ADAPT

STRUCTURAL CONCRETE SOFTWARE

ADAPT-MAT 20

USER MANUAL

adaptsupport@risa.com   www.risa.com
RISA, Tech., 26632 Towne Centre Dr. Ste210, Foothill Ranch, California, USA
Tel: +1 (949) 951-5815, Toll Free: +1 (800) 332-RISA

LIST OF CONTENTS

Chapter 1

OVERVIEW

This Page Left Intentionally

BLANK

ADAPT-MAT is a computer program that enables you to model, analyze, design, and generate structural drawings for ground supported concrete structures that are used to transfer load to the underlain soil in a serviceable and safe manner. The program can handle practically all possible foundation configurations and loads, using a state-of-the-art 3D modeling and Finite Element Technology, and designing in accordance with the US and major international building codes.

Following a short glance at some of the features of the program that are described below, it is recommended that you go through the section on "Quick Start" to familiarize yourself with the operation of the program. Also, it is recommended to follow the **ADAPT-Builder 20 GUI Quick Reference Guide** in conjunction with this manual for detailed descriptions of each ribbon, tool panel and tool associated with the latest version of the program. Next, follow the tutorial, before you start your design project.

Since this program forms a part of the ADAPT-Builder suite, the general graphical interface and modeling techniques are described in the **ADAPT-Modeler and ADAPT-Floor Pro User Manuals**. This User Manual forms part of the ADAPT-MAT software package. It is recommended that you keep the manual handy and refer to it when needed.

If you are already familiar with ADAPT-Floor Pro, you may skip the section on Quick Start, and Modeling and Design Process, since the two programs use essentially the same interface, modeling and design process.

This Page Left Intentionally

BLANK

Chapter 2

BASIC FEATURES

This Page Left Intentionally

BLANK

2.1 OVERVIEW

This chapter explains the basic features of the program.

2.2 GEOMETRY

A foundation mat or raft as it is also referred to, can be faithfully modeled as it is intended for construction. The following describes the structural components that can be modeled and handled by the program as part of a foundation system;

- Slab regions: a foundation mat can consist of one or more slab regions, each with its own shape on plan, and its own thickness. The slab regions can have different elevations, creating steps either at the top or bottom of the foundation system.

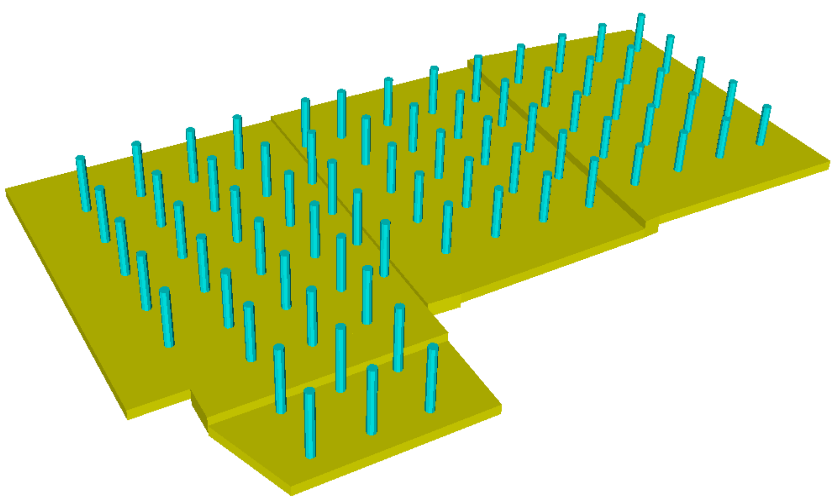


FIGURE 2.2-1 VIEW OF A MAT FOUNDATION CONSISTING OF MORE THAN ONE SLAB REGION WITH STEPS BOTH AT TOP AND BOTTOM

- Grade beams: Grade beams can be in any number, any dimension and orientation. Grade beams can be standalone or be part of a foundation slab. If they are part of the foundations slab, their structural interaction with the slab in resisting the applied loads is automatically accounted for in the analysis and design steps of the software. Further, the program recognizes the elevation of the grade beams with respect to the foundation slab in both its analysis and design stages.

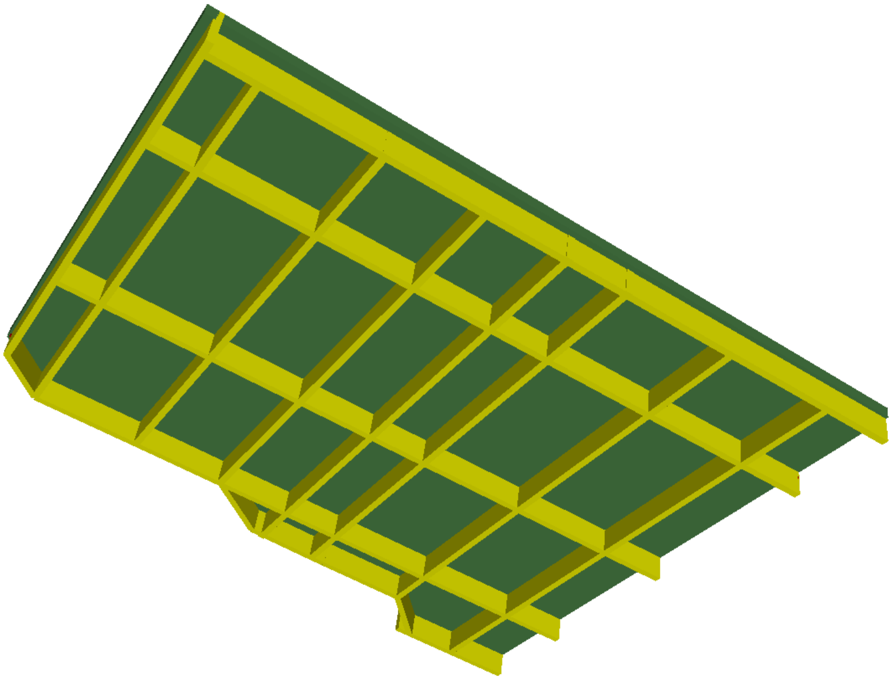


FIGURE 2.2-2 VIEW OF A FOUNDATION SLAB WITH INTEGRATED GRADE BEAMS

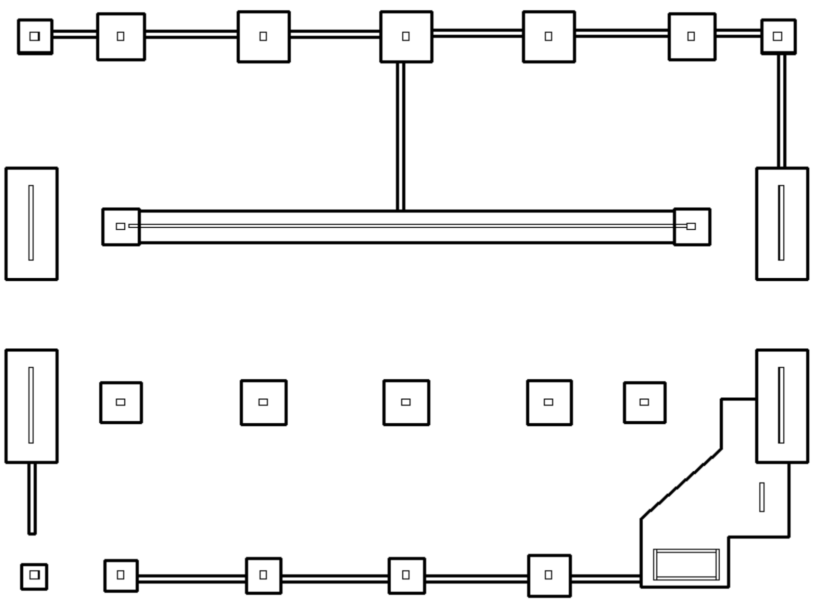


FIGURE 2.2-3 PLAN OF A FOUNDATION SYSTEM WITH ISOLATED FOOTINGS AND GRADE BEAMS

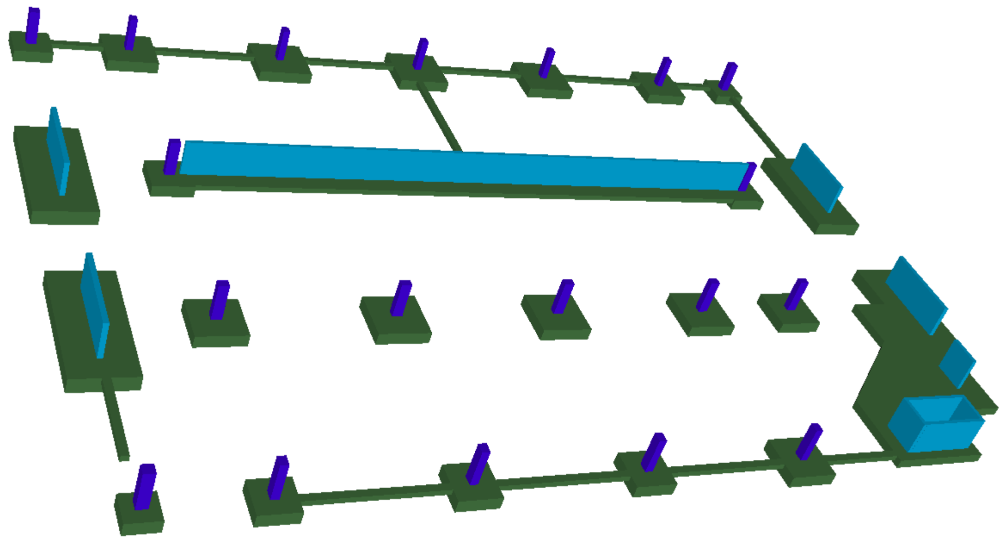
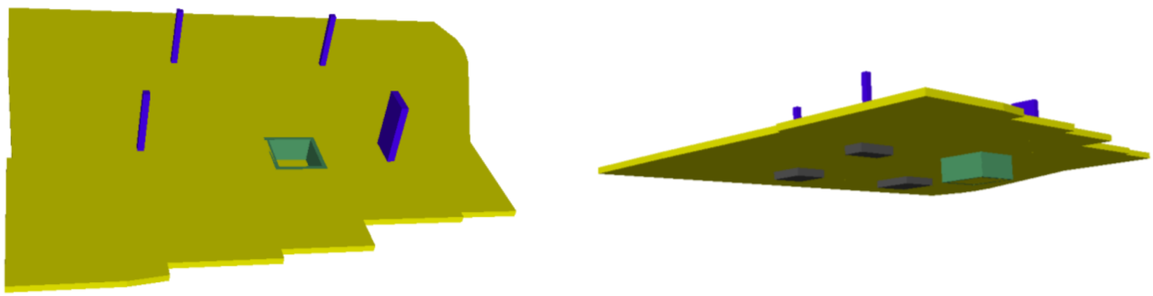


FIGURE 2.2-4 VIEW OF A FOUNDATION SYSTEM WITH ISOLATED FOOTINGS, GRADE BEAMS AND FOOTINGS BELOW WALLS

- Pile caps: Pile caps can be modeled either in isolation, or as part of a foundation mat. When integrated with the foundation mat, their interaction with the mat in resisting the applied load will be automatically accounted for by the program.
- Thickening below slab: Thickenings below a mat slab to resist punching shear below columns can be readily modeled with a column-drop/panel tool. The program accounts for the local stiffening of the foundation slab due to added thickness, as well as the resistance it provides for punching shear.
- Openings: Openings of regular or irregular geometry can be defined in any number and at any location.
- Elevator pits: Significant depressions in foundations slab with perimeter walls, typical of elevator pits can be modeled in the program and analyzed.



(a) elevator shaft model

(b) pads below columns

FIGURE 2.2-5 ELEVATOR SHAFTS AND PADS BELOW COLUMNS AND WALLS, AND ELEVATOR PITS CAN BE MODELED WITH CORRECT GEOMETRY AND ELEVATION

- Walls and columns above foundation mats: One story height of walls and columns can be modeled above a foundation system. The program accounts for the stiffness of these structural components when analyzing the foundation. The degree of stiffness of each of these structural components depends on the fixity defined by you at the far end of a wall or a column. The default setting of the program is freedom to displace and rotate at the far ends of the walls and columns above a foundation. The height of a wall or column above a foundation is taken to be the story height defined by you, but you have the option to modify the height of each wall.
- Upturned beams; Beams can be modeled to be entirely above a foundation slab, or partially above and partially below the slab.

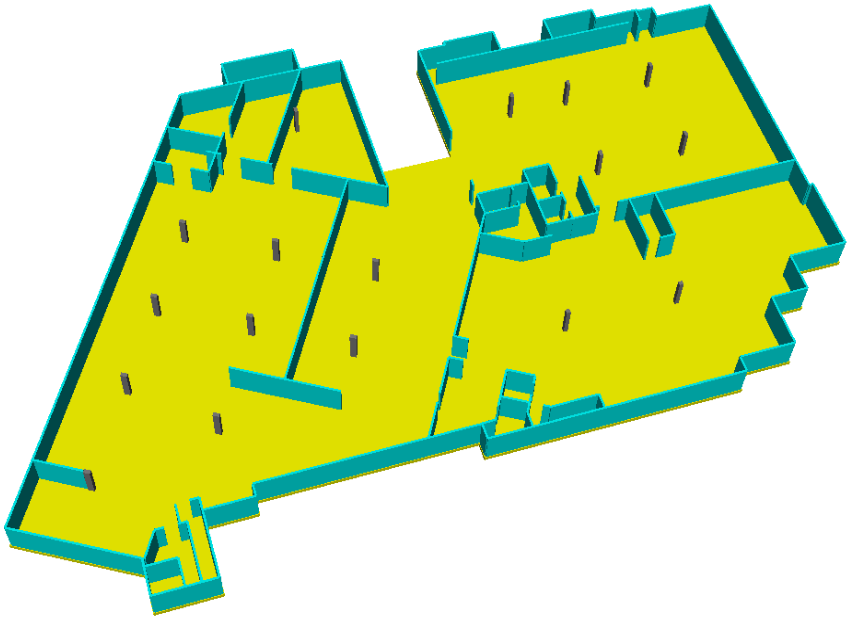


FIGURE 2.1-6 VIEW OF A FOUNDATION SYSTEM INCLUDING THE MODELING OF WALLS AND COLUMNS

2.3 SUPPORT CONDITIONS

A foundation system can be supported partially or wholly, on a variety of support conditions as described below:

2.3.1 Soil Support Area

Foundations can be modeled to rest on more than one type of soil. Each soil type will be specified with its own property and the support area it covers. A supporting soil region can be extended beyond the boundary of a mat and below the openings. The program will consider only the resistance of soil that is immediately below the structural members of the foundation. Soil regions modeled extending beyond the boundary of a mat's structural members and within the openings will not be considered to provide support. Not all the regions of a foundation system need be supported on soil. You may define parts of the foundation to overhang or span unsupported lengths.

The soil is represented by Winkler springs, for which you define the associated bulk modulus as part of your input data. The unit for the soil's bulk modulus is lb/in^3. This value typically varies between 100 to 400 pci (between 0.03 to 0.12 N/mm^3). In the absence of detailed information 200 psi (0.06 N/mm^3) is a reasonable starting point.

2.3.2 Compression Only Soil

You have the option to limit the transfer of force between a foundation member and its underlain soil as compression only. This results in separation between the underlain soil and the foundation member, where tension is likely to occur – hence no-load transfer. Also, you can specify the soil to resist both tension and compression.

The soil region you define provides only up and down support. For a support with capability of resisting forces in the horizontal direction and moments, you will use other options of support, as detailed below.

2.3.3 Soil / Rock Anchors

Soil and Rock anchors are designed to resist tensile forces only. They are used where there is potential of uplift, such as overturning due to high winds, seismic forces, or uplift from raised water table. Under normal conditions, support is provided by soil, but when the load on a foundation results in uplift, the soil/rock anchors will be mobilized to resist the uplift. The tensile force developed in a soil/rock anchor depends on the user defined stiffness. In principle, soil anchors are "tension only" point supports with specified stiffness values. You will use point springs to model soil anchors.

The default setting of the program is that the soil anchors take only tension in the vertical direction. You define their property in terms of (pounds per inch of extension, kN/mm extension, or tons/cm of extension). The program provides you the option to specify stiffness for displacements other than vertical direction.

2.3.4 Grade Beam Support

Grade beams that are integrated with a mat slab do not need additional support definition. The soil region that supports the mat will also support the grade beam. But for grade beams that are isolated (**Fig. 2.3-1**) you need to specify a line support along the beams. The stiffness of the support is defined in terms of displacement of the soil support per unit force placed on unit length of the grade beam [lb/in2; kN/mm2; t/m2). Obviously, the wider the grade beam, the stronger will be the resistance of the supporting soil, since the larger contact area mobilizes a larger volume of soil beneath the beam.

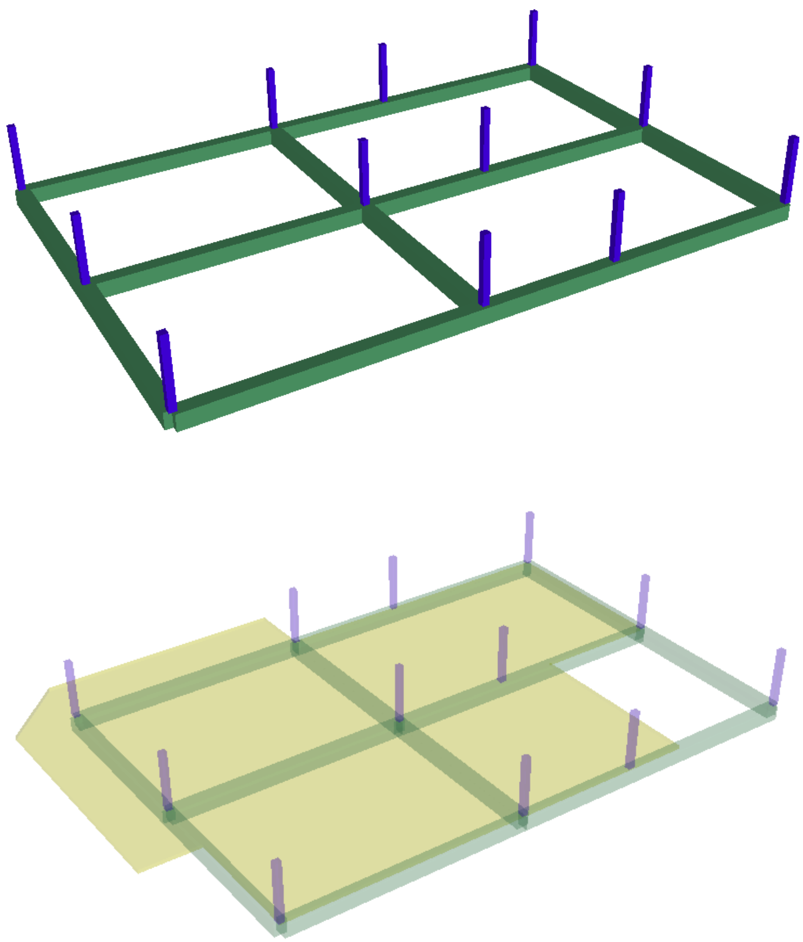


FIGURE 2.3-1 STANDALONE AND INTEGRATED GRADE BEAMS

For example if the bulk modulus of the soil is 200 lb/in3, (0.06 N/mm3 units) and the width of the grade beam is 24 inch (600 mm), the resistance of the soil per unit length of the beam to be specified is : 200x24 = 4,800 lb/in2 length of grade beam (0.06x600 = 36 N/mm2 length).

In the general case, you will use line spring tool with compression stiffness in the vertical direction to model grade beam supports.

2.3.5 Line Springs

Line springs provide you with a more general support condition than the simple support of a member on soil. The support provided by a line spring can be resistance along one or more of the three principal directions, with or without associated rotational stiffness. The stiffness provided along the length of a line spring is constant. Changes of stiffness along a line are defined by several line springs, each with its own stiffness.

2.3.6 Point Springs

These can provide translational or rotational restraints at one or more directions, at one or more locations of your choice on the foundation system. You identify the location of a point spring and specify its stiffness along and about the three principal directions as part of your input data.

2.3.7 Point Supports

You can define a point support anywhere at a foundation system and specify the type of fixity the selected location provides at the selected location. The fixity can be translation along one or more of the principal axes, and/or rotation about each. In addition to location on plan, you define the location of the point support in the vertical direction.

2.3.8 Line Supports

A line support is a more general form of a support condition in which the underlain soil can generally provide for a grade beam. You start by defining the location and length of a line support. Then you specify the type of support that you want the line to provide. This is carried out by assigning restraints to the line support you have defined. The restraints can be translation along one or more principal direction(s), and/or rotation about one or more of the principal direction(s). The vertical location of the line support can be below, above or any other height with respect of the mat foundation.

2.3.9 Piles

Piles are used where the soil is considered inadequate in providing the support needed for the superstructure. A pile-supported mat behaves essentially the same as a column supported slab, since the mat and its load are supported at discrete pile locations similar to a suspended slab supported on columns. There is no design contribution of the soil below the mat in providing resistance and is disregarded. Pile-supported mats can be best modeled and designed using ADAPT-Floor Pro. When using ADAPT-MAT each pile has to be modeled as a point spring having the same stiffness properties as the pile it represents.

2.3.10 Voids in Soil

Where there is no soil support below part of a foundation, such as a foundation overhang of a light building along its perimeter due to loss of moisture in soil, you do not define a soil support. Transfer of force between a foundation and soil can take place only at the locations where you define soil.

2.4 MATERIAL PROPERTIES

Each of the structural components specified, such as slab regions, grade beams, and reinforcement have can be specified with its own material property. Structural components of the same type, such as two columns can each have their own different material properties. You define the properties of the materials to be used in your model in the "Materials" pull-down menu and assign them to the structural components you create.

2.5 LOADS

The complete library and options for definition of loads located in ADAPT-Modeler applies to ADAPT-Floor Pro and ADAPT-MAT. Among many options, you can define point loads, line loads and patch loads (distributed load over a defined area) anywhere on the foundation slab. The loads you define can consist of concentrated forces along each of the principal directions and moments applied about each of the principal directions.

2.5.1 Load Cases

Each load you define is assigned to a "load case." This will enable you to group the loads that are associated with a common source. There is essentially no limitation on the number of loads that you may define, nor is there a limitation on the number of load cases. The program comes with default load cases of DEAD, LIVE, and PRESTRESSING (when ADAPT-MAT (PT) is used) along with several other pre-defined cases. The mat self-weight is automatically accounted for and by default, is included in load combinations for analysis.

