



ADAPT-Modeler 20

USER MANUAL

Copyright© 2020

Contents

1	Getting Started.....	7
2	Quick Reference Guide	9
	2.1 Generate Structural Model	9
	2.2 Analysis Using ADAPT-PT or ADAPT-RC.....	9
	2.3 Analysis Using ADAPT-FLOOR PRO	10
3	Basic Operations and Main Menu	17
	3.1 ADAPT-Modeler Main Screen	17
	3.2 Mouse Function and Operation	18
	3.3 Cursor Function AND Operation	18
	3.4 Operation of <i>Return</i> and <i>Tab</i> Keys	20
	3.5 Operation of Key Combinations	20
	3.6 How to End/Close an Operation	20
	3.7 How to Abandon an Operation	20
4	Toolbars and Dialog Windows.....	21
	4.1 Introduction	21
	4.2 Main User Interface	21
	4.2.1 Top-Left Quick-Access Default Tools (1).....	21
	4.2.2 Top-Right Level Tools (2)	22
	4.2.3 Message Bar (3).....	23
	4.2.4 Bottom Quick Access Tools (4).....	23
	4.2.5 Bottom-Right – Status Bar (5)	24
	4.2.6 The Result Browser (6)	25
	4.2.7 Properties/Colorize Grid (7).....	26
	4.3 File Ribbon	27
	Used to update or create a new file from.....	28
	4.4 Home Ribbon	29
	4.4.1 Display Panel	29
	4.4.2 Selection Mode Panel.....	30
	4.4.3 Selection Tools Panel.....	31
	4.4.4 Draw Panel	31
	4.4.5 Tools Panel	34
	4.4.6 Zoom/Camera Panel.....	34
	4.4.7 Viewport Panel.....	36

4.4.8	WCS/UCS (Coordinate Systems) Panel	37
4.4.9	Scaling Panel	38
4.4.10	Snap Tools Panel	39
4.5	Visibility Ribbon	40
4.5.1	Message Bar Panel	40
4.5.2	Colors Panel	40
4.5.3	Section Panel	41
4.5.4	Render Panel	41
4.5.5	Display Settings Panel	41
4.5.6	Selection Panel	42
4.5.7	Viewing Panel	42
4.5.8	Shading Panel	43
4.6	Modify Ribbon	43
4.6.1	Properties Panel	43
4.6.2	Edit Panel	44
4.6.3	Copy/Move Panel	45
4.6.4	Tools Panel	46
4.6.5	Blocks Panel	47
4.6.6	Add/Remove Points Panel	47
4.6.7	Labels Panel	48
4.7	Criteria Ribbon	48
4.7.1	Material Properties Panel	48
4.7.2	Design Criteria Panel	49
4.7.3	Design Criteria (SOG) Panel	51
4.8	Model Ribbon	52
4.8.1	Wizards Panel	52
4.8.2	Level Assignment Panel	52
4.8.3	Gridline Panel	52
4.8.4	Type Manager Panel	53
4.8.5	Structural Components Panel	54
4.8.6	Isolate Panel	55
4.8.7	Preprocessing Panel	55
4.8.8	Preprocessing (SOG) Panel (in SOG only)	56
4.8.9	Supports Panel	56
4.8.10	Springs Panel	57
4.8.11	Displacements Panel (in SOG only)	58
4.8.12	Visibility Panel	58

4.8.13 Properties Panel	60
4.8.14 Transform Panel	60
4.9 Loading Ribbon	61
4.9.1 Load Combo Panel	61
4.9.2 LL Reduction Panel	63
4.9.3 General Panel	64
4.9.4 Lateral/Building Panel.....	65
4.9.5 Tributary Panel	66
4.9.6 Pattern Panel.....	66
4.9.7 Visibility Panel	66
4.10 Tendon Ribbon.....	68
4.10.1 Criteria Panel.....	68
4.10.2 Settings Panel.....	69
4.10.3 Model Panel	69
4.10.4 Modify Panel	70
4.10.5 Shop Drawing Panel (add-on Module)	71
4.10.6 Visibility Panel	73
4.10.7 Design Panel.....	73
4.11 Rebar Ribbon	74
4.11.1 Design Criteria Panel	74
4.11.2 Model Base Rebar Panel	75
4.11.3 Generate Panel.....	75
4.11.4 Visibility Panel	76
4.11.5 Reporting Panel	76
4.12 Analysis Ribbon	77
4.12.1 Vibration Panel.....	77
4.12.2 Meshing Panel.....	77
4.12.3 Analysis Panel.....	78
4.12.4 Contour Settings Panel	79
4.12.5 Warping Panel	80
4.12.6 Reactions Panel	80
4.12.7 Visibility Panel	81
4.13 Floor Design Ribbon	81
4.13.1 Punching Shear Panel	81
4.13.2 Strip Modeling Panel	82
4.13.3 Section Design Panel	83
4.13.4 Rebar Panel	84

4.13.5	Tools Panel.....	84
4.13.6	Steel FRC Panel (MAT Only).....	85
4.13.7	Strip Results/Visibility Panel.....	85
4.14	PT/RC Export Ribbon.....	87
4.14.1	Material Properties Panel.....	87
4.14.2	Design Criteria/Settings Panel.....	88
4.14.3	Load Factors Panel.....	88
4.14.4	Strip Modeling Panel.....	89
4.14.5	Design Strips Panel.....	90
4.14.6	Export Panel.....	90
4.15	Column Design Ribbon.....	91
4.15.1	Type Manager Panel.....	91
4.15.2	Settings Panel.....	91
4.15.3	Live Load Reduction Ribbon.....	92
4.15.4	Design Panel.....	92
4.15.5	Reports Panel.....	93
4.15.6	Labels Panel.....	94
4.15.7	Visibility Panel.....	95
4.16	Wall Design Ribbon.....	95
4.16.1	Settings Panel.....	95
4.16.2	Sections Panel.....	95
4.16.3	Design Panel.....	96
4.16.4	Reports.....	96
4.16.5	Labels Panel.....	98
4.16.6	Visibility Panel.....	98
4.17	Reports Ribbon.....	99
4.17.1	Print Panel.....	99
4.17.2	Compiled Reports.....	99
4.17.3	Single Default Reports Panel.....	100
4.17.4	Analysis Reports Panel.....	109
5	Structural Modeling Tools.....	113
5.1	Overview.....	113
5.2	Structural Model Parts.....	114
5.2.1	Structural Components.....	114
5.2.2	Common Properties of Structural Components.....	115
5.3	Level Assignments.....	119

