ADAPT-MAT

USER MANUAL

Copyright © September 2018
# LIST OF CONTENTS

## OVERVIEW ................................................................. 1

## BASIC FEATURES .......................................................... 5

| 2.1 | OVERVIEW .................................................................. 7 |
| 2.2 | GEOMETRY ................................................................. 7 |
| 2.3 | SUPPORT CONDITIONS ................................................. 11 |
| 2.3.1 | Soil Support Area ....................................................... 11 |
| 2.3.2 | Compression Only Soil ............................................... 12 |
| 2.3.3 | Soil / Rock Anchors .................................................... 12 |
| 2.3.4 | Grade Beam Support ................................................... 12 |
| 2.3.5 | Line Springs ............................................................... 13 |
| 2.3.6 | Point Springs .............................................................. 14 |
| 2.3.7 | Point Supports ............................................................ 14 |
| 2.3.8 | Line Supports ............................................................. 14 |
| 2.3.9 | Piles ........................................................................... 14 |
| 2.3.10 | Voids in Soil ............................................................. 15 |
| 2.4 | MATERIAL PROPERTIES .................................................. 15 |

## LOADS ........................................................................ 15

| 2.5.1 | Load Cases .................................................................. 15 |
| 2.5.2 | Load Combinations ..................................................... 15 |

## BASE REINFORCEMENT .................................................. 16

## POST-TENSIONING ......................................................... 16

## ANALYSIS ..................................................................... 16

## DESIGN ........................................................................ 18

## GENERATION OF DRAWINGS ......................................... 18

## LINK WITH 3RD PARTY PROGRAMS AND ADAPT DATA EXCHANGE... 19

## QUICK START ............................................................... 21

| 3.1 | OVERVIEW .................................................................. 23 |
| 3.2 | OPENING THE PROGRAM ............................................. 23 |
| 3.3 | QUICK START – STEP-BY-STEP.................................... 24 |
| 3.3.1 | Create a Grid ............................................................... 24 |
| 3.3.2 | Building the Model ..................................................... 25 |
| 3.3.3 | Viewing the Model .................................................... 26 |
| 3.3.4 | Defining the Soil Support ............................................ 27 |
| 3.3.5 | Application of Loads ................................................ 27 |
| 3.3.6 | Mesh and Analyze ..................................................... 29 |
| 3.3.7 | Viewing Analysis Results .......................................... 29 |

## USER INTERFACE ......................................................... 33
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>OVERVIEW OF ADAPT-BUILDER PLATFORM</td>
<td>35</td>
</tr>
<tr>
<td>4.2</td>
<td>ADAPT-BUILDER SPLASH SCREEN</td>
<td>35</td>
</tr>
<tr>
<td>4.3</td>
<td>ADAPT-BUILDER GRAPHICAL USER INTERFACE</td>
<td>37</td>
</tr>
<tr>
<td>4.3.1</td>
<td>Mouse Function and Operation</td>
<td>38</td>
</tr>
<tr>
<td>4.3.2</td>
<td>Main User Interface</td>
<td>40</td>
</tr>
<tr>
<td>4.3.2.1</td>
<td>Top-Left Quick-Access Default Tools (1)</td>
<td>40</td>
</tr>
<tr>
<td>4.3.2.2</td>
<td>Top-Right Level Tools (2)</td>
<td>41</td>
</tr>
<tr>
<td>4.3.2.3</td>
<td>Message Bar (3)</td>
<td>42</td>
</tr>
<tr>
<td>4.3.2.4</td>
<td>Bottom Quick Access Tools (4)</td>
<td>43</td>
</tr>
<tr>
<td>4.3.2.5</td>
<td>Bottom-Right – Status Bar (5)</td>
<td>43</td>
</tr>
<tr>
<td>4.3.3</td>
<td>File Ribbon</td>
<td>44</td>
</tr>
<tr>
<td>4.3.4</td>
<td>Home Ribbon</td>
<td>45</td>
</tr>
<tr>
<td>4.3.4.1</td>
<td>Display Panel</td>
<td>46</td>
</tr>
<tr>
<td>4.3.4.2</td>
<td>Selection Mode Panel</td>
<td>47</td>
</tr>
<tr>
<td>4.3.4.3</td>
<td>Selection Tools Panel</td>
<td>48</td>
</tr>
<tr>
<td>4.3.4.4</td>
<td>Draw Panel</td>
<td>48</td>
</tr>
<tr>
<td>4.3.4.5</td>
<td>Tools Panel</td>
<td>52</td>
</tr>
<tr>
<td>4.3.4.6</td>
<td>Zoom/Camera Panel</td>
<td>52</td>
</tr>
<tr>
<td>4.3.4.7</td>
<td>Viewport Panel</td>
<td>54</td>
</tr>
<tr>
<td>4.3.4.8</td>
<td>WCS/UCS (Coordinate Systems) Panel</td>
<td>55</td>
</tr>
<tr>
<td>4.3.4.9</td>
<td>Scaling Panel</td>
<td>56</td>
</tr>
<tr>
<td>4.3.5</td>
<td>Visibility Ribbon</td>
<td>56</td>
</tr>
<tr>
<td>4.3.5.1</td>
<td>Message Bar Panel</td>
<td>57</td>
</tr>
<tr>
<td>4.3.5.2</td>
<td>Colors Panel</td>
<td>57</td>
</tr>
<tr>
<td>4.3.5.4</td>
<td>Render Panel</td>
<td>58</td>
</tr>
<tr>
<td>4.3.5.5</td>
<td>Display Settings Panel</td>
<td>58</td>
</tr>
<tr>
<td>4.3.5.6</td>
<td>Selection Panel</td>
<td>58</td>
</tr>
<tr>
<td>4.3.5.7</td>
<td>Viewing Panel</td>
<td>59</td>
</tr>
<tr>
<td>4.3.5.8</td>
<td>Shading Panel</td>
<td>59</td>
</tr>
<tr>
<td>4.3.6</td>
<td>Modify Ribbon</td>
<td>60</td>
</tr>
<tr>
<td>4.3.6.1</td>
<td>Properties Panel</td>
<td>60</td>
</tr>
<tr>
<td>4.3.6.2</td>
<td>Edit Panel</td>
<td>61</td>
</tr>
<tr>
<td>4.3.6.3</td>
<td>Copy/Move Panel</td>
<td>61</td>
</tr>
<tr>
<td>4.3.6.4</td>
<td>Tools Panel</td>
<td>63</td>
</tr>
<tr>
<td>4.3.6.5</td>
<td>Blocks Panel</td>
<td>64</td>
</tr>
<tr>
<td>4.3.6.6</td>
<td>Add/Remove Points Panel</td>
<td>64</td>
</tr>
<tr>
<td>4.3.6.7</td>
<td>Labels Panel</td>
<td>64</td>
</tr>
<tr>
<td>4.3.7.1</td>
<td>Material Properties Panel</td>
<td>65</td>
</tr>
<tr>
<td>4.3.7.2</td>
<td>Design Criteria Panel</td>
<td>66</td>
</tr>
<tr>
<td>4.3.7.3</td>
<td>Design Criteria (SOG) Panel</td>
<td>68</td>
</tr>
<tr>
<td>4.3.8</td>
<td>Model Ribbon</td>
<td>69</td>
</tr>
<tr>
<td>Section</td>
<td>Panel Name</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>------------</td>
<td>------</td>
</tr>
<tr>
<td>4.3.8.1</td>
<td>Wizards Panel</td>
<td>69</td>
</tr>
<tr>
<td>4.3.8.2</td>
<td>Level Assignment Panel</td>
<td>69</td>
</tr>
<tr>
<td>4.3.8.3</td>
<td>Gridline Panel</td>
<td>69</td>
</tr>
<tr>
<td>4.3.8.4</td>
<td>Type Manager Panel</td>
<td>70</td>
</tr>
<tr>
<td>4.3.8.5</td>
<td>Structural Components Panel</td>
<td>70</td>
</tr>
<tr>
<td>4.3.8.6</td>
<td>Preprocessing Panel</td>
<td>72</td>
</tr>
<tr>
<td>4.3.8.7</td>
<td>Preprocessing (SOG) Panel (in SOG only)</td>
<td>72</td>
</tr>
<tr>
<td>4.3.8.8</td>
<td>Supports Panel</td>
<td>73</td>
</tr>
<tr>
<td>4.3.8.9</td>
<td>Springs Panel</td>
<td>73</td>
</tr>
<tr>
<td>4.3.8.10</td>
<td>Displacements Panel (in SOG only)</td>
<td>74</td>
</tr>
<tr>
<td>4.3.8.11</td>
<td>Visibility Panel</td>
<td>75</td>
</tr>
<tr>
<td>4.3.8.12</td>
<td>Properties Panel</td>
<td>76</td>
</tr>
<tr>
<td>4.3.8.13</td>
<td>Transform Panel</td>
<td>77</td>
</tr>
<tr>
<td>4.3.9</td>
<td>Loading Ribbon</td>
<td>78</td>
</tr>
<tr>
<td>4.3.9.1</td>
<td>Load Combo Panel</td>
<td>78</td>
</tr>
<tr>
<td>4.3.9.2</td>
<td>LL Reduction Panel</td>
<td>80</td>
</tr>
<tr>
<td>4.3.9.3</td>
<td>General Panel</td>
<td>80</td>
</tr>
<tr>
<td>4.3.9.4</td>
<td>Lateral/Building Panel</td>
<td>81</td>
</tr>
<tr>
<td>4.3.9.5</td>
<td>Tributary Panel</td>
<td>82</td>
</tr>
<tr>
<td>4.3.9.6</td>
<td>Pattern Panel</td>
<td>83</td>
</tr>
<tr>
<td>4.3.9.7</td>
<td>Visibility Panel</td>
<td>83</td>
</tr>
<tr>
<td>4.3.10</td>
<td>Tendon Ribbon</td>
<td>83</td>
</tr>
<tr>
<td>4.3.10.1</td>
<td>Criteria Panel</td>
<td>84</td>
</tr>
<tr>
<td>4.3.10.2</td>
<td>Settings Panel</td>
<td>84</td>
</tr>
<tr>
<td>4.3.10.3</td>
<td>Model Panel</td>
<td>84</td>
</tr>
<tr>
<td>4.3.10.4</td>
<td>Modify Panel</td>
<td>85</td>
</tr>
<tr>
<td>4.3.10.5</td>
<td>Shop Drawing Panel (add-on Module)</td>
<td>86</td>
</tr>
<tr>
<td>4.3.10.6</td>
<td>Visibility Panel</td>
<td>87</td>
</tr>
<tr>
<td>4.3.10.7</td>
<td>Design Panel</td>
<td>87</td>
</tr>
<tr>
<td>4.3.11</td>
<td>Rebar Ribbon</td>
<td>88</td>
</tr>
<tr>
<td>4.3.11.1</td>
<td>Design Criteria Panel</td>
<td>88</td>
</tr>
<tr>
<td>4.3.11.2</td>
<td>Model Base Rebar Panel</td>
<td>89</td>
</tr>
<tr>
<td>4.3.11.3</td>
<td>Generate Panel</td>
<td>90</td>
</tr>
<tr>
<td>4.3.11.4</td>
<td>Visibility Panel</td>
<td>90</td>
</tr>
<tr>
<td>4.3.11.5</td>
<td>Reporting Panel</td>
<td>90</td>
</tr>
<tr>
<td>4.3.12</td>
<td>Analysis Ribbon</td>
<td>91</td>
</tr>
<tr>
<td>4.3.12.1</td>
<td>Vibration Panel</td>
<td>91</td>
</tr>
<tr>
<td>4.3.12.2</td>
<td>Meshing Panel</td>
<td>91</td>
</tr>
<tr>
<td>4.3.12.3</td>
<td>Analysis Panel</td>
<td>93</td>
</tr>
<tr>
<td>4.3.12.4</td>
<td>Contour Settings Panel</td>
<td>94</td>
</tr>
<tr>
<td>4.3.12.5</td>
<td>Warping Panel</td>
<td>94</td>
</tr>
</tbody>
</table>
### 4.3.12.6 Reactions Panel ................................................................. 95
### 4.3.12.7 Visibility Panel ................................................................. 95
### 4.3.13 Floor Design Ribbon ............................................................ 97
  - 4.3.13.1 Punching Shear Panel .................................................... 97
  - 4.3.13.2 Strip Modeling Panel ..................................................... 97
  - 4.3.13.3 Section Design Panel ..................................................... 99
  - 4.3.13.4 Rebar Panel ................................................................. 99
  - 4.3.13.5 Tools Panel ................................................................. 100
  - 4.3.13.6 Steel FRC Panel (MAT Only) ......................................... 100
  - 4.3.13.7 Strip Results/Visibility Panel ........................................ 100
### 4.3.14 PT/RC Export Ribbon ............................................................ 103
  - 4.3.14.1 Material Properties Panel .............................................. 103
  - 4.3.14.2 Design Criteria/Settings Panel ....................................... 103
  - 4.3.14.3 Load Factors Panel ...................................................... 104
  - 4.3.14.4 Strip Modeling Panel ................................................... 104
  - 4.3.14.5 Design Strips Panel ..................................................... 106
  - 4.3.14.6 Export Panel ............................................................... 106
### 4.3.15 Column Design Ribbon .......................................................... 107
  - 4.3.15.1 Type Manager Panel ..................................................... 107
  - 4.3.15.2 Settings Panel ............................................................ 107
  - 4.3.15.3 Live Load Reduction Ribbon .......................................... 107
  - 4.3.15.4 Design Panel .............................................................. 108
  - 4.3.15.5 Reports Panel ............................................................ 109
  - 4.3.15.6 Labels Panel ............................................................... 110
### 4.3.16 Wall Design Ribbon ............................................................. 111
  - 4.3.16.1 Settings Panel ............................................................ 111
  - 4.3.16.2 Sections Panel .......................................................... 111
  - 4.3.16.3 Design Panel .............................................................. 112
  - 4.3.16.4 Reports ................................................................. 112
  - 4.3.16.5 Labels Panel ............................................................... 114
### 4.3.17 Reports Ribbon ................................................................. 114
  - 4.3.17.1 Print Panel ............................................................... 114
  - 4.3.17.2 Compiled Reports ...................................................... 115
  - 4.3.17.3 Single Default Reports Panel ....................................... 115
  - 4.3.17.4 Analysis Reports Panel ............................................... 124
  - 4.3.17.5 Export DWG Panel .................................................... 127