2.5.2 Load Combinations

Depending on the building code you select, the program will automatically generate the primary load combinations of the code. The user has the option to edit the default combinations and/or define additional load combinations. There is practically no limit on the number of load combinations you can define. In addition to reporting the outcome of each load combination, the program has the ability to determine and report the envelope of the analysis results of the load combinations you define.

2.6 BASE REINFORCEMENT

ADAPT-MAT allows the user to pre-define layers of reinforcement at any depth in the slab. The reinforcement is referenced from the top or bottom of the slab and the user is prompted to enter the cover from the top or bottom reference plane. This is known as "base reinforcement" in the program. The reinforcement can be in one or two orthogonal directions that you define. The program considers base reinforcement in the analysis and design and reports the necessary reinforcement in addition to pre-defined base reinforcement.

The base reinforcement you define, can be expressed in terms of (i) bars at given spacing (regular mesh), or (ii) reinforcement areas per unit width of the slab, (iii) or isolated single or spaced bars with given length, size and location, (iv) or a combination of one or more of the above types. Different regions of the mat can be assigned different reinforcement. In other words, you can define different mesh reinforcement specifications for different regions in the mat.

2.7 POST-TENSIONING

ADAPT-MAT features the entire capability of prestressing options that is available in ADAPT-Floor Pro. This includes full flexibility in defining tendon layout, post-tensioning type (un-bonded or bonded), and stressing operations. The **ADAPT-Floor Pro 20 Basic Manual**, Chapter 5 contains a detailed description of tendon modeling in the ADAPT-Builder platform.

2.8 ANALYSIS

Unlike the standard conditions of suspended slabs, the analysis of a mat foundation can be an iterative process. Where there is likelihood of separation of soil from the foundation mat, an iterative solution is required, in order to determine the location and extent of soil/foundation separation.

The analysis process is initiated by assuming full contact of a mat with underlain soil. During each iteration, the program eliminates the regions of the soil/mat contact where uplift occurs, until full equilibrium of the entire structural system through transfer of compressive force between the mat and its underlain soil is achieved. During each iteration, the program re-generates the stiffness matrix of the structure, and obtains a solution. For this reason, and the fact that in such conditions superposition of load cases does not apply, the analysis of mat foundations with potential of uplift takes longer to achieve.

To reiterate, difference between the analysis of a mat foundation and an elevated slab is that, where uplift occurs, the principle of superposition of solutions does not apply, since each solution with uplift relates to a different structural boundary condition of the structure.

Like ADAPT-Floor Pro, the outcome of the analysis is in the form of displacements, forces and moments. When post-tensioning is present in a model, the program will report stresses at the top and bottom fibers of the mat. ADAPT-MAT (both RC and PT versions) generate and report the distribution of soil pressure below the mat and grade beams.

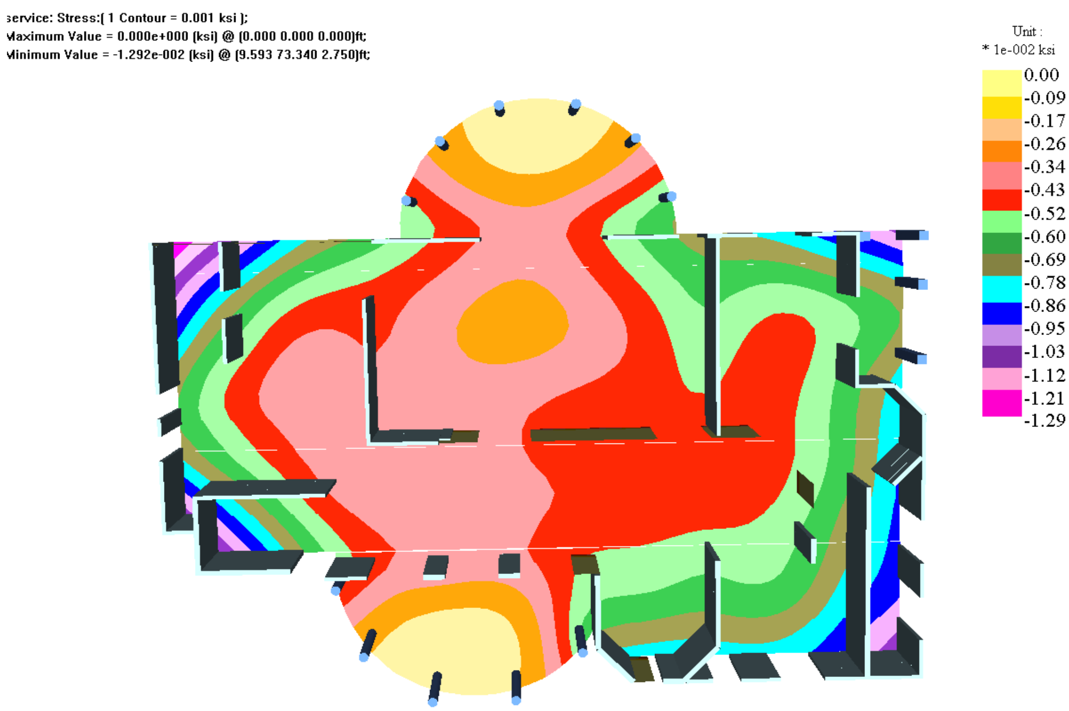
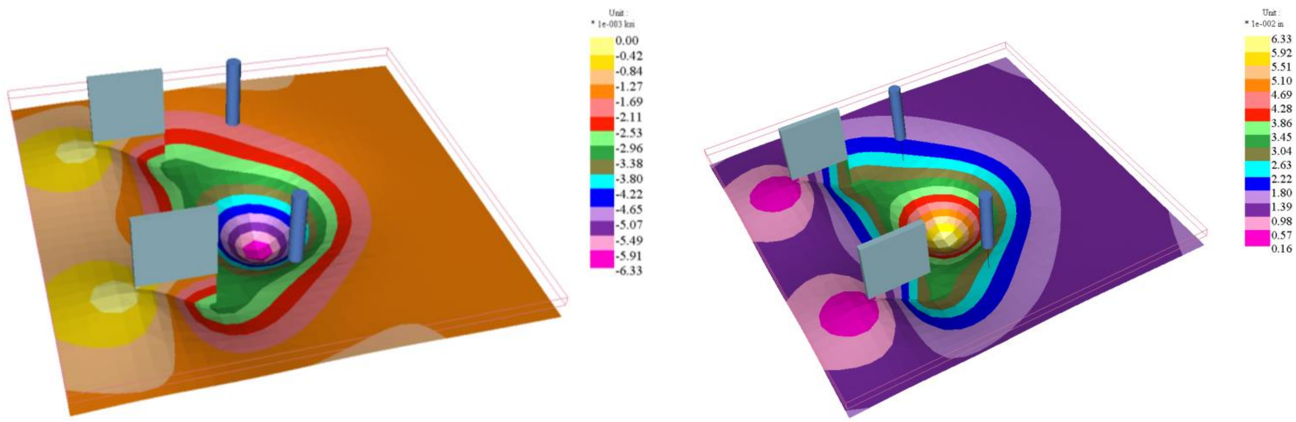


FIGURE 2.8-1 EXAMPLE OF THE DISTRIBUTION OF SOIL PRESSURE BELOW A MAT WITH FULL SOIL/MAT INTERFACE CONTACT



(a) deflected shape **(b) soil pressure**

FIGURE 2.8-2 DISTRIBUTION OF SOIL PRESSURE

The example above shows, (a) the displacement of a foundation mat under a central concentrated load and overturning moments on the walls, and (b) the uplift (soil/foundation separation) at the tip of the walls due to the distribution of soil pressure.

2.9 DESIGN

ADAPT-MAT carries out a design of the mat slab by performing code checks prescriptive of the selected building code. Where required, the program determines and reports reinforcement from the library of bar reinforcement as defined by the user or bar sizes of your choice. The program checks both service (SLS) and strength (ULS) requirements of the selected building code. The reinforcement report of the program includes the quantity, position, and length of each bar in plan, ready to be used in a structural drawing. Where post-tensioning is present, the program provides a detailed stress check as required prescriptive of the selected building code. The stress checks can be reported both graphically and in tabular format.

2.10 GENERATION OF DRAWINGS

The reinforcement plan generated automatically by the program can be readily exported to either a DXF or DWG file format that can be used to combine with the remainder of your work in a construction drawing.

2.11 LINK WITH 3RD PARTY PROGRAMS AND ADAPT DATA EXCHANGE

If the foundation slab you design forms part of a multi-story building for which you have developed an independent model in a commercially available program, and you have the results of the loads from the superstructure, there are several ways to facilitate the transfer of this information to ADAPT -MAT as applied load.

- The common method is to simply enter the load in the program using the loading toolbar. This is referred to commonly as the "manual" method of input.
- Loads from other software can be formatted into the program's data exchange file and be imported to ADAPT-MAT. The program can read and import loads if the information is formatted according to ADAPT's Data Exchange File.
- ADAPT-MAT has the capability of importing solutions directly from other commercially available programs through use of the ADAPT-Integration Console (IC). Through creation of input/output files, this program creates a new ADAPT Data Exchange File that is imported to ADAPT-MAT. In its current form, ADAPT-IC can import applied gravity and lateral loads and reactions. As is often the case with mat foundations, the entire gravity load on a structure would be considered in mat design along with lateral loads. There are current alternate methods that can be used in a third-party program which will allow the total gravity load to be imported to ADAPT-MAT. It is recommended to consult with an ADAPT Technical Support Specialist for addition information. support@adaptsoft.com

This Page Left Intentionally

BLANK

Chapter 3

QUICK START

This Page Left Intentionally

BLANK

3.1 OVERVIEW

This chapter includes a simple example as an introduction to the program. Once you have reviewed the example in this chapter, it is recommended to review **Chapter 6, “Tutorial”** for a step-by-step description of data generation and design.

3.2 OPENING THE PROGRAM

- Open the program to display the splash window shown below. Select “MAT” option, if not already selected. Click OK to open the main program interface.

- A thorough description of the main graphical user interface (GUI) and its menus and tools can be found in the **ADAPT-Modeler & ADAPT-Floor Pro 20 Basic Manuals**. You may refer to them, if needed. This section describes the features that are unique to the ADAPT-MAT program.
- Getting started, a description of commonly used tools enabling the creation of a model are described below:

- Create a grid providing a unique set of dimensions and guide us to create the structure
- Tools to build the mat, such as its geometry and other components
- Tools to view the geometry in three dimensions and facilitate verification of modeling accuracy
- Tools to define the soil spring support below the mat
- Tools to apply loads on the mat
- Tools enabling the user to mesh and analyze the mat
- Tools used to view analysis results

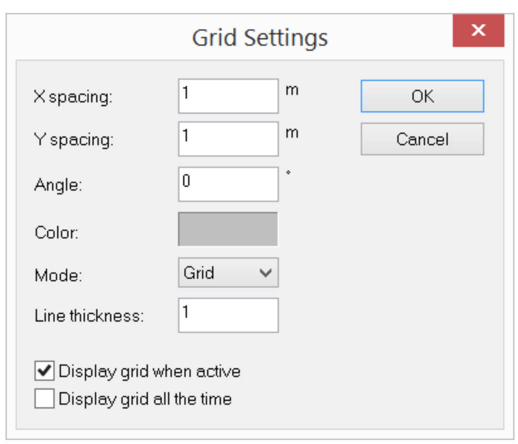
We will introduce and invoke each of the tools listed above one after the other in form of a quick start step-by-step guide.

3.3 QUICK START – STEP-BY-STEP

The purpose of this example is to show a step-by-step procedure to begin using ADAPT-MAT in the modeling of a mat foundation.

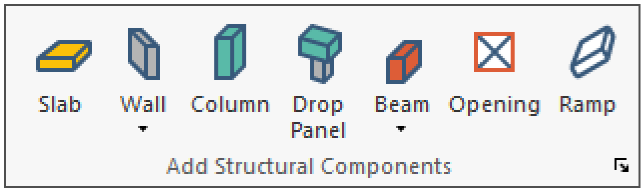
3.3.1 Create a Grid

In the Snap tools from the lower *Quick Access Bar*, select the icon, Grid Settings and set the X and Y grid spacing to the desired dimension.



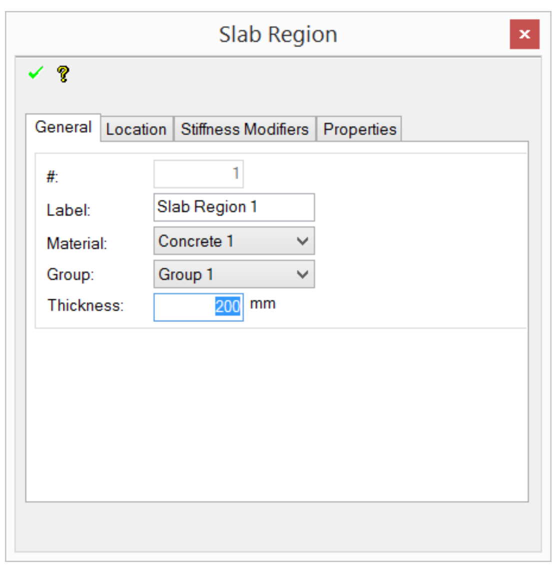
3.3.2 Building the Model

Use *Model→Add Structural Components* to insert structural components such as slab regions, columns, walls, beams, openings, drop caps/panels, base reinforcement or to set the vertical plane dimensions (story heights).



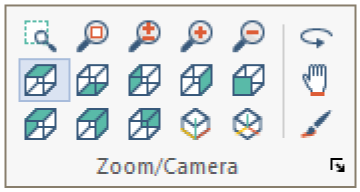
If the model is started with an import of a .dwg or .dxf file (see Section 3.2.1) you can use the *Transform* panel, to transform closed polygons to structural components.

Once a structural component is added, double-click on the any component to modify the default dimension(s) of the component. After changes are made to the component properties, make sure to select the green checkmark for the changes to be updated and active.

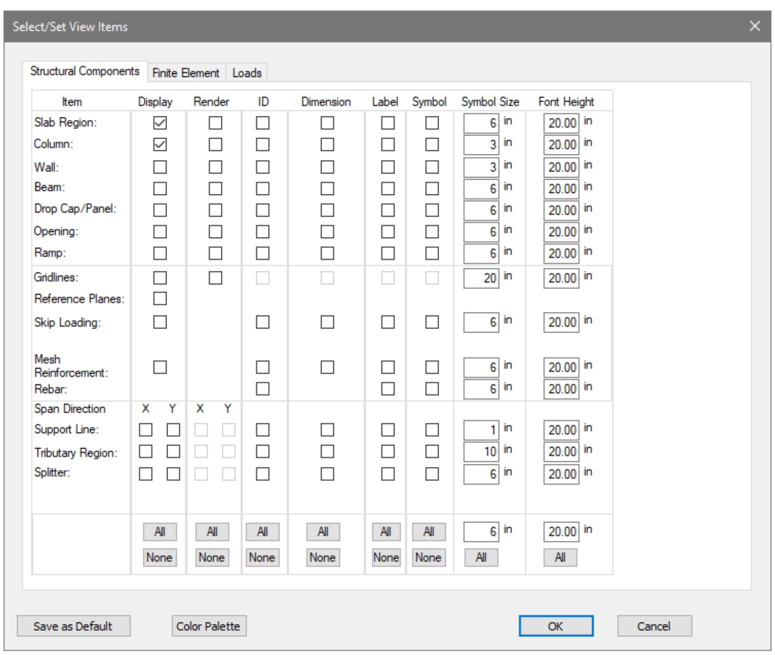


3.3.3 Viewing the Model

Once a model has been successfully built, where all necessary components are added, it is important to validate the accuracy of the model. Doing so will ensure that the model most closely represents the foundation system being analyzed. To make a cursory view of the model in two or three-dimensions, use *Home→Camera/Zoom.* The tools located here allow you to view the model from different elevation and plan views. You can also elect to view the structure as an isometric view. This is useful when verifying more complex geometries and loading. The toolbar contains several selections allowing you to pan, zoom, rotate, and refresh the view.

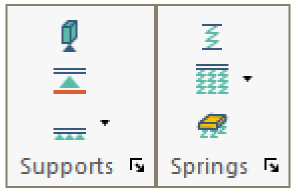


The *Visibility* ribbon gives options to customize the display and what components and loads are shown in the main interface. One of the most commonly used tools is the *View Display* tool When this button is activated, a menu of items will be shown prompting the user to set those components and loads to be viewed along with the option to display component or load ID's, Dimensions (load magnitude or moments), Labels or Symbols. The symbol size and font height of text can also be modified.



3.3.4 Defining the Soil Support

Soil supports defined as point, line or area springs can be added to the model from the *Model→Supports or Springs*.



Once the supports have been added to a model, as is the case with components, the user can double-click on the support to open the support properties. In this dialogue window, the soil stiffness can be changed from the default setting. Note also that line and/or point supports with assumed infinite stiffness can be included in a mat foundation model. These tools are located on the same toolbar.

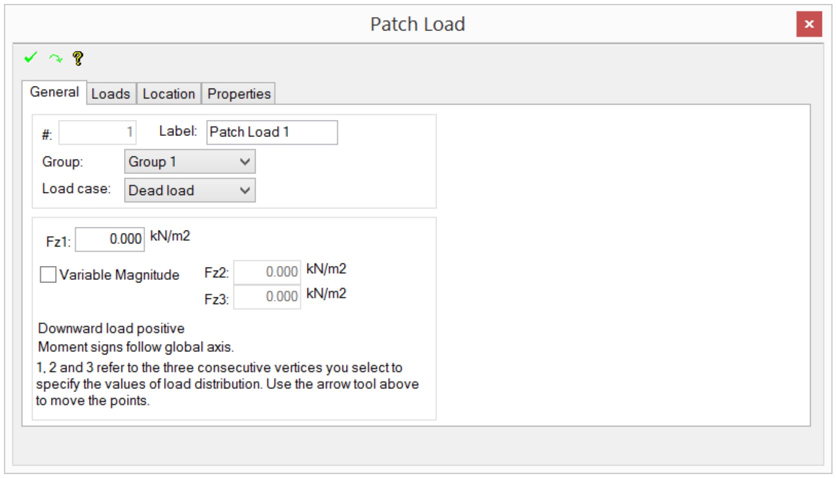
3.3.5 Application of Loads

When a design code is selected from the *Criteria→Design Criteria→Design Code* , the program will automatically generate multiple Strength and Service load combinations. Each combination is composed of default load cases defined as Live, Dead or PT (when post-tensioning is included in a model). See Section 2.4 for additional information related to load case and load combination generation.

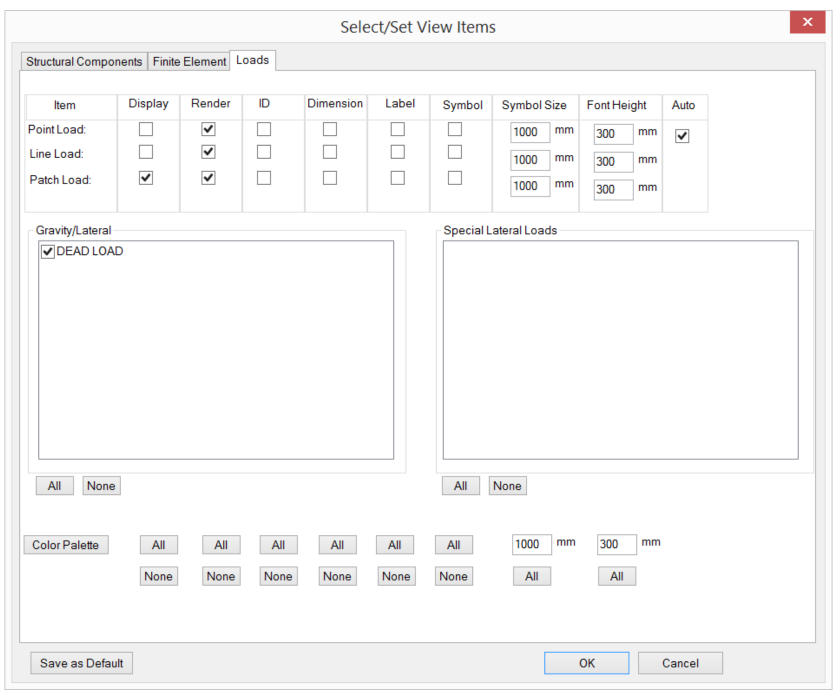
The *Loading→General* panel contains several tools used for the creation of point; line and area (patch) loads. Any type of load can be added for any defined load case. Note that the user can add as many load cases as desired in addition to those default cases created by the program. The Patch and Line Load Wizard tools allow the user to easily create loads for a selected region or slab boundary.

Once a load has been generated, the magnitude or moment values associated with the load type can be changed by double-clicking on the loading symbol and changing the value in the load properties window.

The group assignment and load case can also be modified using the load properties windows. To select a load to modify, it is helpful to view the model in an isometric (top-front-right) view from *Visibility→Camera/Zoom.*



To view loads, the *Loading* ribbon contains the tool from *Visibility→Show Loads* which will toggle load display in the main interface. The *View Load Settings* tool contains a rooted menu allowing the user to isolate the display of both load cases and load types.