5.4	Level Assignment Tool	121
5.5	Organization of the Structural Components Data	122
5.6	Modeling Options	122
5.6.1	Transform Panel	123
5.6.2	Add Structural Components Panel	125
5.6.3	Model→Visibility Panel	130
5.6.4	Loads.....	131
6	Strip Modeling Tools	139
6.1	Overview	139
6.2	Strip Modeling Tools	141
6.3	Strip Generation Tools	151
6.4	Strip visibility Tools	152
6.5	Other Data Specific to ADAPT-PT and ADAPT-RC.....	152
6.5.1	Data Specific to Current Design Strip	152
6.5.2	Data Applicable to the Entire Project.....	153
7	Finite Elements (FLoor Design) Modeling Tools	157
7.1	Overview	157
7.2	Meshing	158
7.2.1	Overview	158
7.2.2	Maximum Mesh Size	158
7.2.3	Meshing Tools	159
7.3	Execute Analysis.....	164
7.4	View Analysis Results	165
7.5	Examine Design Values	166
7.6	Edit/Apply Boundary Conditions	170
7.6.1	Overview	170
7.6.2	How to View and Edit the Boundary Conditions.....	170
7.6.3	How to Apply Boundary Conditions	171
7.6.4	Model Ribbon, Supports and Springs Panels	172
8	Appendix A.....	175
8.1	Treatment of Compound (Interconnected) Wall Assemblies	175
8.2	Structural Modeling	175
8.3	Support Lines and Design Strips.....	175

1 Getting Started

This manual describes the user interface of the Builder software platform, along with the tools you will use, if you want to generate data for ADAPT-PT and ADAPT-RC. Becoming familiar with the program interface and its various tools will serve you well in your modeling and design work. If you have experience with the program and want to refresh your memory before starting a new project, go to **Chapter 2** – Quick Reference Guide.

This manual is divided into several chapters as follows:

- **Chapter 2** is a quick reference guide for those who have used the program, and simply intend to refresh your experience, before starting a new project
- **Chapter 3** describes the basic operations and main menus of the program
- **Chapter 4** walks you through the basic operations tools of the program. You will use these tools in all modeling, analysis, and design work. You will find the tools offer an extensive drafting capability, allowing you to faithfully replicate complex structures
- **Chapter 5** shows you how to build a structural model in three dimensions. It also covers the common situation in which you would use an architect's drawing as the basis of your structural model
- **Chapter 6** targets only those of you who will be generating your structural models in Builder platform to create design strips for export to ADAPT-PT
- **Chapter 7** covers the basic Finite Elements Modeling (FEM) Tools. This chapter introduces FEM and illustrates basic operations of meshing and FEM analysis

For your next step, depending on the way you want to use the program, it is best to go through one or more of the program tutorials¹.

¹ The tutorials can be downloaded from the download email. In addition you may email adaptsupport@risa.com in order to obtain a copy of the tutorial files.

2 Quick Reference Guide

The natural sequence from creating a structural model, to generating structural documents and subsequently fabrication drawings, is listed in the following. There are many short cuts and alternatives in the process of data generation and execution. These, however, are avoided in the following list in favor of a more common and straightforward approach.

2.1 Generate Structural Model

Generate a structural model, either by importing a DWG file and converting it to structural components, or creating your own structural model using the tools of ADAPT-Modeler. Once you are done, additional details for the analysis depend on whether you plan to analyze the model using ADAPT-PT, ADAPT-RC, or ADAPT-Floor Pro. The following is a guide.

2.2 Analysis Using ADAPT-PT or ADAPT-RC

If you want to use the structural model in connection with ADAPT-PT or RC¹

- **Enter loads** (dead and live load cases only)². Skipping of live load, live-load reduction and other load-related issues are handled in the *PTRC Export* ribbon, *Design Criteria* and *Load Factor* panels, which will be covered later, or directly in ADAPT-PT or RC.
- **Go to *PTRC Export* → *Material Properties***, and review/edit the contents of the following icons:
 - Concrete (Strip Method)
 - Mild Steel (Strip Method)
 - Prestressing (Strip Method)
- **Go to *PTRC Export* → *Design Criteria* and *PTRC Export* → *Load Factor***. Review/edit all the input screens, as these will be used as default values for your entire project. You will have the option to modify these individually once data is exported to ADAPT-PT or RC.

¹ The input screens that are specific to ADAPT-PT and ADAPT-RC are marked with a brown background. The input screens that apply to both ADAPT-PT and RC and Floor Pro are identified with a neutral background color.

² PT version 7.xx and RC version 4.xx can accept only dead and live loads, plus the optional inclusion of selfweight. Later versions of these programs can handle more load cases.

You need not go to the load case and load combinations, since these are handled in ADAPT-PT and RC.

- **Generate support lines** in two orthogonal directions. If a support line is intended to rest on anything but a column or wall, enter a point support³.
- **Create design strips**
 - Consider the design strips one by one⁴
 - Select a design strip and view its idealization in 3D viewer
 - Export/open ADAPT-PT or RC
 - Once in ADAPT-PT or RC, view the imported data for accuracy
 - Edit data if necessary
 - Execute data
 - Obtain a report
- Go back to the Modeler environment and consider the next design strip

2.3 Analysis Using ADAPT-FLOOR PRO

If you want to use the structural model in connection with ADAPT Floor-Pro

- **First run for model validation:**
 - Go to *Analysis* → *Meshing* and click on the *Mesh Generation* icon. Accept defaults of the program.
 - Once meshing is complete, go to *Analysis* → *Analysis* and click on the *Execute Analysis* icon.
 - Once analysis is complete, the *Results Browser* will open.
 - In the *Loads* tab of the *Results Browser* the user can choose the load combination, envelope of load combination, or load case of the result the user would like to review. Select the *Service (Total Load)* load combination.
 - Go to the *Analysis* tab of the *Results Browser*, here the user can view result contours and diagrams for components (slabs, beams, columns, walls). Expand the *Slab* tree by clicking on the + sign next to slabs. Expand the *Deformation* tree by clicking on the + sign next to deformation. Check the box for *Z-Translation*. This will display the deflection contour for the *Service (Total Load)* load combination. Without dead and live loads, or post-tensioning added to the model the result should be based on the structure self-weight alone.

