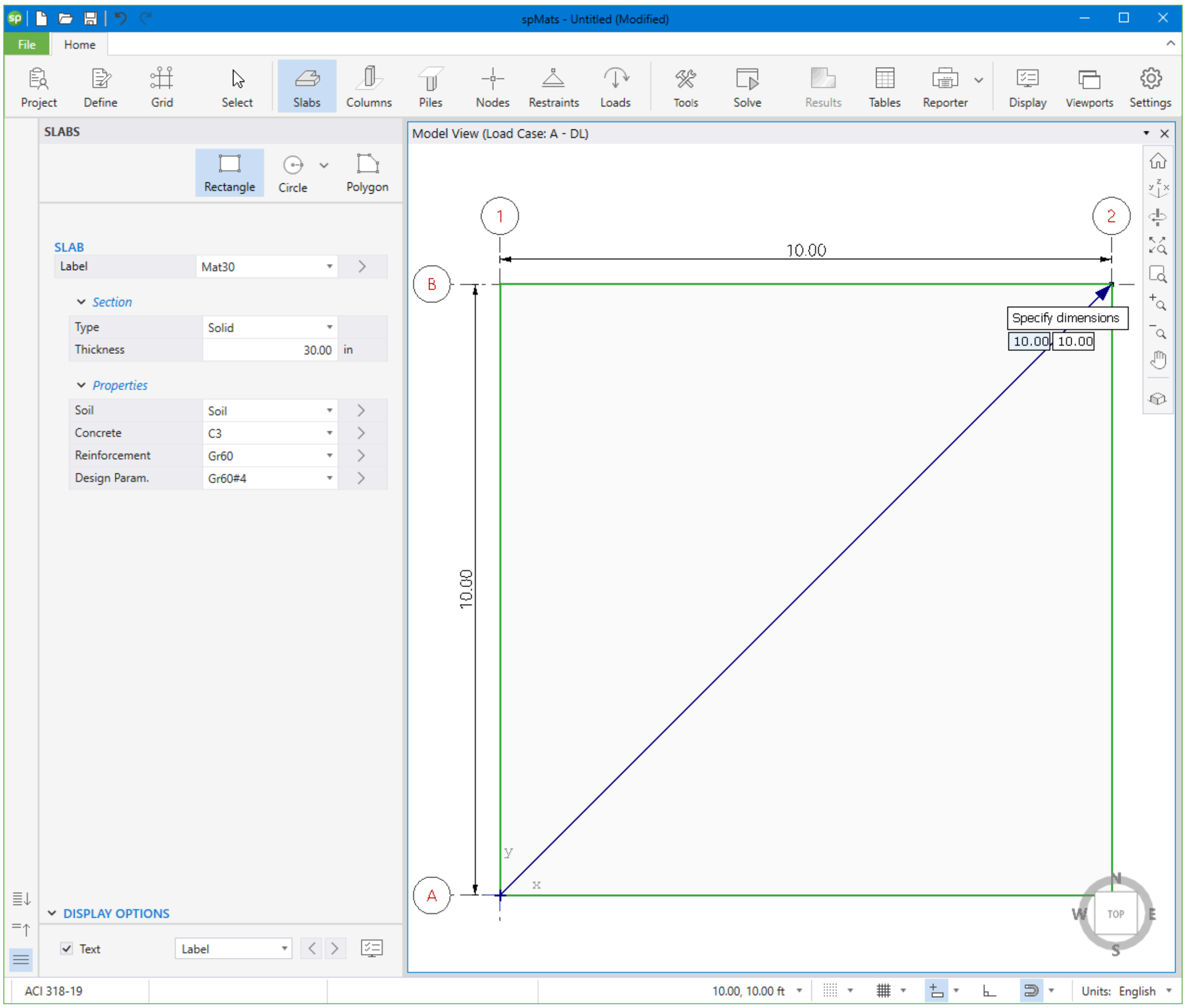
    - Enter the following load combinations shown in the figure below:



8.1.3. Assigning Properties

12. From the **Ribbon**, select **Slabs** command.

- In the left panel, select **Rectangle** then select MAT30 from LABEL.
- In the VIEWPORT, marquee-select the region (A, 1) – (B, 2) to apply the selected slab to the entire foundation.



13. From the **Ribbon**, select **Nodes** command.

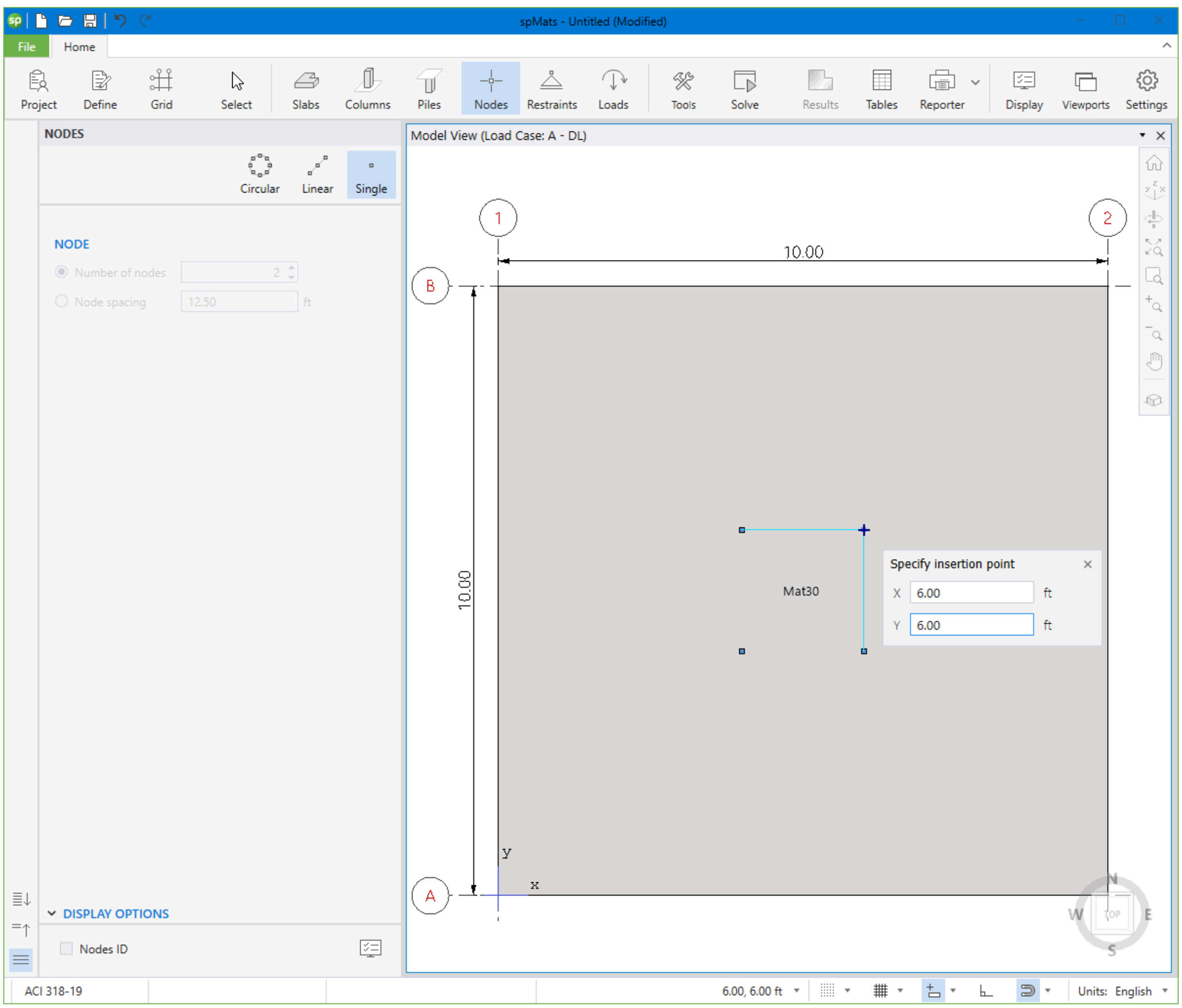
- In the left panel, select **Single**.
- In the VIEWPORT, enter the coordinates of each node using the dynamic input box (to activate the dynamic input box simply start typing):

NODE 1: (4, 4)

NODE 2: (6, 4)

NODE 3: (4, 6)

NODE 4: (6, 6)



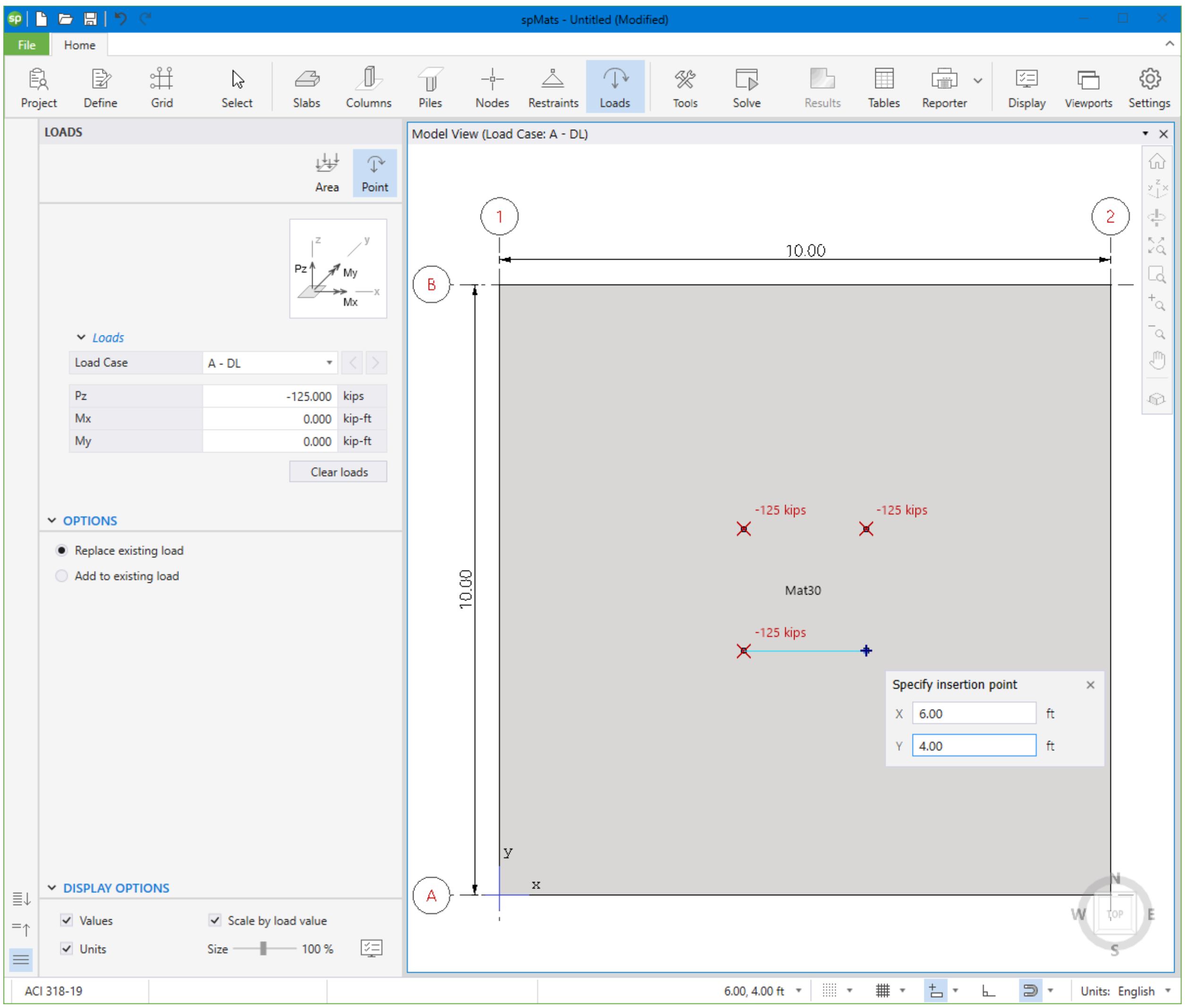
8.1.4. Assigning Loads

14. From the **Ribbon**, select **Loads** command.

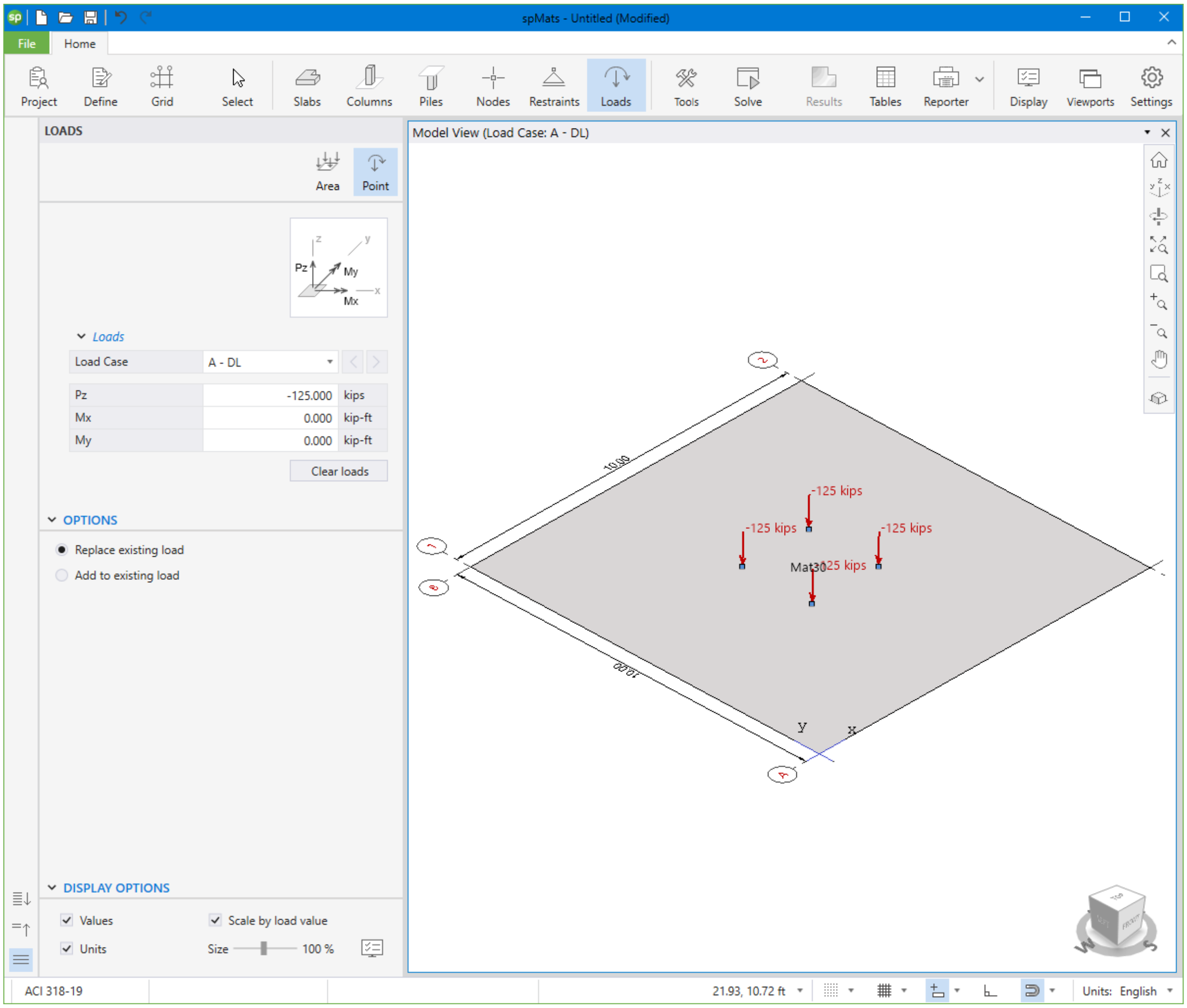
- In the left panel, select **Point** then select A-DL from LOAD CASE and enter the following:

  P_Z: `-125.00` kips

- Apply to all nodes as shown in the figure below.



- Also, you can click on the 3D VIEW icon from **View Controls** (top right of the active viewport) to get a better view of the applied loads.

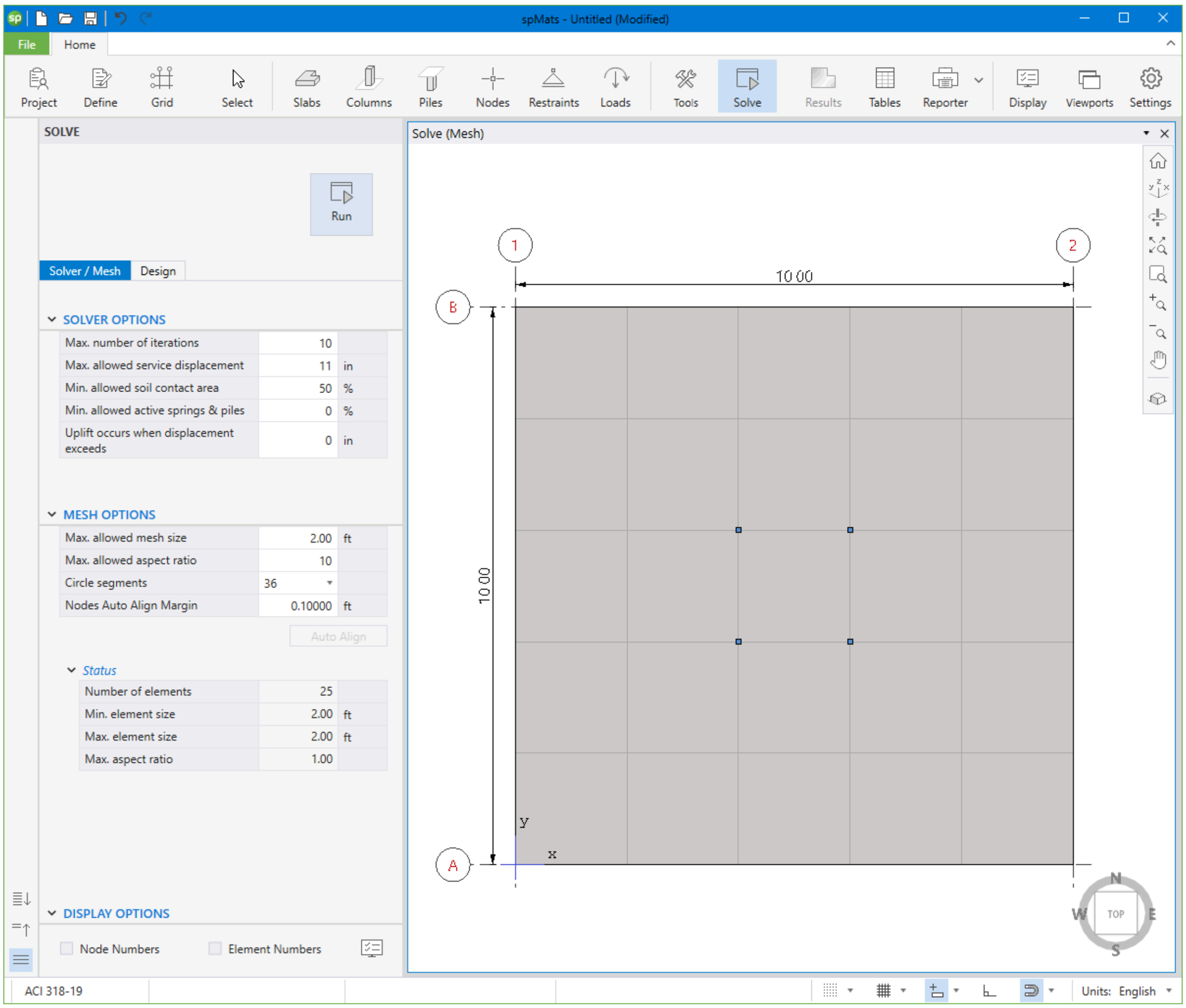


8.1.5. Solving

15. From the **Ribbon**, select **Solve** command.

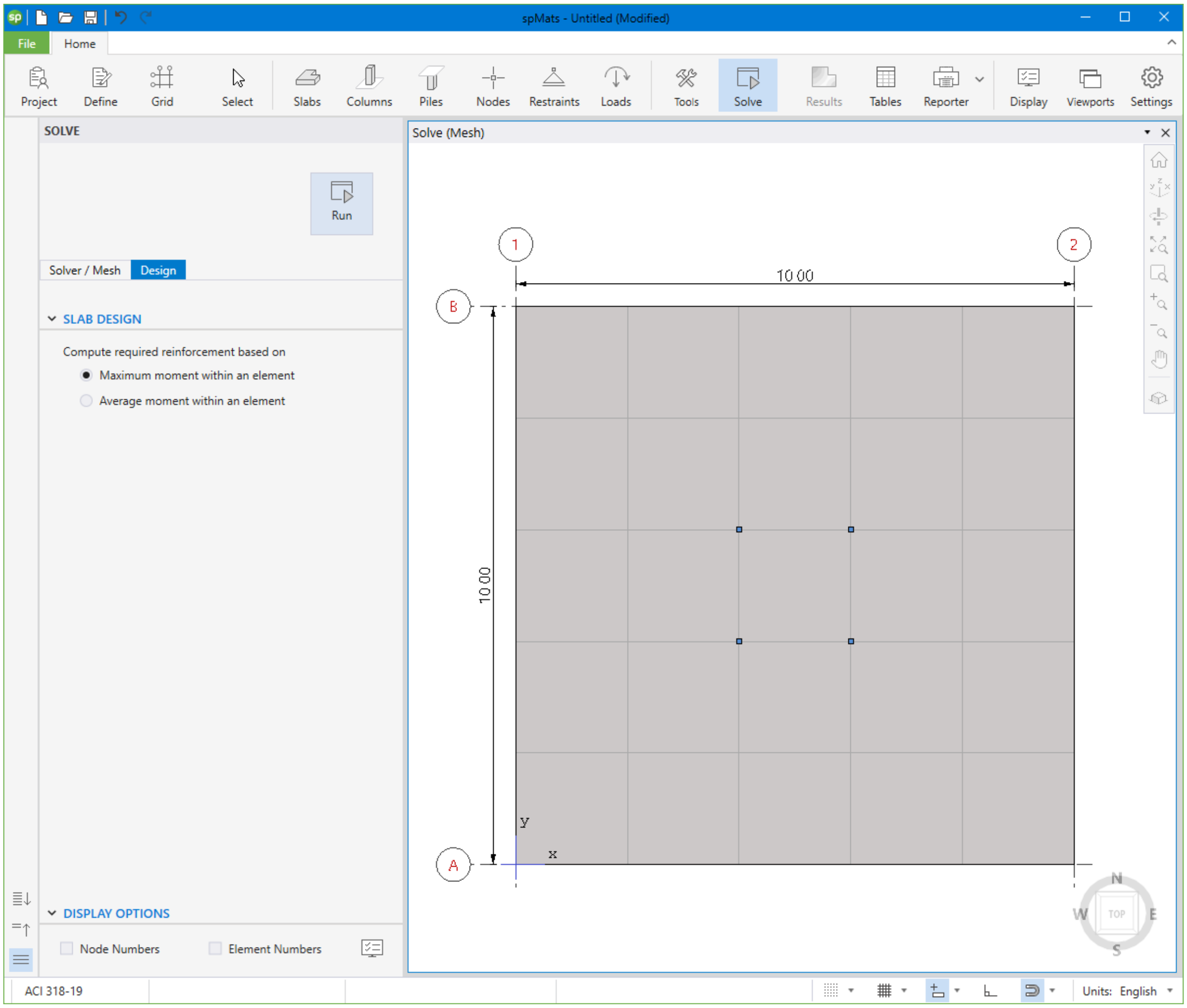
    For **Solver / Mesh Options**:

    - Leave all **Solver Options** to their default settings.
    - Leave all **Mesh Options** to their default settings.