### MODELING AND DESIGN PROCESS ................................................. 129

**5.1 OVERVIEW** ................................................................. 131

**5.2 DESIGN PROCEDURE** ....................................................... 131
  - 5.2.1 Create the Structural Model ............................................. 131
  - 5.2.2 Define Soil Support Conditions ........................................ 132
# LIST OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.2.3</td>
<td>Validate the Structural Model</td>
<td>132</td>
</tr>
<tr>
<td>5.2.4</td>
<td>Complete and Finalize Input Data</td>
<td>132</td>
</tr>
<tr>
<td>5.2.5</td>
<td>Perform Analysis</td>
<td>133</td>
</tr>
<tr>
<td>5.2.6</td>
<td>Prepare to Design</td>
<td>133</td>
</tr>
<tr>
<td>5.2.7</td>
<td>Validate the Code Compliance of the Design</td>
<td>133</td>
</tr>
<tr>
<td>5.2.8</td>
<td>Generate Structural Drawings</td>
<td>134</td>
</tr>
<tr>
<td>5.2.9</td>
<td>Generate Structural Calculation Reports</td>
<td>134</td>
</tr>
</tbody>
</table>

## TUTORIAL

<table>
<thead>
<tr>
<th>TUTORIAL</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>OVERVIEW</td>
</tr>
<tr>
<td>6.2</td>
<td>INTRODUCTION</td>
</tr>
<tr>
<td>6.3</td>
<td>DESIGN SCOPE AND CRITERIA</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Structural Layout</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Material Properties</td>
</tr>
<tr>
<td>6.3.3</td>
<td>Applicable Codes</td>
</tr>
<tr>
<td>6.3.4</td>
<td>Structural Documents</td>
</tr>
<tr>
<td>6.3.5</td>
<td>Design Loads</td>
</tr>
<tr>
<td>6.3.5.1</td>
<td>Dead Load</td>
</tr>
<tr>
<td>6.3.5.2</td>
<td>Live Load</td>
</tr>
<tr>
<td>6.3.6</td>
<td>Load Combinations and Stresses</td>
</tr>
<tr>
<td>6.3.6.1</td>
<td>Strength Load Combinations</td>
</tr>
<tr>
<td>6.3.6.2</td>
<td>Serviceability Load Combinations</td>
</tr>
<tr>
<td>6.3.6.3</td>
<td>Initial Load Combinations</td>
</tr>
<tr>
<td>6.3.7</td>
<td>Deflections</td>
</tr>
<tr>
<td>6.3.8</td>
<td>Cover</td>
</tr>
<tr>
<td>6.3.9</td>
<td>Soil Properties</td>
</tr>
<tr>
<td>6.3.9.1</td>
<td>Equivalent Spring Constant</td>
</tr>
<tr>
<td>6.3.9.2</td>
<td>Soil Pressure</td>
</tr>
<tr>
<td>6.3.9.3</td>
<td>Displacement at Interface of Soil Layers</td>
</tr>
<tr>
<td>6.3.9.4</td>
<td>Numerical Example</td>
</tr>
<tr>
<td>6.4</td>
<td>GENERATION OF 3D STRUCTURAL MODEL THROUGH DWG IMPORT</td>
</tr>
<tr>
<td>6.4.1</td>
<td>First Drawing Import</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Transformation of Structural Components</td>
</tr>
<tr>
<td>6.5</td>
<td>MATERIAL, SOIL SUPPORT, CRITERIA AND LOADINGS</td>
</tr>
<tr>
<td>6.5.1</td>
<td>Set and Assign Material Properties</td>
</tr>
<tr>
<td>6.5.1.1</td>
<td>Set and Assign Multiple Concrete Materials</td>
</tr>
<tr>
<td>6.5.1.2</td>
<td>Set and Assign Mild Steel Material (Rebar)</td>
</tr>
<tr>
<td>6.5.2</td>
<td>Assign Soil Support</td>
</tr>
<tr>
<td>6.5.3</td>
<td>Set Criteria</td>
</tr>
<tr>
<td>6.5.4</td>
<td>Input and Assign Loadings</td>
</tr>
<tr>
<td>6.5.4.1</td>
<td>Patch Load Generation</td>
</tr>
<tr>
<td>6.5.4.2</td>
<td>Line Load Generation</td>
</tr>
</tbody>
</table>
6.5.4.3  Point Load Generation .............................................................. 159
6.5.4.4  Load Combinations................................................................................. 160

6.6  FINITE ELEMENT MESHING, ANALYSIS AND VIEW RESULTS .......... 161
6.6.1  Finite Element Meshing ................................................................................. 161
6.6.2  Analyze Structure ........................................................................................ 162
6.6.3  View Results ................................................................................................. 163
   6.6.3.1  View Deflection ..................................................................................... 163
   6.6.3.2  Review of Soil Pressure ............................................................................ 164

6.7  GENERATION OF SUPPORT LINES AND USE OF SPLITTERS .............. 166
6.7.1  Generation of Support Lines ....................................................................... 166
6.7.2  Use of Splitters .............................................................................................. 168

6.8  PRODUCE AND REVIEW DESIGN RESULTS ........................................... 170
6.8.1  Review Analysis/ Design Options .............................................................. 170
6.8.2  Generate Design Sections .......................................................................... 171
6.8.3  Review Design Strips (Column and Middle Strips) ................................ .. 172
6.8.4  Design the Design Sections ........................................................................ 173
6.8.5  Adequacy Check for the Design Sections .................................................. 173
6.8.6  Generate Rebar Drawing ............................................................................. 176
6.8.7  Specify Base Reinforcement and Re-design .............................................. 178
6.8.8  Punching Shear Check ................................................................................ 179

ANALYTICAL BACKGROUND ........................................................................ 183
7.1  OVERVIEW ..................................................................................................... 185
7.2  STRUCTURAL MODELING ........................................................................... 185
   7.2.1  Analysis .................................................................................................... 185
   7.2.2  Design ..................................................................................................... 186
7.3  MISCELLANEOUS TOPICS ....................................................................... 186
   7.3.1  Soil Pressure ............................................................................................ 186
   7.3.2  Superposition ........................................................................................... 188
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 2018 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.
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.

- 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.
Chapter 2  BASIC FEATURES

(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.
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/in³. This value typically varies between 100 to 400 psi (between 0.03 to 0.12 N/mm³). In the absence of detailed information 200 psi (0.06 N/mm³) 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.
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: $200 \times 24 = 4,800$ lb/in² length of grade beam $(0.06 \times 600 = 36$ 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 lines 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 2018 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
Chapter 2  BASIC FEATURES

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 recommend to consult with an ADAPT Technical Support Specialist for addition information. support@adaptsoft.com
This Page Left Intentionally

BLANK
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 2018 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 $\rightarrow$ 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 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 Display 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 floating dialog 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 default, *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 2018 Basic Manual. These tools are equally applicable to ADAPT-MAT as well as ADAPT-Floor Pro and are universal to the ADAPT-Builder Platform.
Chapter 4

USER INTERFACE
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. The solution'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, and built-in building codes (e.g., American, Canadian, British, European, and Australian), ADAPT's Builder allows for a streamlined workflow.

ADAPT-Builder Express (EX)® Concrete Design Suite

![ADAPT Builder Express Workflow Diagram]

FIGURE 4.1-1 ADAPT BUILDER EXPRESS WORKFLOW

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 particular 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)

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](image)

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.

ADAPT-BUILDER contains a ribbon-based user-interface that contains contextual, customizable tool panels. Each tool on a panel contains a tool-tip 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. Right-click options are context specific and may change depending on the type of component selected while carrying out this operation.
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

4.3.2.1 Top-Left Quick-Access Default Tools (1)

New
Creates a new file from defaults or a template selection.

Open
Opens an existing Builder file.

Save
Saves the active Builder file.

Undo
Undo previous action.

Redo
Redo previously undone action.

Print
Opens printer selection and settings and prints active screen.
Customize Quick Access Toolbar
Customizes the Quick Access Toolbar.

4.3.2.2 Top-Right Level Tools (2)

Level Assignment
Opens the Reference Plane Manager to create, edit or remove model levels and associated level heights and names.

Active Level Up
Sets the active plane to the next level above in vertical sequence.

Active Level Down
Sets the active plane to the next level down in vertical sequence.

Single-Level mode
Sets the modeling and analysis mode to view a single level. This level is the only active level in this mode. Any program function used in Single-Level mode applies only to components at the level shown. Use the Active Level Up and Active Level Down mode to switch to other levels. Analyses run in Single-Level mode consider only the current plane.

Multi-Level mode
Sets the modeling and analysis mode to view the entire modeled structure. Any modeling function used in Multi-Level mode applies only to set current plane shown in the Status Bar. Use the Active Level Up and Active Level Down mode to set the active plane to another level. Analyses run in Multi-Level mode consider the entire structure.
Chapter 4 USER INTERFACE

Help
Gives access to documentation, ADAPT-Builder version information, EULA and ADAPT website.

Help expanded

- **Documentation**
  Opens a folder containing program related documentation in windows explorer.

- **About ADAPT**
  Opens a window providing general contact information for ADAPT Corporation

- **About Builder**
  Gives access to documentation, ADAPT-Builder version information, EULA and ADAPT website.

- **Technical Support**
  Opens a window displaying contact information for ADAPT’s support department.

- **Disclaimer**
  Opens a window displaying a disclaimer related to the usage of the software.

4.3.2.3 Message Bar (3)

Display’s model information or prompts user for input.
4.3.2.4 Bottom Quick Access Tools (4)

See tool descriptions on individual toolbars and use them here for non-snap and grid tools. Leave as-is for snap and grid settings/grid active.

4.3.2.5 Bottom-Right – Status Bar (5)

Cursor coordinates
Shows X, Y and Z global coordinates of the current cursor position.

Units
Shows the current system of units: US, SI or MKS. To change units for a model, export the model using FILE-Export-.INP and import the file from FILE-Import-.INP.

Level
Reports the active plane. Use the Active Level Up and Active Level Down mode to set the active plane.

Layer
Reports the active layer. Select this text to change the active layer.
4.3.3 File Ribbon

New
Creates a new file from defaults or a template selection.

Open
Opens an existing Builder file.

Recent
Lists the last four Builder files saved from the machine.

Save
Saves the active Builder file.

Save As…
Save the active Builder file as a new name.

Save as Version 2012
Saves the files to a previous version for backward compatibility. Stiffness usage cases and customized rebar libraries are reset to program defaults.

Save as Project Template
Saves the criteria, material or load combinations as an ADAPT template .apt file.
**Auto Save Settings**
Opens the Automatic Save Options window to enable auto-save and backup and to set a save time interval.

**Import**
Opens options to import:
- **.INP file (generic ADAPT exchange file)**
  Used to update or create a new file from another Builder .adm file or from the ADAPT-Revit/ADAPT-Etabs/ADAPT-TSD integration links.
- **.DWG/DXF file**
  Used to import an existing Autocad file to create a new Builder .adm file through polygon transformation.
- **.XLS file**
  Used to import point and line loads from a formatted Excel file

**Export**
Opens options to export:
- **.INP file (generic ADAPT exchange file)**
  Used to update or create a Builder .adm
- **.DWG/DXF file**
  Used to export drawing data from Builder to Autocad
- **Revit**
  Used to create a new Revit .rvt file.
- **S-Foundation**
  Used to export loading and reaction data to S-Foundation.
- **Visicon**
  Used to create a new Visicon model for BIM coordination.

**Title Block Info**
Opens window the user can enter general information about the project/model. The information is used in the graphical and tabular reports of the software.

**Print**
Opens printer selection and settings and prints active screen.
Exit
Exits the application.

4.3.4 Home Ribbon

4.3.4.1 Display Panel

Customize User Interface Tools
Includes settings to customize the Builder User Interface and graphical modeling environment.

- **General Settings**
  Settings for delete confirmation, property window behavior when modifying component properties, cursor properties and cursor scroll settings.

- **Text Settings**
  Font type and size of text identifiers.

- **Dynamic Dimension Settings**
  Customize the display of length, angle and reference line for dynamic dimensioning and sets incremental distance and angle (degrees).
- **Folder for Saving Data**
  Sets the default or custom location for saving ADAPT-PT/RC .adb files when using the Strip Method option.