³ If a support line rests on another support line, or is resting on a beam along its length, you must enter a point support at the location where the design strip you are creating is intended to be supported.

⁴ You have the option to use the Strips pull-down menu to generate the input data for the entire set of design strips at one time. But if you are not familiar with the program, it is best to do them one by one, as suggested herein.

- Zoom, rotate, and view the results thoroughly to ensure that the deflected shape under selfweight looks reasonable. Make sure there is no deflection where the structure was intended to have been supported. Correct the structural model if the deflected shape and values under selfweight do not appear reasonable.
 - Alternatively, the user can use the legacy ADViewer to view results. Once analysis is complete, select *View Analysis Results* from the *Analysis* → *Analysis* panel. This opens the 3D viewer of the program to display the solution.
 - Once in 3D viewer, select the *Service (Total Load)* combination and Z-Translation. This is vertical displacement of the structure. Then, click on the tool with two light-bulb graphics. This will display the deflected shape for the *Service (Total Load)* load combination. Without dead and live loads, or post-tensioning added to the model the result should be based on the structure self-weight alone.
 - Zoom, rotate, and view the results thoroughly to ensure that the deflected shape under selfweight looks reasonable. Make sure there is no deflection where the structure was intended to have been supported. Correct the structural model if the deflected shape and values under selfweight do not appear reasonable.
- **Add loads:**
 - Go to *Loading* → *Load Case/Combo* and click on the *Load Cases* icon to add load cases, such as dead load, live load, prestressing, and other load cases that you want to include in your design.
 - In the *Loading* ribbon use the other tools to enter the loading.
 - **Add prestressing:** If the structure has prestressing, add the prestressing tendons.
 - **Edit material properties:** Go to *Criteria* → *Materials Properties*, and enter the material properties for concrete, nonprestressed steel and prestressed steel, if applicable. If there is more than one concrete material, steel or prestressing in your structure, this is the time to give a label to each of the new materials used and define their properties. In your modeling, the program has assumed that all the components of the structure you created have the material names entered on the first line of each of the lists. If you added any new material to the list of existing materials, open the property box of the structural components that must have the new material and change their material name to the one you created. It is recommended not to rename or remove the default material in any material list. If you want to add a specific

material with a specific name, add a new material and modify the label of the new material leaving the default material intact.

- **Review/edit design criteria:** Go to *Criteria* → *Design Criteria*, click the *Design Code* icon and review the default values of each of the tabs in the criteria window that displays. The other icons in this icon panel open the same criteria dialog window but to the icon specified tab. Modify the criteria, as necessary. Make sure that you select the building code of your choice. Once you select/confirm the building code, the program automatically creates the default load combinations of the building code you selected. Note: Changing the code will reset the load combinations (delete any user input load combinations and revert to the default combinations for the selected code) in the model. Before adding to the load combinations, the user should ensure they are in the design code they want to use for the design of the model.
- **Add extra load combinations for validation:**
 - Go to *Loading* → *Load Case/Combo* and click on the *Load Combinations* icon.
 - Create a load case for selfweight only⁵, with *No Code Check* option.
 - View and edit the load cases and load combinations. If you have prestressing, create a load case (PT) for prestressing only.
 - Go to *Analysis* → *Analysis* and click on the *Execute Analysis* icon.
 - Once the analysis is complete use the *Results Browser* or, click the *View Analysis Results* icon within the *Analysis* → *Analysis* panel to check the deflection shape of each load case and load combination, to make sure they look reasonable. If deflections do not appear correctly, go back to the loads, criteria, and prestressing layout, if needed, in order to fix the problem.
- **Design:**
 - Using the support line *Dynamic Editor* from the *Floor Design* → *Strip Modeling* panel, create support lines in two orthogonal directions. Make sure that you assign the support lines in one direction X-direction, and its orthogonal direction Y-direction. (This does not necessarily mean the support lines run in the X and Y direction)

⁵ Once you add new loads, the selfweight load case is likely to become part of other load combinations. That is why you need to create a selfweight load combination. Also, if you plan to have skipping of live load, leave this option to the last, after you have made sure that the model you have created works well.

- Go to Floor Design → Section Design and click on Generate Section New. Save data.
 - From the Floor Design → Section Design and click on Design the Sections.
- **Check punching shear values:**
 - If you have a column-supported slab, go to *Floor Design* → *Punching Shear* and click on *Execute Shear Check*.
 - Once the shear check is complete, go to the *Results Browser*. Navigate to *Load Combos* → *Envelope* → *Envelope Strength*. In the *Analysis* tab of the results browser the user should see a *Punching Shear* tree. Expand the *Punching Shear* tree and check the box next to *Stress Check*. This will turn any supports that fail the punching shear requirements red, supports that need reinforcing will be pink, and supports that pass the shear check with no need for additional shear reinforcement will be green. Note: For shear results to be active and displayed the user must have selected the strength envelope combination or a single strength combination.
 - To view the stress ratios, expand the *Punching Shear* tree in the *Results Browser* and click the check box next to *Stress Ratio*.
- **Generate/view reinforcement:**
 - From the *Floor Design* → *Rebar* panel, click on *Calculated Rebar Plan* icon to compile a rebar drawing.
 - Go to *Rebar* → *Visibility* and click on the *Show Rebar* icon to make the reinforcement visible. This is the tool with yellow circle.
 - Use the capabilities of the other tools on this panel to view and edit the display.
- **Generate rebar drawings:**
 - Using the tools of the *Rebar* → *Visibility Panel*, select the reinforcement that you want to be shown on the structural drawing.
 - Edit/move the reinforcement annotation to make it arrive at a clear presentation.
 - Change the font size to values suitable for printing on the paper size you are going to select.
 - From the *File* menu, select *Print* to examine the features of the drawing you are going to print.
 - Print the drawing from the print preview window, or export it to AutoCad, using the *File* → *Export DWG* tool of the program.