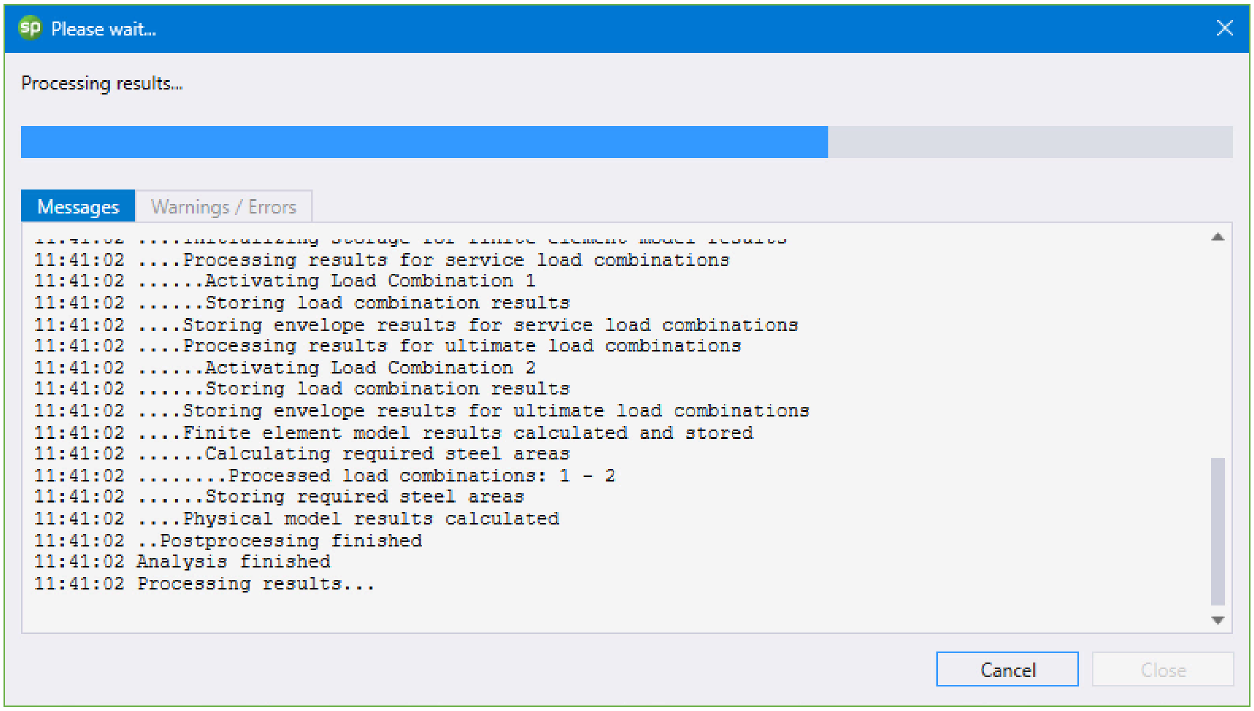


For **Design Options**:

- Leave all **Slab Design Options** to their default settings.

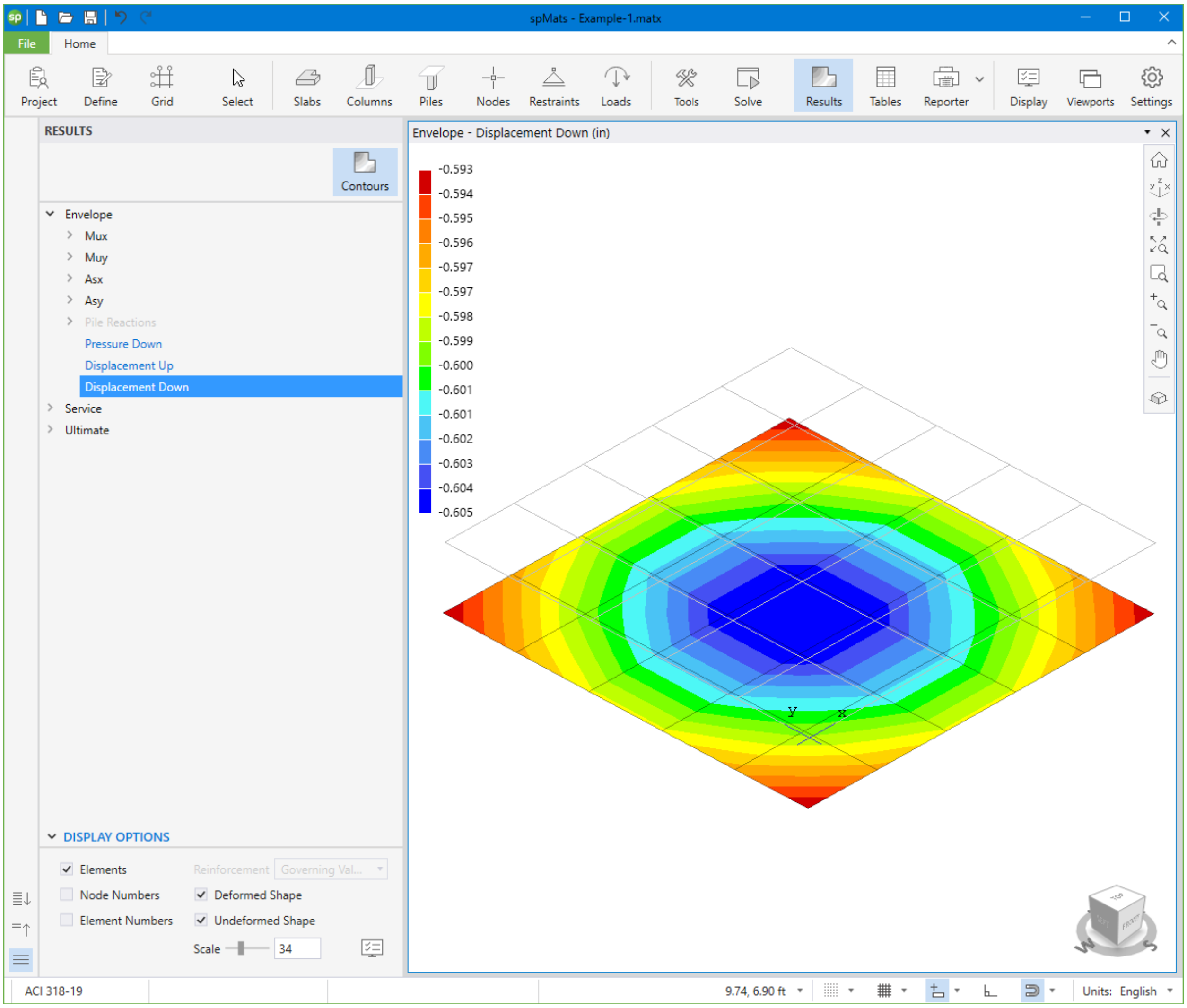


- Click on the **Run** button.
- The spMats **Solver** window is displayed and the solver messages are listed. After the solution is done, the design will be performed and then the focus will immediately be passed to the **Contours** scope.

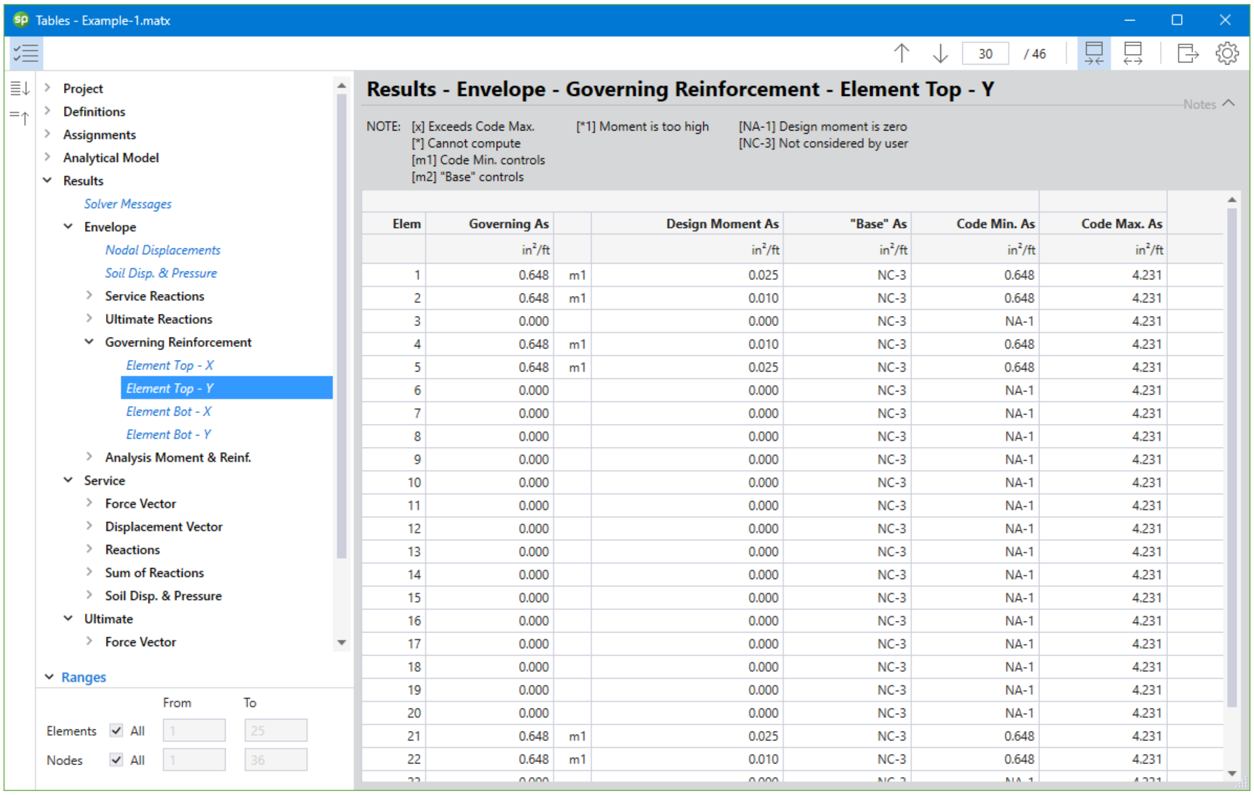


8.1.6. Viewing and Printing Results

16. After a successful run, results can be viewed in a contour form.

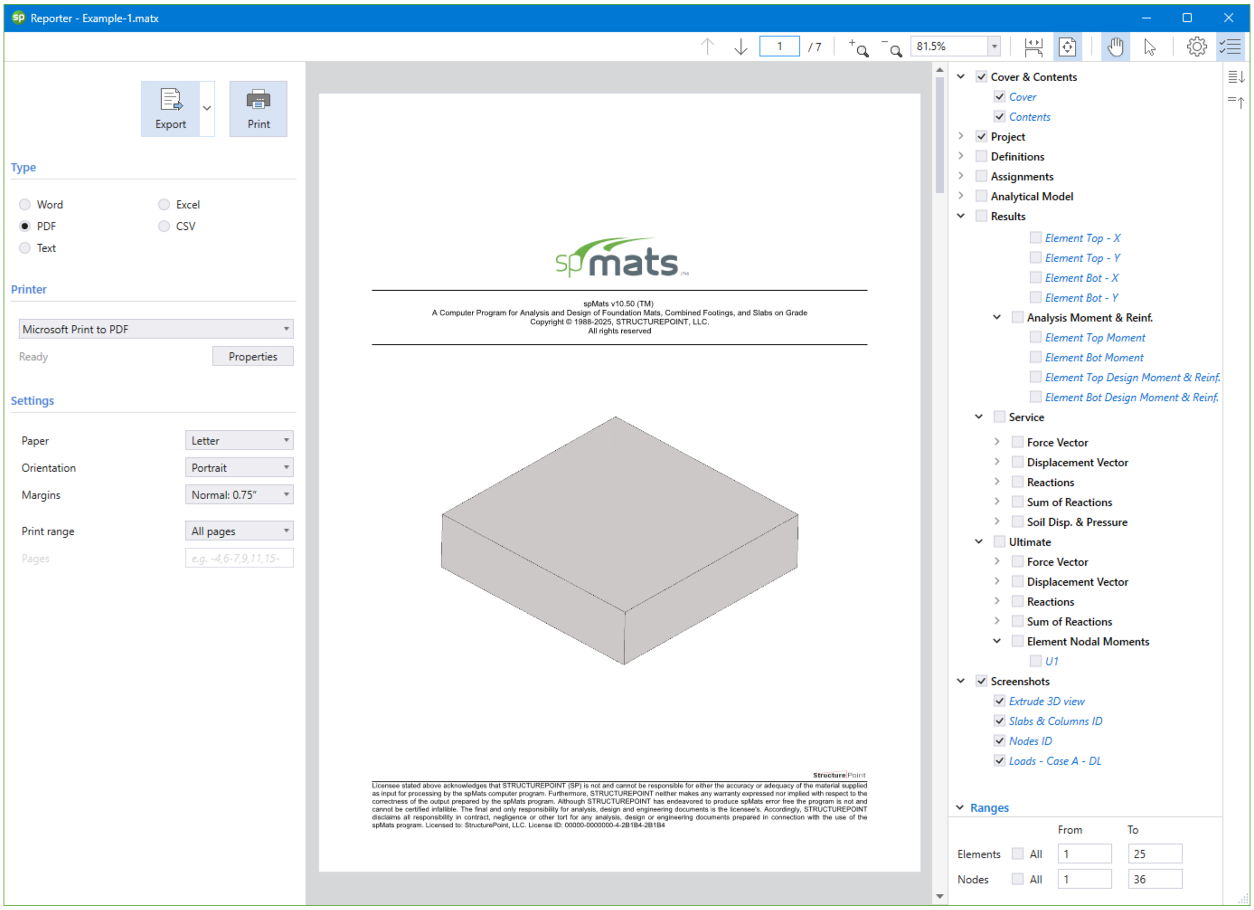


17. Results can be also viewed in table format by selecting the **Tables** command from the **Ribbon.**



Elem	Governing As		Design Moment As	"Base" As	Code Min. As	Code Max. As
	in²/ft		in²/ft	in²/ft	in²/ft	in²/ft
1	0.648	m1	0.025	NC-3	0.648	4.231
2	0.648	m1	0.010	NC-3	0.648	4.231
3	0.000		0.000	NC-3	NA-1	4.231
4	0.648	m1	0.010	NC-3	0.648	4.231
5	0.648	m1	0.025	NC-3	0.648	4.231
6	0.000		0.000	NC-3	NA-1	4.231
7	0.000		0.000	NC-3	NA-1	4.231
8	0.000		0.000	NC-3	NA-1	4.231
9	0.000		0.000	NC-3	NA-1	4.231
10	0.000		0.000	NC-3	NA-1	4.231
11	0.000		0.000	NC-3	NA-1	4.231
12	0.000		0.000	NC-3	NA-1	4.231
13	0.000		0.000	NC-3	NA-1	4.231
14	0.000		0.000	NC-3	NA-1	4.231
15	0.000		0.000	NC-3	NA-1	4.231
16	0.000		0.000	NC-3	NA-1	4.231
17	0.000		0.000	NC-3	NA-1	4.231
18	0.000		0.000	NC-3	NA-1	4.231
19	0.000		0.000	NC-3	NA-1	4.231
20	0.000		0.000	NC-3	NA-1	4.231
21	0.648	m1	0.025	NC-3	0.648	4.231
22	0.648	m1	0.010	NC-3	0.648	4.231

18. Results can be printed or exported in different formats by selecting the **Reporter** command from the **Ribbon.**



8.2. Example 2 – Mat Foundation

8.2.1. Problem Formulation

Analyze the mat shown below. The mat is supported on two types of soil, Soil-1 and Soil-2, as shown in the figure. Use the following data:

Design data

Concrete: f_c' = 3.00 ksi

w_c = 145 pcf

E_c = 3,156 ksi

v (Poisson's ratio) = 0.15

Soil-1: Subgrade modulus = 50 kcf

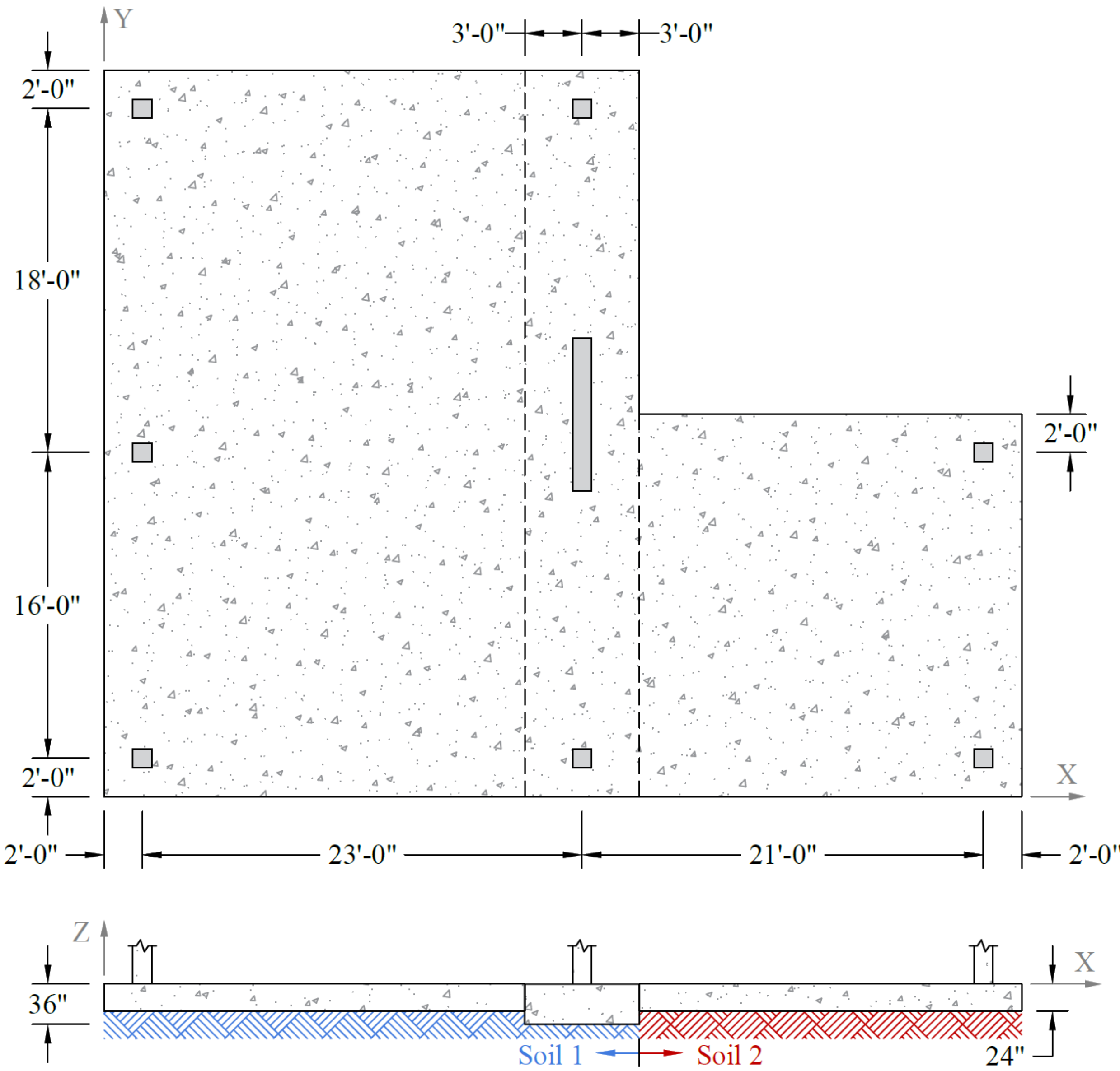
Allowable pressure = 6 kcf

Soil-2: Subgrade modulus = 75 kcf

Allowable pressure = 8 kcf

Steel: f_y = 60 ksi

E_s = 29,000 ksi



Y
3'-0"
3'-0"
2'-0"
18'-0"
16'-0"
2'-0"
2'-0"
2'-0"
23'-0"
21'-0"
2'-0"
X
Z
X
36"
Soil 1
Soil 2
24"

The superstructure was analyzed for wind loads. The resulting reactions for the columns and wall, as well as for the dead and live loads, are as follows:

		Dead	Live	Wind
Column	P_z (kip)	-50	-35	-10
	M_x (kip-ft)	0	0	5
	M_y (kip-ft)	0	0	0
Wall	P_z (kip)	-376	-208	-80
	M_x (kip-ft)	0	0	60
	M_y (kip-ft)	0	0	0

The mat will be analyzed for three service combinations and nine ultimate combinations. Deflections and pressures are to be checked for the service combinations and the mat will be designed (compute required reinforcement) for the ultimate combinations.