**Layer Settings**
Controls layer options to delete or create new layers, modify layer properties and layer display.

**Line Style Settings**
Controls linestyle options to delete or create new linestyles or modify linestyle properties.

### 4.3.4.2 Selection Mode Panel

![Selection Mode](image)

- **Window Selection/Pick Mode**
  Selects all visible items that are picked with the cursor or located within the drawn window. Press CTRL+ to select multiple picked or windowed components.

- **Lasso Selection**
  Selects the items located within the draw lasso.

- **Path Selection**
  Selects the items located on the drawn path polyline.

- **Move Selected Point**
  Enables component handles that can be selected to move components. Can be used to move only the first selected component.

- **Move Selection**
  Enables component handles than can be selected to move components. Can be used to move multiple, selected components.

- **Hint Mode**
Displays the components label, layer and linestyles when the cursor is hovered over the component.

4.3.4.3 Selection Tools Panel

Select by Type
Opens dialog window to select components by type with criteria filters.

Select All
Selects all the visible items.

Select by Layers
Opens dialog window to select components by listed layers.

4.3.4.4 Draw Panel

Create point
Creates a point at the selected location.

Create Line
Creates a line by start and end point.

Create Line Expanded

Continuous Modeling
Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.
Create Polyline
Creates a polyline by specifying multiple points with straight line segments between points.

Create Polygon
Creates a polygon by snap-point input.

Create Polygon Expanded
- Create Polygon
  Creates a polygon by snap-point input.
- Polygon: Vertices, Center, Radius
  Creates a polygon by specifying number of vertices, center and radius.
- Polygon: Vertices, Diameter
  Creates a polygon by selecting number of sides, start point at mid-distance location of first edge, and end point at mid-distance location of second edge.
- Polygon: Vertices, Start Edge, End Edge
  Creates a polygon by selecting number of sides, start point at mid-distance location of first edge, and end point at mid-distance location of second edge.

Circle: Center, Radius
Creates a circle by specifying center and radius.

Circle: Center, Radius Expanded
Circle: Center, Radius
Creates a circle by specifying center and radius.

Circle: Three Points
Creates a circle by specifying three points.

Circle: Diameter
Creates circle by specifying insertion point and diameter.

Arc: Start, End, Radius
Creates an arc by specifying starting, ending, and radius value.

Arc: Start, End, Radius Expanded

Arc: Start, End, Radius
Creates an arc by specifying starting, ending, and radius value.

Arc: Start, Center, End
Creates an arc by specifying start, center and end points.

Arc: Three Points
Creates an arc by specifying three points.
- **Arc: Radius, Angle**
  Creates an arc by specifying radius and angle.

- **Arc: Radius, Chord**
  Creates an arc by specifying radius and chord.

**Select Position, Write Text**
Creates text by specifying starting point, height and direction, then entering text.

**Select Position, Write Text Expanded**

- **Select Position, Write Text**
  Creates text by specifying starting point, height and direction, then entering text.

- **Write Text, Selection Position**
  Creates text by entering text, specifying starting point, height and direction.

**Create Dimension**
Creates a dimension by specifying start and end point and position.

**Create Section Cut**
Creates a section cut by specifying start and end point for the cut line and selecting an insertion point.
4.3.4.5 Tools Panel

Measure Tool
Measure the distance between two points.

Coordinates
Reports the X, Y and Z coordinates of a user-selected point.

Calibrate
Calibrates the graphical model workspace by selecting two points and specifying the distance between the points.

Change Project Origin
Changes the current project origin to a user defined origin position.

4.3.4.6 Zoom/Camera Panel

Zoom Window
Enlarges the items selected in the zoom window.

Top View
Displays the model from a top perspective.

Top-Left View
Displays the model from a top-left perspective.

Zoom Extents
Resets the view to extents of the modeled components.
**Bottom View**
Displays the model from a bottom perspective.

**Top-Right View**
Displays the model from a top-right perspective.

**Dynamic Zoom**
Reduces or enlarges the model view from a user-defined selected point.

**Left View**
Displays the model from a left perspective.

**Rear View**
Displays the model from a rear perspective.

**Zoom In**
Enlarges the model view from a user-defined selected point.

**Right View**
Displays the model from a right perspective.

**Top-Front-Right View**
Displays the model from top-front-right perspective.

**Zoom Out**
Reduces the model view from a user-defined selected point.

**Front View**
Displays the model from a front perspective.

**Top-Back-Side View**
Displays the model from a top-back-side perspective.

**Rotate View**
Rotates the model view from a user-defined selected point and rotation by mouse control.
Pan View
Pans the model view from a user-defined selected point and mouse control.

Redraw
Redraws the current model view.

4.3.4.7 Viewport Panel

Single
Creates a single viewport window. This is the default viewport of the software.

Single Expanded

- Two Vertical
  Creates the viewport shown.

- Two Horizontal
  Creates the viewport shown.

- Three Vertical
  Creates the viewport shown.
o **Three Horizontal**
   Creates the viewport shown.

o **Four Equal**
   Creates the viewport shown.

o **Four Horizontal**
   Creates the viewport shown.

### 4.3.4.8 WCS/UCS (Coordinate Systems) Panel

#### Display WCS
Turns off/on the axes of the global coordinate system at the model origin.

#### Transform UCS to WCS
Transforms the defined User Coordinate System to the World Coordinate System.

#### UCS at last point
Positions the User Coordinate System and origin at a user-defined point.

#### UCS at end-point of last line
Positions the User Coordinate System and origin at the end-point of a user-defined line.

#### UCS: 5 points
Defines User Coordinate System (UCS) by: origin, X start, X end, Y start, Y end.

#### UCS: 3 points
Defines User Coordinate System (UCS) by: origin, X end, Y end.
4.3.4.9 Scaling Panel

**Decrease Scale Factor**
Decreases the scale factor in the selected coordinate direction/s. Object labels are not scaled. Reset scale factor to position objects back to label location.

**Reset Scale Factor**
Resets the scale factor to the program default for the selected coordinate direction/s.

**Increase Scale Factor**
Increases the scale factor in the selected coordinate direction/s. Object labels are not scaled. Reset scale factor to position objects back to label location.

**Modify the X-Direction Scale**
Enables scaling distortion in the X-Direction.

**Modify the Y-Direction Scale**
Enables Scaling distortion in the Y-Direction.

**Modify the Z-Direction Scale**
Enables Scaling distortion in the Z-Direction.

4.3.5 Visibility Ribbon
4.3.5.1 Message Bar Panel

Message Bar

Enables or hides the User Command Bar for X, Y, Z or dimension input. When the bar is disabled, all node point entry must be performed using mouse clicks with Dynamic Dimensioning guidance when active.

4.3.5.2 Colors Panel

Color Palette

Opens the Color Palette menu to set outline, line and fill colors and opacity settings for structural components, analytical elements and loads. Use this option to set the background color. These settings apply to newly-created components. Existing components will retain their set color unless modified by Component Properties or Modify Item Properties.

4.3.5.3 Section Panel

Create Section Cut

Creates a section cut by specifying start and end point for the cut line and selecting an insertion point.
4.3.5.4 Render Panel

Render Model 🛠️
Opens the ADAPT Solid Modeling Viewer. This tool is used to dynamically view the model components, imported CAD drawings, and view displacements and deformations.

4.3.5.5 Display Settings Panel

Default Display 🔄
Restores the view to the user-defined default display

View Settings (Select/Set View Items) 🏕️
Defines the graphical display settings for structural components, analysis elements and loading. Controls font and symbol sizes for all items.

Group Library 🗄️
Opens the Grouping dialog window. This is used to create, delete, enable display of groups by on/off, and manage imported CAD files. If a group is fully displayed in the model the lightbulb icon will appear solid yellow. If the group is partially displayed the icon will appear half-tone. If all components of a group are not displayed the icon will appear gray.

4.3.5.6 Selection Panel

Show All 🎨

Reverts back to the initial model display prior to the Display Selection or Hide Selection being applied.

**Show Selection**
Displays only the items selected in view and hides all other model items.

**Hide Selection**
Hides only the items selected in view and displays all other model items.

### 4.3.5.7 Viewing Panel

![Image of Viewing Panel]

**Show by Type**
Opens a dialog window for selecting items to display by type in the current view.

**Display Labels on Selection**
Displays selected components in the current view. To reset the view, use Select/Set View Items from Visibility-Display Settings.

**Display All Items**
Displays all items in the current view. To reset the view, use Select/Set View Items from Visibility-Display Settings and/or Group Library to turn off displayed and imported CAD files.

**Displays/Hide Labels**
Displays on/off labels of components in the current view.

### 4.3.5.8 Shading Panel

![Image of Shading Panel]

**Solid Fill**
Shades the components in the current view or selected components as a solid model. Control the fill and outline colors, layers and opacity from **Modify-Modify Item Properties** after selecting the components to be modified. To modify single components, double-click on the component and use the **Properties** tab.

**Wire Frame**
Represents the components of the current view or selected components as a wire-frame displaying only the component outline.

### 4.3.6 Modify Ribbon

#### 4.3.6.1 Properties Panel

**Items Properties**
Opens the selected item’s property dialog for modifying material, geometric and analytic properties associated with the component. This tool becomes active when an item is selected or will launch when double-clicking a single component. If multiple items are selected, use **Modify Item Properties**.

**Modify Selection**
Opens the **Modify Item Properties** dialog window to modify or set selected components layer, material, geometric or analytical properties. This tool should be used when more than 1 item is to be modified in the use instance.
4.3.6.2 Edit Panel

Delete ❌
Deletes the current selection.

Paste 🔴
Inserts the clipboard contents at the insertion point.

Cut 📋
Cuts the current selection and saves to clipboard.

Copy 📇
Copies the current selection and saves to clipboard.

Undo ⬅️
Undo previous action.

Redo ⬅️
Redo previously undone action.

4.3.6.3 Copy/Move Panel

Duplicate 🔇
Copies the current selection and prompts the user for translation base reference point and insertion point.

In Plane (Translation) 🔗
Copies or moves the selected entities in the current plane by start and end vector points. The Copy-Move dialog window will open allowing user
input for X and Y offsets, number of copies, and options to **Copy or Move**.

**By Coordinate (Translation)**
Copies or moves the selected entities in the current plane by user offset input. Opens the **Copy-Move** dialog window allowing user input for X and Y offsets, number of copies, and options to **Copy or Move**.

**Rotation**
Rotates the selected entities in the current plane by selection of center of rotation and angle entry in degrees. The **Copy-Move** dialog window will open reporting the selected rotational reference point, allowing entry for rotation angle, and options to **Copy or Move**. Positive entry is counterclockwise rotation.

**Vertical (Translation)**
Copies or moves the selected entities to other defined planes in the Z-direction. Opens the **Copy/Move Vertical** dialog window allowing the user to:

- **Copy**
  To a defined plane or Up/Down a defined number of times per user input.

- **Move**
  To a defined plane

- **Assign**
  To a defined plane

**By Layer (Translation)**
Re-assigns the current selection to a different layer. Opens the **Move to Layer** dialog window for selecting the destination layer. To assign a new layer use **Layer Settings** in the **Home** ribbon.

**Reference Planes**
Copies the current plane and assigned components to create new planes in the model. Use this tool to replicate typical levels vertically without previously creating new planes in **Model – Level Assignment – Story Manager**.
4.3.6.4 Tools Panel

**Scale**
Scales up/down selected polygons by input of X and Y scale factors. This tool applies only to drawing items like polygons or imported CAD polygons. It does not apply to structural components.

**Mirror Selected Items**
Mirrors the selected entities in the current plane by defining a start and end point of the mirror axis.

**Trim Selected Items**
Trims the selected entity in the current plane by defining the line to trim to and the line to be trimmed. These can be lines/edges of the selected components or reference lines.

**Extend Selected Items**
Extends the selected entity in the current plane by defining the line or edge to be extended to.

**Join**
Joins the endpoints of two selected components by extension of the longitudinal axes to the point of intersection. This tool is active when only 2 components are selected.

**Split**
Sub-divides the selected components into user-defined input. Graphical marks are shown on the component and can be snapped to.

**Unsplit**
Restores the selected components by removing all split points.
4.3.6.5 Blocks Panel

**Create Block**
Groups selected items and forms a drawing block. Double-click on the items to open the block properties. Selecting structural components and assigning them to a block will disable their properties as a physical component.

**Explode Block**
Explodes the selected block and restores the drawing and modeling entities to their original state.

4.3.6.6 Add/Remove Points Panel

**Insert Point**
Adds a new point to a selected item on the current plane. This point can be used to control the item shape in the X and Y plane and is only active when a single item is selected.

**Remove Point**
Removes a point on a selected item on the current plane and is only active when a single item is selected.

4.3.6.7 Labels Panel

**Reset Non-User Defined Labels**
Resets the labels of all components in the model back to a counter of 1 with the “Label prefix” for each component type and for components whose label has not been altered by the user.

**Reset All**
Resets the labels of all components in the model back to a counter of 1 with the “Label” prefix for each component type.

**Reset Column/Wall Stack**
Opens the dialog window for **Automatic Labeling of Stacked Supports** and resets the labels of all continuously vertical column and/or walls. Applies the label at the bottom-most component in the stack to all other columns or walls in the stack.