- In the same way, generate other rebar drawings such as top bars on one drawing and bottom bars on another.
- **Generate tendon layout drawings:** On engineering drawings, most engineers group tendons into tendons in one-direction (such as banded tendons) and tendons in other direction (distributed tendons).⁶ If you plan to show the tendons in two drawings, you must first group them, following the instructions below. If this is not the case, go to the next step.
- **Group tendons:**
 - From the *Visibility* → *Display Settings* panel, select the *Group Library* icon. This opens the group library. Add two group names, such as “banded tendons,” and “distributed tendons.”
 - Using *Select/Set View Items*, turn off everything except tendons and the other basic information you need to identify the tendons. In most cases, it is adequate to retain the tendons, slab outline and column supports.
 - Select as many tendons of one group as practical.
 - From the *Modify* → *Properties* panel, select *Modify Selection*.
 - Once the *Modify Item Properties* dialog windows open, select check box next to *Group* on the left side of the window. Select the group dropdown menu and choose the group label from the dropdown menu you want to assign the tendons to.
 - Repeat the above steps, until all tendons are assigned to their respective groups.
 - Go to the *Grouping Dialog Window* and make only one group of tendons visible, such as distributed tendons. Once you have printed the drawing for this group, hide this group and make the next group visible.
- **Generate single report:**
 - From the *Reports* → *Single Default Reports* panel select *Tendon* → *Tendon Plan*.
 - In the dialog window that opens, select the following, and then click OK.
 - Tendon ID
 - Control point heights
 - Number of strands

⁶ For generating fabrication drawings, tendons are grouped more extensively, assigning unique group identification to tendons of same length and profile.

- Elongation (if you selected the option in data generation)
 - Stressing/dead end (if you selected the option in data generation)
- **Generate compiled report:** From the *Reports* → *Compiled Reports* panel, click on *Open Report Compiler* icon. Select the items of your choice and send to printer.

3 Basic Operations and Main Menu

3.1 ADAPT-Modeler Main Screen

Fig. 3.1-1 shows the full-screen display of the ADAPT-Modeler program, with typical features labeled for easy identification.

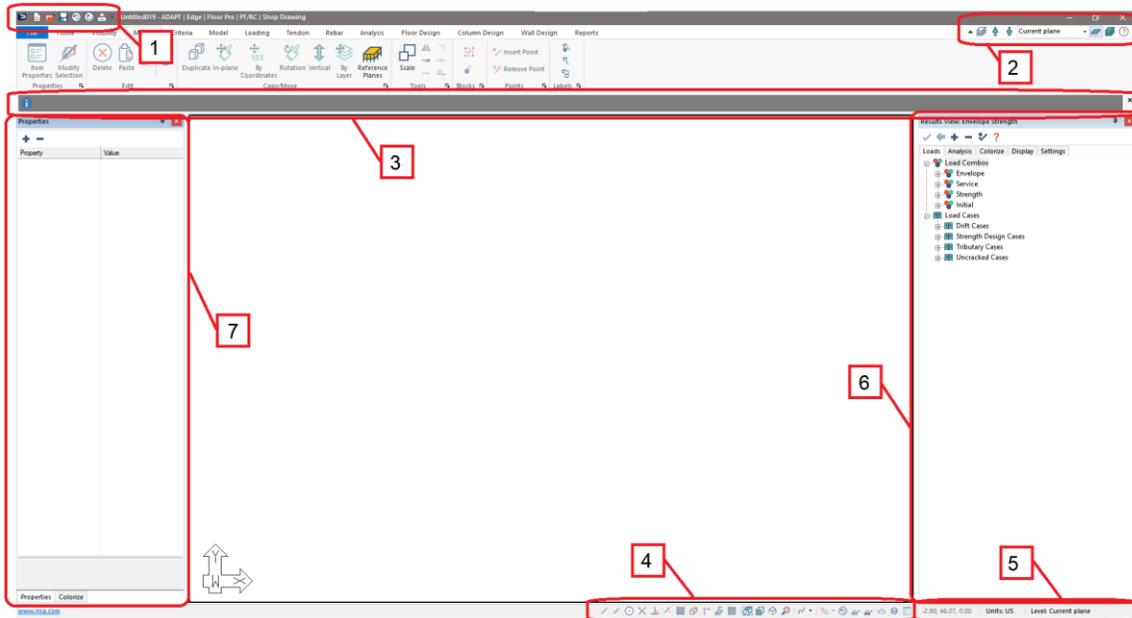


Figure 3.1-1

ADAPT-Modeler operates the same way as other Windows programs. All program tools are accessible through the Ribbon/Panel structure of the Graphical User Interface or through the *Top-Left Quick Access Default Tools (1)*, *Top-Right Level Tools (2)*, and the *Bottom Quick Access Tools (4)* toolbars. Ribbons can be customized by the user. In this document we will use the convention of *Ribbon Name* → *Panel Name* when describing where to access a program tool.

The *Message Bar (3)* displays tool-specific information prompted to the user and any coordinate values that may be typed by the user for specific program procedures.

The *Status Bar (5)* displays such information as the mouse cursor coordinates, current unit system, active level, current layer. A short description of each specific tool also appears in this area when the mouse cursor is placed over the corresponding tool button.

The *Results Browser (6)* displays the contour and diagram results (deflections, moments, shears, etc.) for the component and load combination/load case the

user has selected. The user can also modify the appearance of the results through the *Settings* tab. In addition, the user can set limits for the code checked items of the *Design Strip* results within the *Display* tab.

The *Properties/Colorize Grid (7)* allows the user quick access to component properties for modification. Additionally, the user can use this window to colorize components based on their properties. Lastly the user can also filter selections using these tools as well.