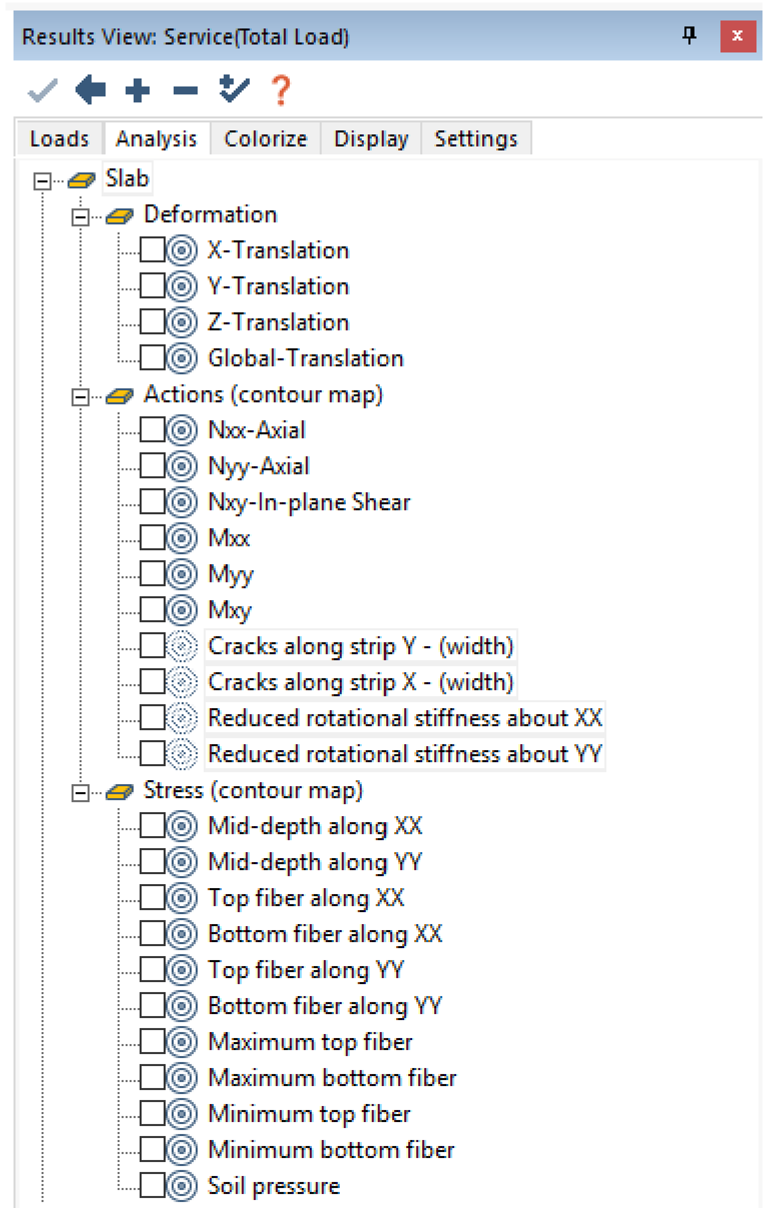


3.3.6 Mesh and Analyze

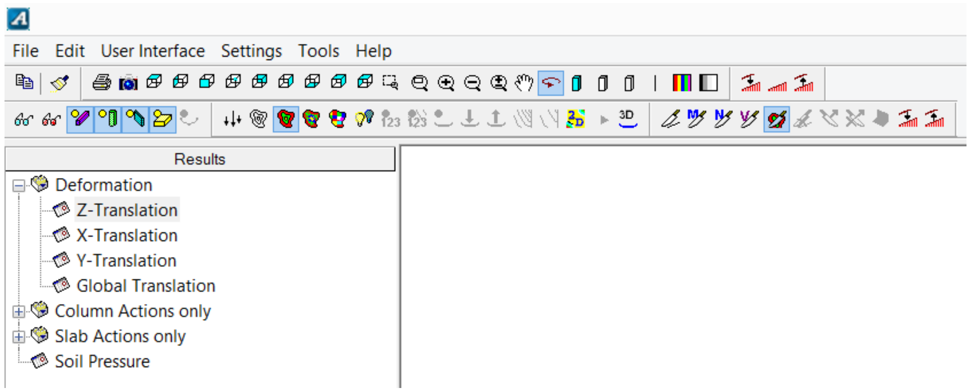
Once the model has been built with components, supports and loads, the model can be meshed and analyzed for a solution. To mesh the structure use *Analyze→Meshing→Mesh Generation*. A dialogue window will appear prompting the user to select a mesh and node consolidation dimension. The default values are set to 3 ft and 1.5 ft. The recommend mesh size is typically between 3-4 times slab thickness and the recommend node consolidation dimension is between 2-3 times slab thicknesses. After meshing the structure, the model can by analyzed by selecting the *Execute Analysis* tool.

3.3.7 Viewing Analysis Results

After the general model analysis has been completed, the analysis results can be viewed graphically and in report format. For a cursory review of results after the first analysis, it is recommended to view the results graphically. Go to *Analysis→Analysis→Result Display Settings.* A docked side window will appear, and graphical viewing selections can be made.



The program also includes a legacy graphical results viewer called *ADViewer* that can be opened in a separate window by selecting *View Results* from *Analysis*→*Analysis* . The image below shows the derfault, *ADViewer* window.



At the left-hand edge of the ADViewer module, the user can select the Result type (e.g. Deformations, Slab Actions, Soil Pressure, etc.) along with the load combination for which the results type is to be viewed. The individual structural and analysis components and component groups can be displayed from the same menu.

A thorough description of the tools located in ADViewer can be found in Chapter 9 of the **ADAPT-Floor Pro 20 Basic Manual**. These tools are equally applicable to ADAPT-MAT as well as ADAPT-Floor Pro and are universal to the ADAPT-Builder Platform.

This Page Left Intentionally

BLANK

Chapter 4

USER INTERFACE

This Page Left Intentionally

BLANK

4.1 OVERVIEW OF ADAPT-BUILDER PLATFORM

Developed from the ground up with ADAPT Building Information Modeling (BIM) Technology, the ADAPT-Builder Platform is a collection of fully integrated design and analysis tools for concrete floor systems, foundations, and beam structures, whether with or without post-tensioning.

With the ADAPT-Edge module, ADAPT-Builder allows for the analysis of gravity and lateral loads on an entire concrete building structure from roof to foundation, including automated load takedown. ADAPT-Edge can be used as a stand-alone product or together with ADAPT's specialized design software for floor and foundation systems – Floor Pro and MAT. In this way, Edge helps to simplify the analysis and design process by using only one model for floor, foundation, and building analysis and design. ADAPT-Edge is specifically designed for the rapid modeling of concrete structures and gives the option of adding prestressing or post-tensioning to any slab or beam member.

ADAPT-Builder’s intuitive and easy-to-use 3D component modeling capabilities allow you to quickly model any structure. In addition, the Builder Platform is the industry's only solution that gives you the flexibility to analyze 3D structural models using either the Finite Element Method (FEM) or the Equivalent Frame Method (EFM). Specialized design tools for concrete beam frames, one-way or column-supported flat slabs, parking structures, mat foundations, ground-supported slabs, column and wall design, as well as built-in building codes (e.g., American, Canadian, British, European, and Australian), ADAPT's Builder platform allows for a streamlined workflow.

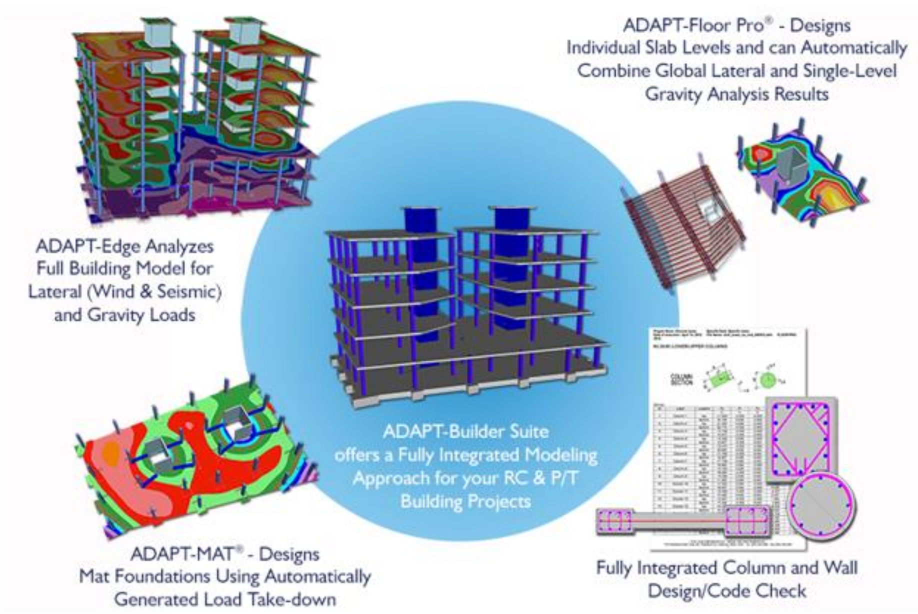


FIGURE 4.1-1 ADAPT BUILDER MODULES

4.2 ADAPT-BUILDER SPLASH SCREEN

ADAPT-Builder is a general platform which contains ADAPT-MAT for the design of mat foundations. While opening the program, the user can choose the configuration as required in the project. ADAPT- Builder's splash screen is shown in the **Figure 4.2-1**.

Then user can select the Structure Type, and choose among the following:

- Elevated Floor Systems, Beam Frames, Grid Frames (FLOOR PRO)
- Mat/Raft Foundation, Grade Beams (MAT)
- Post-Tensioned Slab-On-Ground (SOG)
- Full building Gravity and Lateral Design (EDGE)

Select ADAPT-MAT for the design and analysis of a mat foundation system. To design conventional reinforced concrete structure, select RC for mode. To design a post-tensioned concrete structure, select PT/RC.

FIGURE 4.2-1 ADAPT BUILDER SPLASH SCREEN

Finally, one needs to specify the System of units. SI, US or MKS can be selected in the program. Upon clicking on OK, it will open ADAPT-MAT environment.

4.3 ADAPT-BUILDER GRAPHICAL USER INTERFACE

Figure 4.3-1 shows the full-screen display of the **ADAPT-Builder** program, with typical features labeled for easy identification.

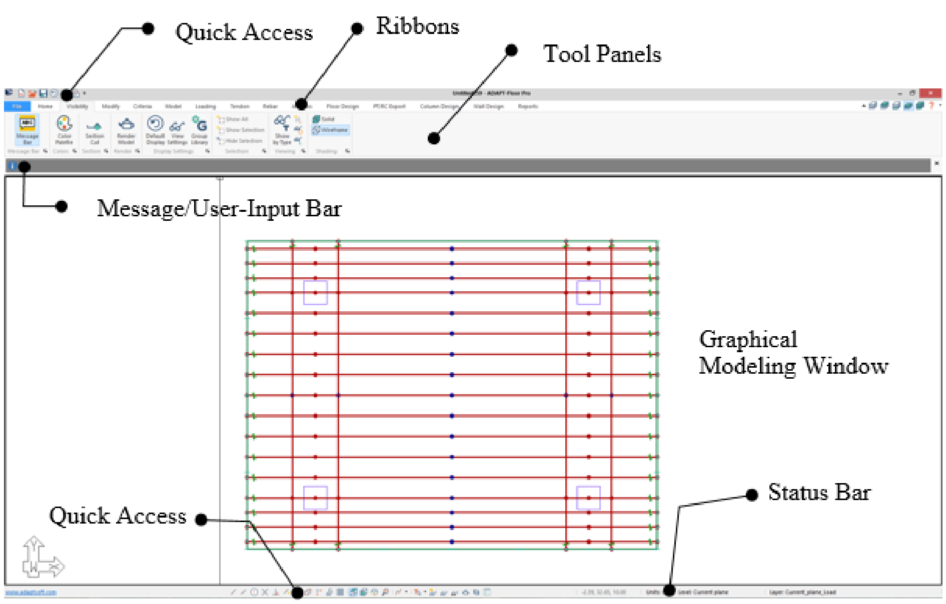


FIGURE 4.3-1 ADAPT BUILDER MAIN SCREEN

ADAPT-Builder contains a ribbon-based user-interface that contain contextual, customizable tool panels. Each tool on a panel contains a tooltip when hovering over the tool icon that provides a brief description of the tool. Each panel can be expanded to show detailed information and descriptions for all tools on the panel. The *Quick Access* panels contain pre-defined and most-commonly used tools for easy access. The top *Quick Access* panel is customizable and can contain user-defined tools. The bottom *Quick Access* panel contains fixed tools that are also available in most of the tool palettes belonging to a ribbon. At the top-right of the upper *Quick Access* panel is the set of *Story Manager* tools. These tools allow the program active mode to be set for *Single-Level* or *Multi-Level* and also navigate vertically between levels. The program ribbons include:

- File – General save, print, import, export functions.
- Home – UI and commonly used tools and settings.
- Visibility – Tools related to graphical visibility options.
- Modify – Tools related to modification of current components in the model.
- Criteria – Material, analysis, and design criteria settings.
- Model – Tools related to modeling of components and supports.
- Loading – Tools related to all loading types, load takedown, LL reduction.
- Tendon – Tools related to modeling, display, modification, and optimization of post-tensioning.
- Rebar – Tools related to modeling, display, and modification of reinforcement.
- Analysis – Tools related to meshing, vibration, cracking and general analysis.
- Floor Design – Tools related to floor design strip generation, design or investigation, display, and results for floors.
- PT/RC Export – Tools related to criteria and strip generation for export to ADAPT-PT/RC.
- Column Design – Tools related to column groups, design parameters and column design and results.
- Wall Design – Tools related to wall piers, wall sections, design parameters and wall design and results.
- Reports – Tabular and graphical reporting.

When the program is open through the splash screen shown in **Figure 4.2-1**, the program will auto-filter and show only those ribbons and tools applicable to the selected state of program module selections.

The *Message/User Input Bar* displays tool-specific information, program prompts, and any values that may be typed by the user for specific program procedures. The *Status Bar* displays such information as the mouse cursor coordinates (location), current unit system, current level, current drawing layer, and gridline spacing and status. A short description of each specific tool also appears in this area when the mouse cursor is placed over the corresponding tool button.

4.3.1 Mouse Function and Operation

The primary function of the mouse is through its left-click. Depending on the mode of the program, as outlined in the next section, the left-click will result in selecting the entity below the cursor, inserting an entity or performing an operation at the location of the cursor.

The right-click of the mouse with cursor on the display portion of the screen will display the window shown in **FIGURE** 4.3-2 **4.3-2.** Right-click options are context specific and may change depending on the type of component selected while carrying out this operation.

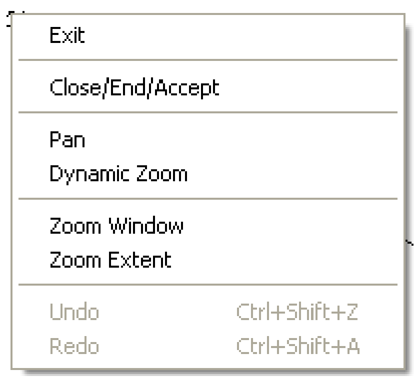


FIGURE 4.3-2 RIGHT-CLICK OPTIONS OF THE MOUSE

Click on a menu item listed to perform the operation described. Functions including layout of poly regions or polylines require the Close/End/Accept option to be selected. Alternately, the user can select the 'C' key on the keyboard to close the operation. If you right-click the mouse while the cursor is outside the Main Window, a list of all available toolbars appears. From this list, you can select the toolbars you want to display.

Double-clicking on an entity opens its properties dialog box.

If more than one item exists in a location in the display screen, left click on the area, and use the *Tab* key on your keyboard to toggle between the multiple items in the same area.

For dynamic rotation of the model view, use the SHIFT key + mouse scroller to control the rotation.

4.3.2 Main User Interface

For a complete description of the Graphical User Interface in ADAPT-Builder please refer to the **ADAPT-Builder 20 GUI Quick Reference Guide**. The manual can be located by going to *Help→Documentation* in the ADAPT-Builder interface.

Chapter 5

MODELING AND DESIGN PROCESS

This Page Left Intentionally

BLANK

5.1 OVERVIEW

This section outlines detailed steps to follow in designing a conventionally reinforced or post-tensioned mat foundation, using ADAPT-MAT. It is assumed that those steps pertaining to post-tensioning do not apply if the mat being designed is conventionally reinforced. Depending on whether you already have an electronic file of the mat geometry or not and whether you are familiar with AutoCAD or not there are different options available to you. Refer to the flow chart and the text that follows for the details.

5.2 DESIGN PROCEDURE

The following steps are intended for completion of a full analysis and design of a mat foundation. The items listed here are discussed in greater detail in the **ADAPT-Modeler 20 Manual**.

The suggested steps for the analysis and design of a mat foundation are:

- Creation of the structural model
- Define the soil support conditions
- Validate the structural model by use of the tools for viewing analysis results
- Complete and finalize input data
- Perform analysis
- Prepare
- Design
- Generate structural drawings
- Generate structural calculation reports

5.2.1 Create the Structural Model

Use one of the following options to create your structural model

- Import an AUTOCAD file of the model (DXF or DWG) and convert it to structural model by transformation.

- Define the foundation slab and loading, using the Build and Loading tools of the program. This is also referred to as "manual" generation of a model.
- Import geometry and load on the model from a multi-story analysis program, such as ETABS, or other programs supported by ADAPT-Builder platform.
- Use the generic data exchange file format of the program to create and import the geometry of the foundation mat.

The common and more accurate method for the generation of geometry of your structural model from the first option above, that is, importing an AUTOCAD file, since most of the commercially available multi-story software does not model the complex foundation geometry with adequate degree of accuracy.

5.2.2 Define Soil Support Conditions

Define the location and properties of the soil support, piles and rock anchors, if any.

5.2.3 Validate the Structural Model

In this step you determine whether the structural model of the foundation slab you have generated, and its support conditions are indeed a faithful representation of your requirements, before proceeding with detailed analysis and design. The steps are:

- Mesh the structure
- Analyze the structure for an arbitrary concentrated load in the central region of the mat and/or self-weight
- View the deflected shape of the structure under self-weight and determine if the results look reasonable in shape and magnitude based on engineering experience and judgment.

5.2.4 Complete and Finalize Input Data

- Add post-tensioning tendons as required if the structure is post-tensioned.

- Review and finalize the design criteria including material properties, analysis/design options, reinforcing bar selections and design code.
- Specify reinforcement mesh to be included in your design, if any.
- Define additional load cases or loads if necessary.
- View the program-generated load combinations and edit if necessary.

5.2.5 Perform Analysis

- Analyze the structure

5.2.6 Prepare to Design

- Create support lines
- Generate design sections automatically
- Review the generated design strips. If necessary, modify the support lines and use splitters to refine the design strips created. Conclude your modifications with a re-creation of automatically generated design strips and sections.

5.2.7 Validate the Code Compliance of the Design

If the foundation system is not pre-stressed, the program automatically provides the adequate amount of reinforcement, where necessary, to meet the requirements of the design code you have selected for Strength and Serviceability.

For pre-stressed foundations, in addition to the reinforcement requirements, the computed stresses must not exceed the code specified threshold. If this condition is not satisfied, the program shows the design sections with dashed lines in purple. At this stage, you must either modify the post-tensioning you have specified, or change other parameters of the foundation, such as thickness and redo the analysis and design. This process continues, until the solution is acceptable and code-specified stress limits are met. To summarize:

- View the outcome of code check for design sections in X-direction, followed by an examination of the same for design sections in Y-direction.
- If there are no purple (dashed) lines the requirements of the building code you selected is satisfied.
- If there are purple (dashed) lines, the code requirements have not been met; investigate and modify the slab until a solution is acceptable.
- Execute the punching shear check – if applicable.
- View the punching shear stress check on the screen, to ascertain that the calculated stresses do not exceed the maximum allowed by code. The program automatically reports these locations to you on the screen with red color and a coded note.

5.2.8 Generate Structural Drawings

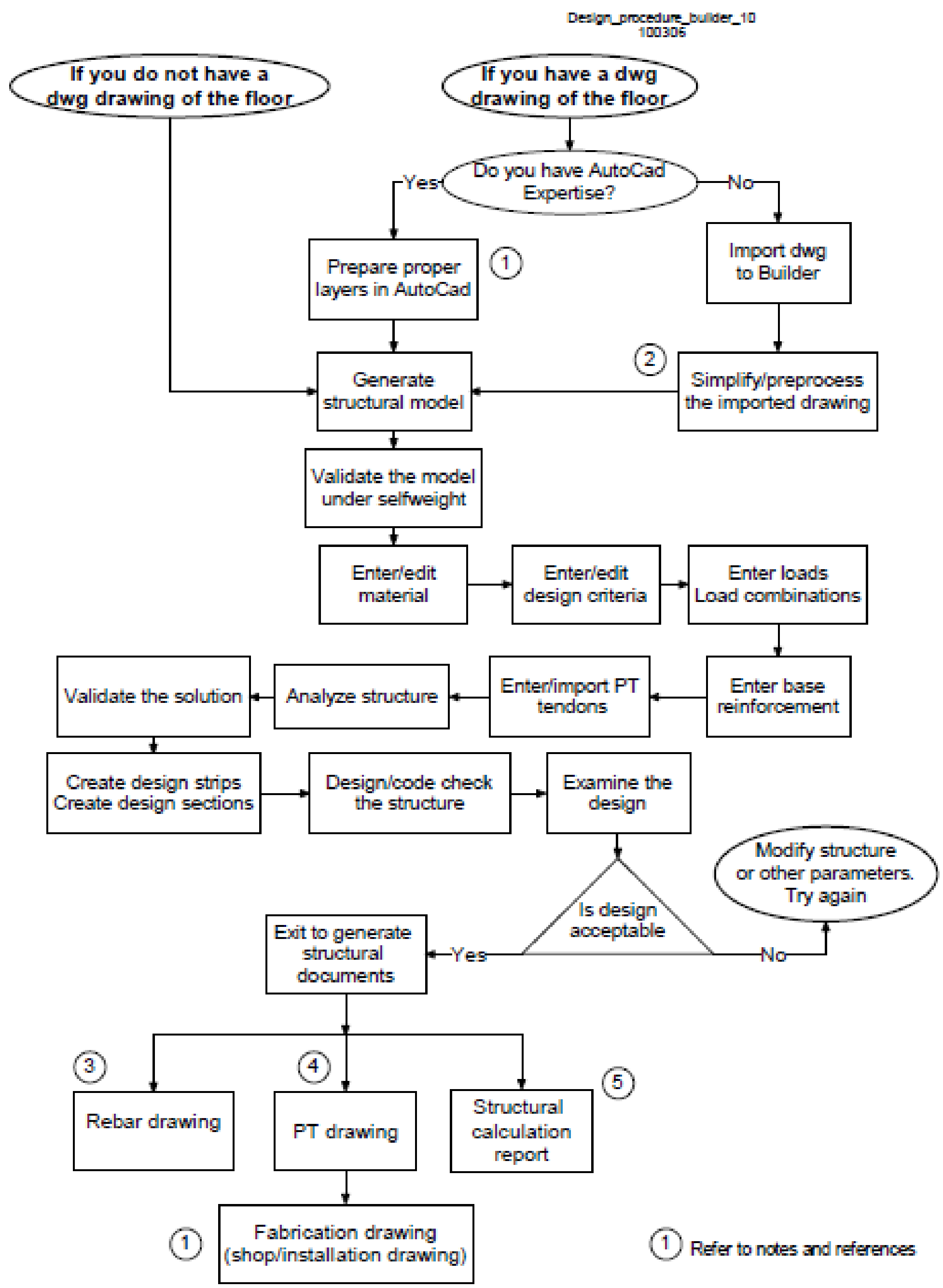
The program provides you with the option to generate structural drawings with detailed reinforcement and post-tensioning (when applicable) information for construction. These tools are described in greater detail in Chapter 10 of the **ADAPT-Floor Pro 20 Basic Manual**. The information contained in this reference applies equally to ADAPT-MAT and ADAPT-Floor Pro as part of the ADAPT-Builder platform.