### 4.3.7 Criteria Ribbon

#### 4.3.7.1 Material Properties Panel

**Concrete**
Opens the **Material** dialog window for defining concrete properties with or without steel fiber.

**Rebar**
Opens the **Material** dialog window for defining non-prestressed reinforcement properties.

**Prestressing**
Opens the **Material** dialog window for defining prestressing properties.

**Fiber**
Opens the **Material** dialog window for defining concrete properties with or without steel fiber.

**Generic**

Opens the **Generic Materials** dialog window for defining generic (non-concrete) material and associated properties. These properties can only be assigned to a generic component defined from **Column Design – Section Type Manager**.

### 4.3.7.2 Design Criteria Panel

#### Design Code

Opens the **Criteria – Design Code** selection options for setting the applicable design code and material constitutive models for the design of slabs and beams for minimum reinforcement requirements, shear and flexure.

#### Analysis/Design Options

Opens the **Criteria – Analysis/Design Options** for defining analytical and design settings. Use this option to define default support conditions for Single- and Multi-Level modes.

#### Allowable Stresses

Opens the **Criteria – Allowable Stresses** menu for defining code required allowable tension and compression concrete stress and non-prestressed and prestressed steel stress. Applies only to the PT design scope when tendons are modeled as part of a slab or beam. For some applicable design codes, input of allowable crack width is also input in this dialog.

#### Shear Design

Opens the **Criteria – Shear Design Options** for defining one- and two-way shear reinforcement type, size, number of legs and rails. For two-way shear define the principal stress check for rr and ss axes separately or combined.
**Tendon Height**
Opens the **Criteria – Tendon Height Defaults** menu for defining the distance from top and bottom slab or beam surface to the CGS of tendons at supports and span. Use this menu to input the incremental distance for adjustment of the CGS from slab soffit for tendon mapping and graphical tendon adjustment in elevation.

**Rebar Cover**
Opens the **Criteria – Rebar Minimum Cover** menu to define the top and bottom cover distance to non-prestressed longitudinal reinforcement. For two-way slabs the distance defined is relative to the outer layer of reinforcement. Open **Support Lines – Design** properties to define outer and inner layers for reinforcement of the strip. For one-way slabs and beams the distance defined is relative to the reinforcement parallel to the support line.

**Rebar Size/Material**
Opens the **Criteria – Preferred Reinforcement and Material** menu to define preferred reinforcement sizes, reinforcement material assignments and select a default rebar library. Rebar libraries can be modified and imported/exported. Define top and bottom flexural bar sizes for one-way, two-way and beam criteria and shear stirrup size for beams.

**Rebar Round-Up**
Opens the **Criteria – Rebar Round Up** menu options for bar length and spacing detailing options. Use this menu to input length and spacing distance rounding, bar spacing limits (RC mode only) and defining Rebar Library Lengths.

**Rebar Lengths**
Opens the **Criteria – Reinforcement Bar Lengths** menu options for longitudinal reinforcement cut-off lengths and bar extension lengths. Cut-off lengths apply to minimum reinforcement over supports and in the span for slabs and beams. Bar extension distance is applied to reinforcement required for Strength combinations (demand moments). Options are available for adjusting bar lengths and position for graphical rebar output.
4.3.7.3 Design Criteria (SOG) Panel

- **Design Code**
  Opens the Criteria – Design Code menu. The design code for SOG mode is hard-coded to PTI/UBC and cannot be changed.

- **Allowable Deflection**
  Opens the Criteria – Allowable Service Deflection for designating the deflection limitation (span/deflection ratio) for the graphical check along design strips. Use Result Display Settings – Design Sections to turn on the deflection display for support lines.

- **Edge Displacements**
  Opens the Criteria – Edge Displacements menu to define parameters for the calculation of applied displacements edge displacements. These values are calculated per the PTI equation for simulating edge lift conditions along the edges of a slab-on-ground.

- **Soil Parameters**
  Opens the Criteria – Soil Parameters menu for input of Edge and Center Lift values for Em (edge moisture distance) and Ym (expected vertical differential movement). These values are user-defined and are only used in calculating the Edge Displacements.

- **Tendon Height**
  Opens the Criteria – Tendon Height Defaults menu for defining the distance from top and bottom slab or beam surface to the CGS of tendons at supports and span. Use this menu to input the incremental distance for adjustment of the CGS from slab soffit for graphical tendon adjustment in elevation.

- **Allowable Stresses**
  Opens the Criteria – Allowable Stresses menu for defining code required allowable tension and compression concrete stresses.
4.3.8 Model Ribbon

4.3.8.1 Wizards Panel

Floor Wizard
Opens the **Floor Wizard** for rapid generation of a Single- or Multi-Level structural model with gravity loading. Post-Tensioning can be applied when in **RC and PT** design scope.

Frame (Beam) Wizard
Opens the **Frame (Beam) Wizard** for rapid generation of Single-Level beam-column frames and continuous beams with gravity loading.

4.3.8.2 Level Assignment Panel

Level Assignment
Opens the **Reference Plane Manager** for management of model levels. Use this tool to define or delete new planes and assign story heights.

4.3.8.3 Gridline Panel

Gridline Wizard
Defines model gridlines by entering vertical and horizontal gridline input for number, distance, angle, label, and base insertion point.
Gridlines expanded

- **Gridline Wizard**
  Defines model gridlines by entering vertical and horizontal gridline input for number, distance, angle, label, and base insertion point.

- **User Defined Gridlines**
  Defines a single gridline with entry of start and endpoint. Double-click on the gridline to change the grid label and properties.

- **Import Gridlines from DWG**
  Opens the **FILE – Import** options for DWG/DXF file used to import an Autocad grid file.

### 4.3.8.4 Type Manager Panel

**Section Type Manager**
Opens the **Type Manager** dialog for creating section types for concrete component design and generic section modeling. Use this tool to create column design sections with reinforcement prior to Code Check/Design of columns.

### 4.3.8.5 Structural Components Panel

**Slab**
Creates a slab region by graphical or coordinate entry of the slab polygon vertices. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command. Double-click on the slab to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

**Wall**

Creates a wall by graphical or coordinate entry of the wall start and endpoint. Select ESC or right-click Exit to close the command. Double-click on the wall to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

- **Continuous Modeling**
  Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.

**Column**

Creates a column by graphical or coordinate entry of the column insertion point. Select ESC or right-click Exit to close the command. Double-click on the column to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

**Drop Panel**

Creates a drop cap/panel by graphical or coordinate entry of the drop cap/panel insertion point. Select ESC or right-click Exit to close the command. Double-click on the drop cap/panel to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

**Beam**

Creates a beam by graphical or coordinate entry of the beam start and endpoint. Select ESC or right-click Exit to close the command. Double-click on the beam to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

- **Continuous Modeling**
  Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.
Opening
Creates an opening region by graphical or coordinate entry of the opening polygon vertices. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command. Double-click on the opening to open the properties or use Modify-Item Properties to define the properties prior to placing the component.

4.3.8.6 Preprocessing Panel

Connect Supports
Establishes automatic top and bottom offsets for columns and walls. Nodes are offset to the connected slabs at top and bottom. Double-click on the component to open Properties-Location to manually adjust offsets.

Connect Beams to Columns and Walls
Auto-shifts the beam to wall centerline or column endpoint. The beam end must be located within the body of the wall or column to take effect.

Align Structural Components
Realigns the selected component based on user-defined position selection input. The program will prompt user to accept the new position before shifting.

Connect Drop/Cap Panel to Column
Shifts the drop/cap panel insertion point to the column endpoint location for analysis.

4.3.8.7 Preprocessing (SOG) Panel (in SOG only)

Align Structural Components
Realigns the selected component based on user-defined position selection input. The program will prompt user to accept the new position before shifting.

4.3.8.8 Supports Panel

Create Supports
Automatically generates supports at far ends of walls and columns. The translation and rotational fixity conditions are defined in Criteria-Analysis/Design Options for Single- and Multi-Level modes. The program generates the supports dependent on the current mode. Use the Story Manager Tools at the upper-right of the Quick Access Toolbar to change analysis modes. To view support symbols use Visibility-Select/Set View Items.

Add Point Support
Creates a point support by graphical or coordinate entry at the support insertion point. Double-click on the support to open the Properties and change fixity conditions.

Add Line Support
Creates a line support by graphical or coordinate entry of the line support start and endpoint. Double-click on the support to open the Properties and change fixity conditions.

- Continuous Modeling
  Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.

4.3.8.9 Springs Panel
Add Point Spring

Creates a point spring by graphical or coordinate entry at the spring insertion point. Double-click on the spring to open the Properties and change sub-grade modulus (Kxx, Kyy, and Kzz) and to define a C or T spring.

Add Line Spring

Creates a line spring by graphical or coordinate entry of the line spring start and endpoint. Double-click on the spring to open the Properties and change subgrade modulus (Kxx, Kyy and Kzz) and to define a C or T spring.

- Continuous Modeling

  Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.

Add Area Spring

Creates an area spring region by graphical or coordinate entry of the area spring polygon vertices. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command. Double-click on the spring to open the Properties and change subgrade modulus (Kxx, Kyy and Kzz) and to define a C or T spring.

4.3.8.10 Displacements Panel (in SOG only)

Point

Creates a point displacement by graphical or coordinate entry at the support insertion point. Double-click on the support to open the Properties and define the displacement value. Negative is upward.

Line

Creates a line support by graphical or coordinate entry of the line support start and endpoint. Double-click on the support to open the Properties and define the displacement value. Negative is upward.
Continuous Modeling
Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.

4.3.8.11 Visibility Panel

Display/Hide Slab Region Panel
Displays on/off slab regions.

Display/Hide Wall (N/A to SOG)
Displays on/off walls.

Display/Hide Column (N/A to SOG)
Displays on/off columns.

Display/Hide Drop Cap/Panel
Displays on/off drop caps and panels.

Display/Hide Beam
Displays on/off beams.

Display/Hide Opening
Displays on/off openings.

Display/Hide Gridlines
Displays on/off gridlines.

Display/Hide Point Supports
Displays on/off point supports. Double-click on support to see fixity conditions.
Display/Hide Line Supports
Displays on/off line supports. Double-click on support to see fixity conditions.

Display/Hide Point Springs (N/A to SOG)
Displays on/off point springs. Double-click on spring to see sub-grade moduli settings for Kxx, Kyy and Kzz and to set compression or tension spring type.

Display/Hide Line Springs (N/A to SOG)
Displays on/off line springs. Double-click on spring to see sub-grade moduli settings for Kxx, Kyy and Kzz and to set compression or tension spring type.

Display/Hide Area Springs
Displays on/off area springs. Double-click on spring to see sub-grade moduli settings for Kxx, Kyy and Kzz and to set compression or tension spring type.

Point Displacement (SOG only)
Displays on/off point displacements.

Line Displacement (SOG only)
Displays on/off line displacements.

4.3.8.12 Properties Panel

Items Properties
Opens the selected item’s property dialog for modifying material, geometric and analytic properties associated with the component. This tool becomes active when an item is selected or will launch when double-clicking a single component. If multiple items are selected, use Modify Item Properties.
4.3.8.13 Transform Panel

Transform Slab
Creates a slab region from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

Transform Beam
Creates a beam component from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

Transform Opening
Creates an opening from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

Transform Column
Creates a column component from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

Transform Drop Cap/Panel
Creates a drop cap/panel component from selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

Transform Lines to Polygon
Creates a polygon from a group of selected lines. The polygon can then be transformed to a component using other transformation tools.

Transform Single Wall
Creates a single wall component from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.
Transform Compound Wall
Creates multiple wall components from a selected polygon. The polygon can be natively drawn using Home-Draw tools or by import of a .DWG or .DXF file.

4.3.9 Loading Ribbon

4.3.9.1 Load Combo Panel

Load Cases
Opens the Load Case Library options for creating, deleting and modifying General/Lateral Loads, Building Loads, and Lateral Load Solution Sets. The following defines each load type:

- **General/Lateral**
  Used for gravity load cases, temperature and shrinkage loads, and applied lateral reactions transferred to a slab or beam. Only general load case types can be assigned as Reducable. These load cases are solved for in Single- and Multi-Level mode.

- **Building**
  These are Wind, Seismic or Generic Lateral Loads applied to a global structure and reacted by the lateral load resisting system. Building loads can be created by use of Loading Wizards, created manually or imported through the ADAPT-Integration Console and are only solved for when in Multi-Level mode. Building load case reactions are available to be applied to a Single-Level analysis.

- **Lateral Load Solution Sets**
  These are imported reactions to a Single-Level model from the ADAPT-Integration Console and checked for placement compatability and equilibrium.
Load Combinations
Opens the Combinations input menu for creating, deleting and modifying load combinations. Default load combinations will be created per the selection of Design Code from the Criteria panel. Each load combination is set an Analysis/Design Options type. The following defines each type:

- **Service**
  Applies the serviceability requirements (allowable stress, minimum reinforcement, etc.) for the selected design code and criteria and to the model design sections for the defined load combination. Multiple service conditions may be evaluated, including Total, Sustained, Frequent, QP, etc.

- **Strength**
  Uses the factored, ultimate design actions for the model design sections and determines reinforcement required necessary to produce section capacity > demand.

- **Initial**
  Applies to models including prestressing. The program checks the concrete stress against the allowable initial stress at transfer and calculates necessary supplemental reinforcement.

- **Cracked Deflection**
  This type is used only to evaluate the slab for cracking and loss of stiffness for the combination. This requires the user to perform analysis, design the design sections and use Floor Design-Cracked Deflection to run the cracked deflection check. This check only applies to Single-Level mode.

- **No Code Check**
  This is used to evaluate results for the load combination but the program will produce no code checks or reinforcement.