3.2 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 **Fig. 3.2-1**.

Exit	
Close/End/Accept	
Pan	
Dynamic Zoom	
Zoom Window	
Zoom Extent	
Undo	Ctrl+Shift+Z
Redo	Ctrl+Shift+A

Figure 3.2-1 - Right-Click Options of the Mouse

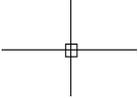
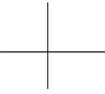
Click on a menu item listed to perform the operation described.

If you right-click the mouse over a structural component the program will bring up the right-click menu shown in **Fig. 3.2-1** along with additional options specific to modification of that component.

If you right-click the mouse while the cursor is outside the display screen, options to customize the ribbons or the bottom quick access panel will appear. From these options you can customize the programs display.

3.3 Cursor Function AND Operation

Depending on the cursor mode, the program responds differently. Before starting an operation, it is important to make sure that the cursor is in the appropriate mode.

Shape	Mode	Description
	<p>Selection/Pick</p>	<p>In this mode, you can select an entity displayed on the screen by placing the cross over it and left-clicking the mouse. Once an entity is selected, its color changes.</p> <p>There are two ways to enable Selection/Pick mode:</p> <ul style="list-style-type: none"> • Right-click the mouse, and select <i>Exit</i> • Go to <i>Home</i> → <i>Selection Mode</i> and click on the <i>Window Selection/Pick Mode</i>  icon.
	<p>Hint</p>	<p>In this mode, the program displays the identification of an entity that the point of the arrow touches. To change to this mode, go to <i>Home</i> → <i>Selection Mode</i> and click on the <i>Hint Mode</i>  icon.</p>
	<p>Creation</p>	<p>In this mode, the program will create an entity, such as a line, column or slab. Place the cross at the location where you want the entity to be created and left-click the mouse. Detailed instruction for creation of each entity will be prompted on the <i>Message Bar</i> at the top of the modeling space.</p> <p>To enable <i>Creation Mode</i>, left-click the mouse on the tool of the entity you intend to create. Then follow the instructions in the <i>Message Bar</i>.</p>
	<p>Snap</p>	<p>In this mode, the magnet indicates that the cursor is in <i>Snap Mode</i> and is searching to snap onto an entity. The cursor will search for one or more entities. Once the cursor becomes close to any of the entities or conditions it is searching for, it will display a yellow sign over the location to be snapped. The shape of the yellow sign displayed identifies the entity for snapping.</p>
	<p>Undefined Creation</p>	<p>In this mode, the program can be requested to create an entity, although the plane on which the entity is to be created is not displayed. You must change the screen view (go to <i>Top View</i> , if you are in a <i>different view</i>) before you can create the entity in mind.</p>

3.4 Operation of *Return* and *Tab* Keys

A special function assigned to the *Return* key is to repeat the last operation performed – when the duplication of such operation is practical. For example, if you use *Copy* command to copy an entity, pressing the *Return* key, will invoke the *Copy* command again.

In addition to its normal function, the *Tab* key can help you select an individual item in a group of similar items that overlap. When you attempt to select items that overlap, such as several beams intersecting at the same location, the mouse-click on its own is not adequate to identify the beam of your choice. The program will select one and change its color. If the program's selection is not the one you intended, click the *Tab* key. The program will select and display the next item. Continue clicking the *Tab* key, until the item of your choice is selected.

3.5 Operation of Key Combinations

Some key combinations act as shortcuts to pre-assigned operations. The key combinations are generally displayed on the computer screen, next to the menu items they represent. The most used combinations are:

Copy *Ctrl* key + C

Move *Ctrl* key + M

The user can also assign hot keys. Hot keys are user defined keyboard shortcuts. The user can assign hot keys using the *Customize Quick Access Toolbar* tool.

3.6 How to End/Close an Operation

To end or close an operation, such as closing a polygon, press the *End* or *C* key

3.7 How to Abandon an Operation

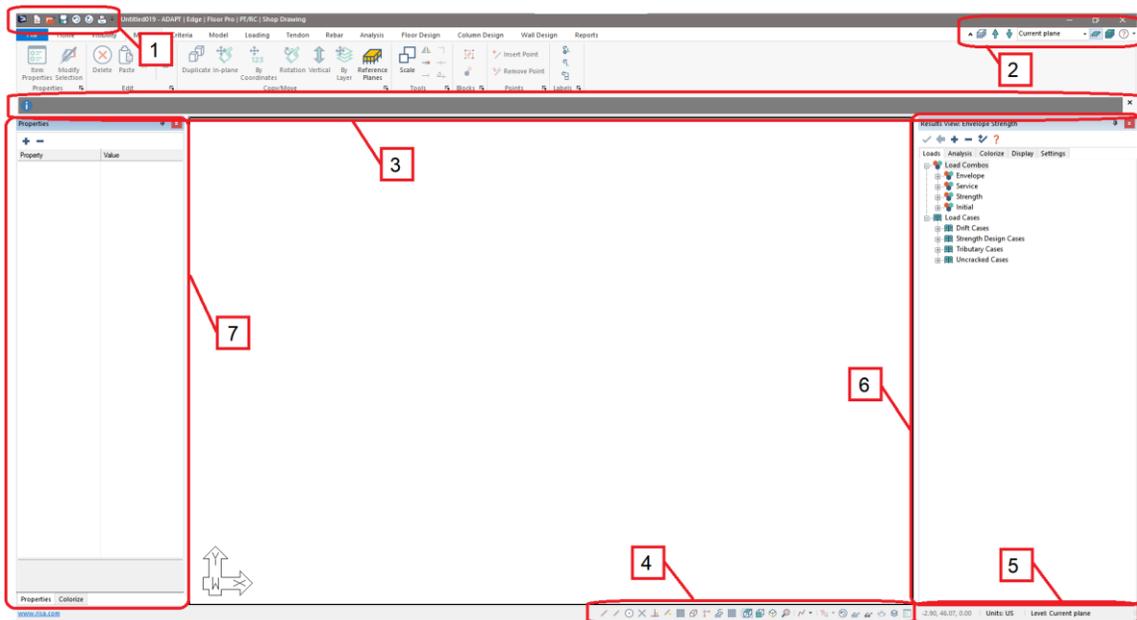
To abandon an operation you have already started, such as drawing a polygon, press the *Esc* key.