- For an expeditious outcome of your design, use the “rebar generation” tool to view rebar along design strips in the X and Y directions. The program will show required, enveloped reinforcing for the top and bottom of the slab and/or beams.
- Review the generated reinforcing and edit the size, orientation, number and length of the bars, if needed.
- Add any reinforcement that you consider necessary to complete the detailing of the structure.
- View/modify the font size and line properties of the drawing suitable for the size of drawing file (.DWG) that will be exported to a CAD format.
- Export the drawing to AUTOCAD for production.

5.2.9 Generate Structural Calculation Reports

Refer to Chapter 11 of the **ADAPT-Floor Pro 20 Basic Manual**. This document contains a detailed explanation of Reports that can be produced by both ADAPT-MAT and ADAPT-Floor Pro along with a sample report. Using this as a guide or reference, prepare a similar report for your project.

FLOW CHART FOR DESIGN OF MAT (RAFT) FOUNDATION



This Page Left Intentionally

BLANK

Chapter 6

TUTORIAL

This Page Left Intentionally

BLANK

6.1 OVERVIEW

This tutorial package is tailored to the needs of design engineers who are seeking to become familiar with the latest developments in design of reinforced concrete MAT /RAFT (foundation slab consisting of extended layers of concrete and usual resting on soft ground) systems. The tutorial covers, in detail, the process of designing reinforced concrete (RC) MAT foundation using the ADAPT-Builder suite of software with focus on ADAPT-MAT. Long regarded as a difficult engineering challenge, designing concrete floor systems is greatly simplified with ADAPT-Builder, which provides significant efficiencies throughout the design process. This tutorial is broken into several sessions, which would likely to take four hours in total.

This tutorial uses the following programs of the ADAPT-Builder Design Suite:

- ADAPT-Modeler® 20
- ADAPT-MAT® 20

6.2 INTRODUCTION

This tutorial walks you through the complete modeling, analysis, design and detailing of a reinforced concrete mat/raft system. It covers the procedure to import a 2D drawing file and transform it to the required model. During the loading procedure we will use efficient options available inside ADAPT Builder environment.

The tutorial concludes with the construction drawing showing general arrangements, structural calculation reports and the non-pre-stressed reinforcement. Along the way, the tutorial will cover more advanced modeling and analysis topics within ADAPT-Builder.

The tutorial is broken down into several sessions, each intended to guide you through a specific aspect of design. The material that follows is intended for a user that has some basic familiarity with ADAPT-Builder. The step-by-step procedures outlined in each section do not contain all intermediate steps.

The raft/mat system selected for the tutorial is specifically developed, to demonstrate salient steps of RC MAT foundation design using ADAPT Builder Environment. Its overall dimensions are approximately 164 x 90 feet. The project data for this tutorial has been generated in US units.

This tutorial is based on AC318-2014 (including provisions from IBC 2015). Note that the bulk of material presented in this tutorial applies to many building codes included in the software, such as EC2, IS, Australian, Canadian and BS8110. Items such as allowable stresses, load combinations and associated factors will change depending on the code you wish to use for future designs.

6.3 DESIGN SCOPE AND CRITERIA

6.3.1 Structural Layout

This outlines the criteria to be used for the structural engineering design of a typical mat system (**Fig. 6.3-1**) of the subject matter project.

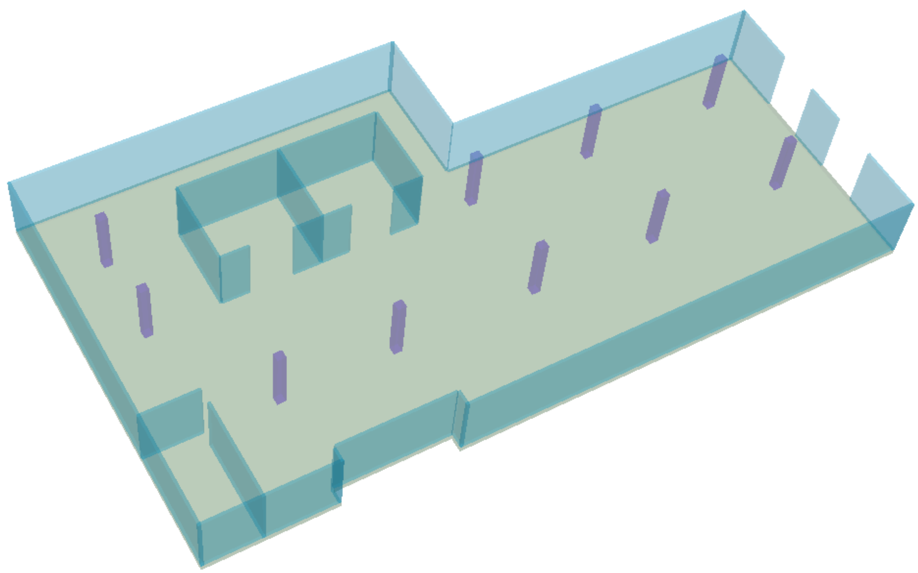


FIGURE 6.3-1 TYPICAL REINFORCE CONCRETE MAT SYSTEM

The concrete outline and the general structural plan with key dimensions are shown below:

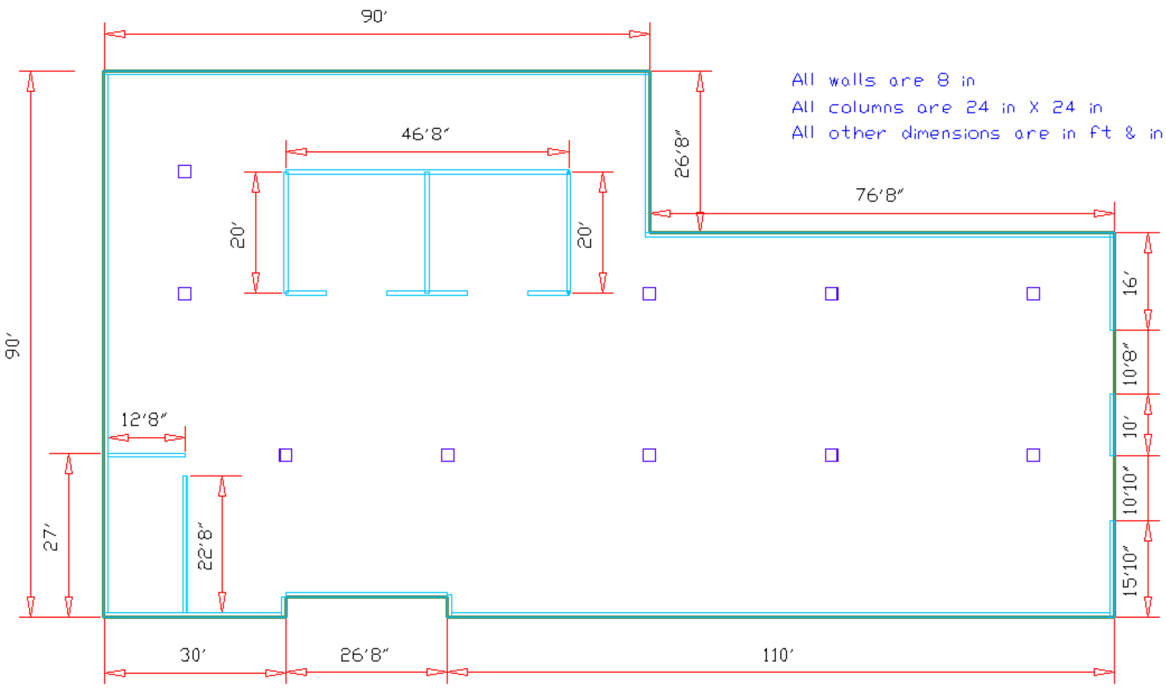


FIGURE 6.3-2 GENERAL STRUCTURAL PLAN (US)

6.3.2 Material Properties

Concrete:

Weight	= 150 pcf
Cylinder Strength (f'_c) at 28 days	= 4000 psi (slab);
	5000 psi (column & wall)
Creep Coefficient	= 2

Non-pre-stressed Reinforcement:

Yield Strength	= 60 ksi
Modulus of Elasticity	= 30,000 ksi

Soil:

Allowable Long-Term Pressure	= 2000 psf

6.3.3 Applicable Codes

The design is based on ACI318-14/IBC 2014.

6.3.4 Structural Documents

The final design should include following:

- Structural Calculation
- General Arrangement Drawings
- Loading Plans
- Design Section Report
- Rebar required and provided at all locations
- Design Section Capacity

6.3.5 Design Loads

6.3.5.1 Dead Load

Self-weight	= based on volume
Superimposed Dead Load	= 0.04 ksf on the entire raft
Line Load along the walls	= 1.37 kip/ft along edge walls = 1.7 kip/ft along other walls
Point Load (Column Reactions) load	= 56.2 kip downward axial = 18 kip along major axis = 9 kip along minor axis

6.3.5.2 Live Load

Uniformly Distributed	= 0.21 ksf

No lateral loading and any other loadings are not considered in this tutorial model. However, one may refer to the other tutorial for further clarification.

6.3.6 Load Combinations and Stresses

The parts and factors of the program's automatically generated load cases and load combinations are listed below. All combinations follow ACI 318 and IBC 2015 stipulations.

6.3.6.1 Strength Load Combinations

The strength requirement for each member is established using the following factored load combinations:

Only for Dead Load:

U = 1.40 x Selfweight + 1.40 x Dead load

For Dead and Live Load:

U = 1.20 x Selfweight + 1.20 x Dead load + 1.60 x Live load

6.3.6.2 Serviceability Load Combinations

Load Combinations for Serviceability Check:

Sustained in-service load combination (stress check)

U = 1.00 x Selfweight + 1.00 x Dead load + 0.30 x Live load

Total in-service load combination (stress check)

U = 1.00 x Selfweight + 1.00 x Dead load + 1.00 x Live load

6.3.6.3 Initial Load Combinations

Load Combinations for Initial staged check:

U = 1.00 x Selfweight

6.3.7 Deflections

The deflections will be calculated for both uncracked (gross moment of inertia) and cracked (effective moment of inertia). Long-term deflections are estimated using a creep coefficient of 2.

For the mat foundation the maximum deflections are maintained below the following values with the understanding that the structure is not attached to nonstructural elements likely to be damaged by large deflections of the slab:

Maximum allowable total long-term deflection = L/240

Maximum allowable live load deflection = L/360

Where, L = length of clear span.

Hence, Load combination for long-term deflection due to creep and the instantaneous action of live load:

U = 3.00 X Dead load + 1.00 x Live load

Load combination for checking deflection under live load:

U = 1.00 Live Load

6.3.8 Cover

Mild Reinforcement clear covers for the Raft are given below:

Cover to top bars	= 1.25 inch
Cover to bottom bars	= 1.50 inch

6.3.9 Soil Properties

Let us take an example of the structural modeling of foundations that rest on multiple soil layers and each have a different spring constant (Winkler constant).

Figure 6.3-3 shows a foundation slab on three layers of soil, each with its own spring constant k1, k2 and k3. The stiffness experienced by the foundation slab at its interface with soil (interface A in the figure) is due to the combined responses of the three underlain soil layers 1, 2 and 3.

User needs to determine the equivalent spring constant that must be specified for the determination of slab deflection and its design. For academia let us also determine the force and displacement at the interface of each of the layers.

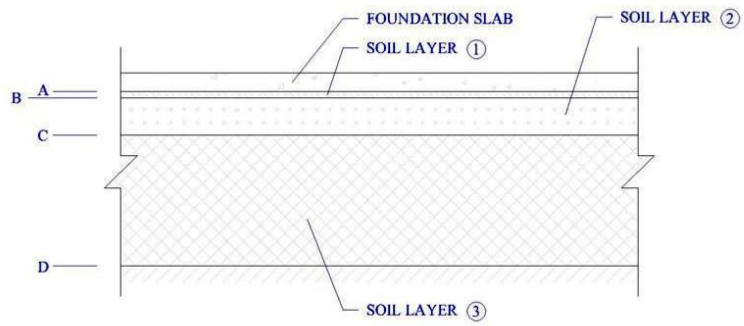


FIGURE 6.3-3 SLAB ON MULTI LAYER SOIL FOUNDATION

6.3.9.1 Equivalent Spring Constant

The equivalent spring constant for design of the foundation is the sum of the inverse of the spring constants of each of the underlain soil layers. For the condition shown in **Figure 6.3-3**, the constant to be used for the analysis of the foundation ke is given by:

1/ke = (1/k1 + 1/k2 + 1/k3)

6.3.9.2 Soil Pressure

The displacement of the foundation at its interface with the soil (interface A in **Figure 6.3-3**) is determined through the analysis of the foundation using ke. For displacement “d” at any given point, the soil pressure “p” is:

p = ke * d

The soil pressure “p” remains the same for the underlain layers. It will be the same for layers A, B and C shown in the figure.

6.3.9.3 Displacement at Interface of Soil Layers

At interface A, the vertical displacement is equal to the value determined from the analysis of the foundation slab, namely “d.”

The reduction (r) in thickness of layer 1 is:

r1 = p/k1

Hence, the vertical displacement of interface B will be:

dB = d – rA

Using a similar procedure, the displacement of the interface between other layers can be determined.

6.3.9.4 Numerical Example

Given:

A foundation slab is supported on the following:

- First layer: 4” synthetic material with spring constant 200 pci
- Second layer: 24” soil with spring constant 250 pci
- Third layer: 7 ft of native soil with spring constant 300 pci

Required:

- Determine the equivalent soil constant for the analysis of the foundation
- If the vertical displacement of the foundation at a point is obtained to be 0.138 inch, determine the force in each of the layers and the vertical displacement at the interface of each.

Solution:

The equivalent soil constant for the analysis of the foundation is:

$$1/k_e = (1/k_1 + 1/k_2 + 1/k_3)$$
$$= (1/200+ 1/250 + 1/300) = 1/81.08$$

$$k_e = 81.08 \text{ pci}$$

For a vertical displacement of 0.138 inch, the soil pressure is:

$$P = k_e * d = 81.08 * 0.138 = 11.19 \text{ psi } (1611 \text{ psf})$$

Using Winkler foundation, the pressure on all the underlain layers will be the same.

Vertical displacement at interface of soil layers:

Displacement at layer A, d_A = 0.138 inch

Compression in thickness of first layer:
$r_1 = p/k_1 = 11.19/200 = 0.056$ inch

Displacement at layer B, $d_B = d_A - r_1 = 0.138 - 0.056 = 0.082$ inch

Compression in thickness of second layer:
$r_2 = p/k_2 = 11.19/250 = 0.04476$ inch

Displacement at layer C, $d_C = d_B - r_2 = 0.82 - 0.0447 = 0.0373$ inch

Compression in thickness of third layer:
$r_3 = p/k_3 = 11.19/300 = 0.0373$ inch

Displacement at layer D, $d_D = d_C - r_3 = 0.0373 - 0.0373 = 0.00$ inch

No displacement value at interface D agrees with the assumptions of the example.

6.4 GENERATION OF 3D STRUCTURAL MODEL THROUGH DWG IMPORT

The steps to follow for the generation of a 3D structural model of the floor system through import of a drawing file are detailed below. After the initial drawing has been transformed into a structural model, the steps to import a revised drawing are outlined. The descriptions of each step are general and should be applied to any model used as an example or tutorial.

6.4.1 First Drawing Import

At this step, the simplified structural or architectural drawing will be imported to the Builder program and converted to a structural model. Follow the steps below:

- Open the Builder program in MAT mode with American unit system (as shown in **Fig. 4.2-1**).
- From the **File** ribbon select **File | Import | DXF/DWG**
- Open the desired .dwg file

- Now Project Calibration Dialog (**Figure 6.4-1**) will appear. Select **Calibrate imported objects** to enter calibration mode. The cursor will default to Snap mode.
- Before you click, make sure the **Snap to End** button is selected from the **Snap Tools** at the **Lower Quick Access Bar.**

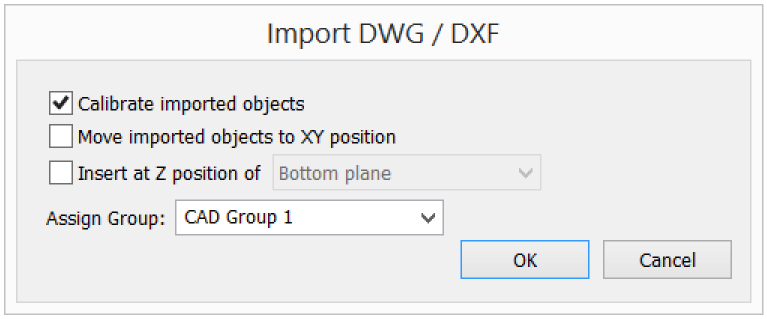
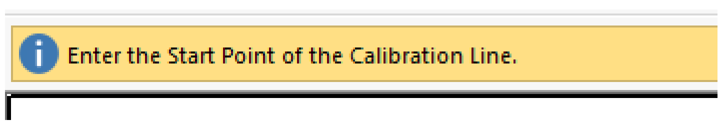


FIGURE 6.4-1 PROJECT CALIBRATION DIALOG

- Calibrate the drawing using any of the dimension lines shown in the drawing or a known distance between two points. The **User Message Bar (UMB)** will ask to **"Enter the Start Point of Calibration Line."** This is the bar below the selected modeling ribbon.



- Click on the first known point. The **UMB** will ask to **"Enter the End Point of Calibration Line."** Click on the second known point. Now it will ask to **"Enter the Correct distance in feet between the two Points you Selected."** Input the proper dimension in feet and hit **enter**. This will complete calibration of the drawing.
- For this tutorial, click **"No"** when asked if you want to change the project origin.

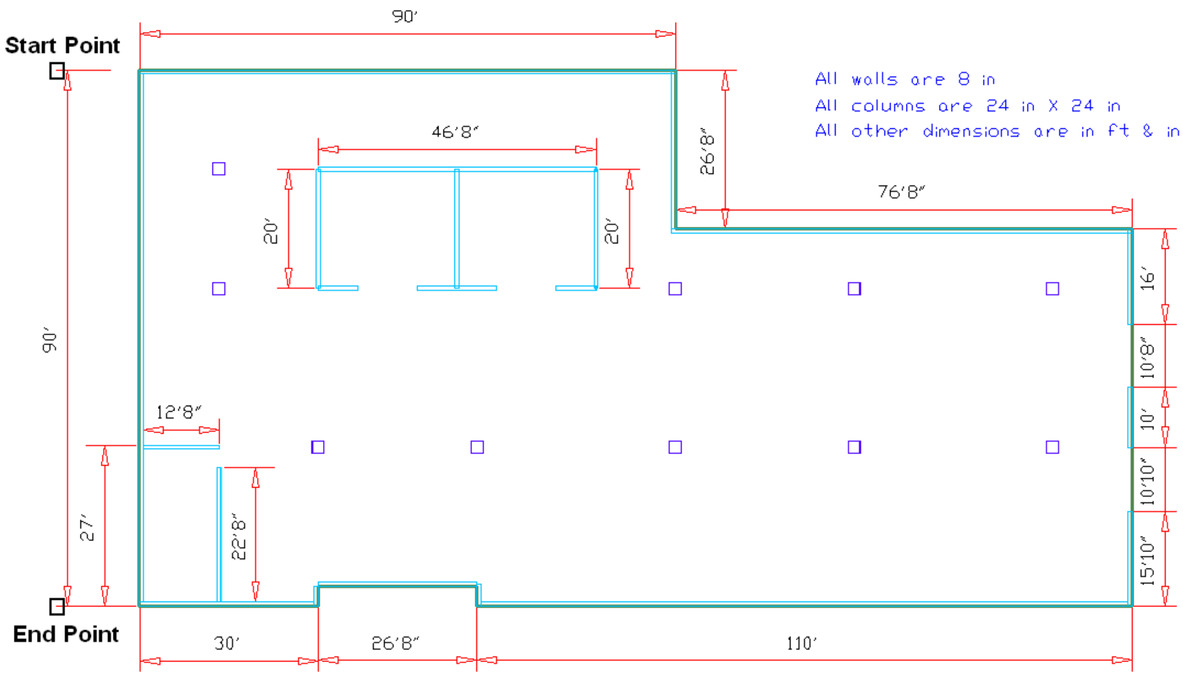


FIGURE 6.4-2 START AND END POINTS OF CALIBRATION LINE

6.4.2 Transformation of Structural Components

- Open the **Transform** tools located on the **Build** ribbon for conversion of the drawing to structural model.

FIGURE 6.4-3 TRANSFORM TOOLS

- Use ribbon item **Home | Display | Layer Settings**, to open the **Layers** dialog box. Click on the button **All Layers Off**. This will turn off all the layers in the drawing. Click on the light bulb icon in the **On/Off** column for the layer representing the structural columns **(Figure 6-4-4)** to turn on only the objects in this layer while the display of other objects remain turned off.

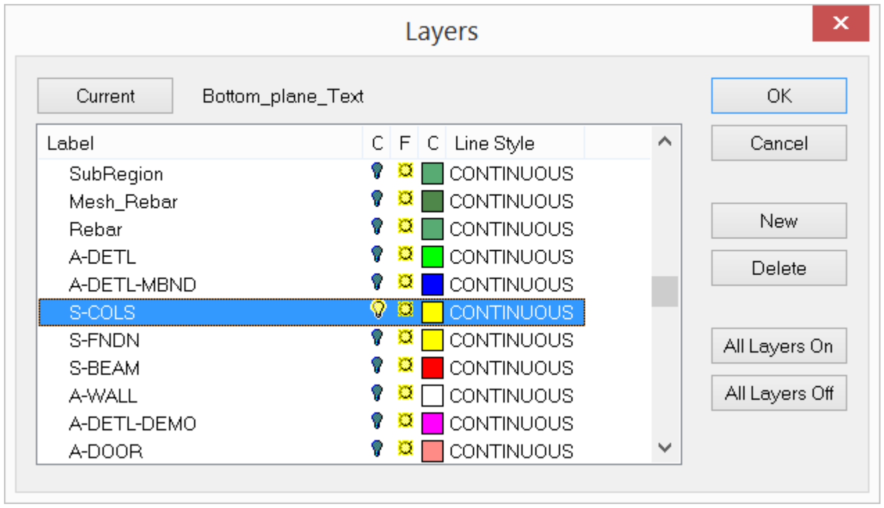


FIGURE 6.4-4 LAYERS DIALOG BOX

- Now only the polygons drawn in selected layer will be displayed in the screen. Select all of them using **Ctrl+A**. Once the column polygons are selected use the **Transform Column** icon from the **Transform tools**.

  IMPORTANT: Only polygons can be transformed using the transformation toolbar in ADAPT-Builder. If your drawing file contains components that are composed of individual lines, this tool will not work.