- **Long-Term Deflection**
  This type is used to combine the effects of other load combinations and produce a result. It is commonly used to combine the results of multiple Cracked Deflection combinations to produce long-term predicted deflections.
4.3.9.2 LL Reduction Panel

Reduction Settings

Opens the Live Load Reduction Factors table for reduction method, area and factors, application settings. These reduction factors apply only to reduction of loads tagged as Reducible from Loading-Load Case Library. The reduction factor calculation is dependent on creation of tributary regions from Loading-Tributary Load Manager. Load Reduction Factors are applied only in two ways:

- **Column Design**
  Go to Column Design-Column Design Options. If set to YES the reduced loads will be applied for column design.

- **Analysis of Single-Level**
  If a Multi-Level analysis has been performed, reactions can be re-applied in Single-Level mode with inclusion of load takedown (Fz reactions). Go to Analysis-Analyze Structure and use the option to Apply Live Load Reduction.

4.3.9.3 General Panel

Add Point Load

Creates a point load by graphical or coordinate entry of the load insertion point. Select ESC or right-click Exit to close the command. Double-click on the load to input the load value or use Modify-Item Properties to define the load prior to placement.

Add Line Load

Creates a line load by graphical or coordinate entry of the load start and endpoint. Select ESC or right-click Exit to close the command. Double-
click on the load to input the load value or use **Modify-Item Properties** to define the load prior to placement.

- **Continuous Modeling**
  Continues the current operation of modeling a component by entering sequential endpoints of each modeled component segment.

**Add Patch Load**
Creates a patch load region by graphical or coordinate entry of the load polygon vertices. Use **ENTER** or right-click **Close/End/Accept**, to close the load region. Select **ESC** or right-click **Exit** to close the command. Double-click on the load to input the load value or use **Modify-Item Properties** to define the load prior to placement.

**Line Load Wizard**
Opens the **Create Line Load Automatically** dialog to select the **General** load case and define the load value. Requires selection of at least one slab or polygon to map load input to the perimeter of the region selected.

**Patch Load Wizard**
Opens the **Create Patch Load Automatically** dialog to select the **General case** load and define the load value. Requires selection of at least one slab or polygon to map load input to the region.

**Import Loads (from XLS file)**
Opens the **XLS Import Wizard** used to import point and line loads into the current plane.

**4.3.9.4 Lateral/Building Panel**

![Wind Load Wizard](image)

**Wind Load Wizard**
Opens the **Wind Load Wizard** dialog for auto-generation or manual input of code-required wind load parameters or surface pressure. Wind pressures are converted to externally applied, modifiable line loads at slab
edges. Wind load cases produced by the wizard are stored as **Building Loads** and solved for when in Multi-Level mode.

### Seismic Load Wizard

Opens the **Seismic Load Wizard** dialog for auto-generation or manual input of lateral-static code-required seismic load parameters, story forces or base shear. When automatic generation is used, the program calculates base shear, story forces and moments. At the time of analysis, the story forces and moments are applied as nodal forces and moments proportioned based on nodal mass over total story mass. Seismic load cases produced by the wizard are stored as **Building Loads** and solved for when in Multi-Level mode.

### Lateral Load Wizard

Opens the **Lateral Load Wizard** dialog for manual input of story forces. At the time of analysis, the story forces and moments are applied as nodal forces and moments proportioned based on nodal mass over total story mass. Generic lateral load cases produced by the wizard are stored as **Building Loads** and solved for when in Multi-Level mode.

### Lateral Reactions

Opens the **Generate Lateral Point Loads** for manual input of column and wall end joint reactions for lateral load cases. This option requires that the loads input be in equilibrium with applied loads and compatible with locations of vertical, lateral-resisting elements. This option applies only to Single-Level mode.

### 4.3.9.5 Tributary Panel

#### Load Takedown

Opens the **Tributary Load Manager** dialog for creation of tributary regions of vertical elements and level and cumulative areas and loads. Tributary Loads are not dependent on the finite element solution and can be re-applied as reactions in Single-Level mode or when designing columns and walls.
4.3.9.6 Pattern Panel

Sub-regions
Use this option to create multiple sub-regions of a slab to select and include as part of a load pattern. Input regions by creating polygons or by select divisible lines in both orthogonal directions. Sub-regions should be created in Single-Level mode but can be applied to multiple planes in a model.

Load Patterns
Opens the Skip Patterns dialog for defining a load case to be patterned and creating load patterns called “Pattern_1, Pattern_2,…Pattern_N.” Use this option in Single-Level mode and select the sub-regions to be included in the pattern and select ADD. Load patterns will be added to the Loading-Load Combinations dialog menu and can be added to combinations.

4.3.9.7 Visibility Panel

Show Loads
Displays on/off loads. If selective loads are in view from Select/Set View Items or Layer Settings, when selected the visible loads will turn off. If selected again, all loads assigned to model will turn on.

4.3.10 Tendon Ribbon
4.3.10.1 Criteria Panel

Prestressing
Opens the Material dialog window for defining prestressing properties.

Allowable Stresses
Opens the Criteria – Allowable Stresses menu for defining code required allowable tension and compression concrete stress and non-prestressed and prestressed steel stress. Applies only to the PT design scope when tendons are modeled as part of a slab or beam. For some applicable design codes, input of allowable crack width is also input in this dialog.

Tendon Height
Opens the Criteria – Tendon Height Defaults menu for defining the distance from top and bottom slab or beam surface to the CGS of tendons at supports and span. Use this menu to input the incremental distance for adjustment of the CGS from slab soffit for tendon mapping and graphical tendon adjustment in elevation.

4.3.10.2 Settings Panel

Tendon Settings
Opens the Tendon Settings dialog menu to control modeling parameters and tolerances for tendons. Use this to disable spline tendon modeling and default to linearized tendon input.

4.3.10.3 Model Panel
Add Tendon
Add a single tendon to the structure by graphical or coordinate input of tendon anchor and high point (control point) locations. The default tendon shape reverse parabolic. Select ESC or right-click Exit to close the command. Double-click on the tendon to modify Tendon Properties.

Map Distributed
Maps a selected “master” tendon over a region by selection of master tendon nodes to user-specified locations. Define the master tendon first, double-click to open the Map Distributed Tendon dialog menu and select coordinates 1 of N for mapping.

Map Banded
Maps tendons to selected design strips for the user-defined criteria set in the Map Banded Tendons dialog menu. Design strips are required to be generated before mapping.

4.3.10.4 Modify Panel

Tendon Editor
Opens the Dynamic Tendon Editor for rapid modification of tendon CGS, number of strands, anchorage, and spline override. Use Top View in Single-Level mode for recommended use.

Trim/Extend Tendons
Auto-adjusts tendon anchor locations to the slab edge. Use Tendon-Tendon Settings to adjust the tolerance distance.

Insert Point
Adds a new control point to the selected tendon on the current plane. Use this to create a new tendon span on an existing tendon.

Remove Point
Removes a control point on a selected tendon on the current plane and is only active when a single tendon is selected. Use this to remove a span from an existing tendon.

**Insert Swerve Point**

Adds a swerve point to the selected tendon on the current plane. Use the swerve point to control lateral curvature in the tendon span where inserted in the XY plane. A maximum of 3 swerve points can be added to a span.

**Remove Swerve Point**

Removes a swerve point on the selected tendon on the current plane. Use the swerve point to control lateral curvature in the tendon span where inserted in the XY plane. A maximum of 3 swerve points can be added to a span.

### 4.3.10.5 Shop Drawing Panel (add-on Module)

**Friction and Elongation**

Calculates friction loss and elongation for tendons set to **Calculated Force** option in **Tendon Properties – Settings**. Select tendons and then the tool to perform the calculations.

**Display Tendon Heights**

Opens the **Tendon Chair** dialog to input display parameters for tendon chair heights. Use **Visibility – Select/Set View Items** to change the symbol and text height.

**Tendon Spacing Tool**

Creates spacing dimensions between selected tendons at a user-specified insertion point.
4.3.10.6 Visibility Panel

Tendon Display Manager
Opens the Tendon Display Information dialog for selection of tendon properties, curvature, stressing and geometry to be displayed. Use Visibility – Select/Set View Items to change the symbol and text height.

Display/Hide Tendons
Displays on/off tendons.

Tendon Intersection Detector
Detects the intersection of two or more tendons in the structure and marks the intersection points.

Display/Hide Radius of Curvature
Displays on/off the radius of curvature of the tendons end, high and low points in display and marks as NG or OK based on minimum radius set in the tendon properties dialog.

Display Tendon Elevation
Creates tendon elevations for the selected tendons at the user-defined insertion point.

4.3.10.7 Design Panel

Tendon Optimizer
Opens the Tendon Optimizer dialog for optimization of tendon design for P/A, %bal load and allowable stresses for the selected tendons within a span. Intended for Single-level use and the level should be analyzed first. Open the Tendon Optimizer and select tendons in the span to be optimized.
4.3.11 Rebar Ribbon

4.3.11.1 Design Criteria Panel

Rebar Cover
Opens the Criteria – Rebar Minimum Cover menu to define the top and bottom cover distance to non-prestressed longitudinal reinforcement. For two-way slabs the distance defined is relative to the outer layer of reinforcement. Open Support Lines – Design properties to define outer and inner layers for reinforcement of the strip. For one-way slabs and beams the distance defined is relative to the reinforcement parallel to the support line.

Rebar Size/Material
Opens the Criteria – Preferred Reinforcement and Material menu to define preferred reinforcement sizes, reinforcement material assignments and select a default rebar library. Rebar libraries can be modified and imported/exported. Define top and bottom flexural bar sizes for one-way, two-way and beam criteria and shear stirrup size for beams.

Rebar Round-Up
Opens the Criteria – Rebar Round Up menu options for bar length and spacing detailing options. Use this menu to input length and spacing distance rounding, bar spacing limits (RC mode only) and defining Rebar Library Lengths.

Rebar Lengths
Opens the Criteria – Reinforcement Bar Lengths menu options for longitudinal reinforcement cut-off lengths and bar extension lengths. Cut-off lengths apply to minimum reinforcement over supports and in the span for slabs and beams. Bar extension distance is applied to reinforcement...
required for Strength combinations (demand moments). Options are available for adjusting bar lengths and position for graphical rebar output.

4.3.11.2 Model Base Rebar Panel

Mesh
Creates a mesh reinforcement region by graphical or coordinate entry of the mesh polygon vertices. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command. Double-click on the mesh symbol to modify mesh properties or use Modify-Item Properties prior to placement. Slabs of varying offsets should be assigned unique mesh reinforcement regions.

Mesh Wizard
Opens the Create Mesh Reinforcement Automatically dialog to select the reinforcement position, CGS, and bar area or size and spacing. Requires selection of at least one slab or polygon to map load input to the region. Slabs of varying offsets should be assigned unique mesh reinforcement regions.

Banded
Creates banded (grouped) rebar by graphical or coordinate entry of the rebar start and endpoint. Select ESC or right-click Exit to close the command. Double-click on the rebar to input properties or use Modify-Item Properties prior to placement.

Distributed
Creates distributed rebar by graphical or coordinate entry of the rebar start and endpoint and distribution range. Select ESC or right-click Exit to close the command. Double-click on the rebar to input properties or use Modify-Item Properties prior to placement.
4.3.11.3 Generate Panel

**Calculated Rebar Plan**
Opens the **Generate Rebar Drawing Options** dialog to select combinations, bar length selection and orientation for graphical display of calculated reinforcement for X and Y design strips at top and bottom locations.

4.3.11.4 Visibility Panel

**Display Manager**
Opens the **Rebar Display Manager** dialog to select rebar graphical display options for calculated and base strip and mesh reinforcement.

**Display/Hide All Rebar**
Displays on/off all reinforcement.

4.3.11.5 Reporting Panel

**Print Settings**
Opens the **Rebar Plot Settings** dialog to configure the rebar appearance settings for printing.

**Insert Details**
Inserts selectable, pre-drawn shear and flexural reinforcement details by graphical or coordinate insertion entry at the current plane.
Export Settings
Opens the DWG/DXF Export Settings For Reinforcement dialog to configure the rebar appearance settings for export to AutoCAD files.

4.3.12 Analysis Ribbon

4.3.12.1 Vibration Panel

Settings
Opens the Vibration Combinations dialog for defining load combinations, number of modes, and direction stiffness contribution factors Rx, Ry, and Rz to be used in evaluating the structure dynamically for modal frequencies, periods and amplitude. This option is used in Single-Level analysis for evaluation of slab sensitive to rhythmic movement. These combinations are also used for determining mass for the Seismic and Lateral Load Wizards.

4.3.12.2 Meshing Panel

Node Proximity Detection
Detects and displays location where the distance between 2 nodes is less than the suggested cell size.

Shift Nodes Automatically
Shifts 2 or more physical component nodes within the specified distance to an analytical node.
Display/Hide Components Representative  
Turns on/off the Component Representative layer. This layer displays physical endpoint, centerline, and edge locations for producing a manual mesh or shifting nodes manually between components.

Shift Nodes Manually  
Shifts the node of a selected component to a selected location. The new location is updated when Automatic Mesh is performed.

Cancel Node Shift  
Resets automatic or manual node shift values of components to zero for the selected mode, Single- or Multi-Level. To modify a single component, double-click and use the FEM tab.