Service Combinations:

S1 = D + L

S2 = D + L + W

S3 = D + W

Ultimate Combinations:

U1 = 1.4 D

U2 = 1.2 D + 1.6 L

U3 = 1.2 D + 1.0 L

U4 = 1.2 D + 0.8 W

U5 = 1.2 D + 1.0 L + 1.6 W

U6 = 0.9 D + 1.6 W

U7 = 1.2 D – 0.8 W

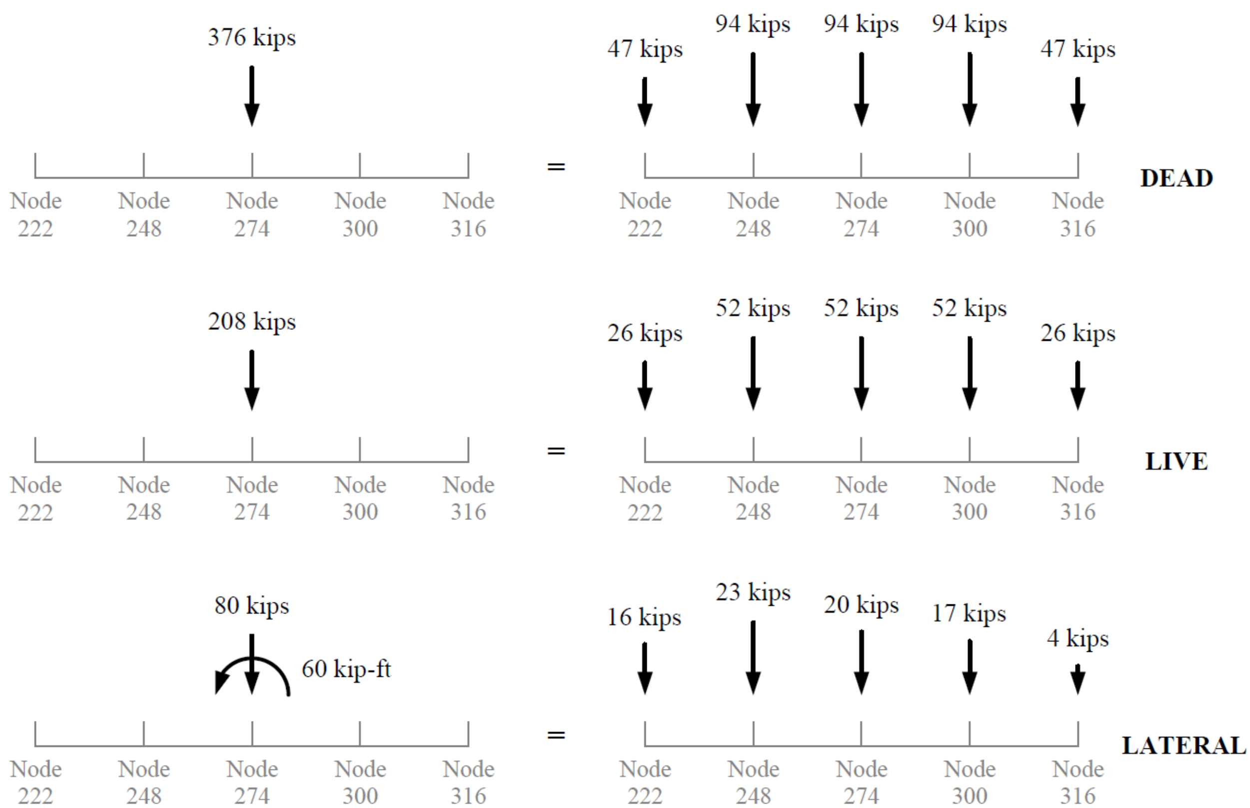
U8 = 1.2 D + 1.0 L – 1.6 W

U9 = 0.9 D – 1.6 W

The lower left-hand corner of the mat is chosen as the origin of the grid system.

Since the wall is spanning over five nodes, (nodes 222, 248, 274, 300 and 316) the wall loads will be divided into equivalent nodal loads depending on the tributary length of each node. Furthermore, the R_x degrees of freedom of these five nodes are slaved to simulate the stiffening effect of the wall.

Wall Loading Conditions

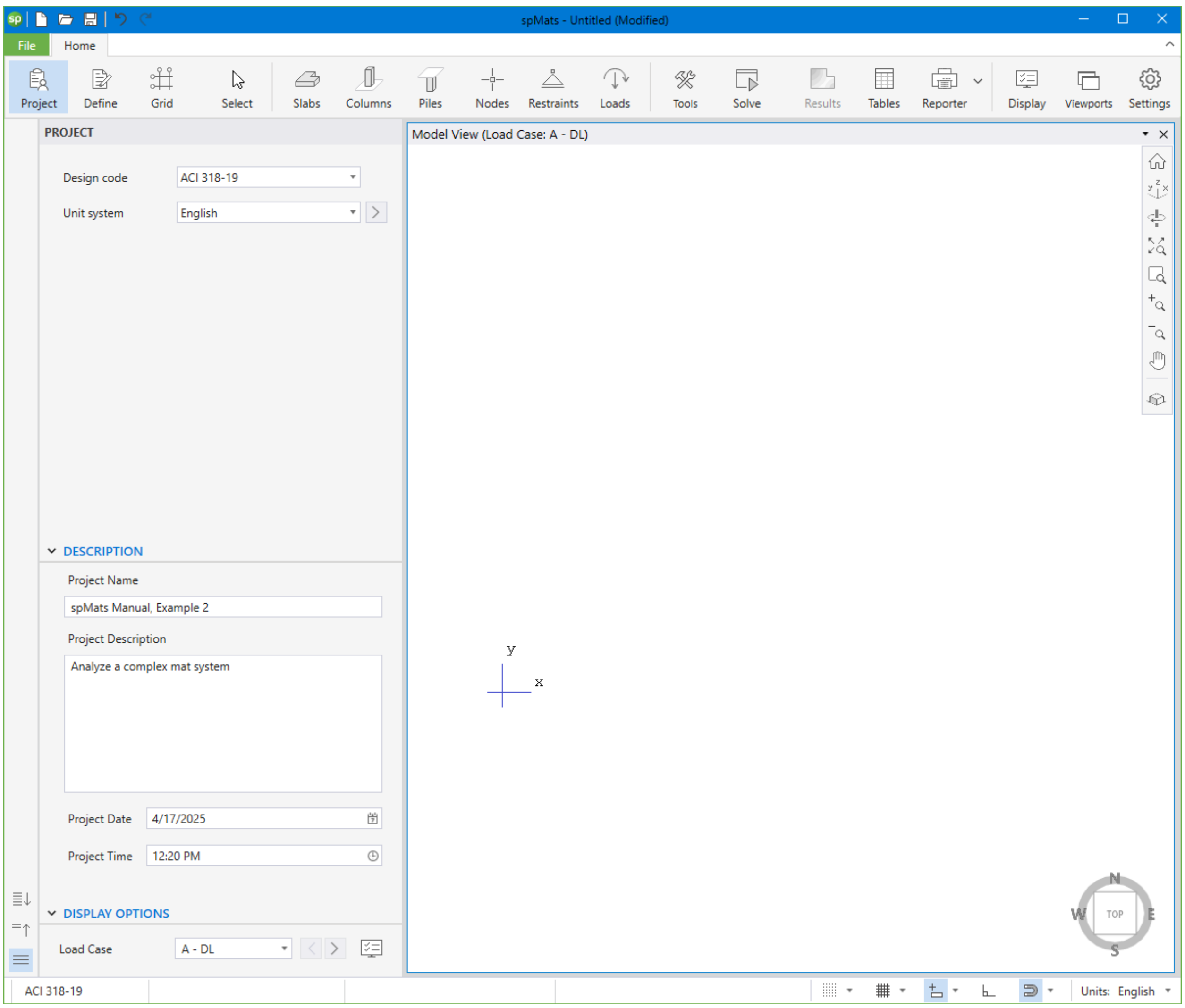


8.2.2. Preparing the Input[3]

1. From the Start screen, select **New Project**.

2. In the **Main Program Window**, select **Project** from the **Ribbon**.

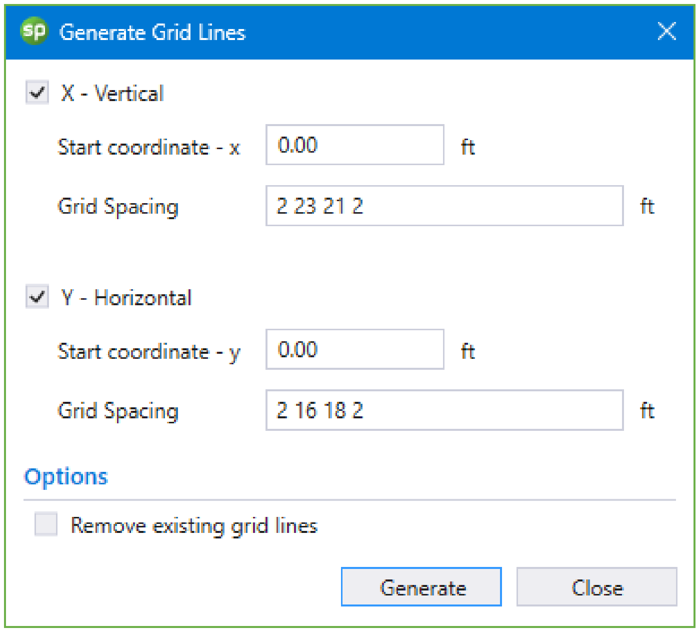
   - Select the DESIGN CODE, UNIT SYSTEM, and enter the PROJECT NAME and PROJECT DESCRIPTION.



[3] All input data entered manually assuming that LOAD DEFAULT DEFINITIONS option is not selected in Startup Defaults.

3. From the **Ribbon**, select **Grid**.

- Click on the **Generate** in the left panel to have the program surface the following:



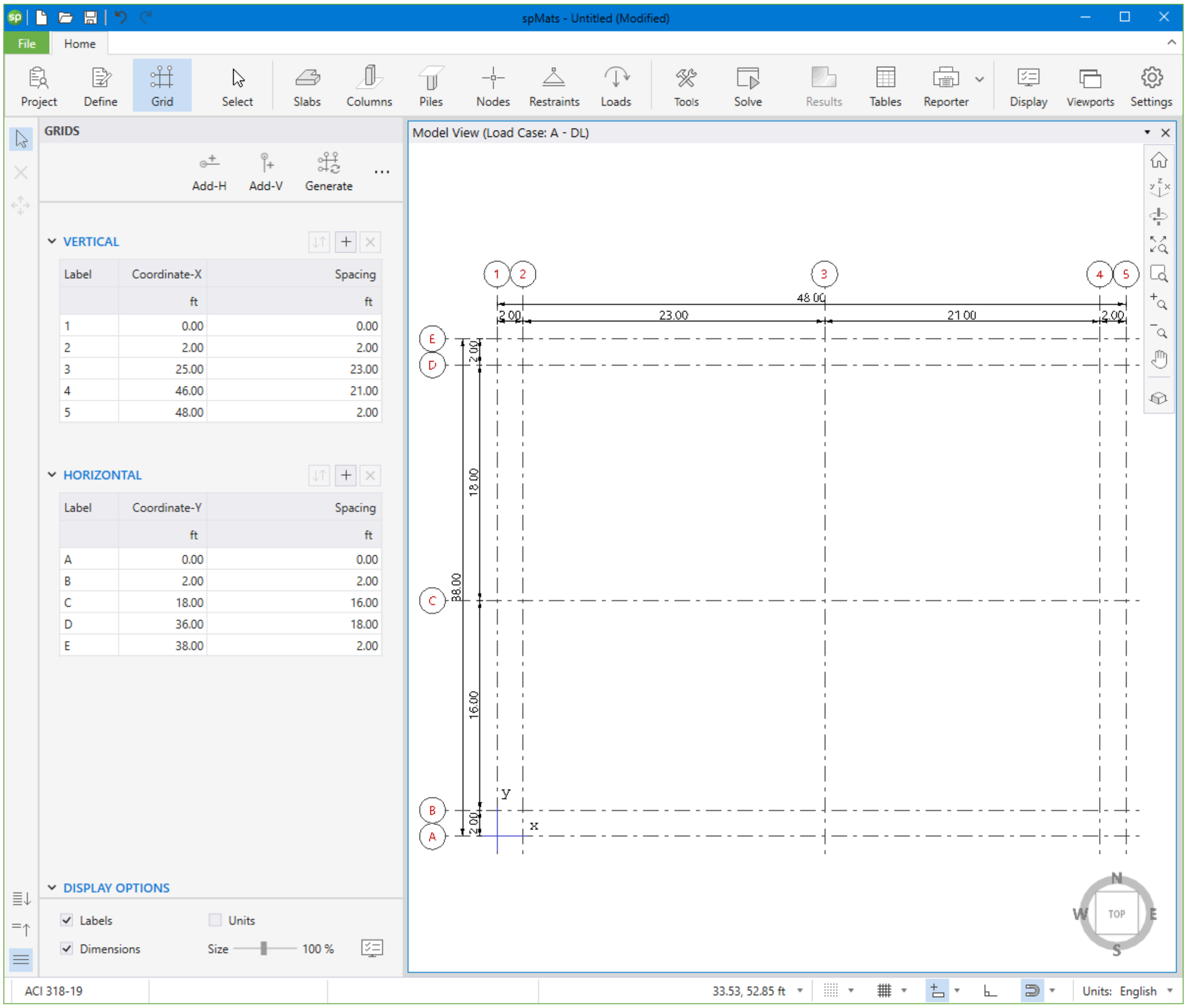
- Place a check mark in the X - VERTICAL box and enter the following values in the corresponding text boxes:

START COORDINATE - X:	0.0
GRID SPACING:	2 23 21 2

- Place a check mark in the Y - VERTICAL box and enter the following values in the corresponding text boxes:

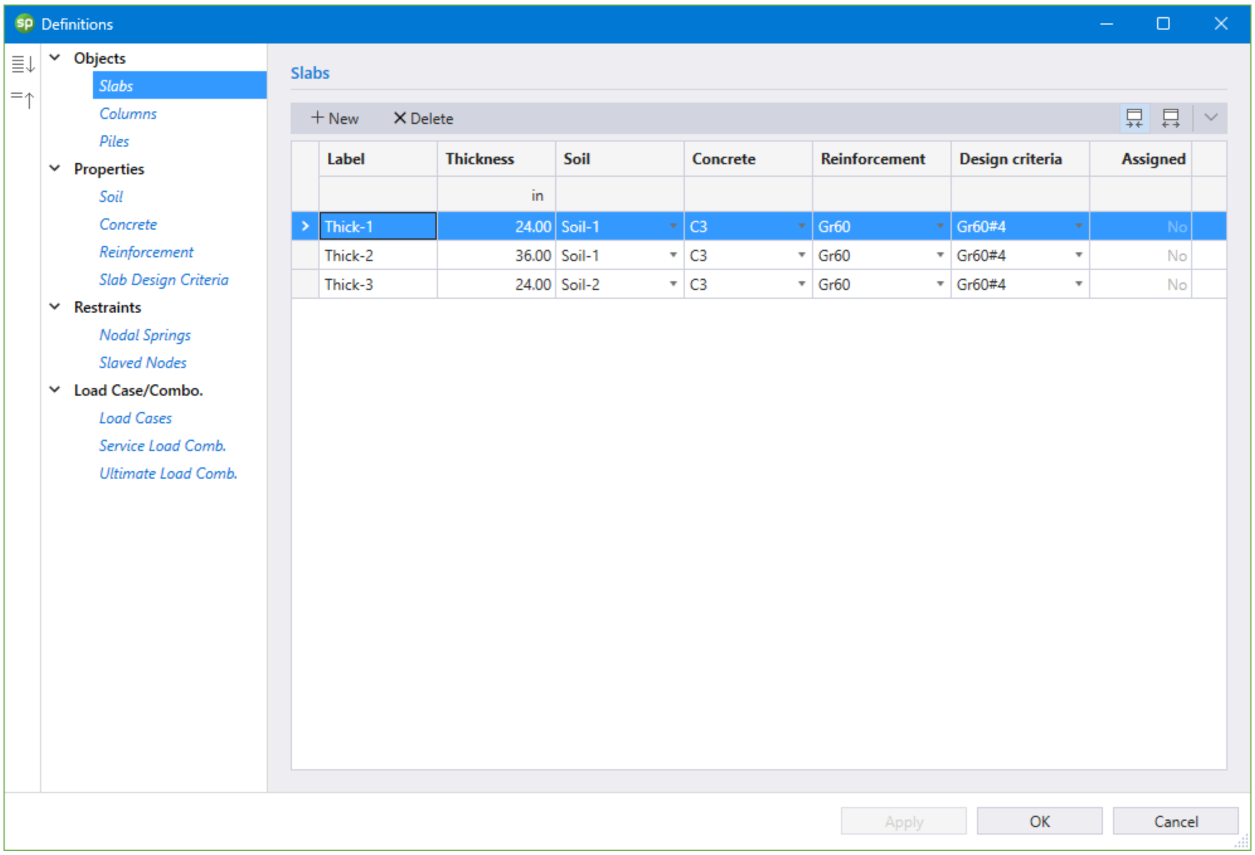
START COORDINATE - Y:	0.0
GRID SPACING:	2 16 18 2

- Click on the GENERATE button to return to the main window. Notice how the VERTICAL and HORIZONTAL grid lines now appear in the VIEWPORT.


spMats - Untitled (Modified)
File Home
Project Define Grid Select Slabs Columns Piles Nodes Restraints Loads Tools Solve Results Tables Reporter Display Viewports Settings
GRIDS
Add-H Add-V Generate
VERTICAL
Label Coordinate-X Spacing
ft ft
1 0.00 0.00
2 2.00 2.00
3 25.00 23.00
4 46.00 21.00
5 48.00 2.00
HORIZONTAL
Label Coordinate-Y Spacing
ft ft
A 0.00 0.00
B 2.00 2.00
C 18.00 16.00
D 36.00 18.00
E 38.00 2.00
DISPLAY OPTIONS
Labels Units
Dimensions Size 100 %
Model View (Load Case: A - DL)
48.00
2.00 23.00 21.00 2.00
38.00
2.00 18.00 16.00 2.00
ACI 318-19
33.53, 52.85 ft
Units: English

4. From the **Ribbon**, select **Define,** then choose **Slabs** from **Objects** to display the **Slabs** dialog box.

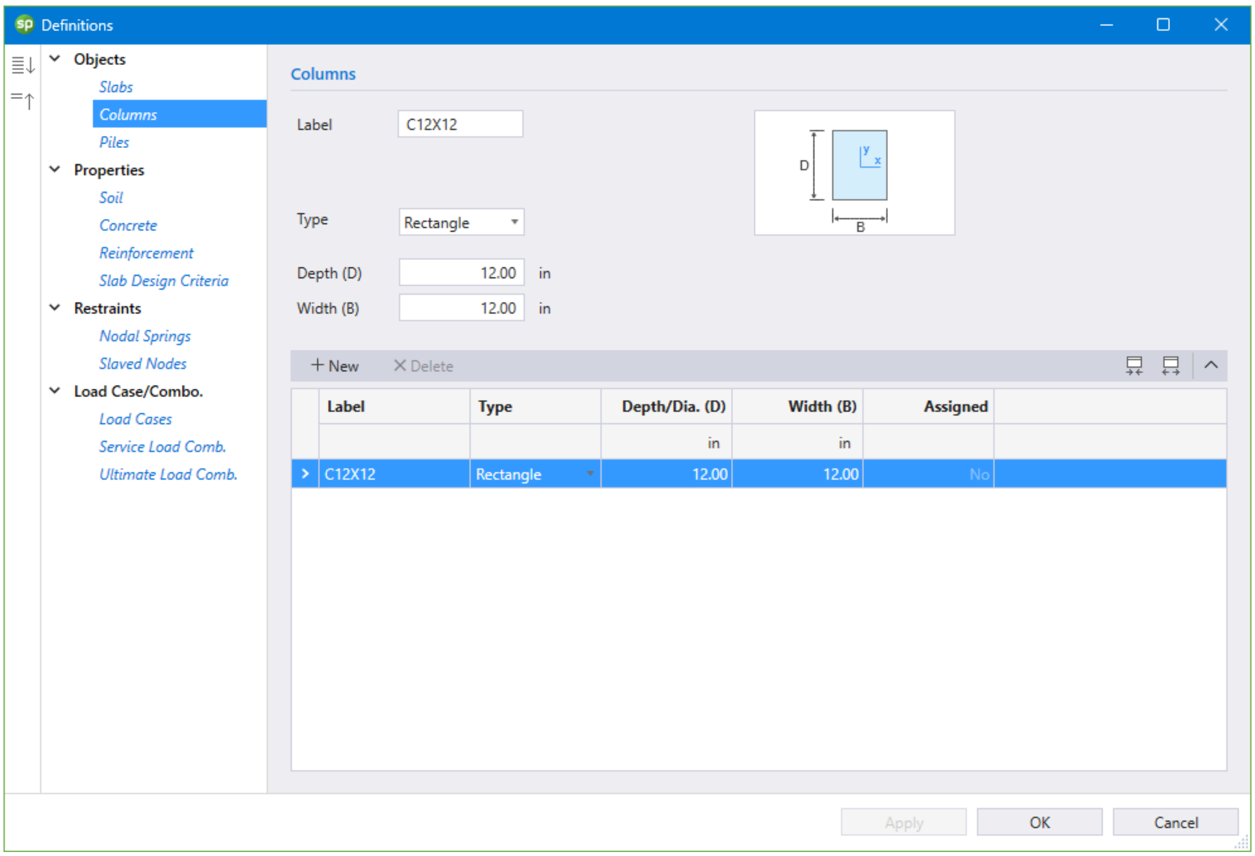
- Input `THICK-1` for LABEL and `24.00` in. for THICKNESS.
- Input `THICK-2` for LABEL and `36.00` in. for THICKNESS.
- Input `THICK-3` for LABEL and `24.00` in. for THICKNESS.



5. Click on **Columns** from **Objects** to display the **Columns** dialog box.

- Enter the following:

LABEL:	`C12X12`
TYPE:	Rectangle
DEPTH/DIAMETER (D):	`12.00` in.
WIDTH (B):	`12.00` in.