- Change the view to an Isometric View by selecting **Top-Front-Right View** from **Home | Camera/Zoom**. You will notice all polygons are changed to a Column entity in the drawing; you may double click on any column to change or view its General Properties, Location, FEM Properties and CAD properties (as shown in **Figure 6.4-5**). Notice that when in ADAPT-MAT mode all columns will be resting on slab, i.e. modeled as **Upper Column** and placed under **Current_Plane_Column** layer.

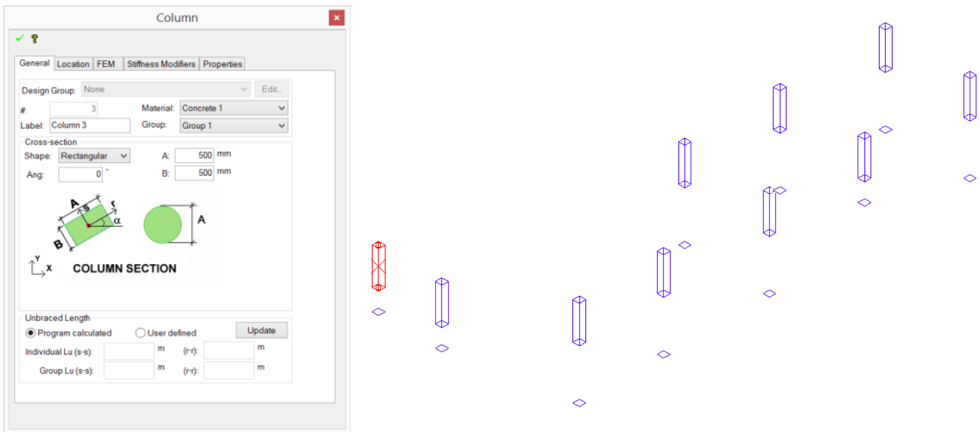


FIGURE 6.4-5 TOP-FRONT-RIGHT VIEW WITH TRANSFORMED COLUMN AND COLUMN DIALOG

- Select the menu item **Home | Display | Layer Settings**, to open *Layers* dialog box. Click on the button "**All Layers Off**". And this time turn **All Layers Off** and turn on that layer representing the structural slab layer (**Figure 6.4-4**) to display only the polygon representing the slab region(s).
- Select the polygon(s) and use **Transform Slab Region** icon from the **Transform** tools to convert the polygon to a slab. The slab will be placed as the **Current_plane_Slab_Region** layer.
- Finally open the **Layer** dialog once again. This time turn **All Layers Off** while turning on only the layer representing any walls.
- Select all the polygons (representing walls) and use the **Transform Wall** icon from the **Transform** tools to convert all polygons as walls. Walls will be placed in the **Current_plane_Slab_Region** layer. All walls will be **Upper Walls** and will be placed under the **Current_plane_Wall** layer.
- Now use the **View Settings** icon from the **Visibility** ribbon.
- This will open the **Select/Set View Items** dialog box. By default, the **Structural Components** tab will be open. Turn on the display of **Slab Region**, **Column** and **Wall** as shown in **Figure 6.4-6** and click on **"OK."** This will display all structural objects in the screen.

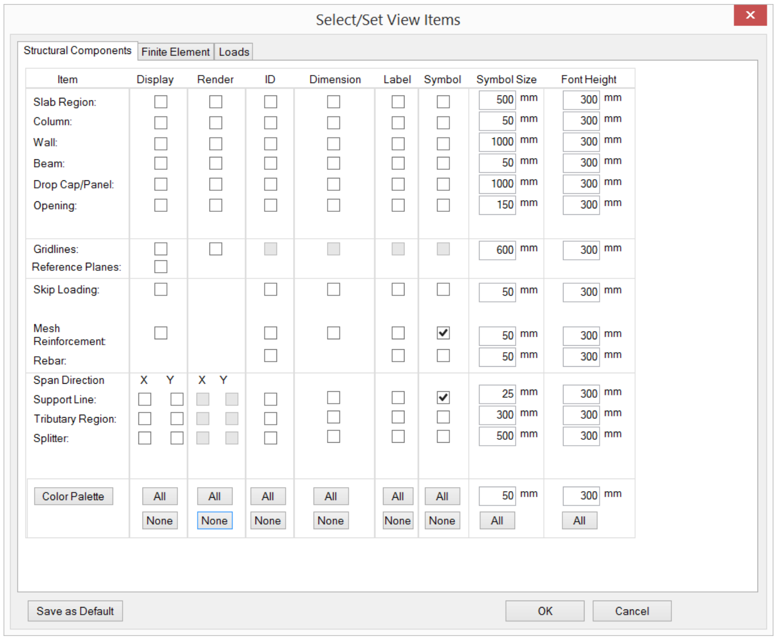


FIGURE 6.4-6 SELECT/ SET VIEW ITEMS DIALOG BOX

- Now save the file. This file contains the structural model created from the drawing file using the ADAPT-MAT environment.

6.5 MATERIAL, SOIL SUPPORT, CRITERIA AND LOADINGS

Open the previously saved file if you have closed it. If you have completed all of the steps successfully as specified in the earlier section, you may continue with the file you created and saved in your hard disk.

6.5.1 Set and Assign Material Properties

6.5.1.1 Set and Assign Multiple Concrete Materials

Since we need to specify two grades of Concrete (one for the slab, another for columns and walls), use the menu item **Concrete | Material Properties | Concrete**. The material dialog box for Concrete will open. By default, there will be one concrete already established. Click on the **Add** button to add another concrete.

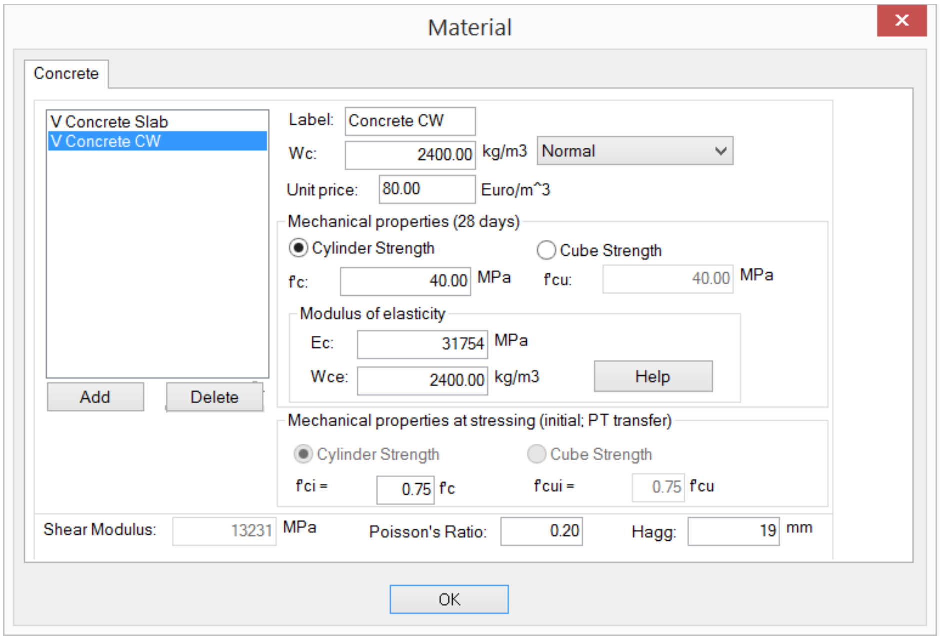


FIGURE 6.5-1 CONCRETE MATERIAL DIALOG BOX

Now select Concrete 1 and rename the label as **Concrete Slab**. Specify **Weight (Wc)** 150 pcf and 28 days **Cylinder Strength (f'c)** as 4000 psi. The **Modulus of Elasticity** of concrete is automatically calculated and displayed by the program using f'c and Wce, and the relationship as mentioned in section 8.5.1 of ACI 318-08 is given below. The user is given the option to override the code value and specify a user defined substitute. The user can specify Wce in place of Wc which will be used only to calculate Ec value.

$Ec = Wc^{1.5} \text{ X } 33 \sqrt{f'c}$ US

$Ec = Wc^{1.5} \text{ X } 0.043 \sqrt{f'c}$ SI

Where,

Ec = modulus of elasticity at 28 days [psi, MPa]

f'c = characteristic cylinder strength at 28 days

Wc = density of concrete [150 lb/ft3, 2400 kg/m3]

Similarly, select Concrete 2 and rename the label as **Concrete CW**. Specify **Weight** 150 pcf and 28 days **Cylinder Strength** as 5000 psi.

Now click on the **Select by Type** tool from the **Home** ribbon. Refer to **Chapter 4** for additional details. Select **Column** and **Wall** and click on **OK**. This will select all columns and walls in the model. Now use the ribbon item **Modify | Properties | Modify Selection**, to open the **Modify Item Properties** dialog box. Turn on **Material** (located in top left corner), select **Concrete CW** from the drop-down list (**Figure 6.5-2**). Click on **OK**, to apply this concrete for all selected entities, i.e. columns and walls.

To change the material for the slab region you can use same procedure to specify the Concrete Slab material. Alternately, double click on the slab. The **Slab Region** dialog box will open. Select the drop down for **Material** and ensure **Concrete Slab** is specified as material (**Figure 6.5-3**).

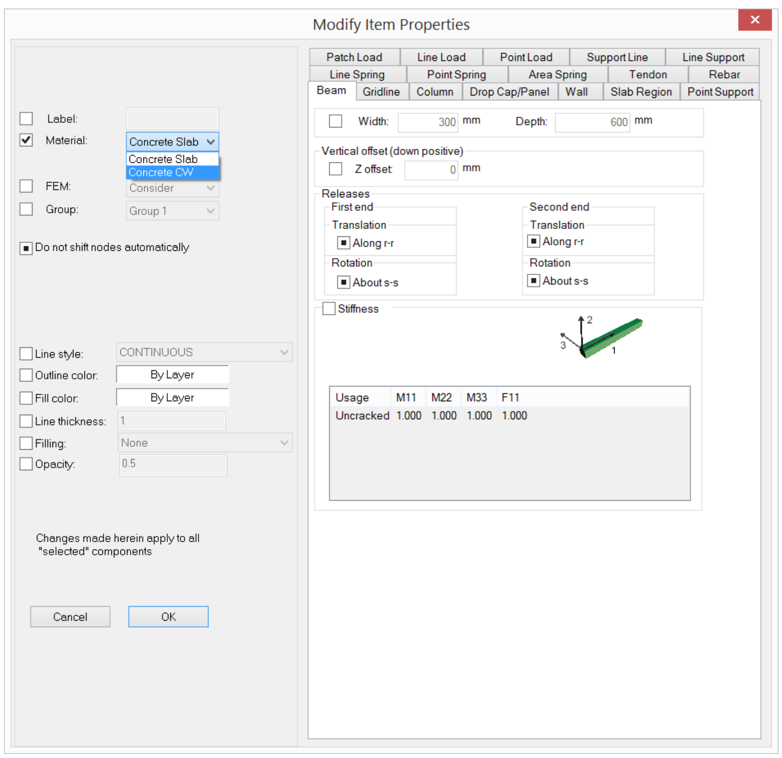


FIGURE 6.5-2 MODIFY ITEM PROPERTIES DIALOG BOX

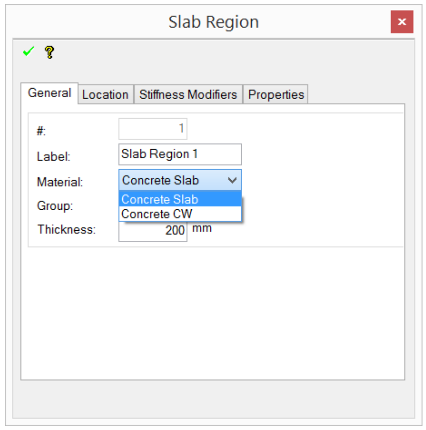


FIGURE 6.5-3 SLAB REGION DIALOG BOX

6.5.1.2 Set and Assign Mild Steel Material (Rebar)

Use the ribbon item **Criteria | Materia Properties | Rebar** to open the **Material** dialog for Mild Steel. Specify the value of fy as **60 ksi** and the value of Es as **29000 ksi**. Click **"OK."** Since this is single entry, this property will be applied for all components.

6.5.2 Assign Soil Support

There are two choices to model soil support. One may snap the corner points of the slab to model the soil support. Otherwise as we want to model soil support of uniform stiffness for the entire foundation, we can model a rectangular or quadrilateral soil support which inscribes the entire foundation. Both will give same result considering soil support below the foundation slab area only. For this tutorial, the second option will be used.

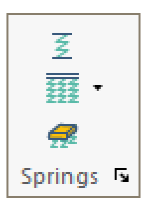


FIGURE 6.5-4 SOIL SUPPORT TOOLS

Use ribbon item **Model | Springs**. The **UMB** will ask to specify four corners. Click on four corners surrounding the slab region and press **C** to *Close/End/Accept*. Ensure that the soil spring boundary inscribes the mat area (**Figure 6.5-5**).

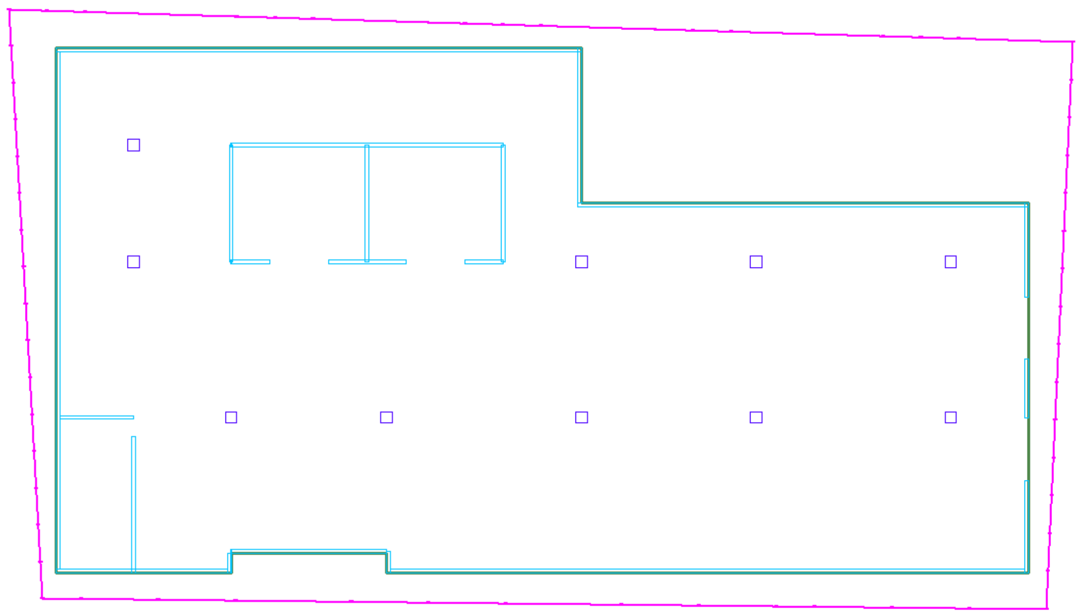


FIGURE 6.5-5 SOIL SUPPORT INSCRIBING THE MAT FOUNDATION

Now double click on the **Soil Support** to open its property box (**Fig. 6.5-6**). As computed in **Section 6.3.9.4**, specify **kza** value **81.08 pci**. Retain the spring type as **Compression Only**.

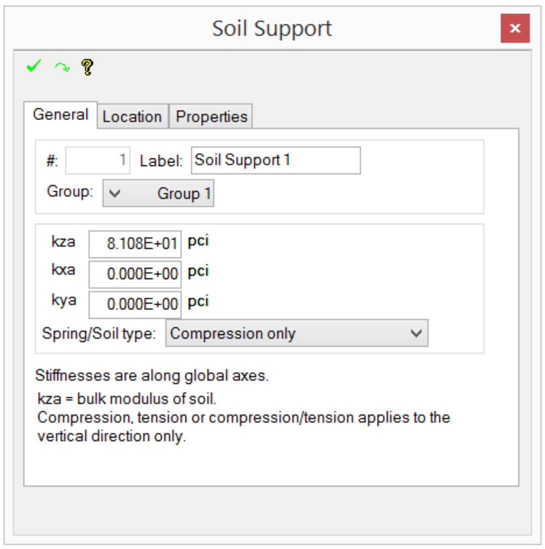


FIGURE 6.5-6 SOIL SUPPORT PROPERTY BOX

6.5.3 Set Criteria

Let us now set design criteria for this project. This should be done before we enter in loadings as automatic load combinations are generated based on the design code selected by the user.

Use the **Criteria** ribbon to open the dialog box. Go to the **Design Code** panel and select **ACI 2014/IBC 2015** for this project (**Figure 6.5-7**). Please note that some of the codes will be unavailable for selection as they do not support the US unit system.

FIGURE 6.5-7 DESIGN CODE CRITERIA

It is recommended to click on all the criteria panels and ensure that the desired parameters are set. Specific to this tutorial, go to **Rebar Size/Material** to specify preferred rebar diameters for the top bar as **#5** and for the bottom bar as **#6**.

6.5.4 Input and Assign Loadings

ADAPT-MAT will automatically consider Self-Weight as we specified Wc for the concrete materials. The program also has two reserve load cases as **Dead Load** and **Live Load**. In this tutorial, we need to specify area loads, line loads and point loads from **Loading | General.**

6.5.4.1 Patch Load Generation

Select the slab and click on the **Patch Load Wizard** tool from the **Loading | General** panel. Specify a value of **0.04 ksf** as Superimposed **Dead Load**. The program will display a confirmation dialog specifying one patch load is applied. This will apply 0.04 ksf uniformly distributed loading on the entire foundation slab.

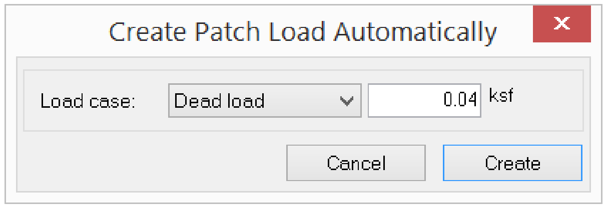


FIGURE 6.5-8 AUTOMATIC PATCH LOAD WIZARD

Again, select the slab and click on the **Patch Load Wizard** tool to specify **0.20 ksf** as **Live Load**.

6.5.4.2 Line Load Generation

Select all the boundary walls located in the model, if any. You may select all walls by using the **Select by Type** tool and then hold down the Ctrl key to de-select any internal walls. Click on the **Line Load Wizard** icon from **Loading | General** to assign **1.37 k/ft** line loading along the boundary walls under the **Dead Load** condition. The program should confirm the number of line loads generated.

Now select only the internal walls and similarly specify **2.0 k/ft** line loading for walls other than the boundary walls.

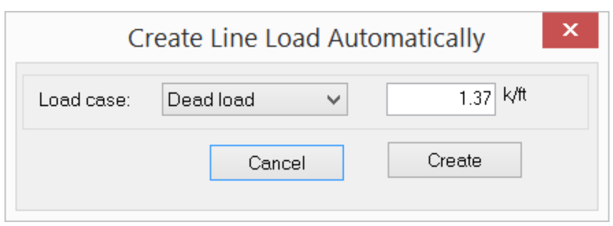


FIGURE 6.5-9 AUTOMATIC LINE LOAD WIZARD

6.5.4.3 Point Load Generation

To apply point loads as column end reactions, use the **Add Point Load** tool. Ensure that the **Snap to Endpoint** tool is activated which will allow you to snap to the column centerline(s). Click on one of the columns which will add a point load without any value. Double click on the point load to open the Point Load dialog box (**Figure 6.5-10**). Input Fz = 56.2 kip inside the **General** tab and go to **Loads** tab to input Fx = 18 kip and Fy = 9 kip. Finally click on the green checkmark to assign the load values. Once done program will remember these values for further generation(s).

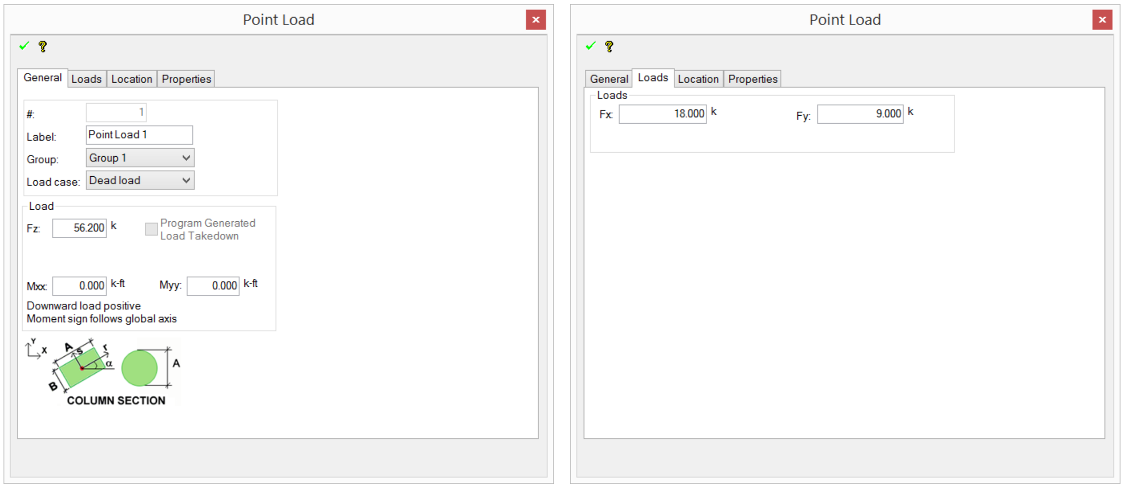


FIGURE 6.5-10 POINT LOAD DIALOG BOX (GENERAL AND LOADS TABS)

Click on the **Add Point Load** tool again and ensure that the **Snap to Endpoint** tool is activated. Snap on all other columns to apply this loading. Where column reactions are different from each other, the user needs to edit its property and change the values accordingly.

6.5.4.4 Load Combinations

Based on the code selected by the user, ADAPT-MAT automatically generates Initial, Strength and Service conditions. The user can add or modify any number of load combinations. However, please note that the program obtains a discrete solution for each load combination; hence, additional load combinations will result in prolonged processing time.

Furthermore, for a structure like this which is supported by a series of "Compression Only" springs with some finite stiffness, superposition of load cases doesn't apply. As an example, the deflection for the dead load and live load cases when calculated individually, may not add up to the deflection calculated for the combined actions.