4 Toolbars and Dialog Windows

4.1 Introduction

The purpose of this section is to act as a reference guide to the menu items and icons in the software. Included with each icon/tool is a short description on the application of the tool in the software.

4.2 Main User Interface



4.2.1 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.

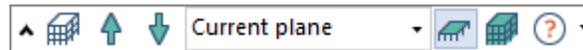
Print

Opens printer selection and settings and prints active screen.

Customize Quick Access Toolbar

Customizes the Quick Access Toolbar.

4.2.2 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.

Active Level Manager

Displays the current active level, and allows the user to navigate to and set levels active through a dropdown style menu.

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.

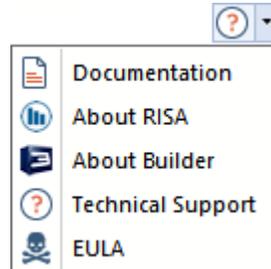
Multi-Level mode

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.

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 RISA** 

Opens a window providing general contact information for RISA, Tech.
- **About Builder** 

Gives ADAPT-Builder version information and support contact information.
- **Technical Support** 

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

Opens a window displaying the End-User License Agreement of the software.

4.2.3 Message Bar (3)



Display’s model information or prompts user for input.

4.2.4 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.

4.2.5 Bottom-Right – Status Bar (5)

53.20, -9.62, 0.00 | Units: US | Level: Current plane | Layer: Current_plane_Tendon

Cursor coordinates

53.20, -9.62, 0.00

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

Units

Units: US

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

Level: Current plane

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

Layer

Layer: Current_plane_Tendon

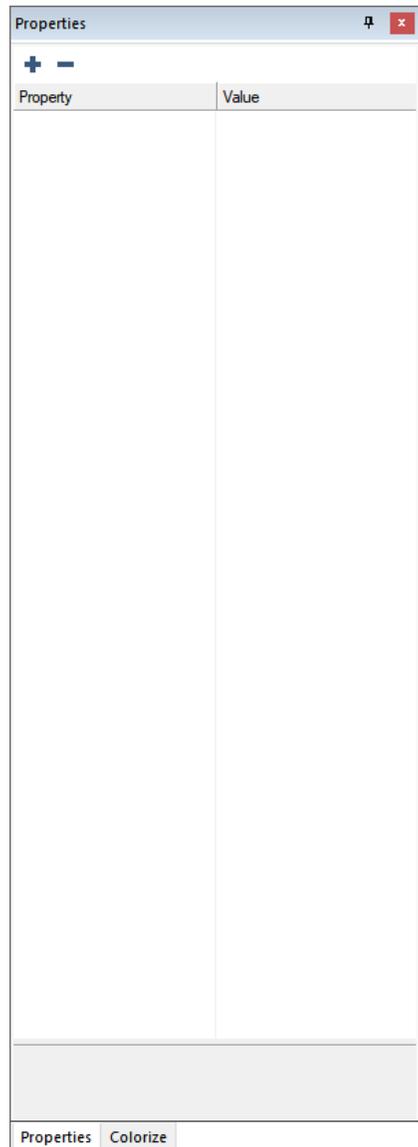
Reports the active layer. Select this text to change the active layer.

4.2.6 The Result Browser (6)



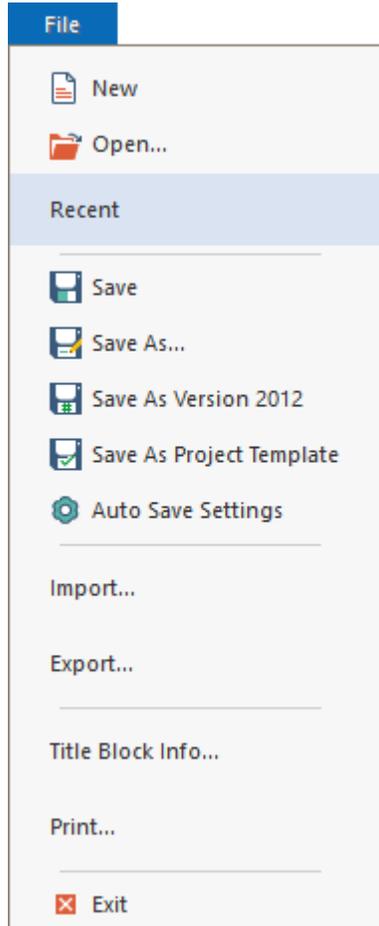
Graphical **Result Browser**. The user can view graphically by load combination/load case the results/actions for Slabs, Beams, Columns, Walls, Design Strips, Punching Shear results, and more. In addition the user can edit the code check limits as well as the display of graphical results within the **Result Browser**.

4.2.7 Properties/Colorize Grid (7)



The added property and colorize grid adds user efficiency to the program. The properties grid allows quick access to the properties of a selected, or multiple selected items. The item’s properties can be modified within the Properties Grid dialog window. The colorize tab allows the user to colorize components based on properties for an easy graphical verification of component properties. Both options also allow filtering of selected items. For detailed information on the Properties/Colorize grid refer to the **ADAPT-Builder 20 New Features Supplement Manual**.

4.3 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

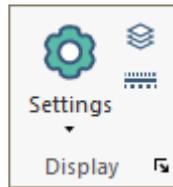
Opens printer selection and settings and prints active screen.

Exit 
 Exits the application.

4.4 Home Ribbon

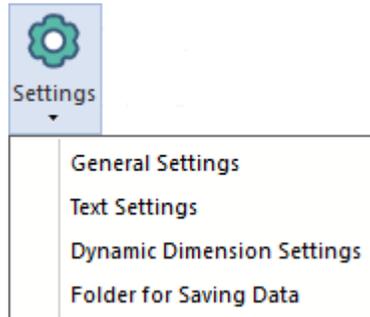


4.4.1 Display Panel



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.