6. Click on **Soil** from **Properties** to display the **Soil** dialog box.

   - Enter the following:

     LABEL: `SOIL-1`

     SUBGRADE MODULUS: `50.00` kcf

     ALLOWABLE PRESSURE: `6.00` ksf

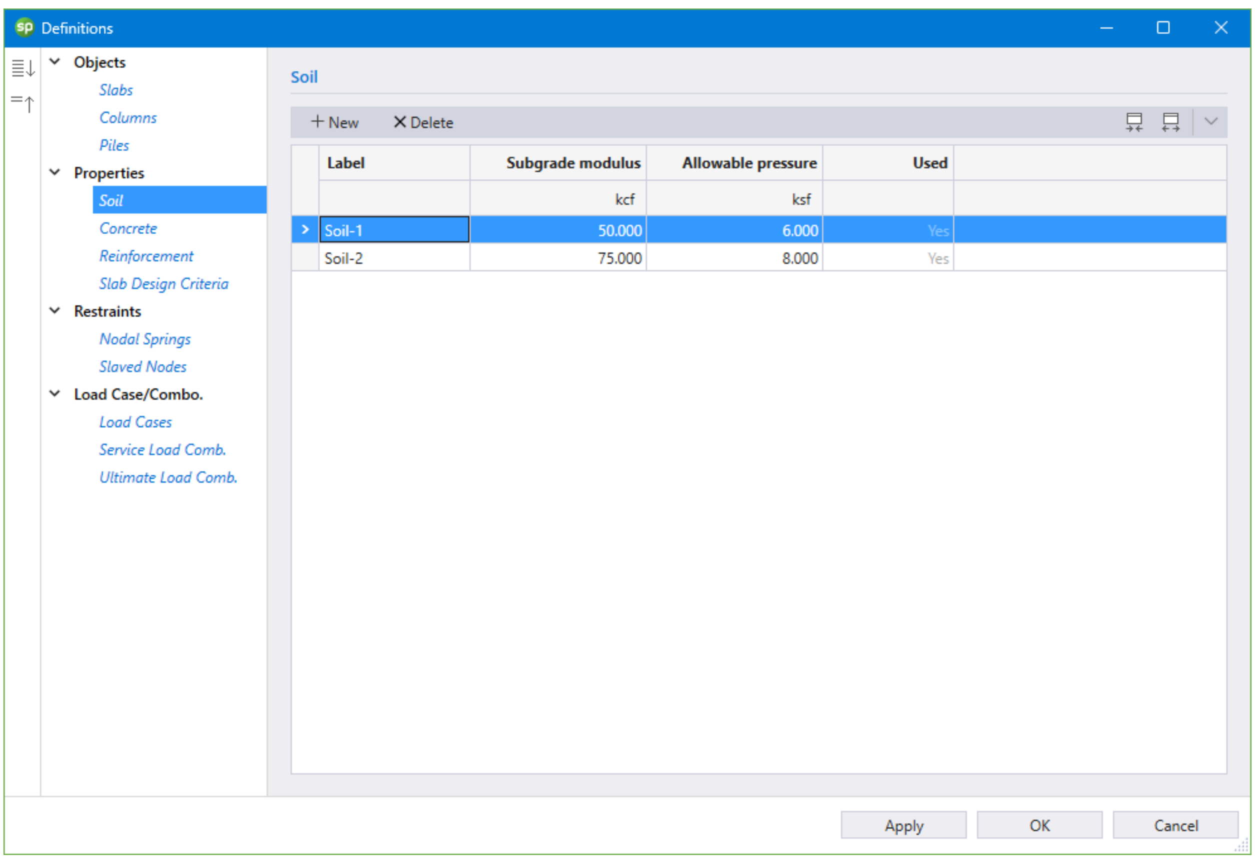
   - Click on the NEW button to add a new entry to the list:

   - Enter the following:

     LABEL: `SOIL-2`

     SUBGRADE MODULUS: `75.00` kcf

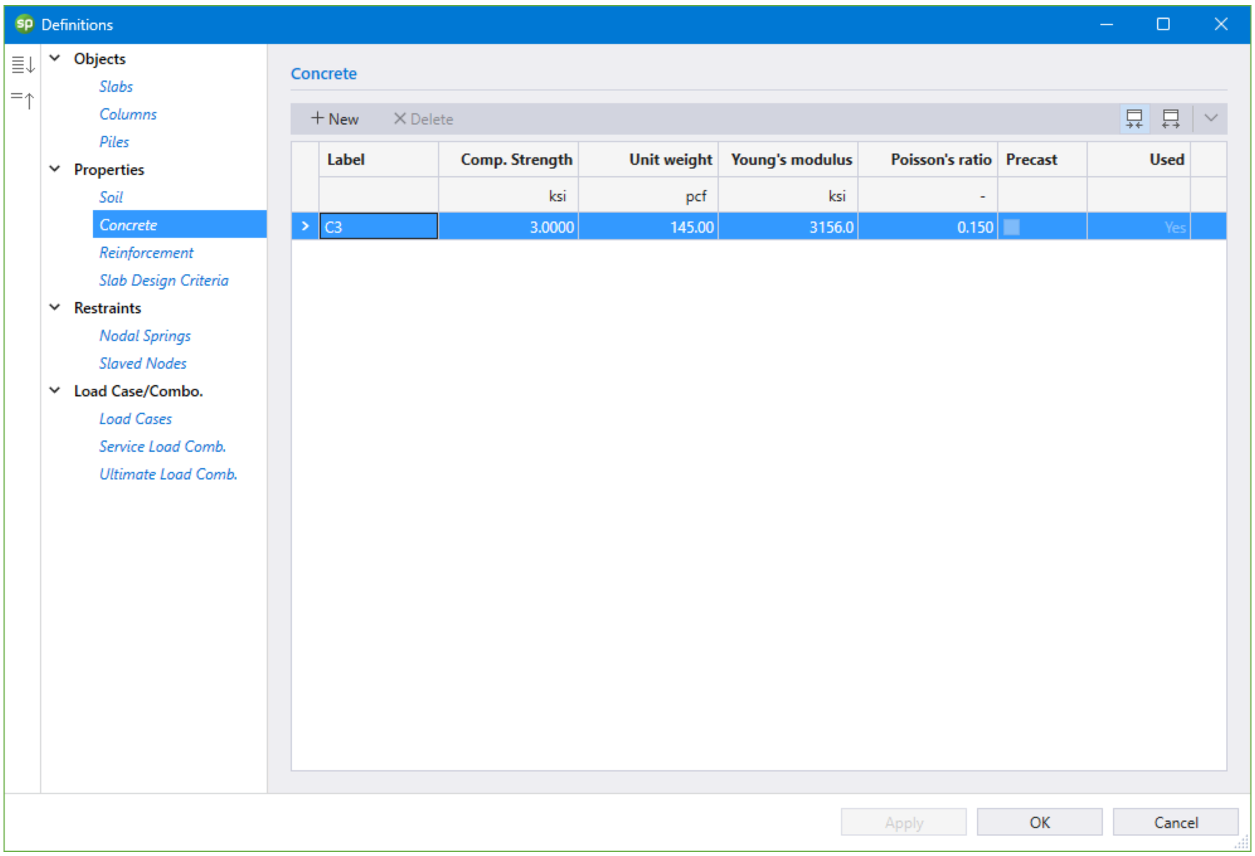
     ALLOWABLE PRESSURE: `8.00` ksf



7. Click on **Concrete** from **Properties** to display the **Concrete** dialog box.

- Enter the following:

LABEL:	C3
COMPRESSIVE STRENGTH:	3.00 ksi
UNIT WEIGHT:	145.00 pcf
YOUNG'S MODULUS:	3156.00 ksi
POISSON'S RATIO:	0.15



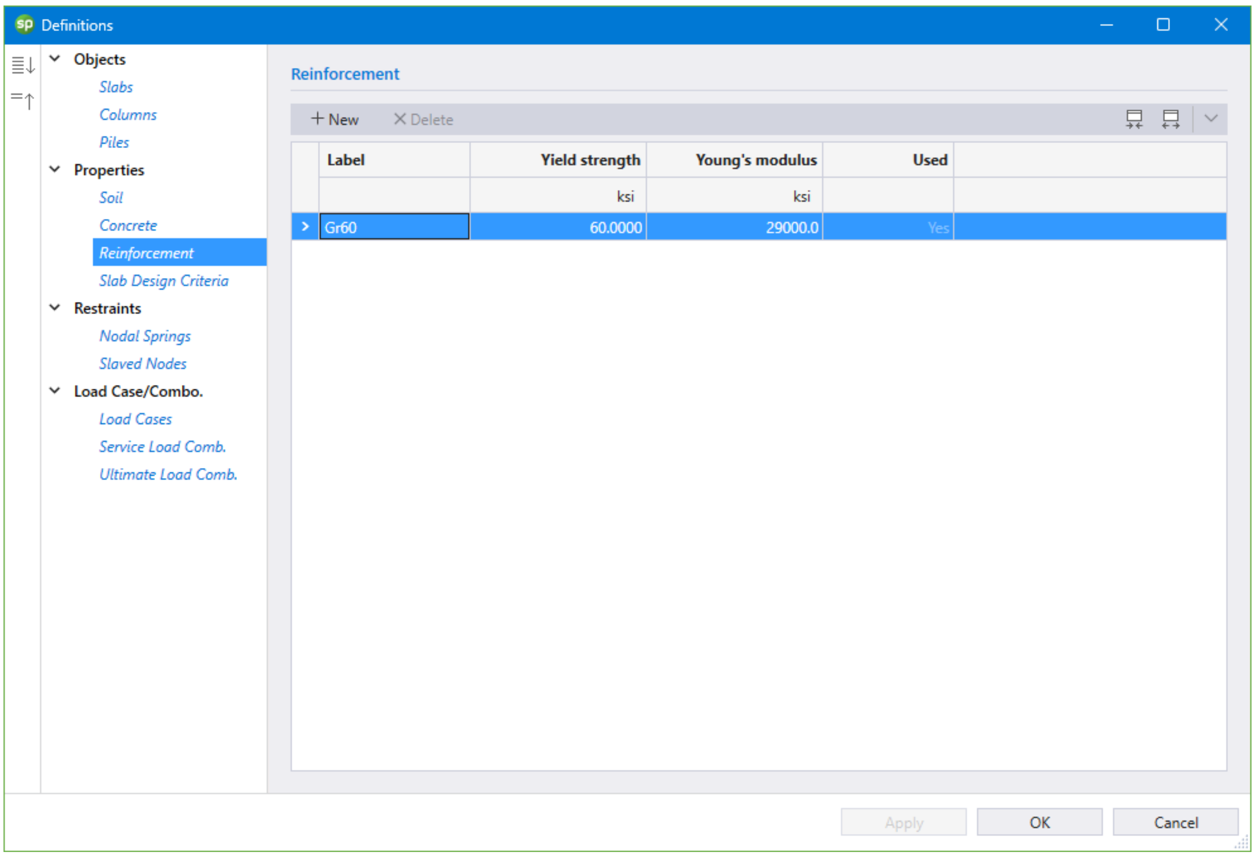
8. Click on **Reinforcement** from **Properties** to display the **Reinforcement** dialog box.

- Enter the following:

LABEL: `Gr60`

YIELD STRENGTH: `60.00` ksi

YOUNG'S MODULUS: `29000.00` ksi



9. Click on **Slab Design Criteria** from **Properties** to display the **Slab Design Criteria** dialog box.

   - Enter the following:

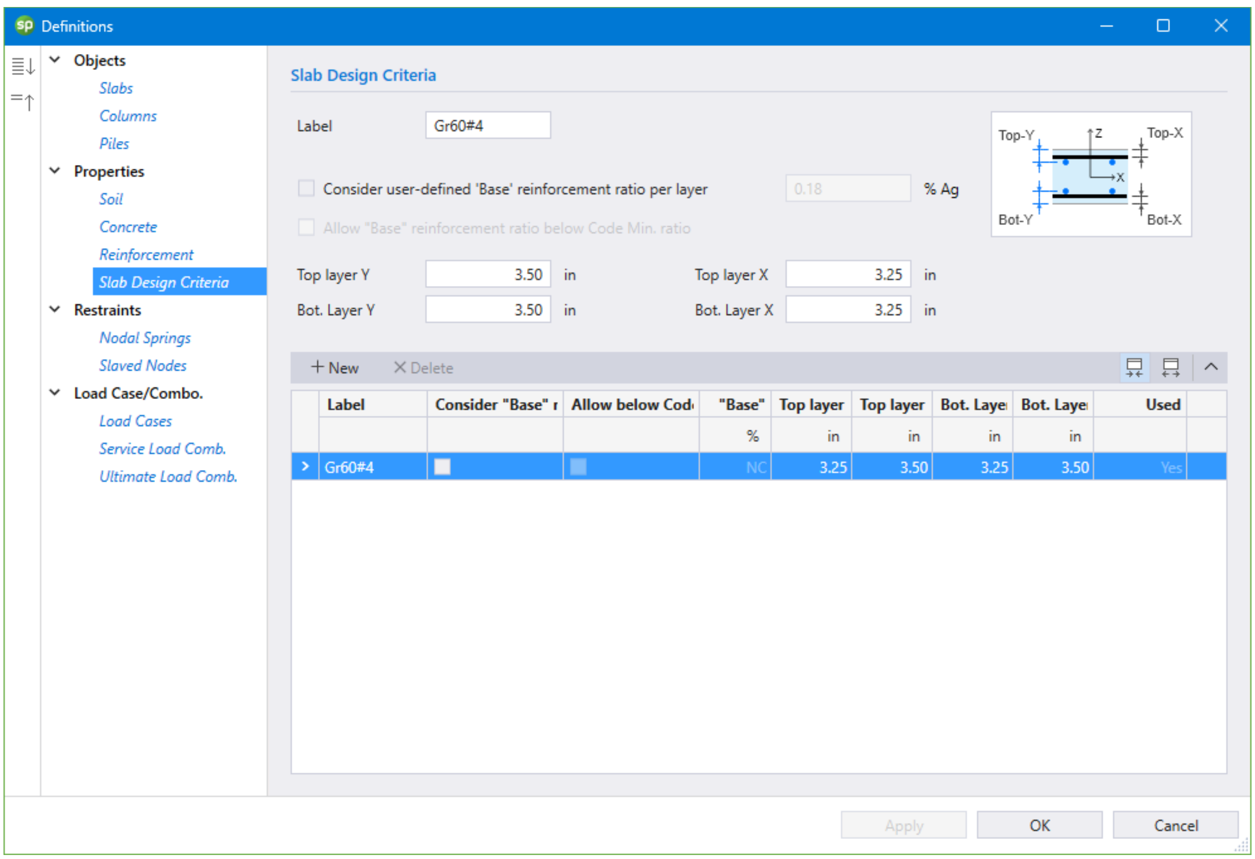
     LABEL: Gr60#4

     TOP LAYER Y: 3.50 in.

     BOTTOM LAYER Y: 3.50 in.

     TOP LAYER X: 3.25 in.

     BOTTOM LAYER X: 3.25 in.

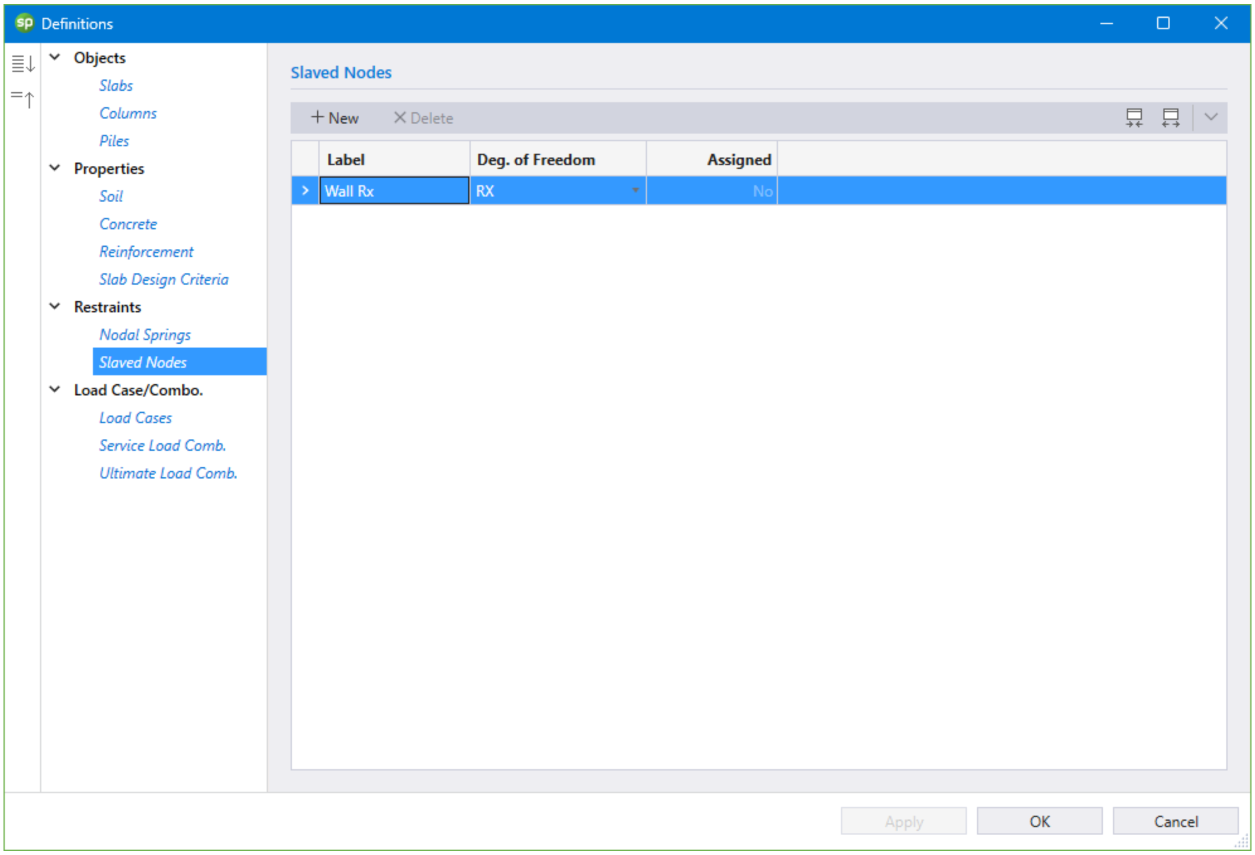


10. Click on **Slaved Nodes** from **Restraints** to display the **Slaved Nodes** dialog box.

- Enter the following:

LABEL: `Wall Rx`

DEGREE OF FREEDOM: Rx



11. Click on **Load Cases** from **Load Case/Combo.** to display the **Load Cases** dialog box.

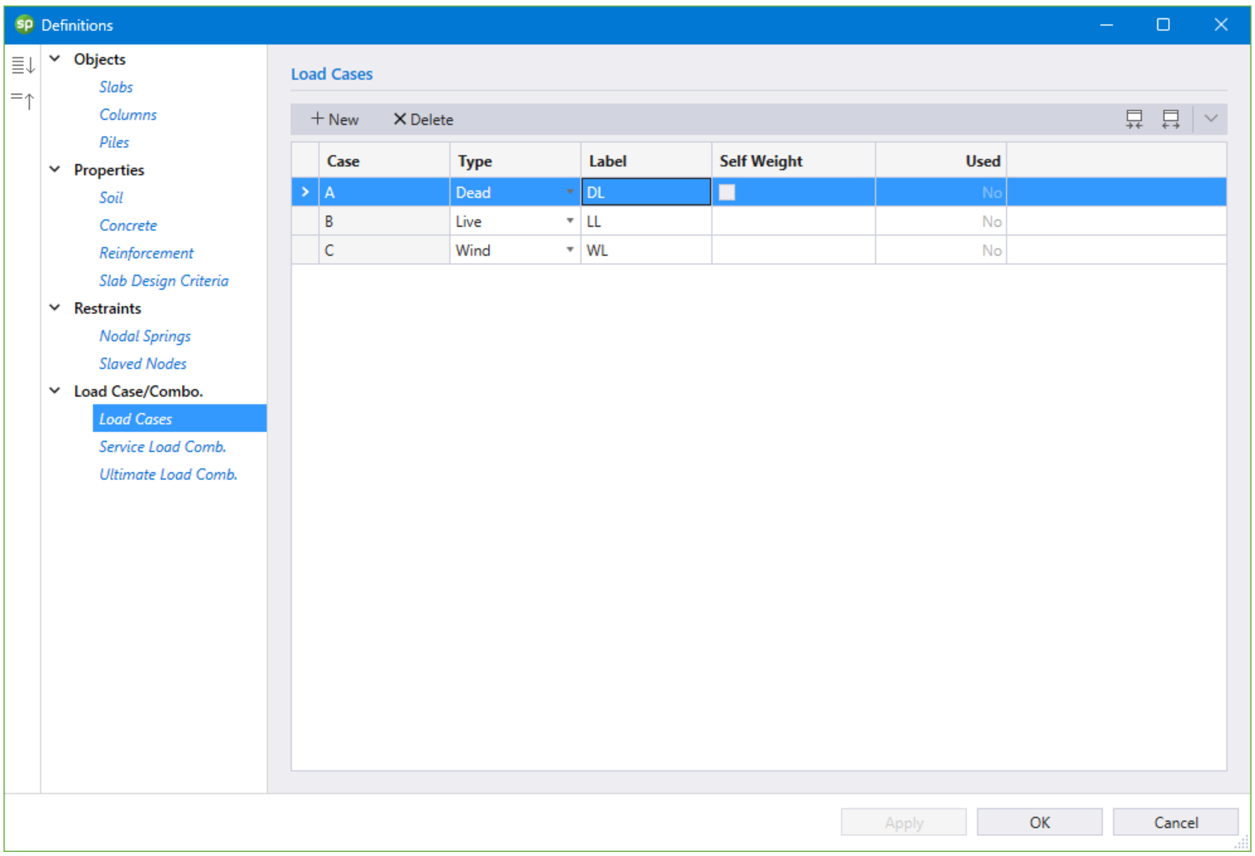
- Enter the following:

  CASE A: DL

  CASE B: LL

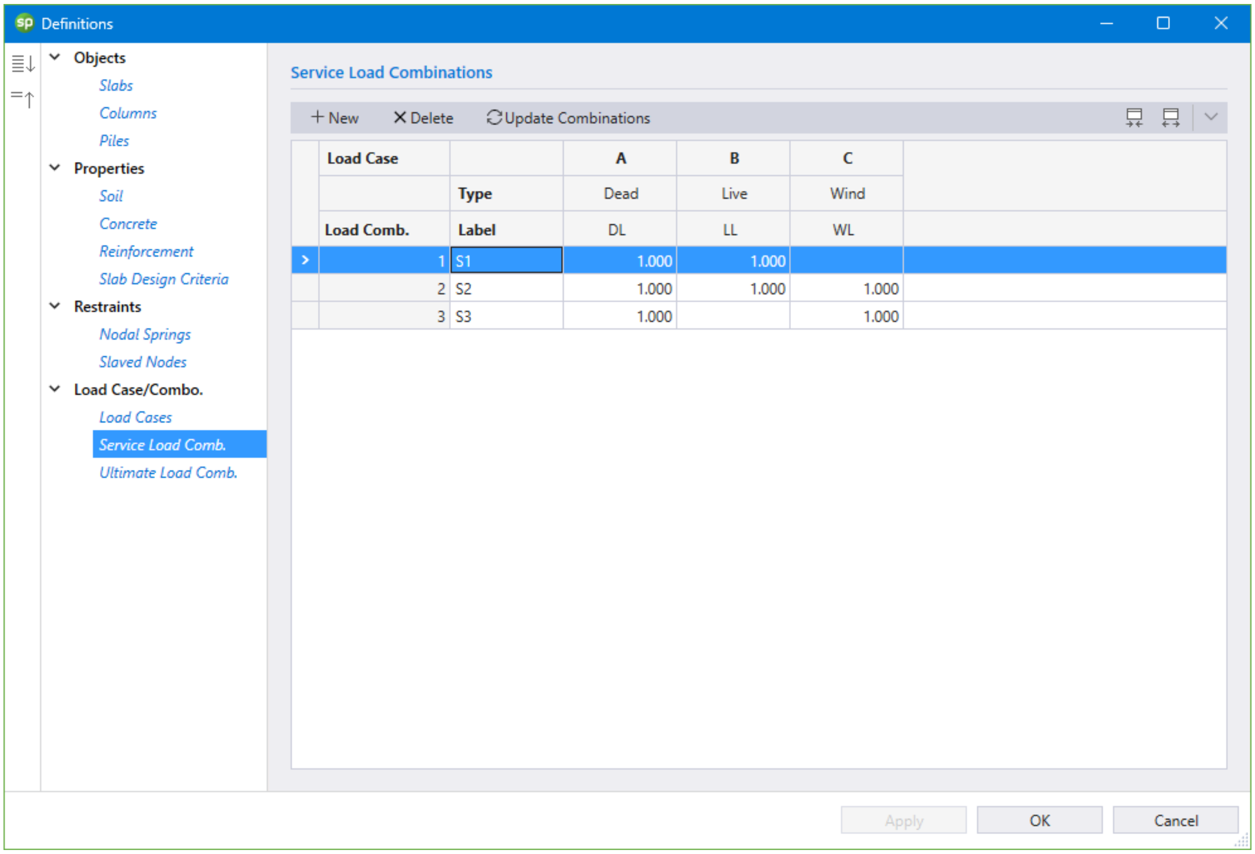
  CASE C: WL

- Uncheck SELF WEIGHT for CASE A.



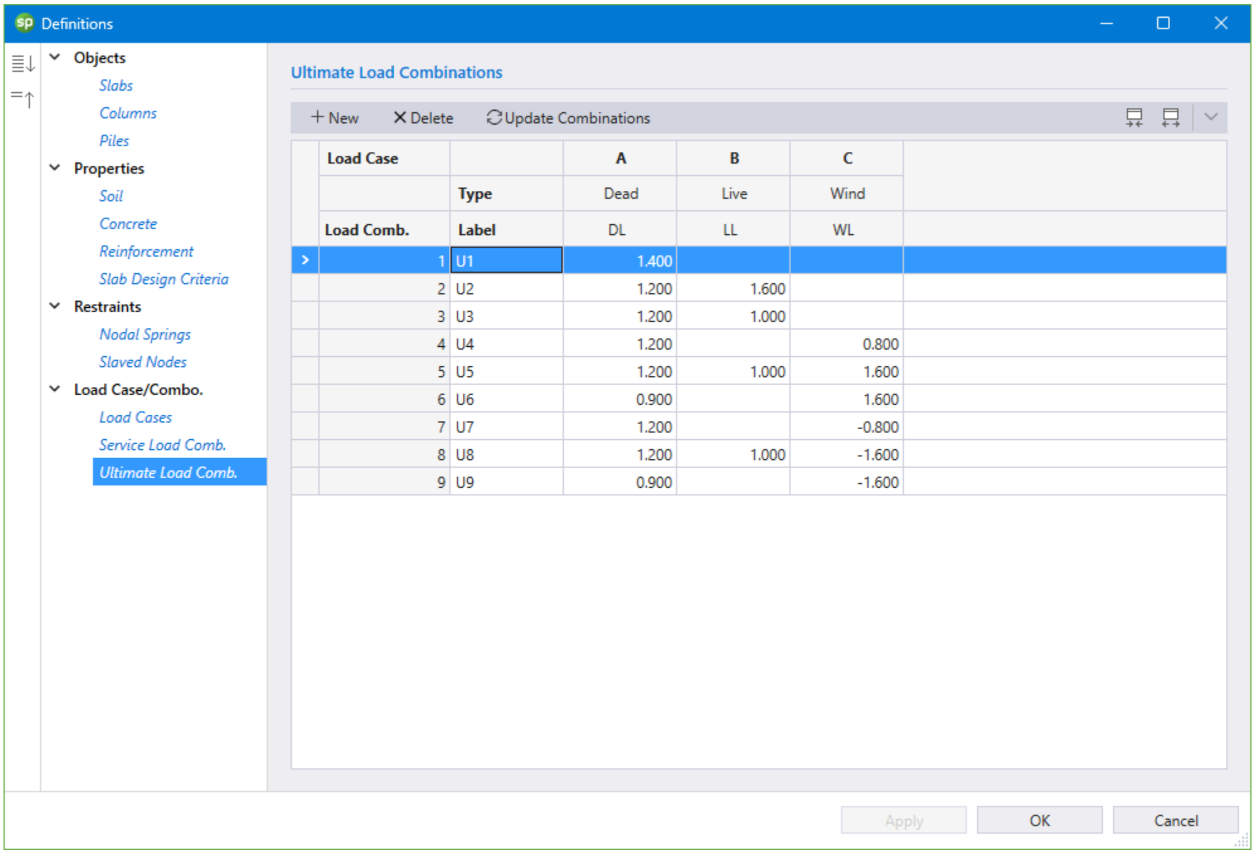
12. Click on **Service Load Combinations** from **Load Case/Combo.** to display the **Service Load Combinations** dialog box.

- Enter the following service load combinations shown in the figure below:



13. Click on **Ultimate Load Combinations** from **Load Case/Combo.** to display the **Ultimate Load Combinations** dialog box.

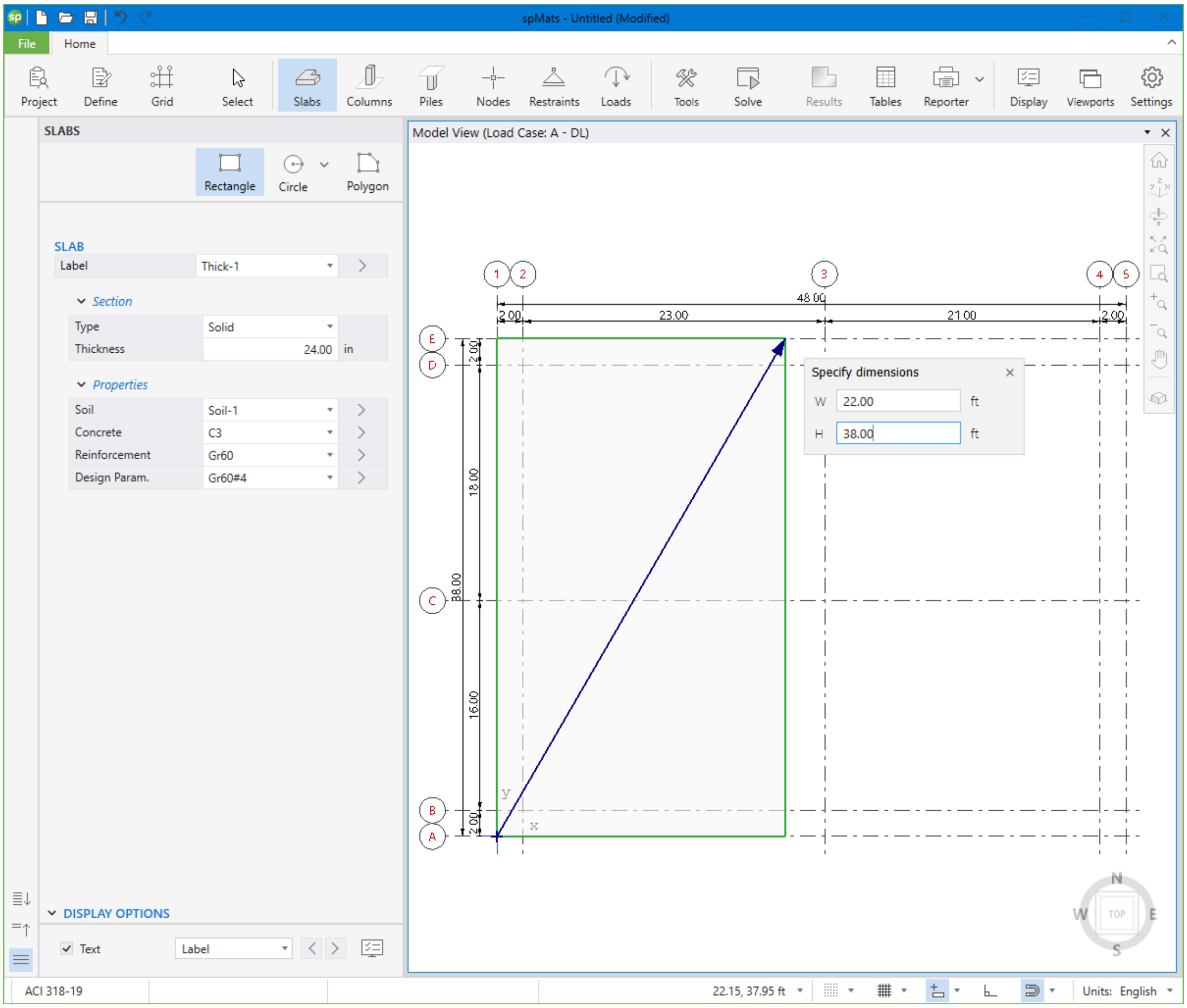
- Enter the following load combinations shown in the figure below:



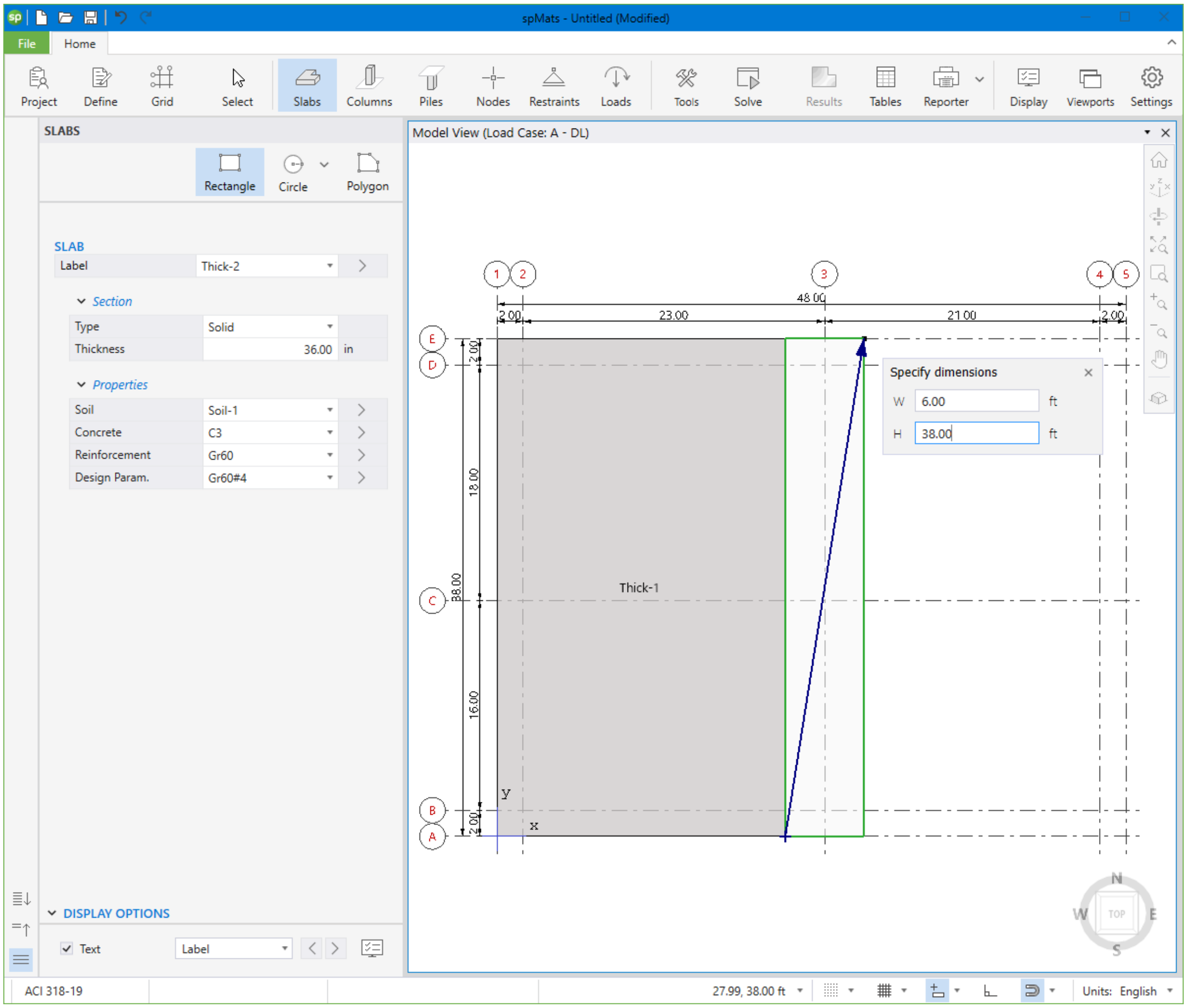
8.2.3. Assigning Properties

14. From the **Ribbon**, select **Slabs** command.

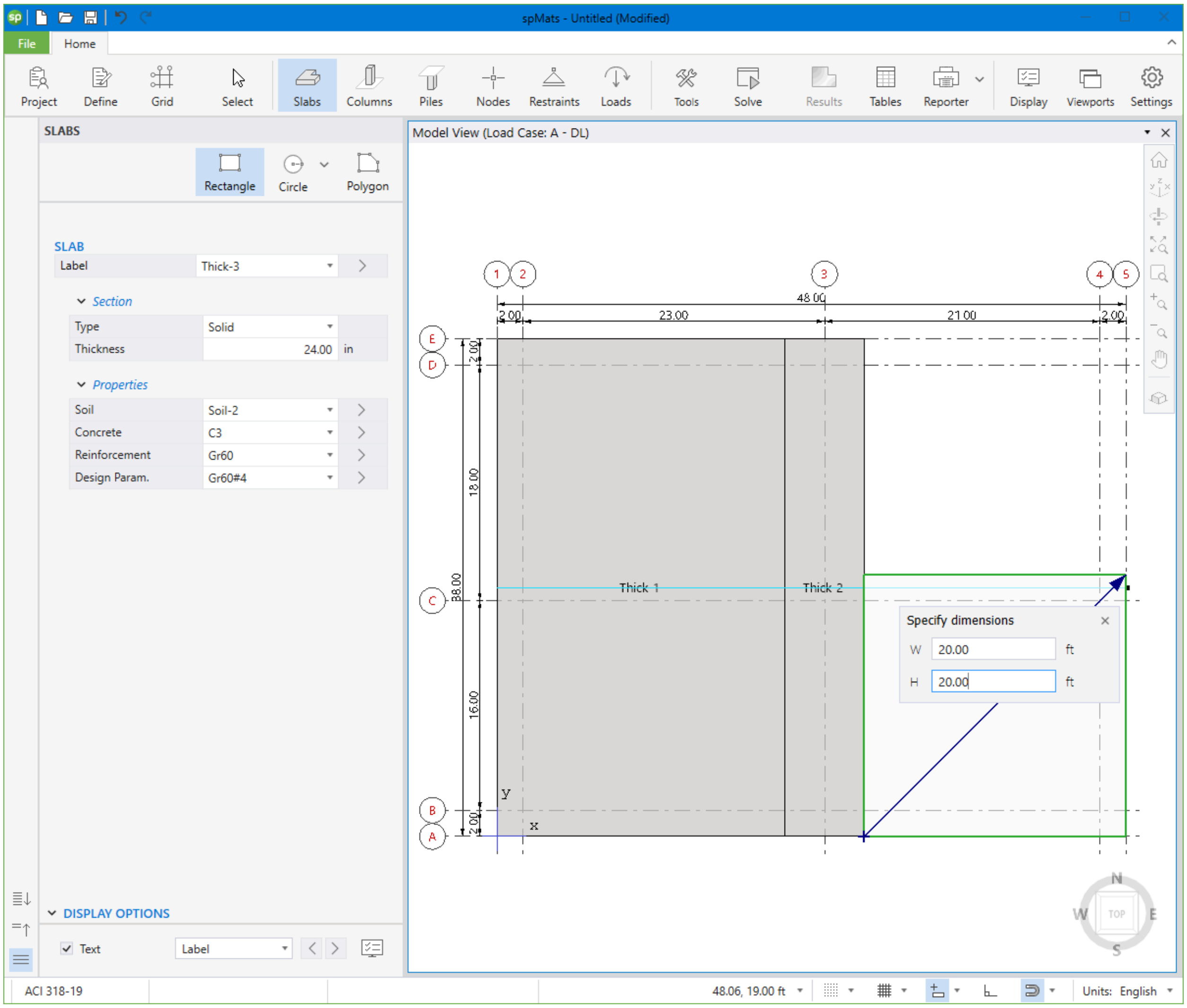
- In the left panel, select **Rectangle** then select THICK-1 from LABEL.
- In the VIEWPORT, marquee-select the region (0, 0) – (22, 38) to apply the selected slab to that part of the footing.



- Select THICK-2 from LABEL.
- In the VIEWPORT, marquee-select the region (22, 0) – (6, 38) to apply the selected slab to that part of the footing.



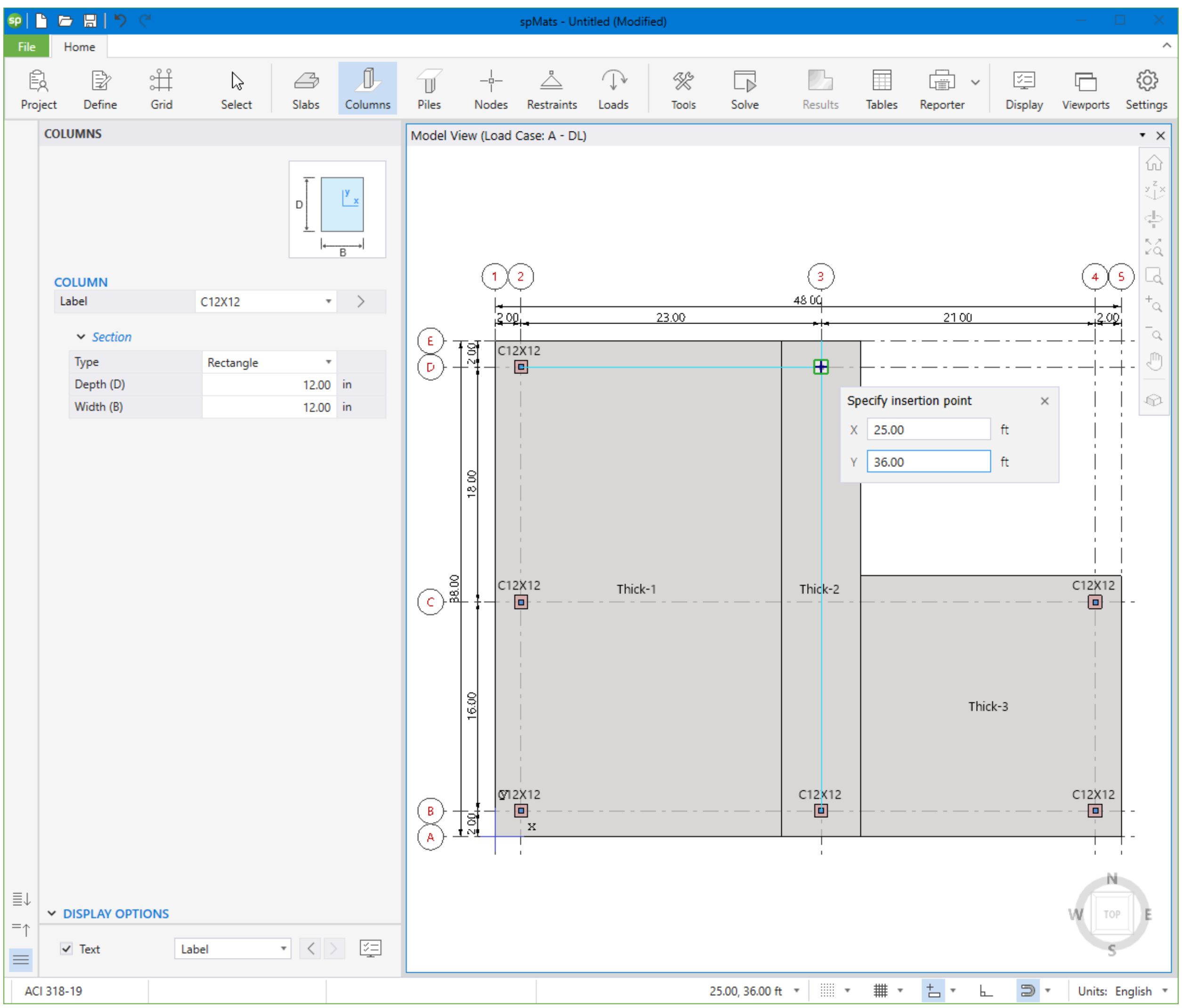
- Select THICK-3 from LABEL.
- In the VIEWPORT, marquee-select the region (28, 0) – (20, 20) to apply the selected slab to that part of the footing.