Use the menu item **Loading | Load Cases/Combo | Load Combination** . The program will contain following load combinations:

Service (Total Load) = 1.00 x Selfweight + 1.00 x Dead load + 1.00 x Live load

Service (Sustained Load) = 1.00 x Selfweight + 1.00 x Dead load + 0.30 x Live load

Strength (Dead and Live) = 1.20 x Selfweight + 1.20 x Dead load + 1.60 x Live load

Strength (Dead Load Only) = 1.40 x Selfweight + 1.40 x Dead load

You need to create the following combinations for checking deflection due to Live Load only and Long-Term Deflection. Specify **NO CODE CHECK** under **Analysis/Design Options** as you don't need to check stress and calculate rebar requirement for these conditions.

LongTerm= 3.00 x Selfweight + 3.00 x Dead load + 1.00 x Live load

LiveLoad = 1.00 x Live Load

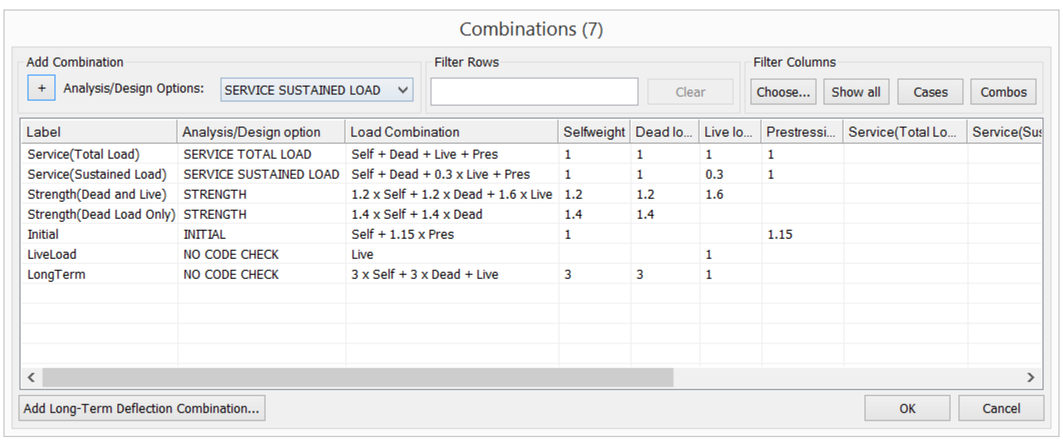


FIGURE 6.5-11 LOAD COMBINATION DIALOG BOX

Now save the file. This file contains the structural model, materials, soil support, design code, rebar specification and loadings with load combinations.

6.6 FINITE ELEMENT MESHING, ANALYSIS AND VIEW RESULTS

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in the earlier sections, you may continue with the file you created and saved in your hard disk.

The model is now ready for analysis. ADAPT-MAT supports the *Object-Oriented Modeling* approach which allows users to model the physical components as they are truly represented. Once done, the program can automatically generate a finite element mesh. From **Analysis | Meshing | Mesh Generation,** the user can specify a sparse or uniform mesh. The default is set to a sparse mesh as it will lead to a faster processing time without compromise or a degraded solution. The user can also specify the suggested cell size and node consolidation parameters. It is recommended to review Chapter 8 of the **ADAPT-Floor Pro 20 Basic Manual** for a thorough description of meshing features in ADAPT-Builder as they are also applicable to ADAPT-MAT.

6.6.1 Finite Element Meshing

The menu item **Analysis | Meshing | Mesh Generation** will be used for creation of the first mesh. In this tutorial the suggested cell size of 3 ft will be used. Turn on the **Shift nodes automatically** option if it is not already set and specify maximum distance as 1.5 ft.

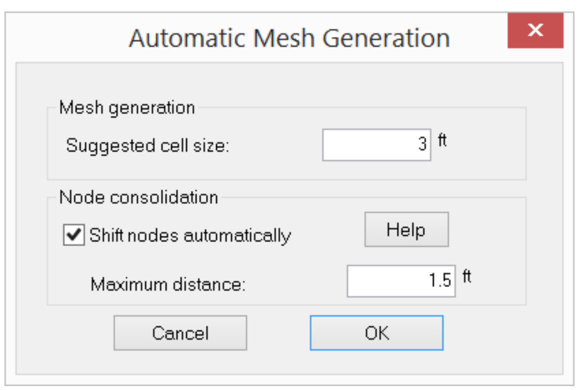


FIGURE 6.6-1 AUTOMATIC MESH GENERATION DIALOG

Click on **OK** to generate finite element mesh as shown below.

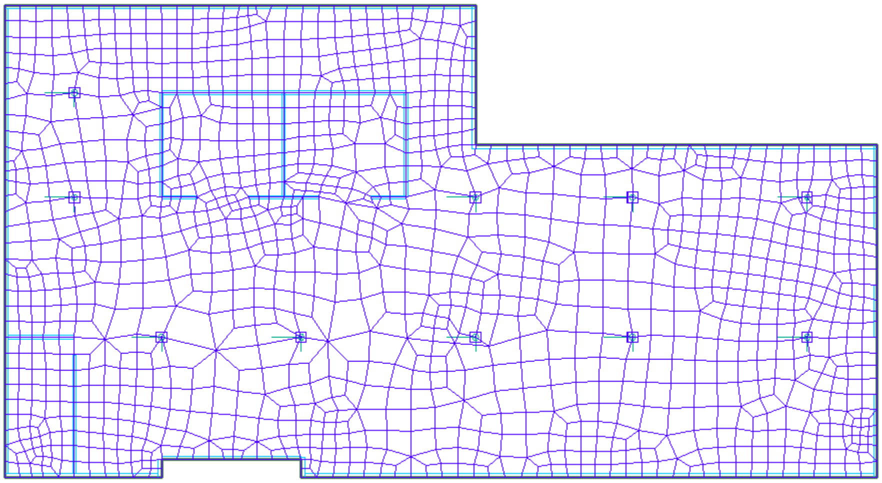


FIGURE 6.6-2 FINITE ELEMENT MESH

6.6.2 Analyze Structure

Use the menu item **Analysis | Analysis | Execute Analysis** and confirm in the subsequent dialog box (**Figure 6.6.3**), that the model is ready for analysis. For this model with soil spring supports, the program will produce iterative solutions for each load combination. Click on **Yes** to save the solution and terminate this dialog.

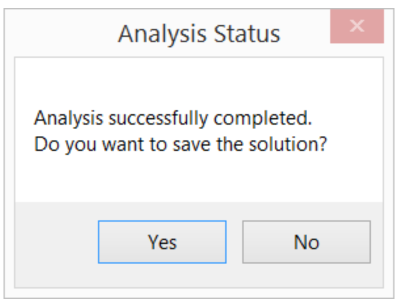


FIGURE 6.6-3 ANALYSIS STATUS DIALOG

6.6.3 View Results

Once the analysis has been successfully completed select **Analysis | Analysis | View Results.**

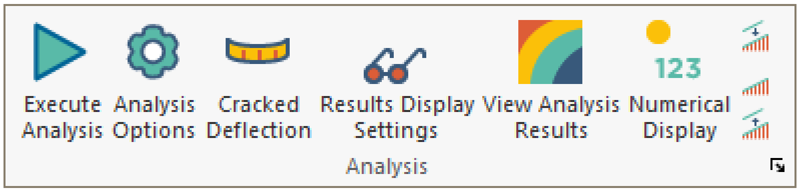
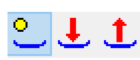


FIGURE 6.6-4 ANALYSIS PANEL

This will open the **ADViewer** (View Results) screen. From the left-hand panel of the screen select the tab **Load Case/ Combinations.**

6.6.3.1 View Deflection

- Select **Service (Total Load)** from the list of the load cases
- On the top of the same left-hand panel, click on **Results**
- From the list of results, select **Deformation** > **Z-translation**. This will display the vertical deflection of the structure.
- Click on the button **Display Results**  icon. This will display the color contour for displacement as shown in **Figure 6.6-5**.
- Use the **Warping**  and **Rotate**  tools on the screen to examine the deflected shape of the model.

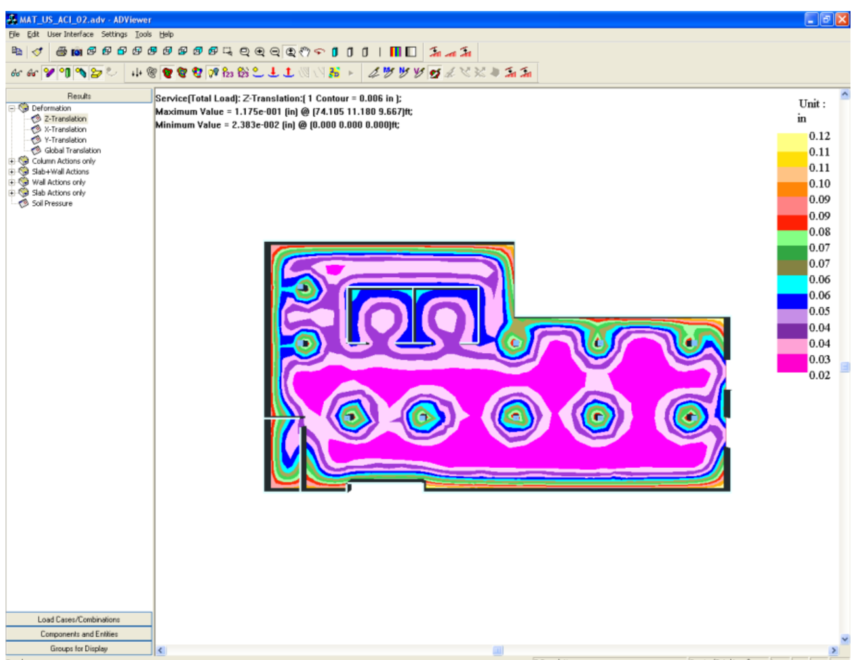


FIGURE 6.6-5 DISPLAY SCREEN OF FEM SOLUTIONS

6.6.3.2 Review of Soil Pressure

- Select Soil Pressure under the Results tab. This will display soil pressure for the selected Load Combination. Use the tab Load Case/ Combinations from left-hand panel to scroll through all combinations specified in the model. The soil pressure is reported as ksi.

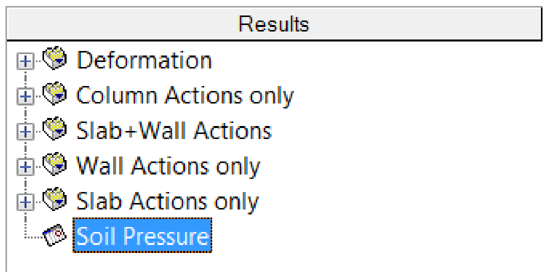


FIGURE 6.6-6 SOIL PRESSURE UNDER RESULT TAB

Important Note: *The allowable soil pressure does not apply to the pressure reported at a "point" in a contour plot, such as in* ***Figure 6.6-7****. The allowable soil pressure is intended for the average pressure over a minimum area, such as a square or circle having a diameter or side value between three to four times of the slab thickness. In the design check of this tutorial, if the point pressure is within the allowable value, the design is considered acceptable. Otherwise, using the pressure contour, the average pressure over the preceding minimum area would have had to be calculated and checked with the allowable value.*

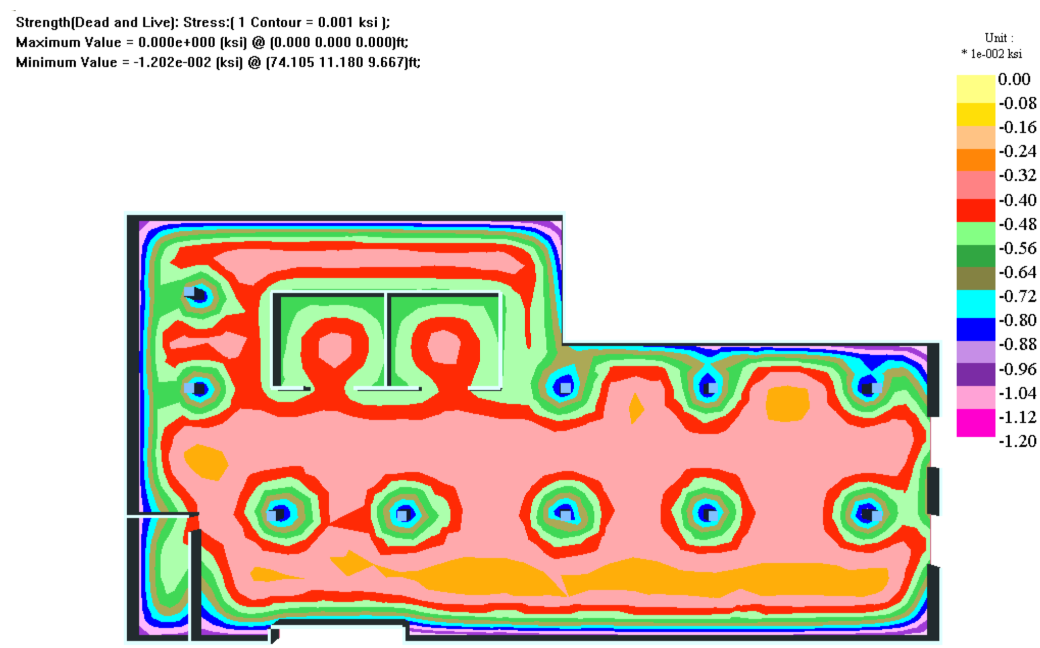


FIGURE 6.6-7 SOIL PRESSURE FOR STRENGTH CONDITION

- The user may also view other actions like Bending Moment of the slab about the X or Y global axes for different conditions. This gives an indication of the regions with maximum moments in a particular direction and possible line of cracking/ failure for the concrete slab. This might be used as a guideline to define support line for design.
- Close the ADViewer screen by using menu item File | Exit or the red X at the top-right corner. This will close the results viewer and return to the main screen of ADAPT-MAT.

6.7 GENERATION OF SUPPORT LINES AND USE OF SPLITTERS

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in earlier section, you may continue with the file you created and saved in your hard disk.

The model is now ready to be designed. The first step is to create Support Lines in both X and Y directions. These are required to establish design strips (i.e. tributary regions) for the generation of design sections. In some cases, the use of splitters is required to generate a more refined tributary region. Splitters are used when support lines do not terminate at slab or opening boundaries. Once support lines and splitters are created, the program can successfully generate design tributaries and automatically create design sections. Design sections are required to check the adequacy of the slab as it relates to the selected code for Strength and Serviceability requirements. Punching Shear (two-way shear) will also be checked for the slab. Finally, we will review the results and produce a results report.

6.7.1 Generation of Support Lines

- Support lines in the X-direction will be drawn first. Use the **Create X-Support Line** tool from **Floor Design | Strip Modeling.** Click on the **Item's Properties** icon of the **Bottom Quick Access** toolbar. A dialog box, as shown in **Figure 6.7-1,** will be displayed. Make sure the intended direction of the Support Line is set to **X-Direction**. Click the green check mark to accept the modification if not. Click the X to close the window.

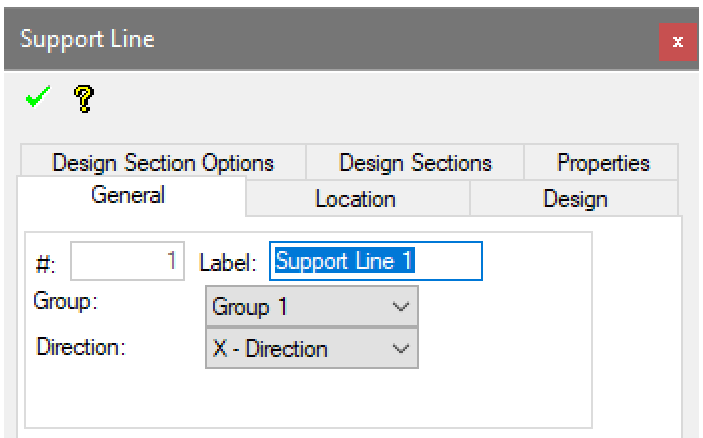


FIGURE 6.7-1 SUPPORT LINE DYNAMIC EDITOR

- Click on the slab edge where the wall end point ends at the slab edge at the top of the model. Move your mouse over to the next support in you line of supports. Keep clicking at supports until you get to the end slab boundary. Make sure to click at the centroid of columns, end points of walls and beams, centerline of transverse walls and beams. This is done to define the span lengths along the support line. Click the final vertex for the support line at the opposite slab edge, from where you started the support line, and click C to close the modeling of that support line.

- Repeat the operation to create other support lines in the X-direction. After closing the support line modeling using the **C** key on your keyboard you may start modeling your next support line. Once you have finished modeling the support lines you can press **ESC** on your keyboard to exit out of the support line modeling tool. Once you have finished entering support lines in the example model your model should look as shown in **Figure 6.7-2**

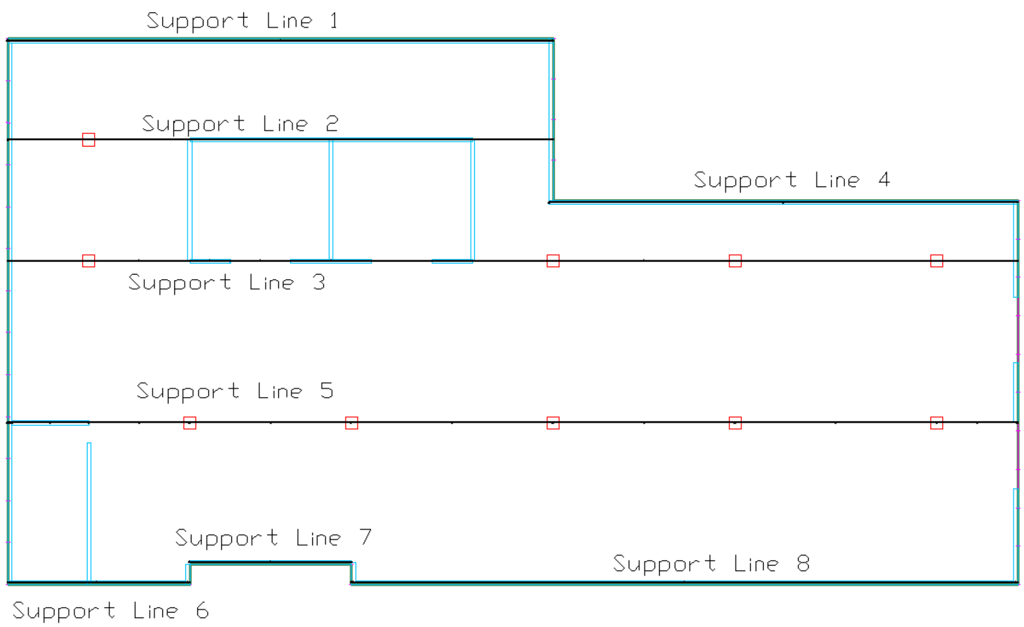


FIGURE 6.7-2 SUPPORT LINES IN X-DIRECTION

- Using the approach described above for the X-direction, draw support lines in **Y-direction**. **Figure 6.7-3** shows the layout of the Y-direction support lines once they have been entered in the model.

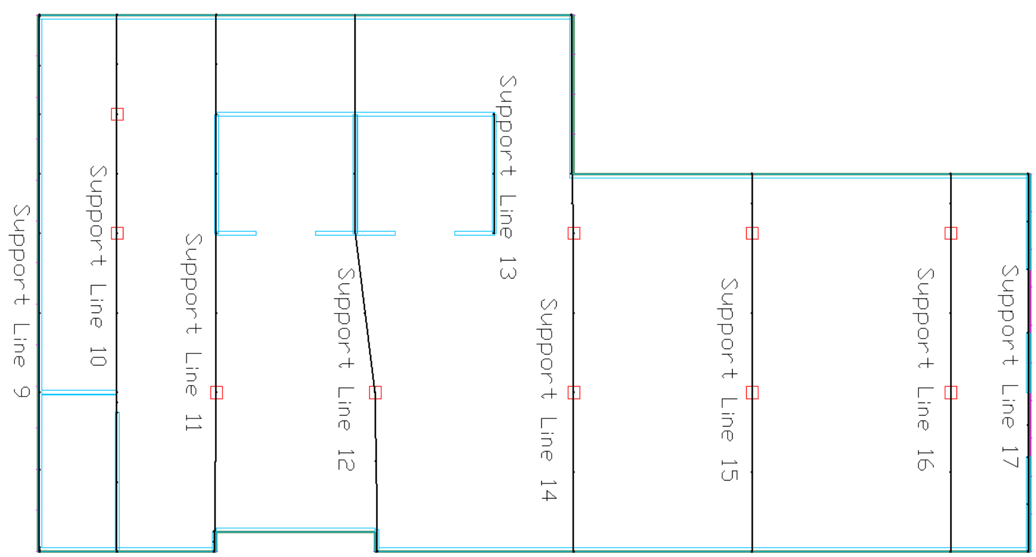


FIGURE 6.7-3 SUPPORT LINES AND SPLITTERS IN Y-DIRECTION

- Save the file. This file contains the structural model, materials, soil support, design code; rebar specification, loadings with load combinations, finite element mesh, analysis result, support lines and splitters.

6.7.2 Use of Splitters

The use of splitters changed in ADAPT-Builder 2019. The use of splitters has been simplified to creating a boundary that the tributary edge extends when the width is required to be limited to either side. Splitters are no longer required for any other purpose as the program has been improved to recognize support line nodes at any location and properly generate strips. Therefore, the use of splitter is no longer necessary in the bounds of this tutorial. For more information on the new use of splitters please refer to the **ADAPT-Builder 2019 New Features Supplement Manual**. You can find this manual by going to *Help→Documentation* in the main user interface.

6.8 PRODUCE AND REVIEW DESIGN RESULTS

Open the previously saved file if you have closed it. If you have completed all steps successfully as specified in earlier section, you may continue with the file you created and saved in your hard disk. This model is ready to proceed for design.

6.8.1 Review Analysis/ Design Options

Since the model was opened under the RC-only mode (Conventionally reinforced only), the user would have to generate column strips and middle strips to remain compliant with ACI design protocols. In ADAPT-Builder 2019, due to improvements in support line generation tools, the program no longer generated middle strips automatically. In ADAPT-Builder 20 we have reintroduced the automatic middle strips option in a different manner than previously implemented in the software. For more information on the new middle strip modeling tools refer to the **ADAPT-Builder 20 New Features Supplement Manual**. Go to **Criteria | Design Criteria | Analysis/Design Options** to open the dialog box shown in **Figure 6.8-1**.