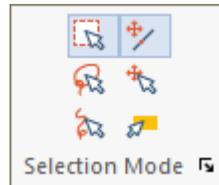
Layer Settings

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.

4.4.2 Selection Mode Panel



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.

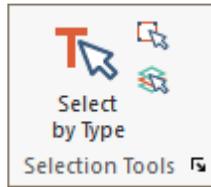
Move Selection

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 linestyle when the cursor is hovered over the component.

4.4.3 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.

4.4.4 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** 

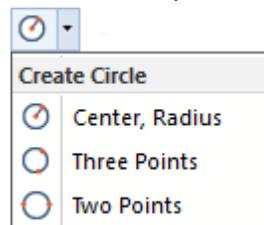
Creates a polygon by selecting number of sides, start point at mid-distance location of first edge, and end point at mid-distance location of second edge.
- Polygon: Vertices, Start Edge, End Edge** 

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



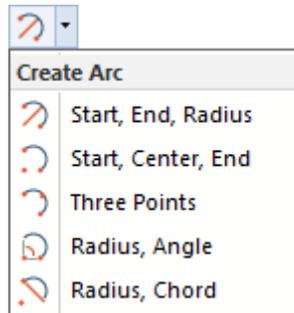
- Circle: Center, Radius** 

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.

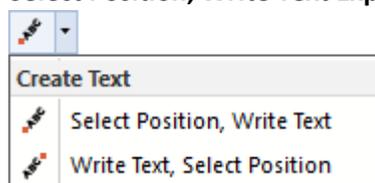
Arc: Start, End, Radius Expanded



- **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.

4.4.5 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.

4.4.6 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.

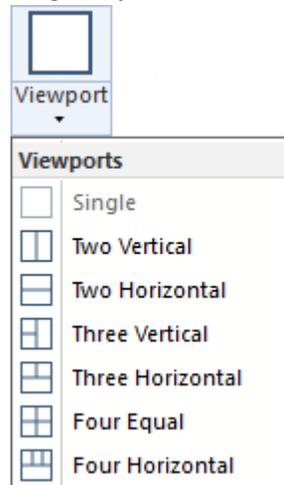
4.4.7 Viewport Panel



Single

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

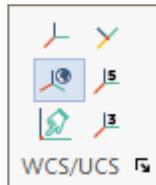
Single Expanded



- **Two Vertical** 
Creates the viewport shown.

- **Two Horizontal**  Creates the viewport shown.
- **Three Vertical**  Creates the viewport shown.
- **Three Horizontal**  Creates the viewport shown.
- **Four Equal**  Creates the viewport shown.
- **Four Horizontal**  Creates the viewport shown.

4.4.8 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.

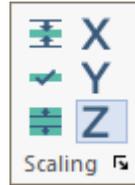
UCS: 5 points

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.

4.4.9 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 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.

4.4.10 Snap Tools Panel



Snap to Endpoint 
Snaps to the closest endpoint.

Snap to Midpoint 
Snaps to item's midpoint.

Snap to Center 
Snaps to the center of a radial item.

Snap to Intersection 
Snaps to closest point of intersection.

Snap to Perpendicular 
Snaps to the perpendicular point on the item that is currently under the mouse cursor.

Snap to Nearest 
Snaps to the nearest position of the considered item.

Snap to Grid 
Snaps to the closest point on the grid.

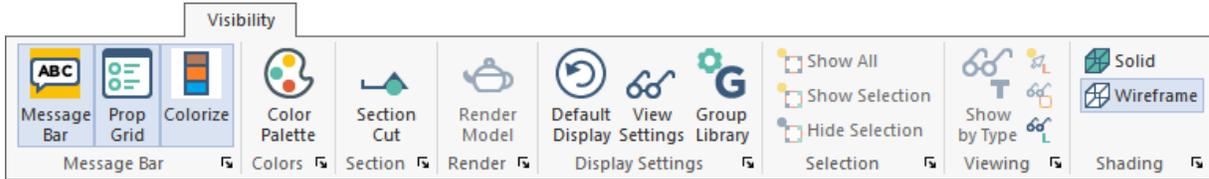
Snap to Vertices of Components 
Enables snapping to the vertices of a structural component.

Snap Orthogonal 
Toggles Ortho mode.

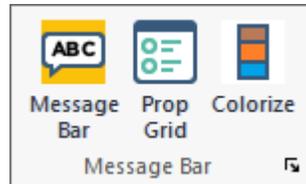
Snap Settings 
Opens the Snap Settings dialog box.

Grid Settings 
Opens the Grid Settings dialog box

4.5 Visibility Ribbon



4.5.1 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.

Prop. Grid

Enables or hides the Properties Grid. Allows the user quick access to selected component properties. Modifications to the properties of components can be made in this window. When disabled, properties for a component must be modified through the components property window. The Properties Grid window can be detached and docked to any side of the UI or left free floating.

Colorize

Enables or hides the Colorize Grid. Allows the user to colorize components based on component properties. The colorize window can be detached and docked to any side of the UI or left free floating.

4.5.2 Colors Panel

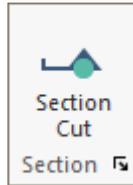


Color Palette

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**.

4.5.3 Section Panel



Create Section Cut 

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

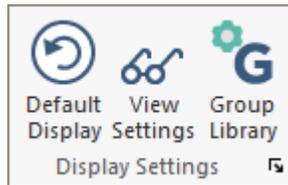
4.5.4 Render Panel



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.

4.5.5 Display Settings Panel



Default Display 

Restores the view to the user-defined default display

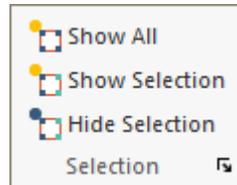
View Settings (Select/Set View Items) 

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.

4.5.6 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.

4.5.7 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**.

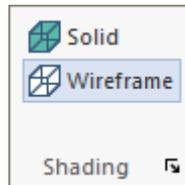
Display All Items 

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.