15. From the **Ribbon**, select **Columns** command.

- In the left panel, select C12X12 from LABEL.
- In the VIEWPORT, enter the coordinates of each node using the dynamic input box (to activate the dynamic input box simply start typing):

NODE 1: (2, 2)	NODE 2: (25, 2)
NODE 3: (46, 2)	NODE 4: (2, 18)
NODE 5: (46, 18)	NODE 6: (2, 36)
NODE 7: (25, 36)	



16. From the **Ribbon**, select **Nodes** command.

- In the left panel, select **Single**.
- In the VIEWPORT, enter the coordinates of each node using the dynamic input box (to activate the dynamic input box simply start typing):

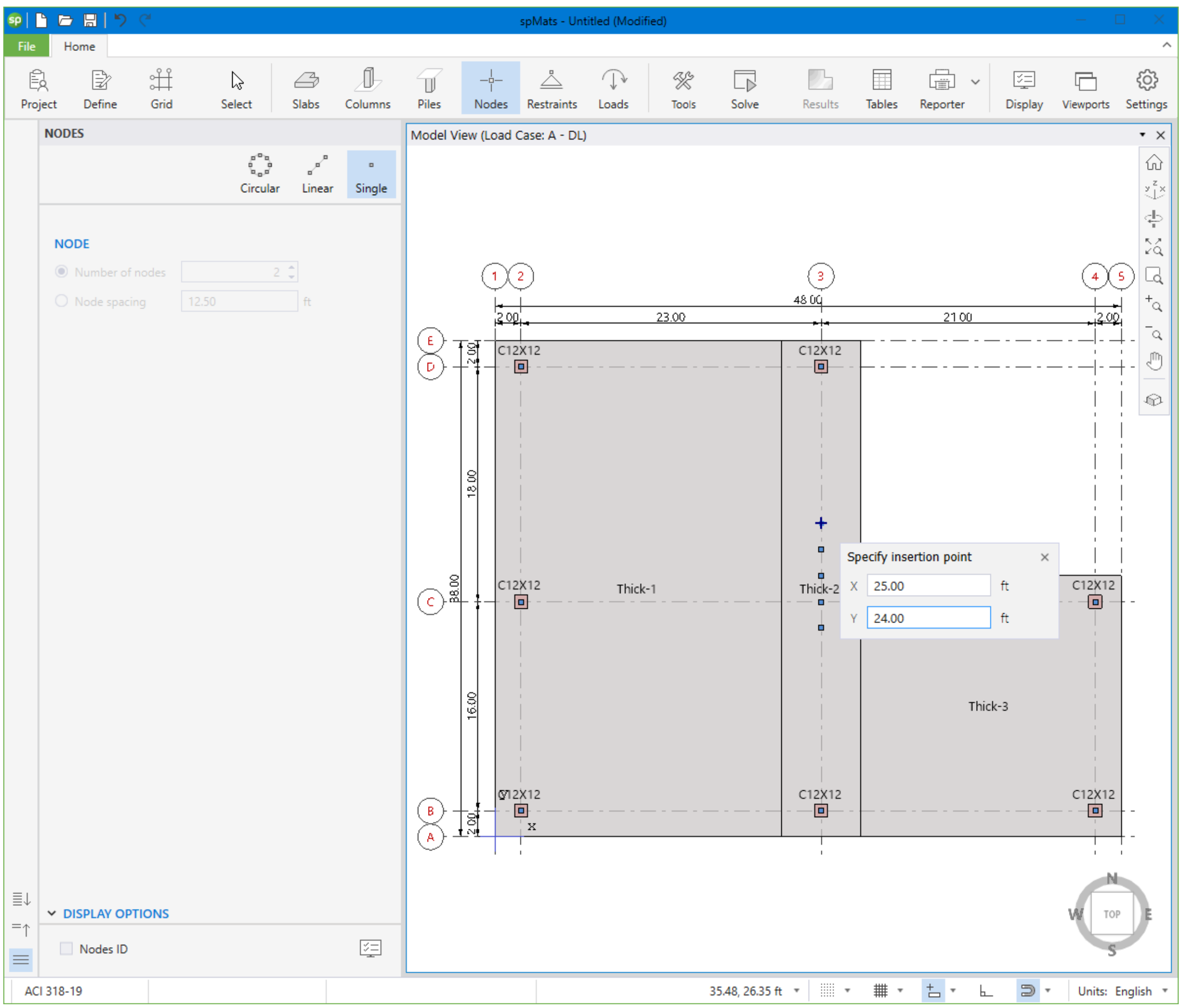
NODE 1: (25, 16)

NODE 2: (25, 18)

NODE 3: (25, 20)

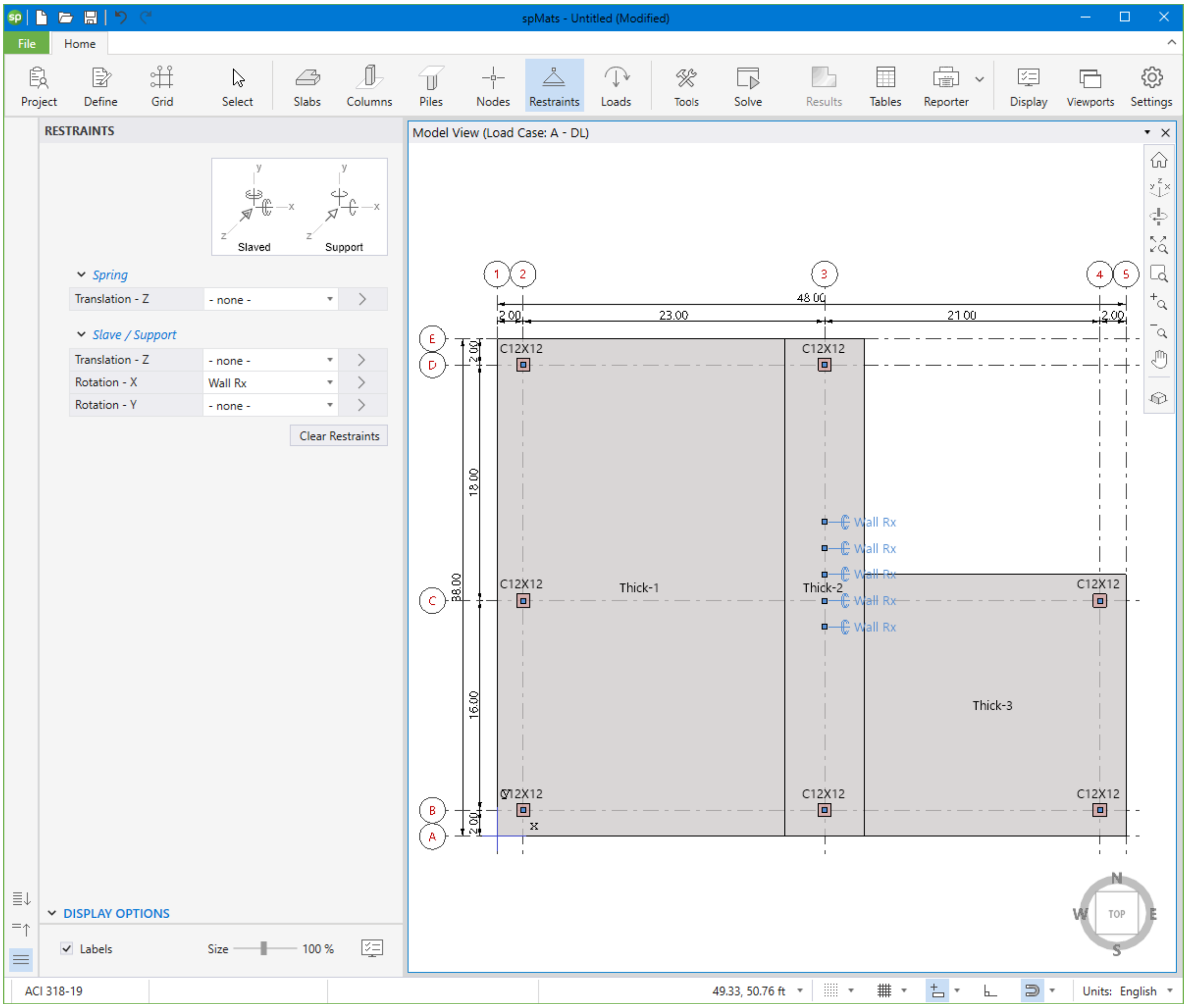
NODE 4: (25, 22)

NODE 5: (25, 24)



17. From the **Ribbon**, select **Restraints** command.

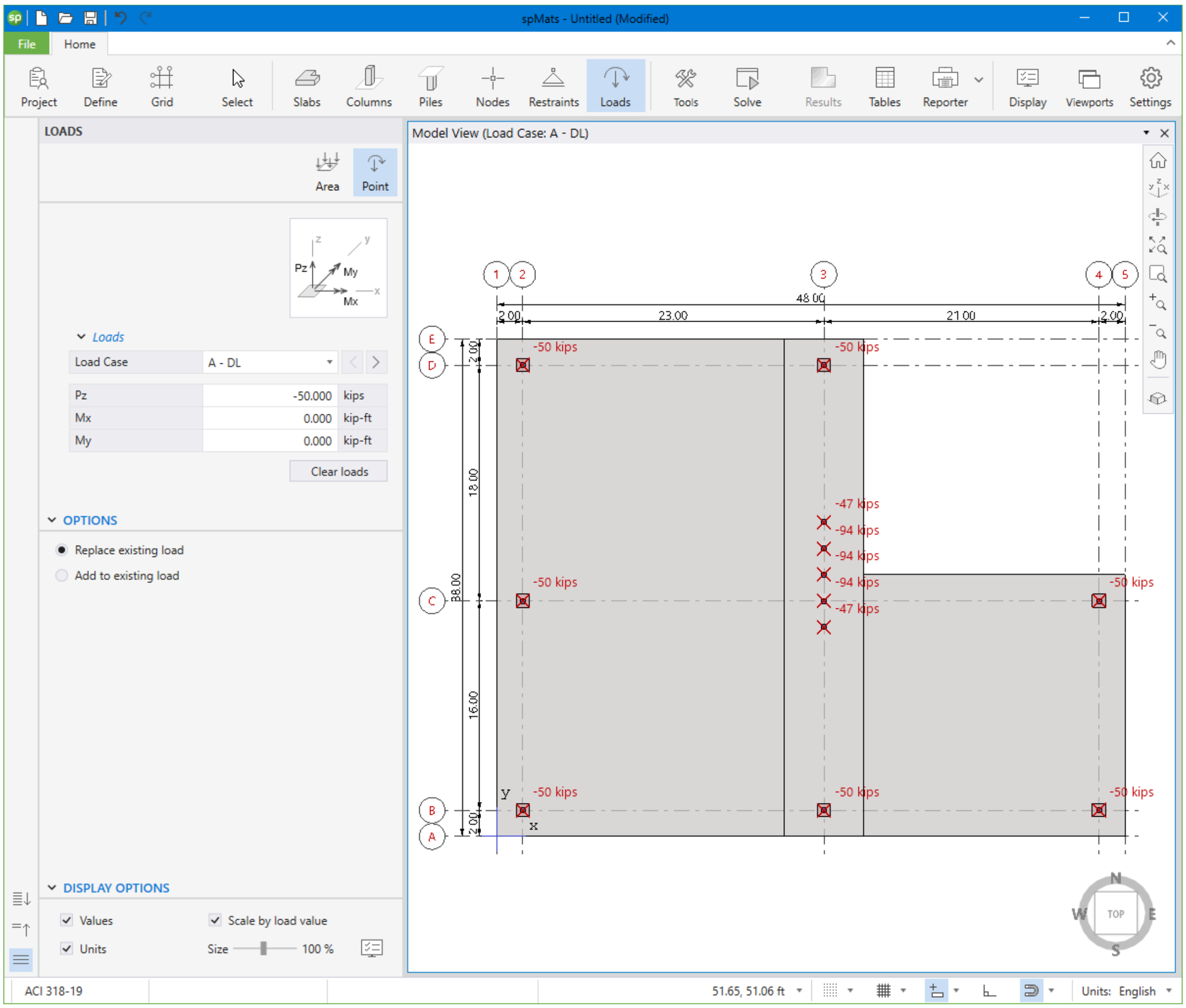
- In the left panel, select WALL RX from ROTATION-X for SLAVE/SUPPORT.
- Apply WALL RX slave/support to all wall nodes as shown in the figure below.



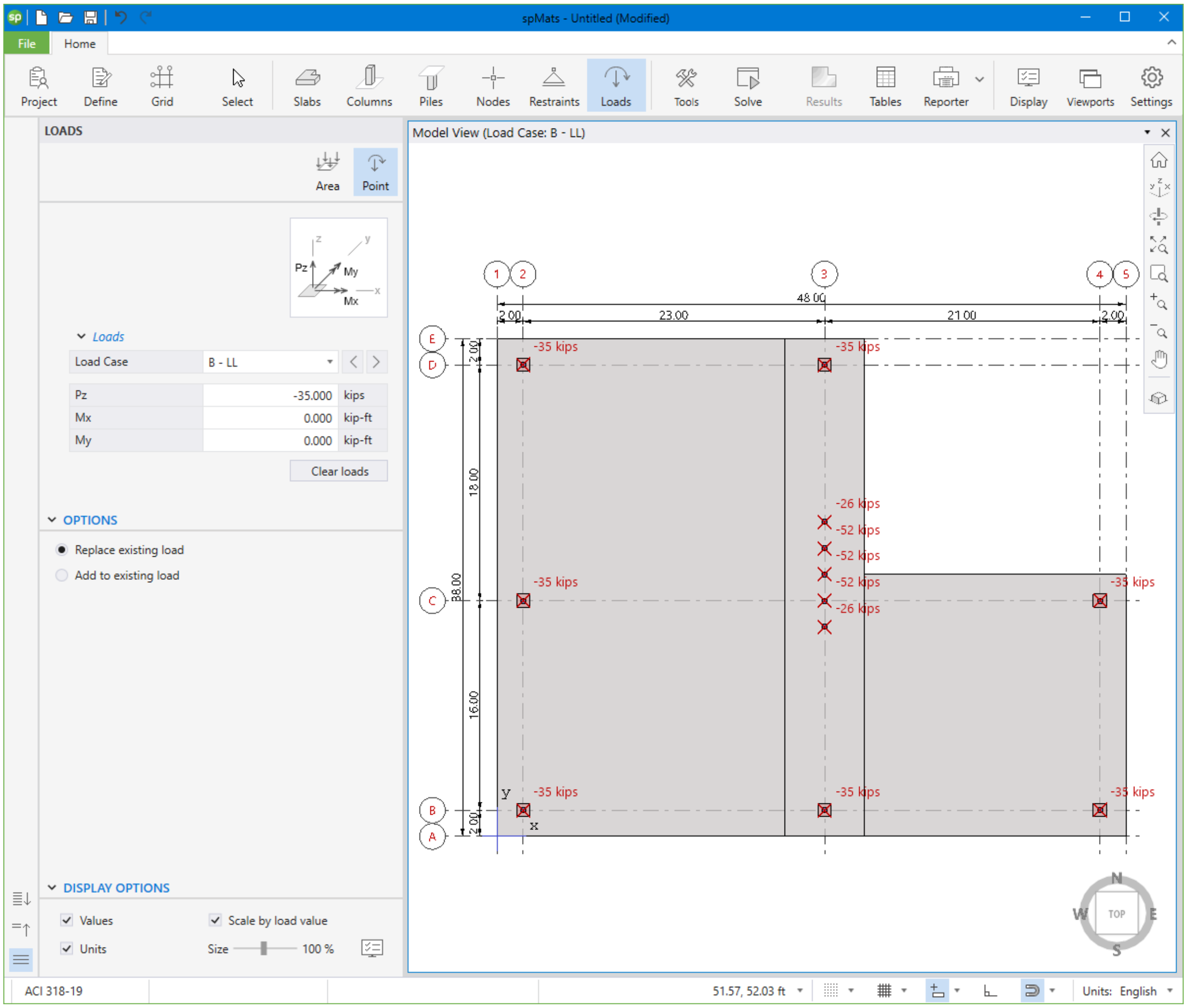
8.2.4. Assigning Loads

18. From the **Ribbon**, select **Loads** command.

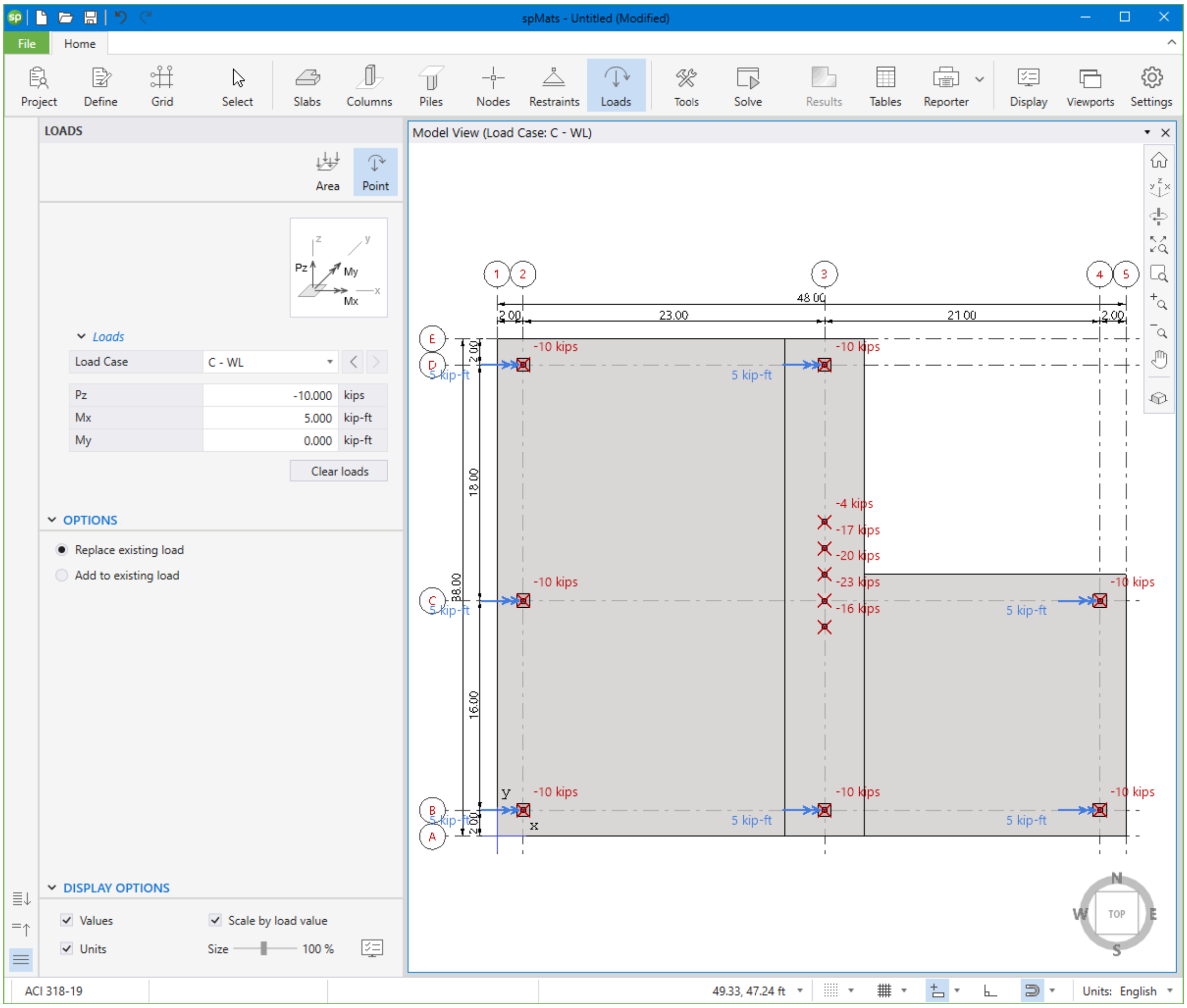
- In the left panel, select **Point** then select A-DL from LOAD CASE and enter the loads as shown in the figure below.