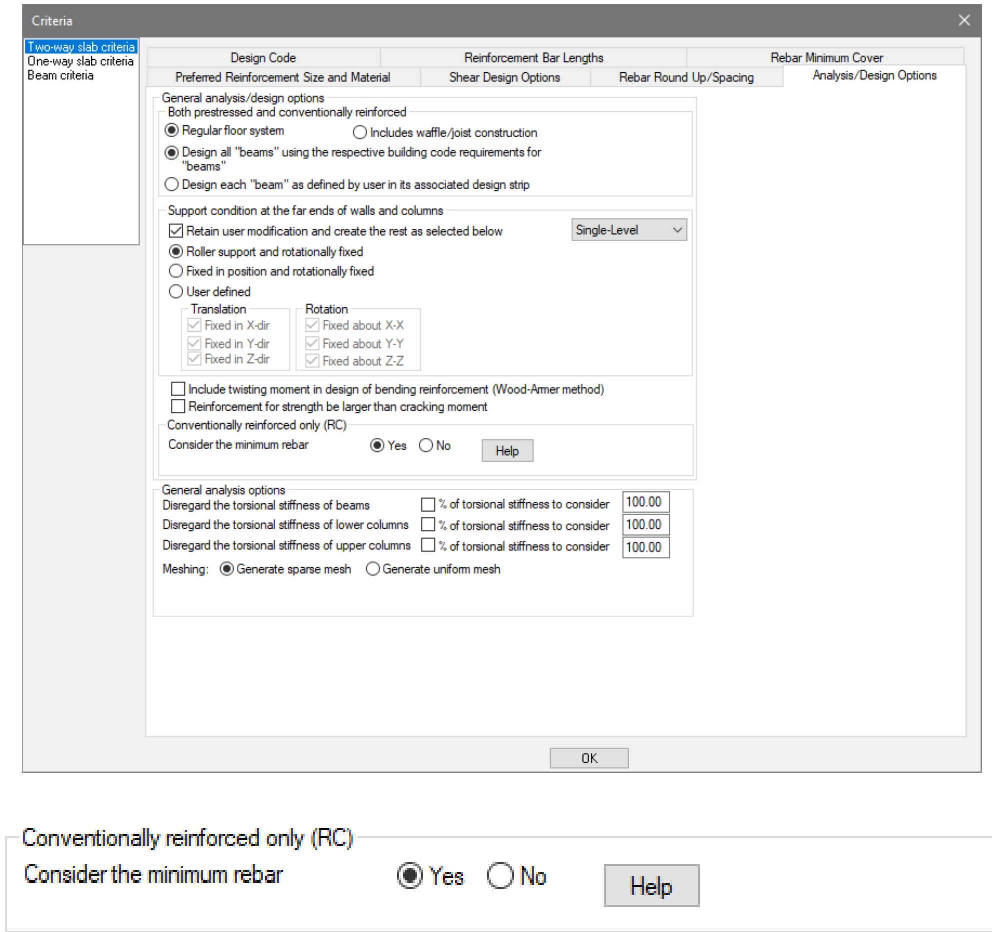


FIGURE 6.8-1 ANALYSIS/DESIGN OPTIONS FOR RC DESIGN

- Ensure that the option is selected as **Yes** from the **Conventionally reinforced only (RC)** option. The option of minimum rebar refers to the code required reinforcement for creep, shrinkage, and temperature. If this item is not selected, the program reports the reinforcement needed to satisfy those requirements for the strength limit state of the structure only.

6.8.2 Generate Design Sections

After entering the column strips in the model go to **Floor Design | Strip Modeling | Dynamic Editor** to open the dynamic editor window shown in **Figure 6.8-2**.

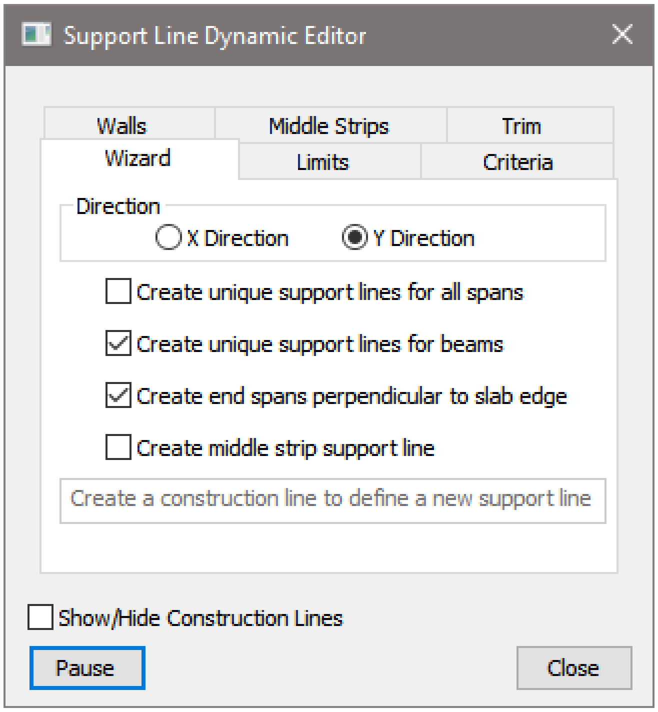


FIGURE 6.8-2 SUPPORT LINE DYNAMIC EDITOR WIZARD TAB

Click on the *Middle Strip* tab to show the middle strip options. Click on the Create Middle Strips button. The program will show middle strips that it will generate in blue as shown in **Figure 6.8-3**.

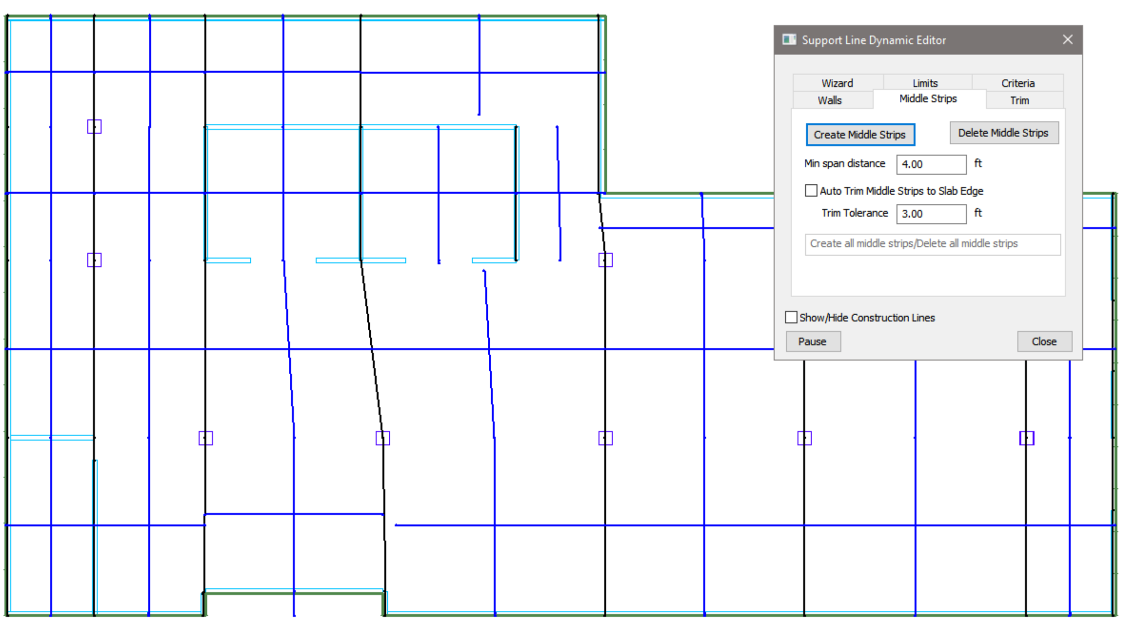


FIGURE 6.8-3 VIEW OF AUTOMATICALLY GENERATED MIDDLE STRIPS

Click the close button to accept and create the middle strips. To generate design sections at each of the support lines, use the menu item **Floor Design | Section Design | Generate Sections New** .

6.8.3 Review Design Strips (Column and Middle Strips)

- Use **Floor Design | Strip Results/Visibility | Display/Hide X- or Y-Tributaries** to view the generated column strips and middle strips in either direction as shown in **Figure 6.8-4**. Middle and column strips are shown hatched in **Figure 6.8-5** for this example.

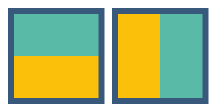


FIGURE 6.8-4 DESIGN STRIPS IN X- AND Y- DIRECTION

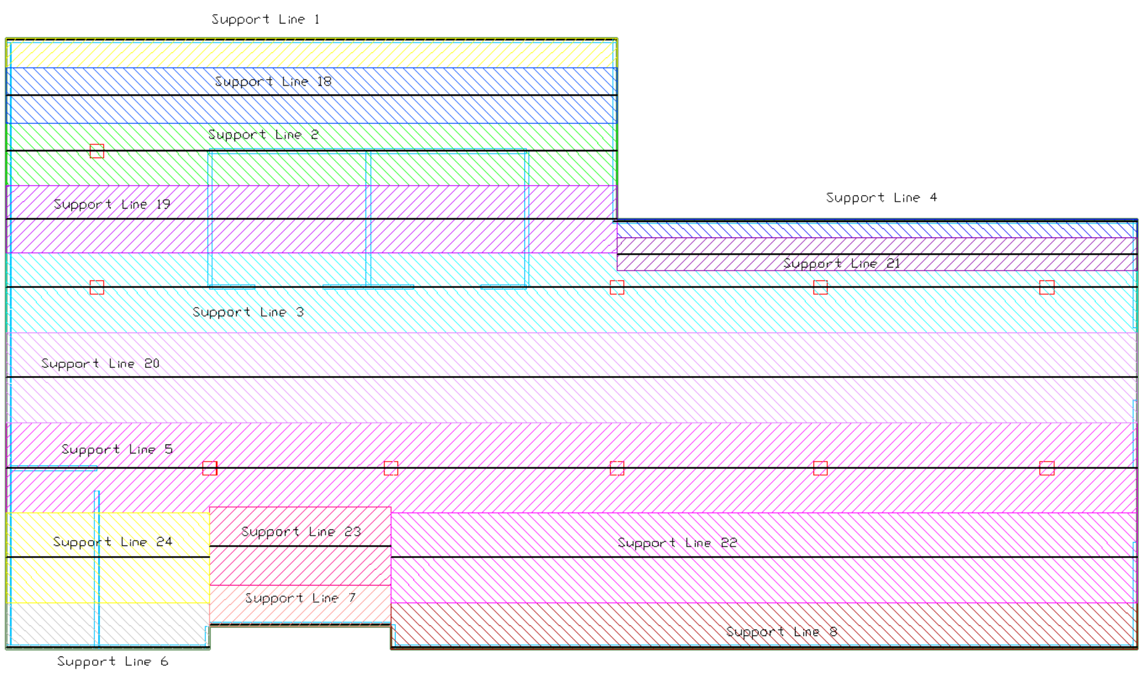


FIGURE 6.8-5 COLUMN AND MIDDLE STRIPS IN X-DIRECTION

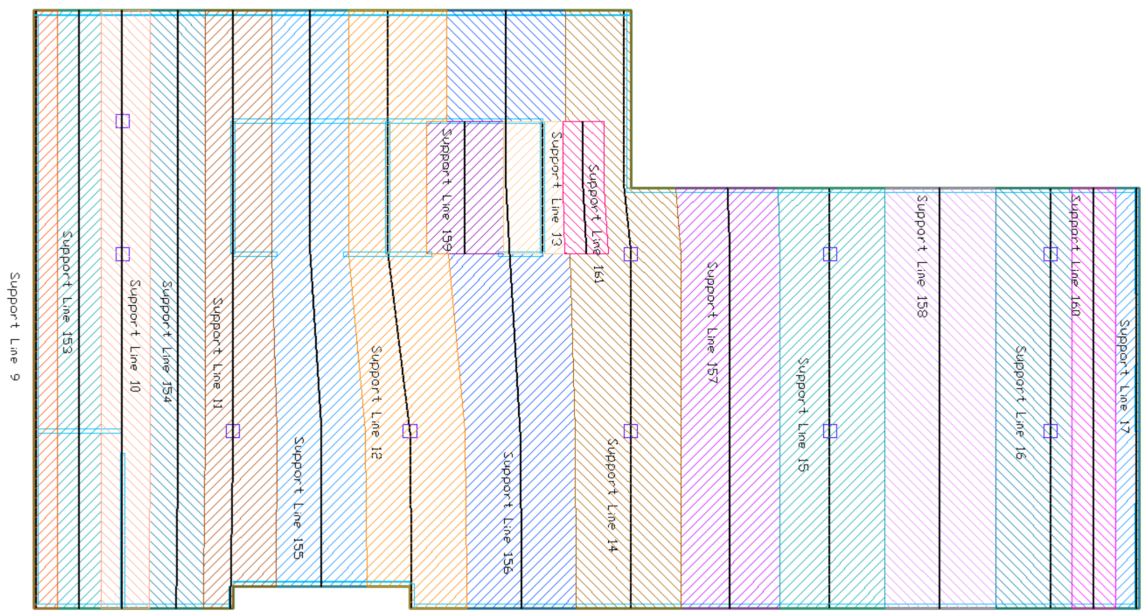


FIGURE 6.8-5 COLUMN AND MIDDLE STRIPS IN Y-DIRECTION

6.8.4 Design the Design Sections

Use **Floor Design | Section Design | Design the Design Sections** to process and produce design section actions and results.

Once the design has been successfully completed, the program will display a confirmation dialogue box as shown in **Figure 6.8-6**. Click on **Yes** to save the results and close this message.

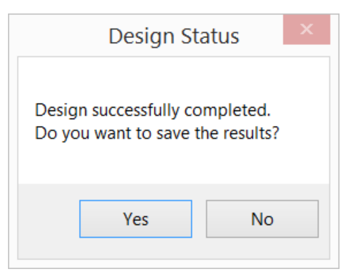


FIGURE 6.8-6 DESIGN STATUS DIALOG BOX

6.8.5 Adequacy Check for the Design Sections

- Now click on **Yes** to save the result and dismiss the **Design Status** dialog box.

- Go to **Floor Design | Strip Results/Visibility | Display Design Sections** . To view results for either design strip direction, utilize the X and Y direction strip display tools on the same tool panel.

- Click each of them to view support lines and design sections, once for the X-direction and once for the Y-direction.

- Make a cursory review of the support line results in both directions. Where design sections are shown in a green color, the sections are found adequate for the specific design check selected from the Results Browser by clicking on the **Results display Settings** icon. Any section shown as a dashed pink line has not met the required code checks and is inadequate. Typically, in RC design, for strength checks design sections will be shown as **OK** for design status (green color) as the program will design the required amount of reinforcement to satisfy the demand actions.

- For serviceability it is important to make a check for deflections in each direction. Select **Service (Total Load)** from the *Loads* tab of the *Results Browser* as shown in **Figure 6.8-7**. Change to the *Display* tab of the *Results Browser*. Set the *Maximum Span/Deflection Ratio. L/* value **240**. Change to the *Analysis* tab of the *Results Browser*. Expand the *Design Sections→Deformation* tree and select *Z-Translation*.

- From **Floor Design | Strip Results/Visibility | Display Graphically**, turn on the support lines in either direction, the program will report the deflection ratio (X/L) for each span as shown in **Figure 6.8-8**. Note also that the deflection value for each design section is shown. The same display of results can be produced for any action (i.e. moment, shear, axial) when the result to be displayed is set for an **Action** and the **Display Graphically** tool is active.

  Note: *When the program is opened in* ***RC and PT*** *mode, additional results will become active in the* ***Results Browser*** *dialogue box. The program will give the option to report top and bottom fiber stresses for any service combination as well as average precompression.*

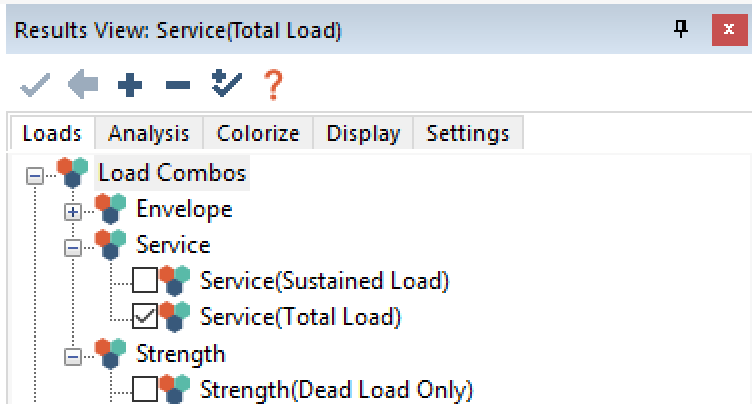


FIGURE 6.8-7 RESULT DISPLAY SETTINGS DIALOGUE BOX

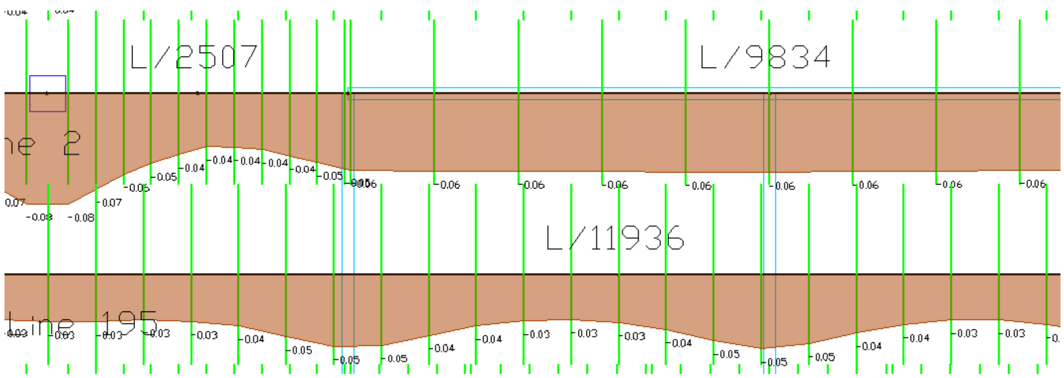


FIGURE 6.8-8 DEFLECTION RESULTS ALONG SUPPORT LINE

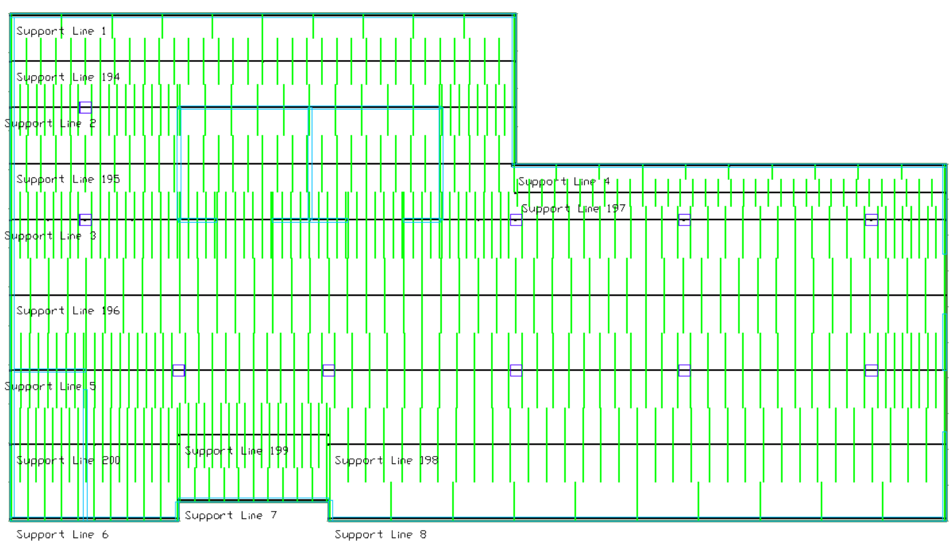


FIGURE 6.8-9 SUPPORT LINES AND DESIGN SECTIONS RESULTS IN X-DIRECTION

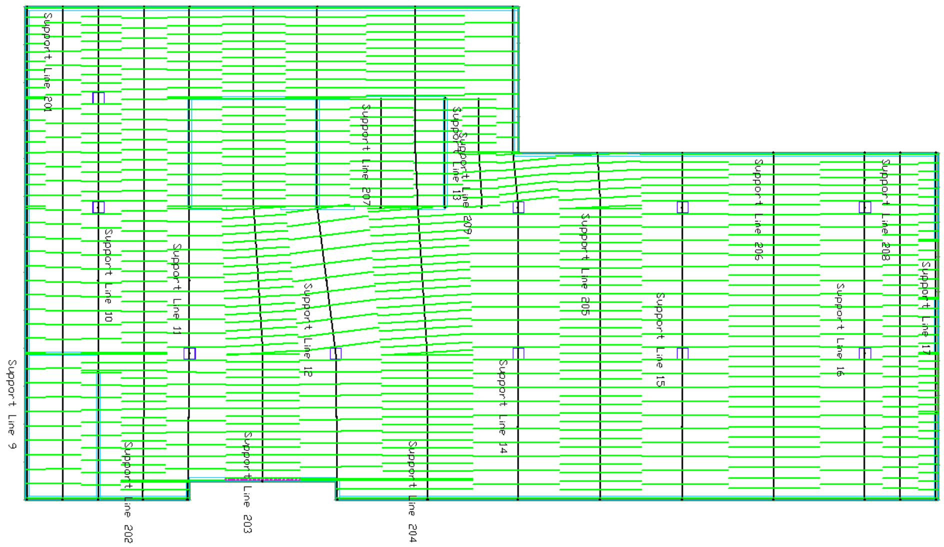


FIGURE 6.8-10 SUPPORT LINES AND DESIGN SECTIONS RESULTS IN Y-DIRECTION

- **Figures 6.8-9** and **6.8-10** shows support line example results for the graphical code check.

6.8.6 Generate Rebar Drawing

- Once the design of sections is complete, use the menu item **Floor Design | Rebar | Calculated Rebar Plan**  to display the rebar required to meet demand for serviceability, strength or envelope. The dialog box as shown in **Figure 6.8-11** will be displayed.