Exclude Meshing  
Creates region by graphical or coordinate entry of the polygon vertices to exclude or include-only for automatic meshing. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command. Double-click on the polygon to for the excluder properties.

Manual Mesh Generation  
Use for manually meshing the slab by input of element edges on horizontal and vertical sides. Select the slab region and select the option. Use in Single-Level mode.

Erase Mesh  
Removes the automatic or manual mesh currently in place in Single- or Multi-Level mode.

Mesh Generation  
Opens the Automatic Mesh Generation Dialog dialog to select meshing method, cell size and shape, and node consolidation options. Select Mesh Slabs to produce slab mesh before analysis. Walls are meshed as part of the analysis run.
4.3.12.3 Analysis Panel

**Execute Analysis**
Opens the **Analysis Options** dialog to run the analysis for the selected mode of operation in Single- or Multi-Level mode. You can specify **Analysis Options** in this window prior to analysis.

**Analysis Options**
Opens the **Analysis Options** dialog to select and save analysis options, load combinations to be processed, usage case to be considered and Global (Multi-Level) analysis results to be considered when analyzing in Single-Level mode.

**Cracked Deflection**
Processes the cracked deflection check for combinations set to **Cracked Deflection**. This requires the user to perform analysis and design the design sections. This check only applies to Single-Level mode.

**Result Display Settings**
Opens/Closes the **Result Display Settings** panel for load selection and settings of graphical analysis and design results.

**View Analysis Results**
Opens the ADViewer for graphical FEM analysis results.

**Numerical Display**
Displays numerical values at each element node or design section for the selected graphical result in **Result Display Settings**.

**Results Scale Down**
Reduces the graphical results scale ordinate.

**Default Results Scale**
Resets the graphical results scale ordinate to the program default relative to max/min values on the element or design strip.

**Results Scale Up**
Increases the graphical results scale ordinate.

### 4.3.12.4 Contour Settings Panel

**Display Contours**
Displays on/off line contours for FEM slab and wall graphical results.

**Increase Number of Contour Lines**
Increases the number of contour lines for the graphical display of FEM slab and wall graphical results. Active only when gradient contours are active from *Result Display Settings – Settings – Contour Settings*.

**Decrease Number of Contour Lines**
Decreases the number of contour lines for the graphical display of FEM slab and wall graphical results. Active only when gradient contours are active. *Result Display Settings – Settings – Contour Settings*.

**Display/Hide Text on Line Contours**
Displays on/off text values for contour lines of FEM slab and wall results.

### 4.3.12.5 Warping Panel

**Display/Hide Contour Warping**
Displays on/off a warped view of the selected FEM slab or wall graphical contour results.
Increase Warping Scale
Increases the warping scale of the selected warped graphical result.

Decrease Warping Scale
Decreases the warping scale of the selected warped graphical result.

4.3.12.6 Reactions Panel

Stored Reactions
Opens the Reaction Manager to view or delete stored Global (Multi-Level) and Single-Level load case and combination reactions for each Usage Case.

4.3.12.7 Visibility Panel

Display/Hide Analysis Elements
Displays on/off shell and frame elements and analysis nodes for the current view. The model requires meshing before display is available.

Display/Hide Nodes
Displays on/off all analysis nodes in the current view. The model requires meshing before the display is available.

Display/Hide Frame Elements
Displays on/off frame elements for columns and beams for the current view. The model requires meshing and saved analysis results before display is available.

Display/Hide Shell Elements
Displays on/off shell elements for slabs and walls for the current view. The model requires meshing before display of slab elements is available and analysis before display of the wall elements is available.

**Display/Hide Excluder**
Displays on/off slab excluder polygons for the current view. To view excluder with mesh inside or outside of excluder region, mesh the slab and use **Display/Hide Shell Elements**.
4.3.13 Floor Design Ribbon

4.3.13.1 Punching Shear Panel

Execute Shear Check
Performs the punching shear check for columns and qualifying walls at the current plane. This option is only operable in Single-Level mode after Single- or Multi-Level analysis has been performed.

Display Punching Shear Design Outcome
Graphically displays and color codes the outcome of the punching shear check as OK (blue), NA (blue), Exceeds Code (red) or Reinforce (red).

Numerical Display
Graphically reports the maximum punching shear stress ratio for the local axis directions and/or combined stress (when selected in Criteria – Shear Design Options).

4.3.13.2 Strip Modeling Panel

Create X-Direction Support Line
Adds a support line to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the support line to modify Support Line Properties and assign the support line in X-direction.

Create Y-Direction Support Line
Add a support line to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the support line to modify Support Line Properties and assign the support line in Y-direction.

**Support Line Wizard**
Opens a dialog window for setting parameters for auto-generation of support lines. This requires the user to specify the start and endpoint of a scanning band for selection of support line vertices.

**Create X-Direction Splitter**
Adds a splitter to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the splitter to modify Support Line Properties and assign as X-direction. X-Direction is indicated by circular symbol on vertices.

**Create Y-Direction Splitter**
Adds a splitter to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the splitter to modify Support Line Properties and assign as Y-direction. Y-Direction is indicated by square symbol on vertices.

**Extend Support Lines to Slab Boundaries**
Auto-adjusts the selected support line/s end-points to the nearest slab edge.

**Create Support Line Tributary Region**
Creates a tributary region assigned to a support line by graphical or coordinate entry of the polygon vertices. Select the support line and then the tool. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command.

**Create Manual Design Section**
Creates a manual design section by graphical or coordinate entry of the load start and end-point.
4.3.13.3 Section Design Panel

Generate Sections New
Auto-generates tributary regions and design sections relative to the modeled support lines and associated properties. This option will erase manual tributary regions.

Generate Sections Existing
Generates design sections for user-defined tributary regions and modified tributary regions auto-generated by the program.

Design the Sections
Processes the design of manual and auto-generated design sections for the selected design code and criteria and selects Calculated Reinforcement to satisfy code checks for the processed combinations.

Investigate the Sections
Processes the flexural capacity of manual and auto-generated design sections for user-defined Base Reinforcement including post-tensioning when modeled.

Delete Design Strips
Erases auto-generated and manually created design strips (tributary regions) and design sections.

4.3.13.4 Rebar Panel

Generate Rebar
Opens the Generate Rebar Drawing Options dialog to select combinations, bar length selection and orientation for graphical display of
calculated reinforcement for X and Y design strips at top and bottom locations.

4.3.13.5 Tools Panel

Open BuilderSum
Open the support lines results in the results interface BuildSum for viewing combination and envelope stresses, force and reinforcement diagrams and design summary.

4.3.13.6 Steel FRC Panel (MAT Only)

FRC Non-Linear Parameters
Opens the dialog windows for calculating non-linear parameters for load transfer over joints, load regions and joint spacing. Applies to MAT mode only when a fiber material is assigned in Criteria-Concrete and the Non-Linear option is selected.

4.3.13.7 Strip Results/Visibility Panel

Display Strip Results
Opens the ADAPT Solid Modeling Viewer. This tool is used to dynamically view the model components, imported CAD drawings, and view displacements and deformations.

Results Scale Down
Reduces the graphical results scale ordinate.
Default Results Scale
 Resets the graphical results scale ordinate to the program default relative to max/min values on the element or design strip.

Results Scale Up
 Increases the graphical results scale ordinate.

Numerical Display
 Displays numerical values at each element node or design section for the selected graphical result in Result Display Settings.

Display Max/Min Values
 Shows the maximum and minimum values of the selected results.

Perpendicular Projection
 When selected, displays the selected graphical design strip or beam element result from Result Display Settings in the XY plane. Use Top View for results display.

Display Design Sections
 Displays off/on design sections along support lines and is dependent on the selection made for All Section, X direction or Y direction within this tool panel. To generate design sections, go to Floor Design-Section Design-Generate Design Sections. This display is ON by default when sections are generated.

Display/Hide Support Lines
 Displays on/off support lines assigned to X and Y directions. If Display Design Sections is active and sections have been generated from Floor Design-Section Design, support lines and design sections will appear. Support lines and sections will be displayed by default for both directions when they are generated.

Display/Hide Support Lines in X-Direction
 Displays on/off support lines assigned to X direction. If Display Design Sections is active and sections have been generated from Floor Design-Section Design, support lines and design sections will appear.

Display/Hide Support Lines in Y-Direction
Displays on/off support lines assigned to Y direction. If **Display Design Sections** is active and sections have been generated from **Floor Design-Section Design**, support lines and design sections will appear.

**Display/Hide Splitters**
Displays on/off splitters assigned to X and Y directions.

**Display/Hide Splitters in X-Direction**
Displays on/off splitters assigned to X direction.

**Display/Hide Splitters in Y-Direction**
Displays on/off splitters assigned to Y direction.

**Display/Hide Support Line Tributaries**
Displays on/off tributary regions for support lines with colored polygons and is dependent on the selection made for X and Y direction, X direction only or Y direction only within this tool panel. Tributaries are generated when design sections are generated from Floor Design-Section Design-Generate Design Sections, but are hidden in display by default.

**Display/Hide Support Line Tributaries X-direction**
Displays on/off tributary regions assigned to X direction. If Generate Design Sections from Floor Design-Section Design has been performed, tributary regions will not be shown by default.

**Display/Hide Support Line Tributaries Y-direction**
Displays on/off tributary regions assigned to Y direction. If Generate Design Sections from Floor Design-Section Design has been performed, tributary regions will not be shown by default.
4.3.14 PT/RC Export Ribbon

4.3.14.1 Material Properties Panel

Concrete Material Properties for Strip Method
Opens the Concrete (Strip Method) dialog window for defining concrete properties exported to ADAPT-PT/RC.

Mild Steel Material Properties for Strip Method
Opens the Mild Steel (Strip Method) dialog window for defining mild steel properties exported to ADAPT-PT/RC.

Prestressing Material Properties for Strip Method
Opens the Prestressing (Strip Method) dialog window for defining post-tensioning properties exported to ADAPT-PT/RC.

4.3.14.2 Design Criteria/Settings Panel

Tendon Height Defaults
Opens the Criteria – Tendon Height Defaults for Strip Method menu for defining the distance from top and bottom slab or beam surface to the CGS of tendons at supports and span exported to ADAPT-PT/RC.

Effective Width
Gives option to consider the effective flange width in ADAPT-PT/RC when Beam design criteria is set in the Criteria-Design Code.
**Loading Treatment Options**
Sets the loading treatment options for LL skipping, Self-weight consideration and live load reduction when exporting strips to ADAPT-PT/RC.

**PT or RC Design Options**
Sets general PT or RC (depending on scope selected in opening splash screen) design options when exporting a strip to ADAPT-PT/RC.

### 4.3.14.3 Load Factors Panel

![Load Factors Panel]

**Load Combinations for Export to ADAPT-RC (RC Only Mode)**
Sets the load combination factors to be used when exporting a strip to ADAPT-RC.

**Load Combinations for Export to ADAPT-PT (PTRC Mode)**
Sets the load combination factors to be used when exporting a strip to ADAPT-PT.

### 4.3.14.4 Strip Modeling Panel

![Strip Modeling Panel]

**Support Line Wizard**
Opens a dialog window for setting parameters for auto-generation of support lines. This requires the user to specify the start and endpoint of a scanning band for selection of support line vertices.

**Create X-Direction Support Line for Strip Method**
Adds a support line to the structure by graphical or coordinate input of the support line vertices. Select **ESC** or right-click **Exit** to close the command. Double-click on the support line to modify **Support Line Properties** and assign the support line in X-direction.
Create Y-Direction Support Line for Strip Method
Adds a support line to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the support line to modify Support Line Properties and assign the support line in Y-direction.

Create X-Direction Splitter for Strip Method
Adds a splitter to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the splitter to modify Support Line Properties and assign as X-direction. X-Direction is indicated by circular symbol on vertices.

Create Y-Direction Splitter for Strip Method
Adds a splitter to the structure by graphical or coordinate input of the support line vertices. Select ESC or right-click Exit to close the command. Double-click on the splitter to modify Support Line Properties and assign as Y-direction. Y-Direction is indicated by square symbol on vertices.

Extend Support Lines to Slab Boundaries
Auto-adjusts the selected support line/s end-points to the nearest slab edge.

Create Support Line Tributary Region for Strip Method
Creates a tributary region assigned to a support line by graphical or coordinate entry of the polygon vertices. Select the support line and then the tool. Use ENTER or right-click Close/End/Accept, to close the polygon. Select ESC or right-click Exit to close the command.

Connect Support Lines to Columns and Walls
Auto-connects support line vertices to walls and columns if a vertex is located on a column or wall. For Strip Method export to PT/RC support lines vertices must be at a support centroid or end-point.

Create Strip Method Load Transfer
Creates a point support at user-defined location for a selected support line by graphical or coordinate entry where one support line is supporting another and both are exported to ADAPT-PT/RC by Strip Method. This would be used for beam-supporting beam.
4.3.14.5 Design Strips Panel

Generate Design Strips for Strip Method
Auto-generates tributary regions (design strips) relative to the modeled support lines for export to ADAPT-PT/RC.

Delete Design Strips
Erases auto-generated and manually created design strips (tributary regions) and design sections.

4.3.14.6 Export Panel

Execute ADAPT-PT/RC
Opens the selected design strip in ADAPT-PT/RC.

Import Tendons from ADAPT-PT
Imports tendons for design strips exported to ADAPT-PT and solved for without unlocking the PT model. Imports for design strips or all design strips.