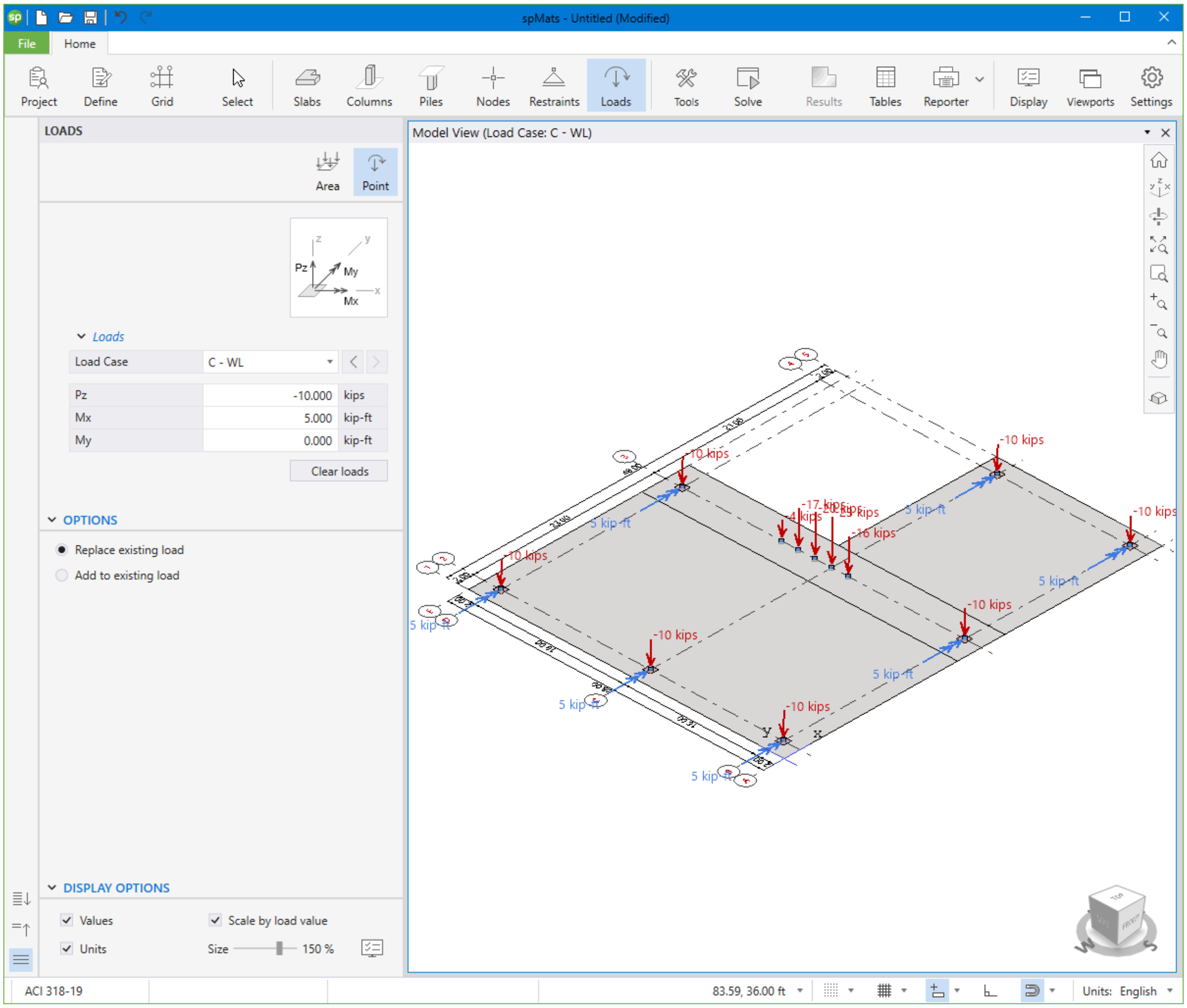
- In the left panel, select B-LL from LOAD CASE and enter the loads as shown in the figure below.



- In the left panel, then select C-WL from LOAD CASE and enter the loads as shown in the figure below.



- Also, you can click on the 3D VIEW icon from **View Controls** (top right of the active viewport) to get a better view of the applied loads.

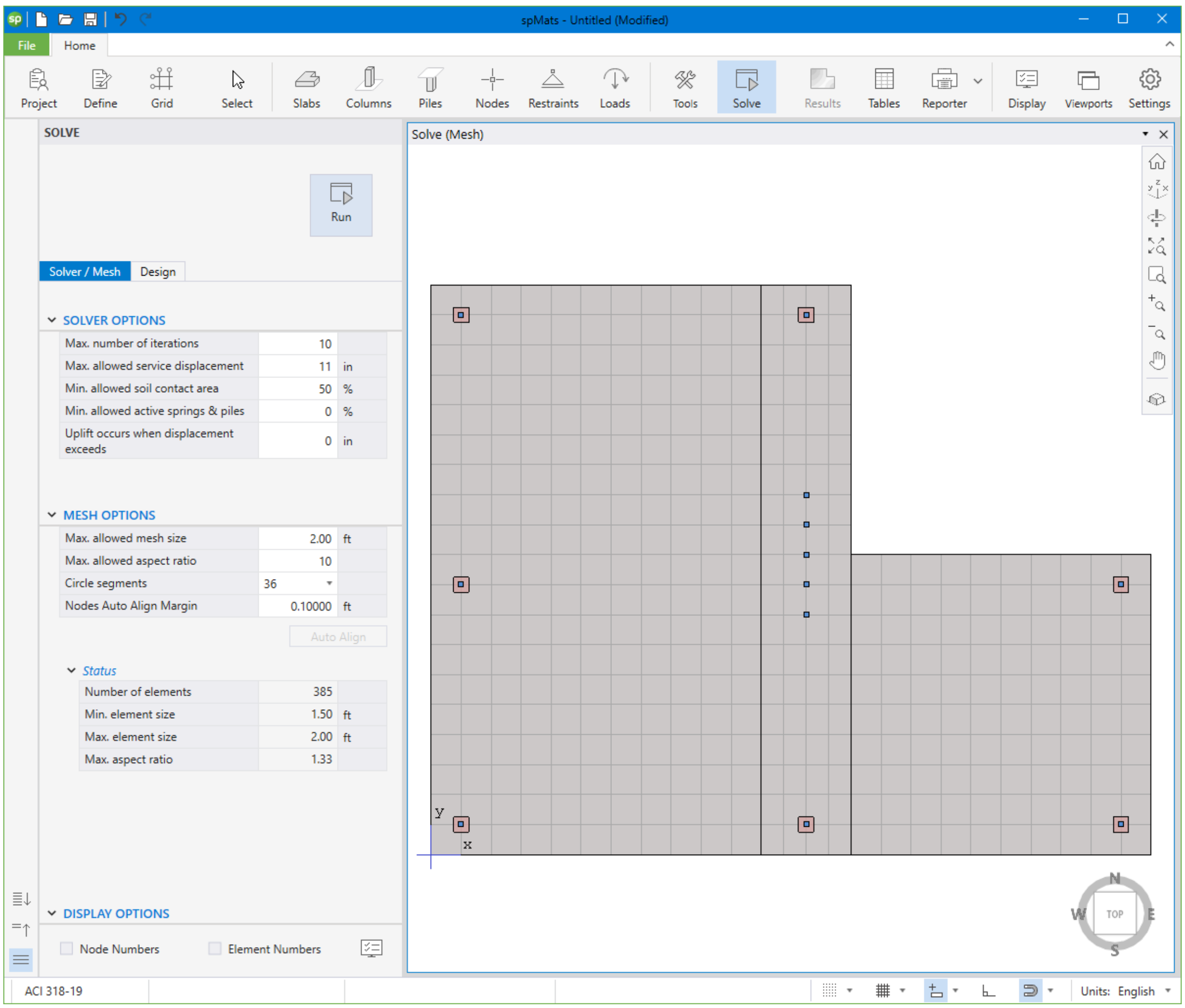


8.2.5. Solving

19. From the **Ribbon**, select **Solve** command.

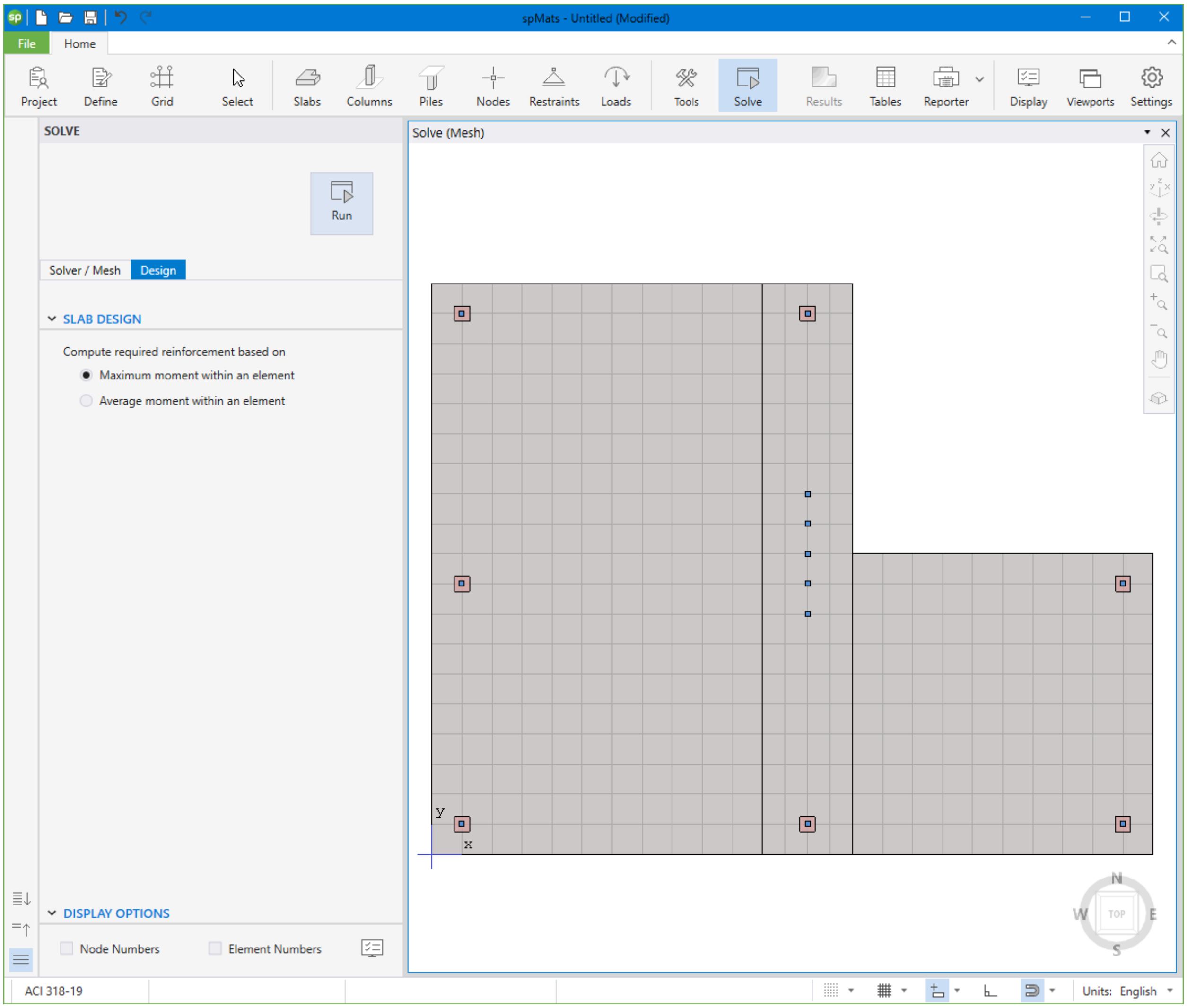
    For **Solver / Mesh Options**:

    - Leave all **Solver Options** to their default settings.
    - Leave all **Mesh Options** to their default settings.

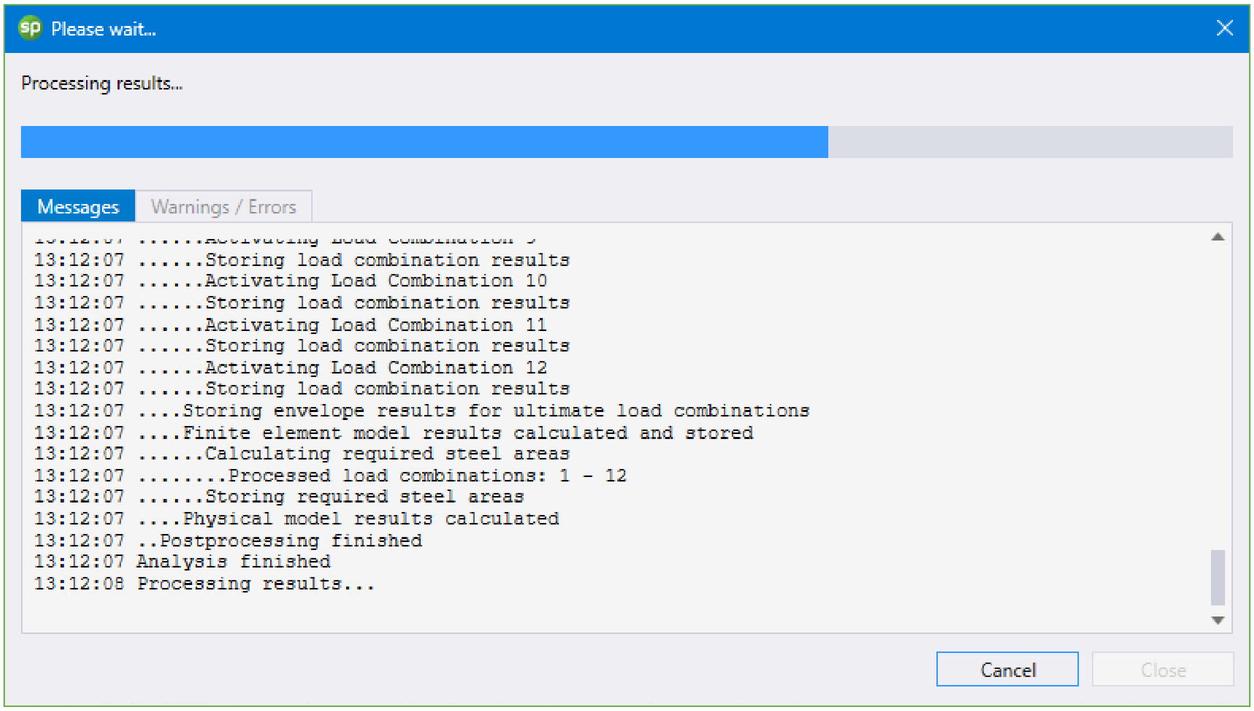


For **Design Options**:

- Leave all **Slab Design Options** to their default settings.

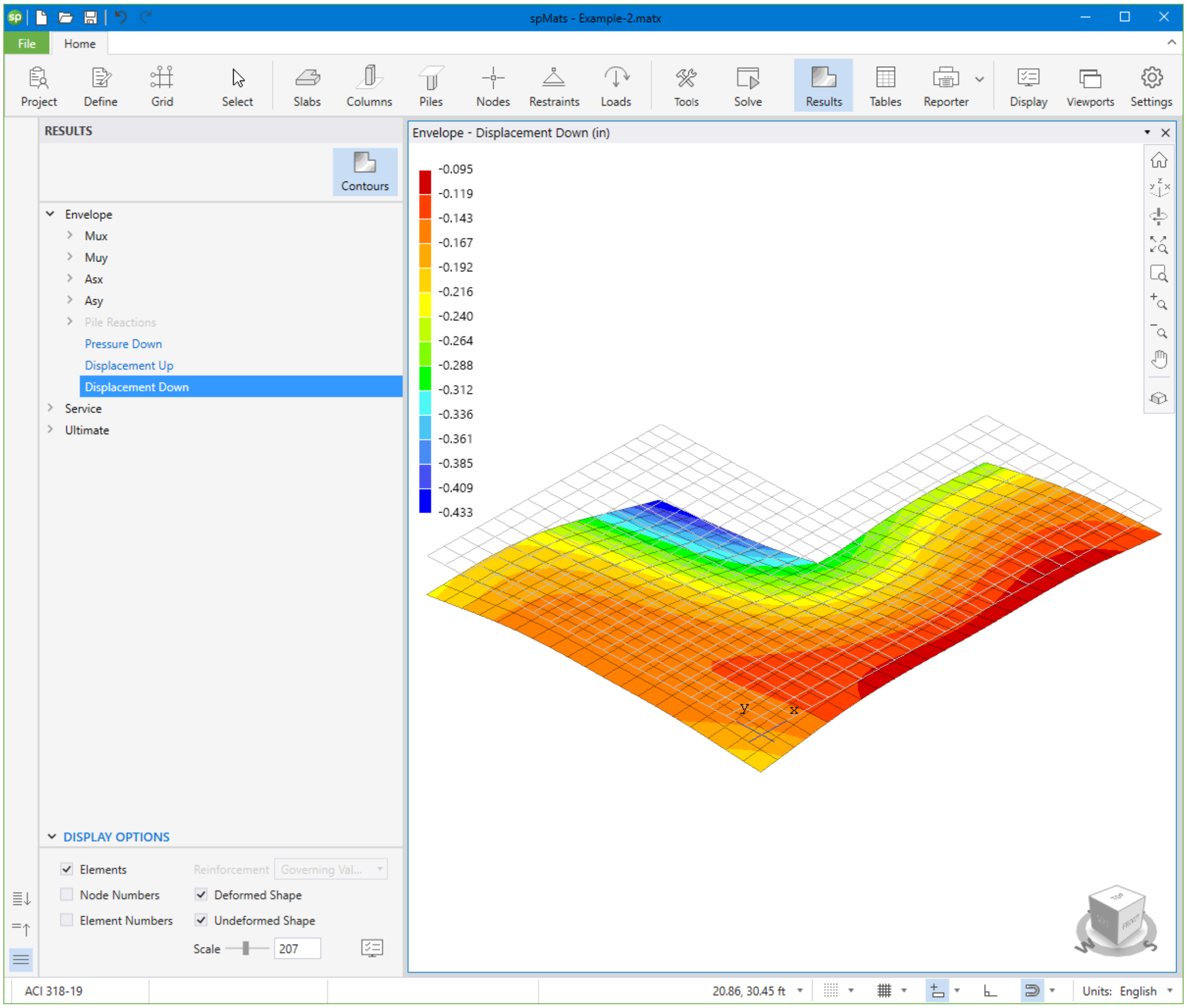


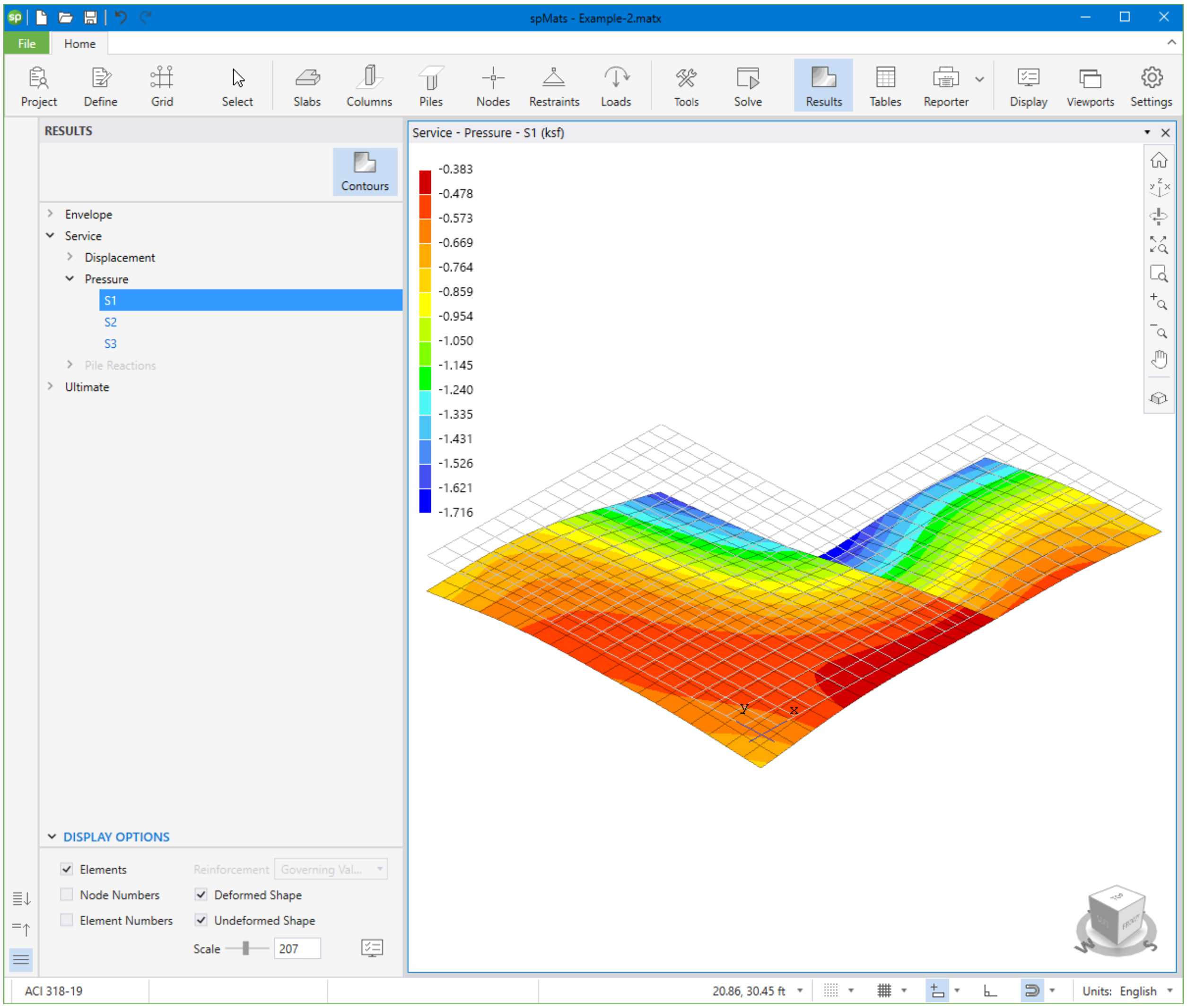
- Click on the **Run** button.
- The spMats **Solver** window is displayed and the solver messages are listed. After the solution is done, the design will be performed and then the focus will immediately be passed to the **Contours** scope.