- The user can select the any of the load combinations defined in the model to view rebar for. Note that when the ACI318 code is selected, no rebar will be generated for serviceability conditions as the code waives temperature and shrinkage reinforcement for soil supported slabs. You may choose the **Bar Length Selection** and the **Bar Orientation**, and then click **OK**. If the orientation of the bars is **Along support lines** the reinforcing is aligned parallel to the support lines even if they are not in the X-Y directions. The selection of an angle generates rebar layouts for those directions. The **Dynamic Rebar Module** calculates the required reinforcement for the direction selected. For this tutorial, specify **Library Length** and to have desirable rebar length and bar orientation along global X and Y axis as shown in **Figure 6.8-11.**

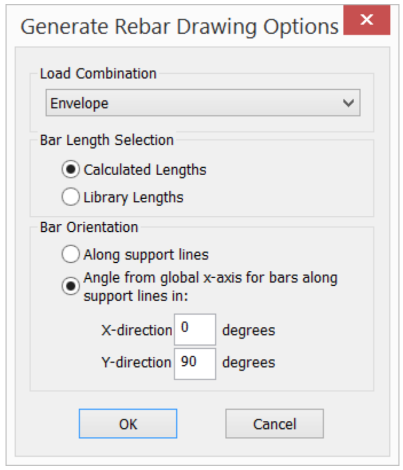


FIGURE 6.8-11 GENERATE REBAR DRAWING OPTIONS

- The program will display the requirement of rebar at the top and bottom faces of the slab and/or beams at all positions for the enveloped condition (considering strength requirement and minimum rebar for service condition when required per code) as shown in **Figure 6.8-12**.

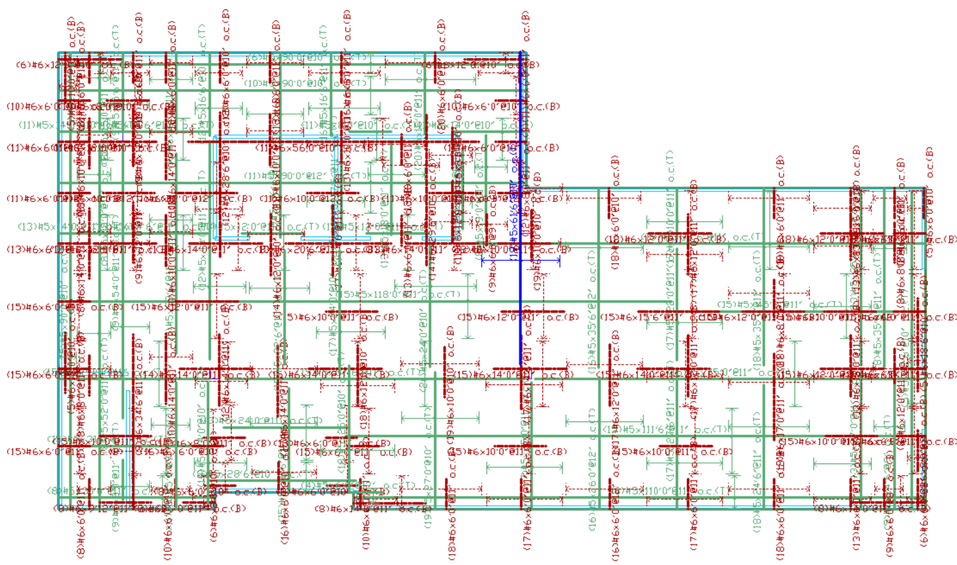


FIGURE 6.8-12 INITIAL REBAR ARRANGEMENT

- Use **Rebar | Visibility | Display Manager** to display/ hide rebar object in different layers and different directions.

6.8.7 Specify Base Reinforcement and Re-design

- It is often impractical and uneconomical to provide a design or rebar layout showing different sizes and spacing of rebar at different positions for construction. In review of the initial rebar layout for this tutorial model, it is important to consolidate size and spacing for groups of bars and determine a uniform rebar mesh, both the top and bottom layers, that will satisfy the initial requirement. Once this is done, the slab can be redesigned.

- Select the slab and click on the **Mesh Reinforcement Wizard**  tool from **Rebar | Base.**

- A dialog box as shown in **Figure 6.8-13** will be displayed. Specify the layer of reinforcement you are going to define as **Bottom** and use the **Bar size** option. For this example, specify **#6 @ 10 in c/c** in both the directions. Click on the **Create** button to add this rebar as user defined base reinforcement. Not that if you are using a different slab for the tutorial, determine what spacing and size of reinforcement is required as mesh rebar to satisfy the initial reinforcement output from the program.

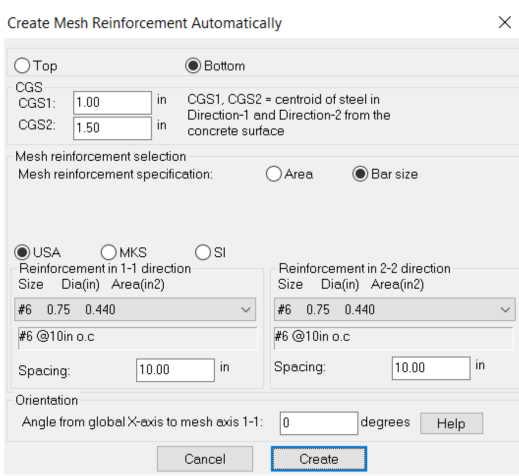


FIGURE 6.8-13 AUTOMATIC MESH REINFORCEMENT DIALOG

- Similarly, add **#5 @ 9 in c/c** as **Top** base reinforcement or what is required for the model you are using for this tutorial.

- After introducing new base reinforcing, the slab needs the design sections to be redesigned and the reinforcement layout must be generated again. The program will produce a new rebar drawing and display any additional rebar to the base reinforcing where required as shown in **Figure 6.8-14**.

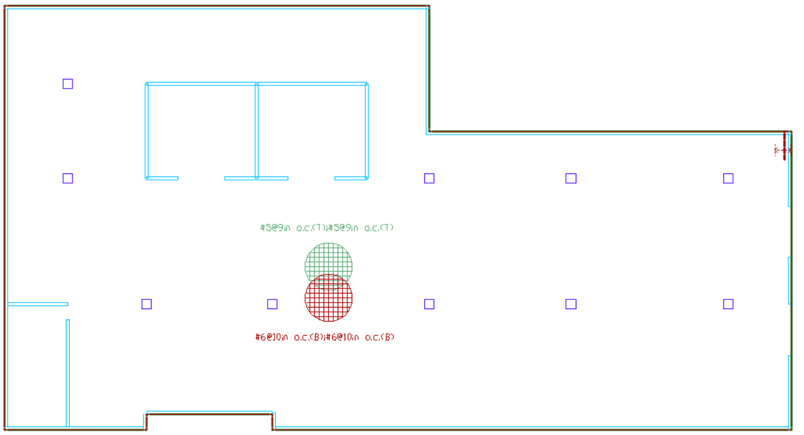


FIGURE 6.8-14 REBAR IN EXCESS OF BASE REBAR

6.8.8 Punching Shear Check

Before we perform a punching shear check we need to set the punching shear settings to be used at each column. These settings are accessible through the new Properties Grid. Select the columns in the model and expand the punching shear section of the Properties Grid to reveal the settings that can be set. **Figure 6.8.15** shows the Punching Shear section of the Property Grid for wall and column components. For more details on these options refer to the **ADAPT-Builder 20 New Features Supplement Manual**.

Punching Shear	
Method	Auto
Condition	NA
Precompression	0.125 ksi
Reinforcement type	Studs
Stud diameter	0.375 in
Number of rails along r	...
Number of rails along s	...
Spacing of studs	Uniform
Reduction of critical se...	Auto
Total Angle	0.00 °

FIGURE 6.8.15 PROPERTIES GRID PUNCHING SHEAR OPTIONS

- Specify the stud diameter as 0.375 in (#3); shear reinforcement type is stud and number of rails per side is 2. The user can modify this per column or select all columns to modify all columns at once.

- Use the menu item **Floor Design | Punching Shear | Execute Punching Shear** to perform a check of punching (two-way) shear for this slab. The program will show the following dialog when done.

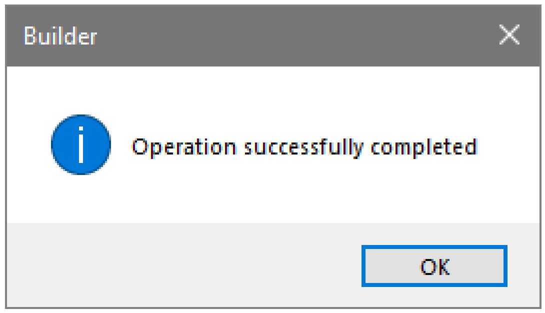


FIGURE 6.8-16 PUNCHING SHEAR CHECK COMPLETION DIALOG

- Use **Analysis | Analysis | Results Display Settings** to view the results on screen. In the *Loads* tab of the **Results Browser** select **Load Combos | Envelope | Strength Envelope.** In the *Analysis* tab of the **Results Browser** select **Punching Shear | Stress Ratio** to view the stress ratios for either local axis (‘r’ or ‘s’) direction. There are 4 cases the program can report for design status. They are as follows:

  - “OK” – calculated stresses are below allowable stresses
  - “Reinforce”- calculated stresses exceed allowable stresses and design code requires reinforcement
  - “Exceeds Code” – design code requirements are not satisfied
  - “NA”- punching shear design is not applicable

- In addition, the program will report the controlling load combination and controlling section in the design.
- The punching shear results for this model are shown in **Figure 6.8-17**.

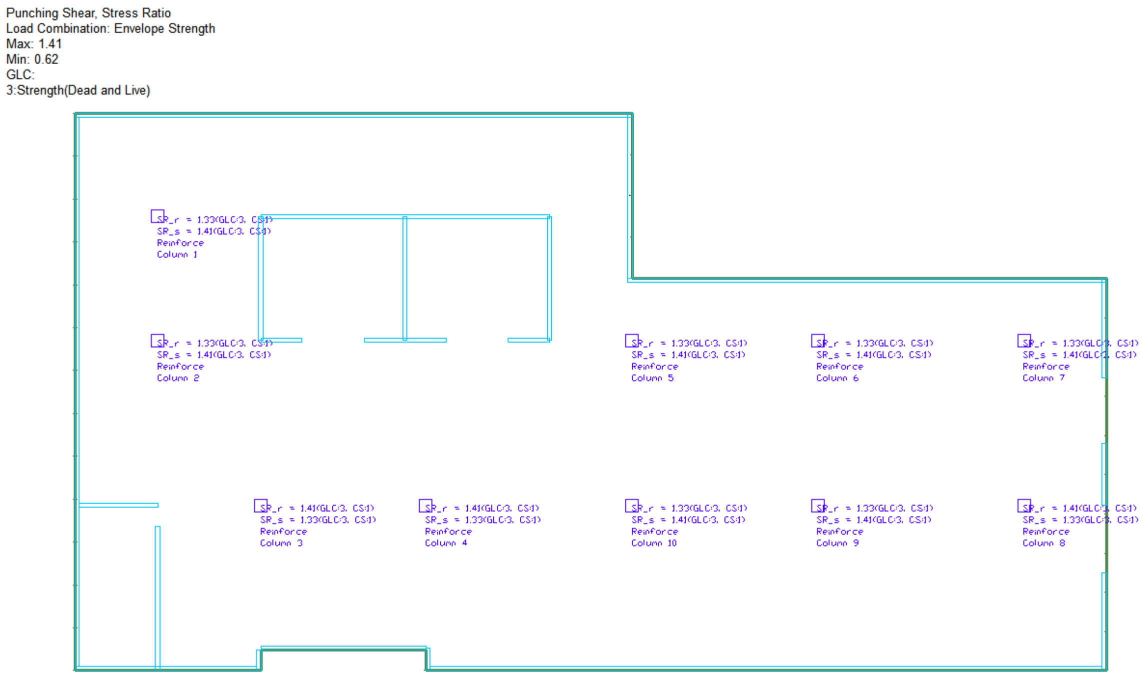


FIGURE 6.8-17 PUNCHING SHEAR DESIGN OUTCOME

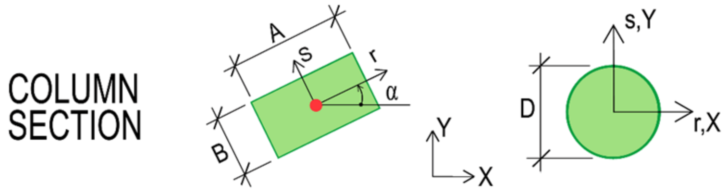
- Further information related to punching shear including design parameters, actions, actual stress from shear and moments and allowable stress are included in the tabular output. The following is a description of how to generate this output.

- From the menu select **Reports | Single Default Reports | Punching Shear.**

- The user has the following options:

    - **Punching Shear Stress Check Result**
    - **Punching Shear Stress Check Parameters, and**
    - **Punching Shear Reinforcement**

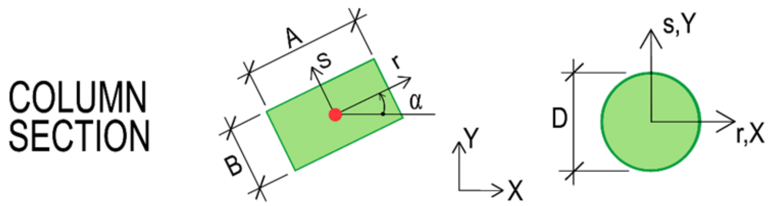
- An example of each is given in **Figures 6.8-18** to **6.8-20.**
- In addition, there is an excel spreadsheet that has more detailed punching shear results that can be accessed by selecting **Reports | Single Default Reports | Punching Shear | Punching Shear Results**.



Load Combination:Strength(Dead and Live)

Label	Condition	Axis	Factored shear	Factored moment	Stress due to shear	Stress due to moment	Total stress	Allowable stress	Stress ratio
			k	k-ft	ksi	ksi	ksi	ksi	
Column 1	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 1	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 2	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 2	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 3	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 3	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 4	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 4	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 5	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 5	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 6	Interior	rr	-74.639	0.000	0.101	0.000	0.101	0.190	0.53
Column 6	Interior	ss	-74.639	0.000	0.101	0.000	0.101	0.190	0.53

FIGURE 6.8-18 PUNCHING SHEAR STRESS CHECK RESULTS



Label	Condition	Axis	Effective depth	Design length rr	Design length ss	Design area	Section constant	Gamma
			in	in	in	in2	in4	
Column 1	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 1	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 2	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 2	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 3	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 3	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 4	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 4	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 5	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 5	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 6	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 6	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 7	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 7	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 8	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 8	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 9	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 9	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 10	Interior	rr	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40
Column 10	Interior	ss	6.13	30.13	30.13	7.38E+002	1.13E+005	0.40

FIGURE 6.8-19 PUNCHING SHEAR STRESS CHECK PARAMETERS

In this example since the stress ratio for all columns are less than 1, shear reinforcement is not required. To obtain a solution which requires shear reinforcement, in order to display shear reinforcement output, the Strength combination load factors were increased. The results from this design are shown in **Figure 6.8-20**.

180.90 SCHEDULE OF STUD RAILS FOR PUNCHING SHEAR REINFORCEMENT

Load Combination: Envelope Strength

Label	Column ID	Number of rails per side(rr,ss)	Number of rails	Stud diameter (in)	Rail Length(in)	#Studs@spacing(in)
Column 1	1	2,2	8	0.38	25.50	17@1.3
Column 2	2	2,2	8	0.38	25.50	17@1.3
Column 3	3	2,2	8	0.38	25.50	17@1.3
Column 4	4	2,2	8	0.38	25.50	17@1.3
Column 5	5	2,2	8	0.38	25.50	17@1.3
Column 6	6	2,2	8	0.38	25.50	17@1.3
Column 7	7	2,2	8	0.38	25.50	17@1.3
Column 8	8	2,2	8	0.38	25.50	17@1.3
Column 9	9	2,2	8	0.38	25.50	17@1.3
Column 10	10	2,2	8	0.38	25.50	17@1.3

Notes:

10@100 = rail placement at face of support.

10@100* = rail placement at face of drop.

Refer to details for arrangement.

FIGURE 6.8-20 PUNCHING SHEAR REINFORCEMENT

Chapter 7

ANALYTICAL BACKGROUND

This Page Left Intentionally

BLANK

This section provides information on several aspects of the analysis and design features of ADAPT-MAT.

7.1 OVERVIEW

ADAPT-MAT is a standalone computer program within the umbrella of the ADAPT-Builder software platform. It has been entirely developed by ADAPT's team of structural engineers and software developers in providing a complete analysis and design solution for mat and raft foundations. The program is the most comprehensive three-dimensional (3D) finite element (FEM) analysis and design tool for conventionally reinforced or post-tensioned mat foundations, with or without grade beams. Using ADAPT's intuitive and easy-to-use Component Technology, it is the only solution available that also calculates the required reinforcement at all locations of the foundation automatically.

Using ADAPT-MAT, you can quickly generate a graphical model of a ground supported slab mat with or without grade beams. The model can be generated using either an available DWG or DXF file of the foundation, or through user input. The mat can be of any irregular shape and subject to any kind of loading from above. Using an adaptive meshing and finite elements, the program analyzes the mat and determines the location, amount, and length of all the reinforcement needed by computation. Where applicable, under horizontal loading or moments, the program accounts for the separation of a mat from the soil.

7.2 STRUCTURAL MODELING

7.2.1 Analysis

The analysis processor of ADAPT-MAT is based on the finite element formulation developed and implemented in the ADAPT-Builder platform, with some modifications to cover the features that are specific to mat foundation analysis. The program uses almost entirely well-proportioned quadrilateral flat shell elements with bending and membrane degrees of freedom. Details of the formulation are given in the **ADAPT-Floor Pro 20 Basic Manual** and its references. Walls are also represented by the same type of shell elements.

Beams and columns are modeled as beam (stick) elements with six degrees of freedom at each node.

Unlike many other commercially available programs, complete compatibility of displacement is established over the entire foundation system and among all its components. For example, grade beams below the foundations are modeled eccentric to the slab = as they appear in real life = but are solved with full compatibility of displacement at their interface with the slab (equal strains in beam stem and slab at a common interface).

Post-tensioned tendons, where present, are discretized into segments associated with each shell element they traverse. When force calculation is invoked by you, the force along each tendon varies as passes from one shell element into the next.

The advanced and unique features of ADAPT-MAT have become possible due to a finite element formulation specifically developed for analysis of complex concrete structures, including post-tensioning [Aalami, 2003].

7.2.2 Design

The design involves (i) the calculation of "design values" (demand), (ii) the comparison with allowable limits of the building code selected by the user, and (iii) provisions of reinforcement, where applicable.

The design values are determined in a manner like an elevated slab and are dependent on the generation of design sections associated with design strips (tributary regions) evolving from support lines. The procedure for creation of support lines and design sections is explained in detail in the **ADAPT-Floor Pro 20 Basic Manual**. The important point to note is that for each design section the builder platform determines the design values from the equilibrium of the finite element nodes, as opposed to the common practice of using the integration of stresses along a cut at the section. As a result, accurate design values are obtained for a relatively coarse finite element mesh [see ADAPT Technical Note TN302, **"Evaluation of Design Values at Design Sections Using ADAPT-Builder Platform"**].

7.3 MISCELLANEOUS TOPICS

7.3.1 Soil Pressure

In addition to the values reported for slabs (stresses, moments, etc.), as in ADAPT-Floor Pro, the program also reports the distribution of soil pressure below a slab as indicated in **Figure 7.3-1**. The following in the interpretation and evaluation of soil pressure is noteworthy.

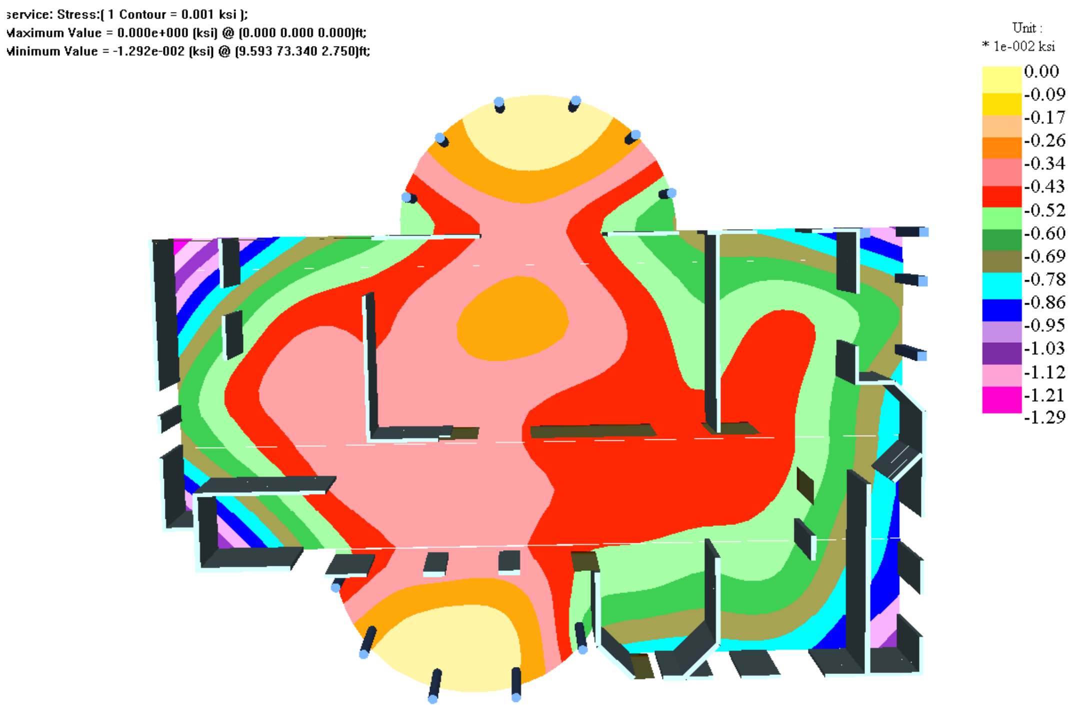


FIGURE 7.3-1 EXAMPLE OF THE DISTRIBUTION OF SOIL PRESSURE BELOW A MAT WITH FULL SOIL/MAT INTERFACE CONTACT

The raw data obtained from a finite element analysis, as shown in the figure above, is the distribution of stress at "points" below the foundation slab. From a practical point of view, however a high or low value of soil pressure at a "point" does not reflect the likely response of the soil that is of interest to design engineer. For an engineering evaluation, when dealing with a reinforced concrete slab resting freely on soil, one considers the average pressure over a minimum area of design significance. For concrete slabs resting on common soil[1] a minimum diameter four to five times the slab thickness should be considered. In other words, at the location of design check, the distribution of soil stress reported below the slab, should be integrated over a "design" significant area to determine the total force. The total force over the "design" patch when divided over the area of the patch will yield the design stress to be compared with the allowable soil pressure for the soil.

[1] Bulk modulus 100 to 200 pci

7.3.2 Superposition

In the general case, the principle of superposition of solutions obtained for different load cases does not apply to mat foundation slabs. Each solution obtained for a load combination is unique since a different set of boundary conditions applies to each. Even though the solutions obtained for the mat foundations are based on elastic material properties, the different amount of separation of soil from the mat between any two load cases creates a difference between the structural systems that carries the load in each of the two load cases. Once the structural systems changes, superposition does not apply.