Generate Input Data for ADAPT-PT/RC
Auto-generates and stores the .adb input files for all design strips to be executed within ADAPT-PT/RC.
4.3.15 Column Design Ribbon

4.3.15.1 Type Manager Panel

**Section Type Manager**
Opens the *Type Manager* dialog for creating section types for concrete component design and generic section modeling. Use this tool to create column design sections with reinforcement prior to Code Check/Design of columns.

4.3.15.2 Settings Panel

**Set Path for S-Concrete**
Sets the file path to the location of the S-Concrete application executable for design integration with ADAPT-Builder.

**Column Design Options**
Opens the *Design Options* dialog for selection of design load combinations, column design parameter and design constraints.

4.3.15.3 Live Load Reduction Ribbon
Live Load Reduction\textsuperscript{LLR}

Opens the Live Load Reduction Factors table for reduction method, area and factors, application settings. These reduction factors apply only to reduction of loads tagged as \textbf{Reducible} from Loading-Load Case Library. The reduction factor calculation is dependent on creation of tributary regions from Loading-Tributary Load Manager. Load Reduction Factors are applied only in two ways:

- **Column Design**
  Go to Column Design-Column Design Options. If set to YES the reduced loads will be applied for column design.

- **Analysis of Single-Level**
  If a Multi-Level analysis has been performed, reactions can be re-applied in Single-Level mode with inclusion of load takedown (Fz reactions). Go to Analysis-Analyze Structure and use the option to Apply Live Load Reduction.

4.3.15.4 **Design Panel**

\textbf{Code Check}\hspace{1cm} Performs a column code check for the defined section type and reinforcement assigned to the selected columns or design groups. The code check is performed for the defined Design Code set in Column Design – Column Design Options. Right-click on a column to view the design summary or use Results – Result Display Settings – Column – Individual Column Design Results.

\textbf{Design Columns}\hspace{1cm} Performs the design for the selected column design groups per the set design code and presents the proposed design in a summary for acceptance. If the design is accepted and updated, the Section Type Manager will be updated to reflect the change. Right-click on a column to view the design summary or use Results – Result Display Settings – Column – Design Group Results.
Open in S-Concrete
Opens the selected column in S-Concrete for code check or design.

Column Design Summary
Opens the Design Summary window to display the current and proposed results for designed column groups.

Result Display Settings
Opens/Closes the Result Display Settings panel for load selection and settings of graphical analysis and design results.

4.3.15.5 Reports Panel

Column Reporting
NO TOOL TIP

Column Reporting Expanded

- Column Reactions – Tabular
  Produces column reactions at top and bottom column locations for solved load combinations.
- **Column Reactions - Graphical**
  Opens the Column Reactions dialog to produce graphical column reactions.

- **Column Design**
  Opens a .XLS file containing column design information.

- **Column Tributary**
  Opens a .XLS file containing tributary area, load and FEM reaction information for solved load cases.

- **Tributary Load Check**
  Opens a .XLS file containing tributary load validation check for equilibrium.

- **Column Design Summary**
  Opens the Design Summary window to enable selection of View Report for the HTML summary page for design groups.

### 4.3.15.6 Labels Panel

![Labels Panel](image)

- **Reset Non-User Defined Labels**
  Resets the labels of all components in the model back to a counter of 1 with the “Label prefix” for each component type and for components whose label has not been altered by the user.

- **Reset All**
  Resets the labels of all components in the model back to a counter of 1 with the “Label” prefix for each component type.

- **Reset Column/Wall Stack**
  Opens the dialog window for Automatic Labeling of Stacked Supports and resets the labels of all continuously vertical column and/or walls. Applies the label at the bottom-most component in the stack to all other columns or walls in the stack.
4.3.16 Wall Design Ribbon

4.3.16.1 Settings Panel

Define Pier Labels
Creates pier labels for assignment to walls or wall groups.

4.3.16.2 Sections Panel

Generate Wall Sections
Creates wall design sections at the top and bottom of each wall assigned as a pier. Sections are created for each wall leg associated to a pier when the angle with adjacent walls is more than 10deg.

Update Wall Sections
Updates existing wall sections when material properties or wall length or thickness have changed.
4.3.16.3  Design Panel

**Wall Design Manager**
Opens the **Wall Design Manager** for creating wall section reinforcement, parameters, selection of load combinations, processing code checks or design and reporting interaction and wall intersection diagrams.

**Wall Design Manager Expanded**

- **Locate S-Concrete**
  Sets the file path to the location of the S-Concrete application executable for design integration with ADAPT-Builder.

**Wall Design Summary**
Opens the HTML summary page for the selected wall design sections.

**Result Display Settings**
Opens/Closes the **Result Display Settings** panel for load selection and settings of graphical analysis and design results.

4.3.16.4  Reports

**Wall Reporting**
NO TOOL TIP

**Wall Reporting Expanded**
- **Wall Reactions - Tabular**
  Produces wall reactions at top and bottom column locations for solved load combinations.

- **Column Reactions - Graphical**
  Opens the *Wall Reactions* dialog to produce graphical wall reactions.

- **Pier Reactions**
  Opens a .XLS file containing pier geometry, properties and reactions for solved load combinations.

- **Wall Design Sections**
  Opens a .XLS file containing wall design section reinforcement, geometry, and design loads and utilization checks.

- **Wall Design Summary**
  Creates a combined PDF summary report for the selected wall design sections.

- **Wall Intersection Summary**
  Creates a combined PDF summary report for joint intersection details of designed wall sections.
4.3.16.5  Labels Panel

Reset Non-User Defined Labels  
Resets the labels of all components in the model back to a counter of 1 with the “Label prefix” for each component type and for components whose label has not been altered by the user.

Reset All  
Resets the labels of all components in the model back to a counter of 1 with the “Label” prefix for each component type.

Reset Column/Wall Stack  
Opens the dialog window for Automatic Labeling of Stacked Supports and resets the labels of all continuously vertical column and/or walls. Applies the label at the bottom-most component in the stack to all other columns or walls in the stack.

4.3.17  Reports Ribbon

4.3.17.1  Print Panel

Print  
Opens printer selection and settings and prints active screen.
4.3.17.2 Compiled Reports

Open Report Compiler
Opens the Report Generation Manager for selecting tabular and graphical reports and screenshots to be generated as a compiled report. After making selections, use FILE-Generate Compiled Report.

Take Report Screenshot
Saves the graphical screen with user-defined name to the Report Generation Manager. Use this option to produce custom images for a compiled report.

4.3.17.3 Single Default Reports Panel

General
Report/Project Title Settings
Set the General and Specific title and information lines.

Report Cover
Generates a .RTF file with report title settings.
Material and Design Criteria
Gerates a .RTF tabular report with material and design criteria settings for the model.

Quantity and Cost
Generates a .RTF tabular report with material quantity and cost for the model components for the current mode (Single- or Multi-Level).

Structural

Slab Regions Detailed Report
Generates a .RTF file with slab vertices coordinates for slabs in the current mode.

Slab Regions Summary Report
Generates a .RTF file with slab material, area and volume for slabs in the current mode.

Openings Detailed Report
Generates a .RTF file with opening vertices coordinates for openings in the current mode.
Beams Detailed Report
Generates a .RTF file with beam material, geometry, volume and length for beams in the current mode.

Columns Detailed Report
Generates a .RTF file with column material, geometry, volume and length for columns in the current mode.

Drop Caps/Panels Detailed Report
Generates a .RTF file with drop cap/panel material and geometry for drop caps/panels in the current mode.

Walls Detailed Report
Generates a .RTF file with wall material, geometry, volume and length for walls in the current mode.

Point Springs Detailed Report
Generates a .RTF file with spring translational and rotational subgrade modulus values for and location coordinates for point springs in the current mode.

Point Springs Summary Report
Generates a .RTF file with spring subgrade modulus for Kzz, location coordinates and spring type (C or T) for point springs in the current mode.

Area Springs Detailed Report
Generates a .RTF file with spring subgrade modulus for Kzz and location coordinates for area springs in the current mode.

Releases and Restraints Detailed Report
Generates a .RTF file with point and line support coordinates and translational and rotational fixity in the current mode.

Plan Geometry
Displays a plan view of all modeled components and dimensions for the current level.


**Load Cases and Combinations**
Generates a .RTF file with load cases and combinations defined in the model.

**Skip Pattern**
Generates a .RTF file listing pattern loads with included sub-regions, skip factors and load reduction factors.

**Applied Loads**
Generates a .RTF file with coordinates, load magnitudes and parameters (Seismic) for all applied loads in the model.

**Dead Load**
Displays a plan view of all modeled Dead loads and magnitude for the current level.

**Live Load**
Displays a plan view of all modeled Live loads and magnitude for the current level.

**User Defined Load Cases**
Displays a plan view of all modeled user-defined General load case loads and magnitude for the current level.
Column Tributary
Opens a .XLS file containing tributary area, load and FEM reaction information for solved load cases.

Tributary Load Check
Opens a .XLS file containing tributary load validation check for equilibrium.

Tendon

Tendon Report
Generates a .RTF file with individual tendons geometry, system type and friction parameters for tendons in the current mode.

Tendon Group Report
Generates a .RTF file with tendon group geometry, system type and friction parameters for the model.

Tendon Elongation, Individual Report
Generates a .RTF file with individual tendons jacking force, seating distance, and elongation for tendons in the current mode. Requires use of Calculated Force option for stressing in the tendon properties.

Tendon Elongation, Group Report
Generates a .RTF file with tendon group jacking force, seating distance, and elongation for the model. Requires use of Calculated Force option for stressing in the tendon properties.
**Tendon Totals**
Generates a .RTF file with tendon totals for strand and duct length, number of strands, weight and stressing ends for tendons in the current mode.

**Tendon Plan**
Opens the **Tendon Report** dialog for display selection and creates a plan view of tendons for the current level.

**Rebar**

**Rebar List**
Generates a .RTF file with size, quantity, and length for calculated and base reinforcement in the current mode.

**Rebar Legend**
Generates a .RTF file for a rebar legend with like reinforcement for size, length and placement (T or B) in the current mode.

**Rebar Totals**
Generates a .RTF file with total size, quantity, length and cost for all longitudinal reinforcement in the current mode.

**Design Section Rebar Tabular Report**
Generates a .RTF tabular report for required flexure and shear reinforcement at design sections in the current mode.

**Punching Shear Reinforcement**
Generates a .RTF file with schedule for stud rails or links at column locations requiring shear reinforcement.

**Rebar Plan**
Opens the Rebar dialog for display selection and creates a plan view of reinforcement for the current level.

**Punching Shear**

**Stress Check Parameters**
Generates a .RTF file with a summary of column punching shear parameters for the critical section producing the controlling stress ratio.

**Stress Check Results**
Generates a .RTF file with a summary of column punching shear results for the critical section producing the controlling stress ratio.

**Punching Shear Reinforcement**
Generates a .RTF file with schedule for stud rails or links at column locations requiring shear reinforcement.
Column

Column Reactions - Tabular
Produces column reactions at top and bottom column locations for solved load combinations.

Column Reactions – Graphical
Opens the Column Reactions dialog to produce graphical column reactions.

Column Design
Opens a .XLS file containing column design information.

Column Tributary
Opens a .XLS file containing tributary area, load and FEM reaction information for solved load cases.

Tributary Load Check
Opens a .XLS file containing tributary load validation check for equilibrium.

Column Design Summary
Opens the Design Summary window to enable selection of View Report for the HTML summary page for design groups.
Wall

Wall Reactions - Tabular
Produces wall reactions at top and bottom column locations for solved load combinations.

Column Reactions - Graphical
Opens the Wall Reactions dialog to produce graphical wall reactions.

Pier Reactions
Opens a .XLS file containing pier geometry, properties and reactions for solved load combinations.

Wall Design Sections
Opens a .XLS file containing wall design section reinforcement, geometry, and design loads and utilization checks.

Wall Design Summary
Creates a combined PDF summary report for the selected wall design sections.

Wall Intersection Summary
Creates a combined PDF summary report for joint intersection details of designed wall sections.
4.3.17.4 Analysis Reports Panel

Design Section/Strip

RTF Reports

Design Section Forces
Design Section Rebar
Design Section Moment Capacities
Design Section Dimensions
Design Section Geometry

Graphical Reports

Support Lines X-direction
Support Lines Y-direction
Design Strips X-direction
Design Strips Y-direction
Design Check in X-direction
Design Check in Y-direction

Design Section Forces Tabular Report
Generates a .RTF file reporting design section actions and stresses (when tendons included) at the current level.

Design Section Rebar Tabular Report
Generates a .RTF file for required flexure and shear section reinforcement at the current level.

Design Section Moment Capacities Tabular Report
Generates a .RTF file for positive and negative capacities for design sections in the current level.

Design Section Dimensions Tabular Report
Generates a .RTF file for design section dimensions for sections at the current level.

Design Section Geometry Tabular Report
Generates a .RTF file for geometric properties for sections at the current level.

**Support Lines X-Direction Plan**
Displays X-direction support lines and labels in plan view.

**Support Lines Y-Direction Plan**
Displays Y-direction support lines and labels in plan view.

**Design Strips X-Direction Plan**
Displays X-direction design strips and labels in plan view.

**Design Strips Y-Direction Plan**
Displays Y-direction design strips and labels in plan view.

**Design Check X-Direction Plan**
Displays X-direction design strips graphical stress check in plan view.

**Design Check Y-Direction Plan**
Displays Y-direction design strips graphical stress check in plan view.

**Analysis Data**

**Equilibrium Check**
Generates a .RTF file with gravity and lateral load equilibrium checks for solved applied loads and reactions.