8.2.6. Viewing and Printing Results

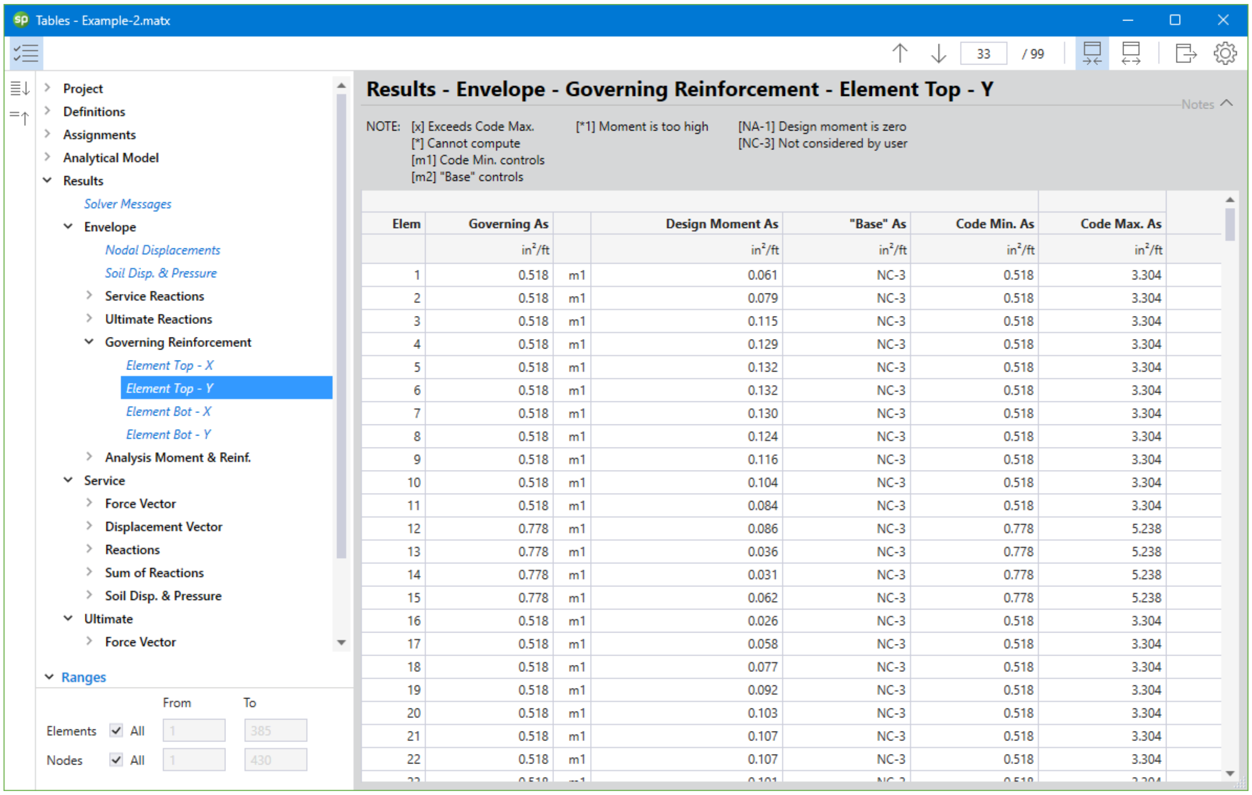
20. After a successful run, results can be viewed in a contour form.





Soil pressure values in the contour view above differ along dissimilar soil boundaries. More information about Winkler's Foundation can be found in Section 2.3.2.1.

21. Results can be also viewed in table format by selecting the **Tables** command from the **Ribbon.**

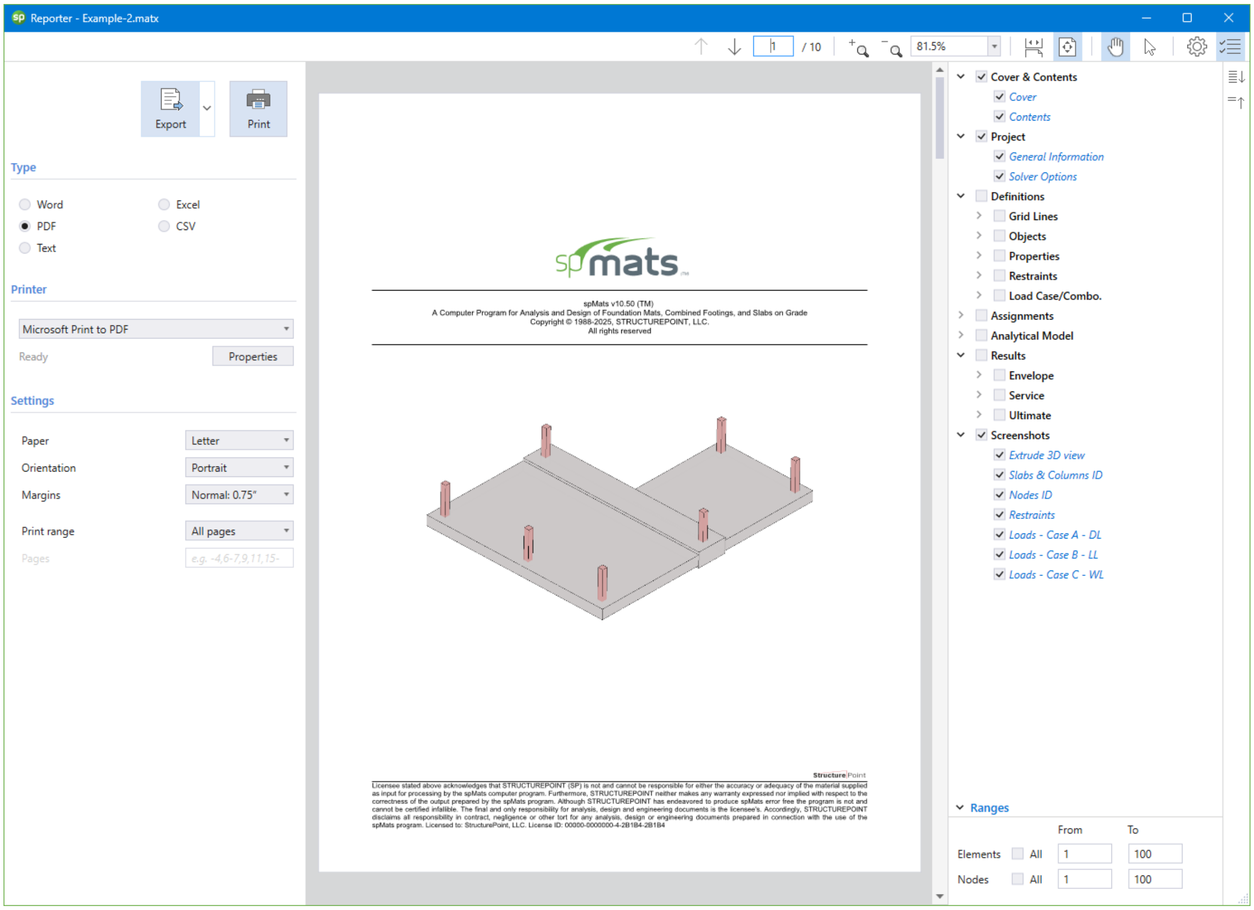


Results - Envelope - Governing Reinforcement - Element Top - Y

NOTE: [x] Exceeds Code Max. [*1] Moment is too high [NA-1] Design moment is zero
[*] Cannot compute [NC-3] Not considered by user
[m1] Code Min. controls
[m2] "Base" controls

Elem	Governing As		Design Moment As	"Base" As	Code Min. As	Code Max. As
	in²/ft		in²/ft	in²/ft	in²/ft	in²/ft
1	0.518	m1	0.061	NC-3	0.518	3.304
2	0.518	m1	0.079	NC-3	0.518	3.304
3	0.518	m1	0.115	NC-3	0.518	3.304
4	0.518	m1	0.129	NC-3	0.518	3.304
5	0.518	m1	0.132	NC-3	0.518	3.304
6	0.518	m1	0.132	NC-3	0.518	3.304
7	0.518	m1	0.130	NC-3	0.518	3.304
8	0.518	m1	0.124	NC-3	0.518	3.304
9	0.518	m1	0.116	NC-3	0.518	3.304
10	0.518	m1	0.104	NC-3	0.518	3.304
11	0.518	m1	0.084	NC-3	0.518	3.304
12	0.778	m1	0.086	NC-3	0.778	5.238
13	0.778	m1	0.036	NC-3	0.778	5.238
14	0.778	m1	0.031	NC-3	0.778	5.238
15	0.778	m1	0.062	NC-3	0.778	5.238
16	0.518	m1	0.026	NC-3	0.518	3.304
17	0.518	m1	0.058	NC-3	0.518	3.304
18	0.518	m1	0.077	NC-3	0.518	3.304
19	0.518	m1	0.092	NC-3	0.518	3.304
20	0.518	m1	0.103	NC-3	0.518	3.304
21	0.518	m1	0.107	NC-3	0.518	3.304
22	0.518	m1	0.107	NC-3	0.518	3.304

22. Results can be printed or exported in different formats by selecting the **Reporter** command from the **Ribbon.**



CHAPTER

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 pressure[1]. The user should introduce additional service level load combinations as required, e.g., for checking displacement limits[2].

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.

[1] IBC 2009, 1806.1; IBC 2006, 1804.1; IBC 2003, 1804.1; IBC 2000, 1804.1; NBCC, 4.2.4.4

[2] Two input files may be needed, one with allowable stress design load factors for foundation pressure check and one with load factors for checking displacements to avoid enveloping results for different sets of load factors under service level combinations.

A.1.1. For ACI 318-19/14/11

- Service load combinations:

Service Load Combinations – ACI 318 – 19 / 14 / 11							
Load Combo	Dead D	Live L	Snow S	Wind W	EQ E	Principal 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			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
	1.00	0.75	0.75	0.45		D, S, W	L
S10	1.00	0.75	0.75	-0.45		D, L, W	S
	1.00	0.75	0.75	-0.45		D, S, W	L
S11	1.00	0.75	0.75		0.525	D, L, E	S
	1.00	0.75	0.75		0.525	D, S, E	L
S12	1.00	0.75	0.75		-0.525	D, L, E	S
	1.00	0.75	0.75		-0.525	D, S, E	L
S13	0.60			0.60		D, W	-
S14	0.60			-0.60		D, W	-
S15	0.60				0.70	D, E	-
S16	0.60				-0.70	D, E	-

- Ultimate load combinations:

Ultimate Load Combinations – ACI 318 – 19 / 14 / 11							
Load Combo	Dead D	Live L	Snow S	Wind W	EQ E	Principal 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	-

A.1.2. 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	Principal 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
	1.00	0.75	0.75	0.75		D, S, W	L
S10	1.00	0.75	0.75	-0.75		D, L, W	S
	1.00	0.75	0.75	-0.75		D, S, W	L
S11	1.00	0.75	0.75		0.525	D, L, E	S
	1.00	0.75	0.75		0.525	D, S, E	L
S12	1.00	0.75	0.75		-0.525	D, L, E	S
	1.00	0.75	0.75		-0.525	D, S, E	L
S13	0.60			1.00		D, W	-
S14	0.60			-1.00		D, W	-
S15	0.60				0.70	D, E	-
S16	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	Principal 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	-

A.1.3. 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	Principal 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
	1.00	1.00	1.00	1.00		D, S, W	L
S5	1.00	1.00	1.00	-1.00		D, L, W	S
	1.00	1.00	1.00	-1.00		D, S, W	L
S6	1.00	1.00	1.00		0.70	D, L, E	S
	1.00	1.00	1.00		0.70	D, S, E	L
S7	1.00	1.00	1.00		-0.70	D, L, E	S
	1.00	1.00	1.00		-0.70	D, S, E	L
S8	0.60			1.00		D, W	-
S9	0.60			-1.00		D, W	-
S10	0.60				0.70	D, E	-
S11	0.60				-0.70	D, E	-

- Ultimate load combinations:

Ultimate Load Combinations – ACI 318 – 02							
Load Combo	Dead D	Live L	Snow S	Wind W	EQ E	Principal 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	-

A.1.4. For CSA A23.3-19/14/04/94

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 (References), the user should use appropriate load factors in conjunction with service level combinations to determine soil pressure for factored loads[3].

- 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	Principal 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	0.75	0.75		D, L, W	S
	0.75	0.75	0.75	0.75		D, S, W	L
S5	0.75	0.75	0.75	-0.75		D, L, W	S
	0.75	0.75	0.75	-0.75		D, S, W	L
S6	0.75	0.75	0.75		0.50	D, L, E	S
	0.75	0.75	0.75		0.50	D, S, E	L
S7	0.75	0.75	0.75		-0.50	D, L, E	S
	0.75	0.75	0.75		-0.50	D, S, E	L
S8	1.00			1.00		D, W	-
S9	1.00			-1.00		D, W	-
S10	1.00				0.667	D, E	-
S11	1.00				-0.667	D, E	-

[3] Two input files may be needed, one with load factors for foundation pressure check, and one with load factors for checking displacements to avoid enveloping results for different sets of load factors under service-level combinations.

- Ultimate load combinations for CSA A23.3-19/14:

Ultimate Load Combinations – CSA A23.3 – 19 / 14							
Load Combo	Dead D	Live L	Snow S	Wind W	EQ E	Principal 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	Principal 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	Principal Loads	Companion Loads
U1	1.25					D	-
U2	1.25	1.50	1.50			D, L	S
	1.25	1.50	1.50			D, S	L
U3	0.85	1.50	1.50			D, L	S
	0.85	1.50	1.50			D, S	L
U4	1.25	1.05	1.05	1.05		D, L, W	S
	1.25	1.05	1.05	1.05		D, S, W	L
U5	1.25	1.05	1.05	-1.05		D, L, W	S
	1.25	1.05	1.05	-1.05		D, S, W	L
U6	0.85	1.05	1.05	1.05		D, L, W	S
	0.85	1.05	1.05	1.05		D, S, W	L
U7	0.85	1.05	1.05	-1.05		D, L, W	S
	0.85	1.05	1.05	-1.05		D, S, W	L
U8	1.25			1.50		D, W	-
U9	1.25			-1.50		D, W	-
U10	0.85			1.50		D, W	-
U11	0.85			-1.50		D, W	-
U12	1.00				1.00	D, E	-
U13	1.00				-1.00	D, E	-
U14	1.00	0.50	0.50		1.00	D, L, E	S
	1.00	0.50	0.50		1.00	D, S, E	L
U15	1.00	0.50	0.50		-1.00	D, L, E	S
	1.00	0.50	0.50		-1.00	D, S, E	L

A.2. StructurePoint Text Exchange Format – spTX

StructurePoint Software can also use spTX (StructurePoint Text Exchange) format to save and input data files. spTX files are plain text files and can be edited using the built in editor in the programs or by any other text editing software.

A.2.1. Data Blocks

Data in a spTX file is organized into blocks. A data block starts with the keyword BLOCK and ends with the keyword END. Blocks can be placed in any order within the spTX file i.e., there is no any particular sequence. Data inside a block can be in the form of a list or a table. A block can have several tables and also have one or more sub-blocks. Sub-blocks within a block can also be placed in any order. Comments can be placed as desired by preceding them with two forward slashes //.

A.2.2. Lists

Lists are a data structures used to organize data inside the blocks. Lists contain data in the form of a key/value couple. Each key and its corresponding value must be in the same line and be separated by at least one or more spaces (blanks).

```
    // sample List
    // key----Value
       Code    ACI19
       Units   English
```

A.2.3. Tables

Just like lists, tables are also data structures used to organize data inside the blocks. A table starts with the keyword TABLE and ends with the keyword END. A block can have one or more table and they can be placed in any order. A table must contain at least one row and two columns. All keys are in the first row. Each subsequent row contains the corresponding values for the keys listed in the first row. All items of a row should be in the same line and be separated by at least one or more spaces (blanks).

```
// sample Table
TABLE ServiceLoadCombinations
    Label      A      B
       S1  1.000  0.000
       S2  1.000  1.000
END
```

A.2.4. spTX General Format

```
BLOCK Main                              // Main Block
  BLOCK BlockName                       // Block only contains List
      Key       value
      Key       value
      Key       value
  END

  BLOCK BlockName                       // Block contains List, internal Block and Table
      Key       value
      Key       value
      BLOCK BlockName                   // Internal Block
         Key       value
         Key       value
         Key       value
      END
      TABLE TableName                   // Internal Table
           Key      Key      Key
         value    value    value
         value    value    value
         value    value    value
      END
  END
END
```

A.3. spMats Text Exchange (MTX) File Format

spMats is able to read data from three file formats MA8, MATX and MTX and save its input data into two file formats, MATX and MTX.

spMats Text Exchange (MTX) file format is the spTX (StructurePoint text Exchange) format adapted for spMats. MTX files follow all the rules, organization and formatting of spTX files. MTX files are plain text files and can be edited using the built in MTX editor in spMats program or by any other text editing software.

MTX files are designed to be human-readable. The user can easily read/modify an MTX file without using the spMats manual. The best way to create a MTX file is by using the spMats GUI. Users can either select the MTX file type in the Save As menu command to save the file as an MTX or open the spMats MTX editor and use the "From spMats" command to convert an existing project file into the MTX format. Creating an MTX file using the spMats GUI has an added advantage as MTX files created this way also list the valid values for each parameter and other required information as comments.

```

BLOCK ProjectProperties
    Code    ACI19    // Valid values: ACI02, ACI05, ACI08, ACI11, ACI14, ACI19, CSA94, CSA04, CSA14, CSA19
    Units   English  // Valid values: English, Metric
END

```

A.3.1. MTX File Organization

Data in an MTX file is organized into Blocks. All Blocks should be placed inside the "Main" Block. Blocks can be placed in any order within the MTX file i.e., there is no any particular sequence. An MTX file contains the following Blocks

```
BLOCK Main
    BLOCK Description
    BLOCK ProjectProperties
    BLOCK Units
    BLOCK SolveOptions
    BLOCK GridLines
    BLOCK Definitions
    BLOCK LoadCases
    BLOCK LoadCombinations
    BLOCK Objects
    BLOCK Loads
END
```

Also, in an MTX file the order of keys and the number of blanks between keys and values are not important. Therefore, the following examples are equivalent.

```
BLOCK ProjectProperties
    Units   English
    Code    CSA14
END
```

```
BLOCK ProjectProperties
    Code    CSA14
    Units   English
END
```

A.3.2. MTX Import File Formats

The MTX format can also be used to import data into the spMats program. Grid, point load, and load combination data can be imported from an MTX file. To import data from an MTX file the **FileType** of the file used for import must be Exchange.

```
BLOCK Main
    Version    10.50
    Program    spMats
    FileType   Exchange
```

The best way to create an import file is to export grid, load, or load combination data and adjust the file as required.

A.3.2.1. Sample Load Import File

```
BLOCK Main
    Version   10.50
    Program   spMats
    FileType  Exchange

    BLOCK ProjectProperties
        Units     English  // Valid values: English, Metric
    END

    BLOCK Units
        TABLE UnitsPrecisions
                        Category    Unit  Precision
            Structure_Dimensions      ft          2    // Valid Units: yd, ft, in, ft-in, m, cm
                    Force_Forces    kips          3    // Valid Units: kips, lbf, kN, N
                   Moment_Forces  kip-ft          3    // Valid Units: kip-ft, lbf-ft, kip-in, lbf-in, kNm, Nm, Nmm
        END
    END

    BLOCK LoadCases
        TABLE LoadCase
            // Type - Valid values: Dead, Live, Snow, Wind, EQ, Others
            // SelfWeight - Valid values: Yes, No
            Case  Type  Label  SelfWeight
               A  Dead     DL         Yes
               B  Live     LL          --
        END
    END

    BLOCK Objects
        TABLE Nodes
            Id        X      Y
             1    16.00  30.00
             2    16.00  20.00
        END
    END

    BLOCK Loads
        TABLE PointLoads
            Node    Case          Pz       Mx        My
               1       A    -100.000    0.000     0.000
               2       A     -50.000   25.000    25.000
        END
    END
```

A.3.2.2. Sample Grid Import File

```
BLOCK Main
    Version   10.50
    Program   spMats
    FileType  Exchange

    BLOCK ProjectProperties
        Units     English  // Valid values: English, Metric
    END

    BLOCK Units
        TABLE UnitsPrecisions
                        Category    Unit  Precision
            Structure_Dimensions      ft          2    // Valid Units: yd, ft, in, ft-in, m, cm
        END
    END

    BLOCK GridLines
        TABLE VerticalGridLines
            Label  Coordinate
              "1"        0.00
              "2"       16.00
              "3"       36.00
        END

        TABLE HorizontalGridLines
            Label  Coordinate
                A        0.00
                B       20.00
        END
    END
END
```

A.3.2.3. Sample Load Combination Import File

```
BLOCK Main
    Version   10.50
    Program   spMats
    FileType  Exchange

    BLOCK LoadCases
        TABLE LoadCase
            // Type       - Valid values: Dead, Live, Snow, Wind, EQ, Others
            // SelfWeight - Valid values: Yes, No
            Case  Type  Label  SelfWeight
               A  Dead     DL         Yes
               B  Live     LL          --
        END
    END

    BLOCK LoadCombinations
        TABLE ServiceLoadCombinations
            Label         A        B
               S1  1.00000  0.00000
               S2  1.00000  1.00000
        END

        TABLE UltimateLoadCombinations
            Label         A        B
               U1  1.40000  0.00000
               U2  1.20000  1.60000
         END
    END
END
```

A.4. Conversion Factors – English to SI

To convert from	To	Multiply by
in.	m (1,000 mm)	0.025400
ft	m	0.304800
lb	N (0.001 kN)	4.448222
kip (1,000 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.5. Conversion Factors – SI to English

To convert from	To	Multiply by
m (1,000 mm)	in.	39.37008
m	ft	3.28084
N (0.001 kN)	Lb	0.224809
kN	kip (1,000 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.6. Material Strength Value Limits

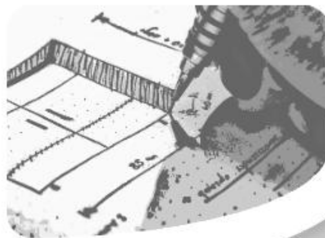
f_y				
Codes	English (ksi)		Metric (MPa)	
	Min	Max	Min	Max
ACI 318-14 & Older	40	80	275	550
ACI 318-19	40	100	275	690
CSA A23.3-19 & Older	40	72.5	275	500

f'_c				
Codes	English (ksi)		Metric (MPa)	
	Min	Max	Min	Max
ACI 318-19 & Older	0.5	50	3	300
CSA A23.3-19 & Older	0.5	50	3	300

A.7. Technical Resources

Bookstore

Technical Articles

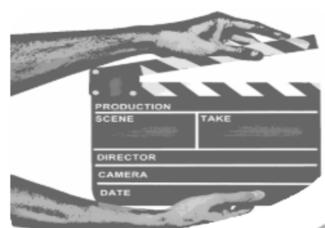


Design Examples

Case Studies

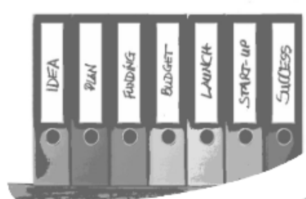
StructurePoint Wiki

Video Tutorials



Web Applications

Technical Manuals



A.8. Contact Information

Web Site: www.StructurePoint.org

E-mail: info@StructurePoint.org

support@StructurePoint.org

licensing@StructurePoint.org

Mailing Address: 1520 Artaius Pkwy #44

Libertyville, IL 60048

USA

Phone: +1-847-966-4357

Fax: +1-847-966-1542

A.9. Technical Manual Revision History

Revision Number	Revision Approval Date	Revision Description
10.50	08-19-2025	Support spMats v10.50 release