**Solution Statistics**
Generates a .RTF file reporting total number of nodes, elements and solved load cases and combinations for the current analysis.
**Analysis Nodes**
Generates a .RTF file for analysis nodes and location coordinates for the current analysis.

**Analysis Elements**
Generates a .RTF file for shell and frame elements with connected analysis nodes for the current analysis.

**Analysis Mesh Plan**
Displays a plan view of the current analysis finite element mesh.

**Reactions**

![Analysis Reactions](image)

**Point Support Reactions**
Generates a .RTF file with point support label and reactions for Fxx, Fyy, Fzz, Mxx, Myy, and Mzz.

**Line Support Reactions**
Generates a .RTF file with point support label and reactions for Fxx, Fyy, Fzz, Mxx, Myy, and Mzz.

**Point Spring Reactions**
Generates a .RTF file with point support label and reactions for Fxx, Fyy, Fzz, Mxx, Myy, and Mzz.

**Line Spring Reactions**
Generates a .RTF file with point support label and reactions for Fxx, Fyy, Fzz, Mxx, Myy, and Mzz.

**Pier Reactions**
Opens a .XLS file containing pier geometry, properties and reactions for solved load combinations.
4.3.17.5 Export DWG Panel

Export DWG/DXF
Opens the dialog for selecting the AutoCAD version and tendon type (linear or spline) for export of the current view as a .DWG or .DXF file.
Chapter 5

MODELING AND DESIGN PROCESS
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 User 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.
Chapter 5  MODELING AND DESIGN PROCESS

- 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 2018 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 2018 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
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® 2018
- ADAPT-MAT® 2018

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 transforming 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 a number of 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 the majority of 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:
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:

\[ U = 1.40 \times \text{Selfweight} + 1.40 \times \text{Dead load} \]

For Dead and Live Load:

\[ U = 1.20 \times \text{Selfweight} + 1.20 \times \text{Dead load} + 1.60 \times \text{Live load} \]

6.3.6.2 Serviceability Load Combinations

Load Combinations for Serviceability Check:

Sustained in-service load combination (stress check)
\[ U = 1.00 \times \text{Selfweight} + 1.00 \times \text{Dead load} + 0.30 \times \text{Live load} \]

Total in-service load combination (stress check)
\[ U = 1.00 \times \text{Selfweight} + 1.00 \times \text{Dead load} + 1.00 \times \text{Live load} \]

6.3.6.3 Initial Load Combinations

Load Combinations for Initial staged check:

\[ U = 1.00 \times \text{Selfweight} \]

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:

\[
\text{U} = 3.00 \times \text{Dead load} + 1.00 \times \text{Live load}
\]

Load combination for checking deflection under live load:

\[
\text{U} = 1.00 \times \text{Live Load}
\]

### 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 \( k_1, k_2 \) and \( k_3 \). 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.
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 $k_e$ is given by:

$$\frac{1}{k_e} = \frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_3}$$

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 $k_e$. For displacement “$d$” at any given point, the soil pressure “$p$” is:

$$p = k_e \times d$$

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:

$$r_1 = \frac{p}{k_1}$$
Hence, the vertical displacement of interface B will be:

\[ d_B = d - rA \]

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:

\[
\frac{1}{k_e} = \left(\frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_3}\right) \\
= \left(\frac{1}{200} + \frac{1}{250} + \frac{1}{300}\right) = 1/81.08
\]

\[ k_e = 81.08 \text{ pci} \]

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

\[ P = k_e \times d = 81.08 \times 0.138 = 11.19 \text{ psi} \]

(1611 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:
\[ r_1 = \frac{p}{k_1} = \frac{11.19}{200} = 0.056 \text{ inch} \]

Displacement at layer B, \( d_B = d_A - r_1 = 0.138 - 0.056 = 0.082 \) inch

Compression in thickness of second layer:
\[ r_2 = \frac{p}{k_2} = \frac{11.19}{250} = 0.04476 \text{ inch} \]

Displacement at layer C, \( d_C = d_B - r_2 = 0.82 - 0.0447 = 0.0373 \) inch

Compression in thickness of third layer:
\[ r_3 = \frac{p}{k_3} = \frac{11.19}{300} = 0.0373 \text{ inch} \]

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 into 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.
6.4.2 Transformation of Structural Components

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

![Transform tools icon]

**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.
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 **Display View** 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.
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.
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.

\[
Ec = Wc^{1.5} \times 33 \sqrt{f’c} \quad \text{US}
\]

\[
Ec = Wc^{1.5} \times 0.043 \sqrt{f’c} \quad \text{SI}
\]

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](image-url)
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 $f_y$ as **60 ksi** and the value of $E_s$ 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.

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).
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.

![Soil Support Property Box](image)
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 of 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.

Go to the **Shear Design** tab to specify the preferred bar size as #3 (0.375 in); shear reinforcement type is stud and number of rails per side is 2. Finally click on OK to apply the changes.
6.5.4 Input and Assign Loadings

ADAPT-MAT will automatically consider Self-Weight as we specified \( W_c \) 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](image)

**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 of 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.
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 $F_z = 56.2$ kip inside the **General** tab and go to **Loads** tab to input $F_x = 18$ kip and $F_y = 9$ kip. Finally click on the green checkmark to assign the load values. Once done program will remember these values for further generation(s).

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
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 2018 Basic Manual for a thorough description of meshing features in ADAPT-Build 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.
Click on OK to generate finite element mesh as shown below.

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.
6.6.3 View Results

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

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.
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.

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.

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 back to 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 Support Line Wizard tool from Floor Design | Strip Modeling. A dialog box, as shown in Figure 6.7-1, will be displayed. Specify the intended direction of the Support Line as the X-Direction.

- Set the band width to 1 ft. The program will scan between this width for supports (columns, walls or beams). A support line vertex (click) will be created at each support. The distance or length between vertices is known as a support line “span.” Click on the OK button to create the support line.
Repeat the operation to create other support lines in the X-direction. You may hit Enter to repeat last operation in the ADAPT-MAT environment. For the example model (see Figure 6.7-1) it is possible to create the first 4 support lines from the top edge of slab in the X-direction. However you need to use Create Support Line X or Y tool from the Floor Design | Strip Modeling to complete support line 5, 6 and 7 as they are not continuous along the full length of slab. Notice also that Support Line 6 does not extend to a slab or opening boundary as all other support lines do. In this case, we need to use vertical splitters to generate a proper tributary region for this support line and those adjacent to it. It is recommended to scan the model you are using for the tutorial and determine which support lines may require the use of splitters.

FIGURE 6.7-1 SUPPORT LINE WIZARD

FIGURE 6.7-2 SUPPORT LINES IN X-DIRECTION
6.7.2 Use of Splitters

- For this example, the **Create X- or Y-Direction Splitter** tool from Floor Design | Strip Modeling is used to draw two vertical splitters on each side of Support Line 6. Both splitters will extend from the endpoint of Support Line 6 to the bottom slab edge and from the endpoint of Support Line 6 vertically such that they terminate perpendicular to Support Line 4.

- Once you click on the **Create X- or Y-Direction Splitter** tool, you cursor will be in **Snap mode**. Use **Modify | Item’s Properties** to specify the property of the splitters you are going to create. The **Splitter** dialog box, as shown in, **Figure 6.7-3** will be displayed.

- Specify the direction for the support line for which this splitter will be associated. This is the **X-direction** for this example. The splitters as described above are shown in **Figure 6.7-4**. If you’re model is different than the example for this tutorial, input splitters in a like manner for all necessary support lines.

![Splitter Dialog Box](image)

**FIGURE 6.7-3 SPLITTER DIALOG BOX**
• Splitters 1 and 2 in this example are shown in Figure 6.7-4.

• Using the approach described above for the X-direction, draw support lines in **Y-direction**. From Figure 6.7-5, note that Support Lines 8-15 were created using the **Support Line Wizard** tool. Support Line 16 requires use of the **Create Support Line X or Y** tool since it only contains two supports at the interior of the slab in the width and length of the scanning band where the **Support Line Wizard** is used.

• In this example, for Support Line 16, we need to create two more splitters to correctly generate tributary regions for this design strip. These splitters are shown in Figure 6.7-5. In your model, review the support lines and identify those which require the use of splitters. If you are using this model for the tutorial, input the splitters as shown below. The Y-direction splitters need to be associated with Y-direction support lines. Make sure that the splitters properties reflect this.
• 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.

6.8 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 can specify to automatically generate column and middle strips. Go to Criteria | Design Criteria | Analysis/Design Options to open the dialog box shown in Figure 6.8-1.
6.8.2 Generate Design Sections

To generate design sections at each of the support lines, use the menu item

**Floor Design | Section Design | Generate Sections New**. This will introduce a middle strip in between the main support lines (column strips) and generate design strips in X and Y directions.
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 the either direction as shown in Figure 6.8-2. Middle and column strips are shown hatched in Figure 6.8-3 for this example.

![FIGURE 6.8-2 DESIGN STRIPS IN X- AND Y- DIRECTION](image)

![FIGURE 6.8-3 COLUMN AND MIDDLE STRIPS IN X-DIRECTION](image)

- Notice that the program will introduce intermediate support lines in between the user-defined support lines identifying column strips. These strips are referred to as “middle strips” and are generated due to the selection being made from **Criteria | Design Criteria | Analysis/Design Options** menu. Had this item been selected as **No**, the program would have created design strips associated with user-defined support lines only. Also notice the generation of strips where splitters were used for this example.
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 6.8-5. Click on Yes to save the results and close this message.

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 **Result Display Settings**. 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 pull-down box in **Result Display Settings** as shown in **Figure 6.8-6**. Set the result option to **Deflection** and set the maximum allowable to **L/240**. Select **Apply** and then **OK**. From **Floor Design | Section Results/Visibility | Display Graphically**, making sure that the support lines in either direction are turned on, and the program will report the deflection ratio (X/L) for each span as shown in **Figure 6.8-7**. 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 **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 **Result Display Settings** dialogue box. The program will give the option to report top and bottom fiber stresses for any service combination as well and average precompression.*
FIGURE 6.8-6 RESULT DISPLAY SETTINGS DIALOGUE BOX

FIGURE 6.8-7 DEFLECTION RESULTS ALONG SUPPORT LINE

FIGURE 6.8-8 SUPPORT LINES AND DESIGN SECTIONS RESULTS IN X-DIRECTION
Figures 6.8-8 and 6.8-9 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 | Calculate Rebar Plan** to display the rebar required to meet demand for serviceability, strength or envelope. The dialog box as shown in Figure 6.8-10 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 6.8-10.
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 6.8-11.

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 6.8-12 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 6.8-12 Automatic Mesh Reinforcement Dialog](image)

**FIGURE 6.8-12 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 6.8-13.

![Figure 6.8-13 REBAR IN EXCESS OF BASE REBAR](image)

FIGURE 6.8-13 REBAR IN EXCESS OF BASE REBAR

6.8.8 Punching Shear Check

- Use the menu item **Floor Design | Punching Shear | Execute**

  ![Punching Shear](image)

  to perform a check of punching (two-way) shear for this slab. The program will show the following dialog when done.

FIGURE 6.8-17 PUNCHING SHEAR CHECK COMPLETION DIALOG

- Use **Floor Design | Punching Shear | Display Punching Shear**

  ![Design Outcome](image)

  to view the results on screen. The **Numerical Display** tool must be active 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:

  o “OK” – calculated stresses are below allowable stresses
  o “Reinforce”- calculated stresses exceed allowable stresses and design code requires reinforcement
  o “Exceeds Code” – design code requirements are not satisfied
  o “NA”- punching shear design is not applicable
• The punching shear results for this model are shown in Figure 6.8-14.

FIGURE 6.8-14 PUNCHING SHEAR DESIGN OUTCOME

• Further information related to punching shear including design parameters, actions, actual stress from shear and moments and allowable stress is 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:
  
  o Punching Shear Stress Check Result
  o Punching Shear Stress Check Parameters, and
  o Punching Shear Reinforcement

• An example of each is given in Figures 6.8-15 to 6.8-17.
In this example since the stress ratio for all columns are less than 1, shear reinforcement is not required. In order to obtain a solution which requires shear reinforcement in order to display shear reinforcement output, the
Strength combination was increased to include a factor of 2.0. The results from this design are shown in Figure 6.8-21.

180.90 SCHEDULE OF STUD RAILS FOR PUNCHING SHEAR REINFORCEMENT

<table>
<thead>
<tr>
<th>Column ID</th>
<th>Number of studs/rail</th>
<th>Stud diameter (in)</th>
<th>Studs</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>0.50</td>
<td></td>
<td>3.00</td>
<td>6.25</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Notes:
Dimensions are in (in)
Distance = distance of the vertical bars from the respective column face
(7) 16.5 indicates layer 7 at distance 16.50 in from face of support.
(7) 16.5" indicates layer 7 at distance 16.50 in from face of support.
** = exceeds maximum allowable by code
Refer to details for arrangement.

FIGURE 6.8-17 PUNCHING SHEAR REINFORCEMENT
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 User 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 similar to 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 User 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-Buildr Platform”].

7.3 MISCELLANEOUS TOPICS

7.3.1 Soil Pressure

In addition to the values reported for slabs (stresses, moments, etc.), similar to 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.