4.5.8 Shading Panel



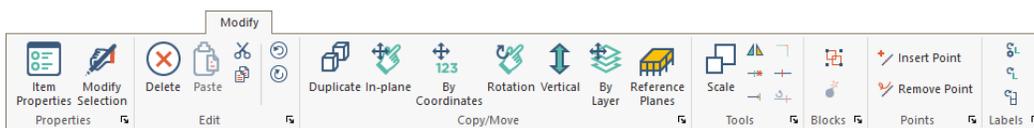
Solid Fill 

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.

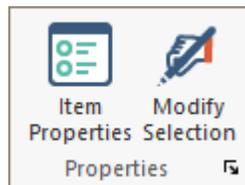
Wire Frame 

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

4.6 Modify Ribbon



4.6.1 Properties Panel



Items Properties 

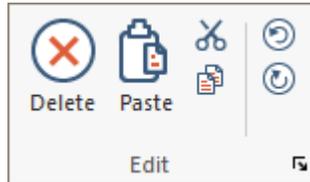
Opens the selected item's property dialog for modifying material, geometric and analytic properties associated with the component. This tool becomes active when an item is selected or will launch when

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.

4.6.2 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.

4.6.3 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 counter-clockwise 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

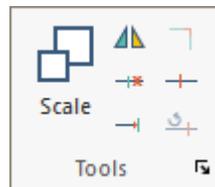
By Layer (Translation)

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**.

4.6.4 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.

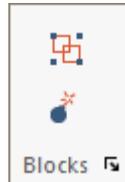
Split 

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.

4.6.5 Blocks Panel



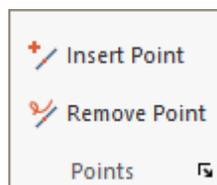
Creat Block 

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.

4.6.6 Add/Remove Points Panel



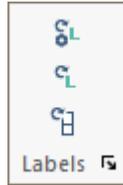
Insert Point 

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.

4.6.7 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.

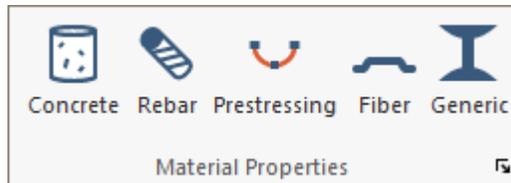
Reset Column/Wall Stack

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.

4.7 Criteria Ribbon



4.7.1 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**.

4.7.2 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

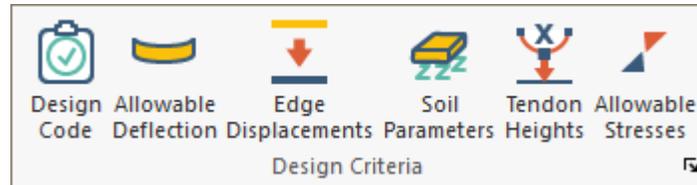
Opens the **Criteria – Rebar Round Up** menu options for bar length and spacing detailing options. Use this menu to input length and spacing distance rounding, bar spacing limits (RC mode only) and defining Rebar Library Lengths.

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.

4.7.3 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 E_m (edge moisture distance) and Y_m (expected vertical differential movement). These values are user-defined and are only used in calculating the **Edge Displacements**.

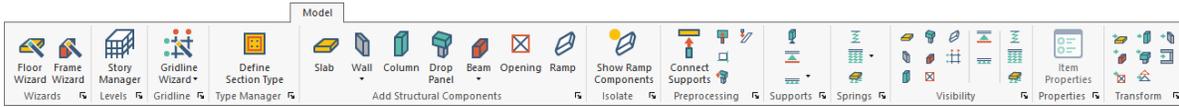
Tendon Height

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.

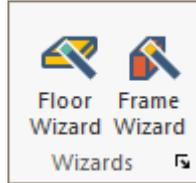
Allowable Stresses

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

4.8 Model Ribbon



4.8.1 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.

4.8.2 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.

4.8.3 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



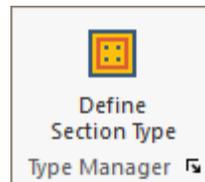
- **Gridline Wizard** 

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.

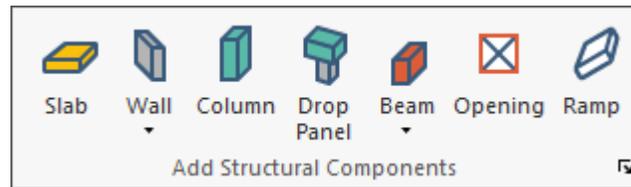
4.8.4 Type Manager Panel



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.

4.8.5 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.

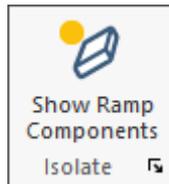
Opening 

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.

Ramp 

Creates a ramp by graphical or coordinate entry of the ramp vertices. The first two vertices entered apply to the active plane and the last two vertices entered locate the ramp at the plane below active.

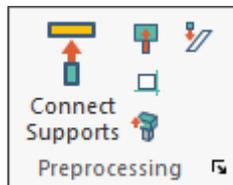
4.8.6 Isolate Panel



Show Ramp Components 

Isolates the graphical view to show only ramps in the model. Use **Visibility>Default Display**, **Visibility>View Settings**, or **Model>Visibility** to restore the view of other components.

4.8.7 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.

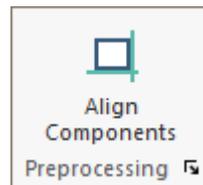
Connect Drop/Cap Panel to Column

Shifts the drop/cap panel insertion point to the column endpoint location for analysis.

Connect with Ramp

Truncates columns to below the ramp. Select columns and then select the tool. Offsets beams that are parallel or transverse to the ramp to be aligned along the ramp or offset below the ramp.

4.8.8 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.

4.8.9 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.

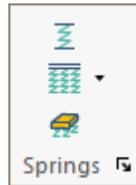
Add Line Support 

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.

4.8.10 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.

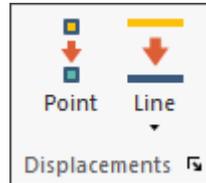
- **Continuous Modeling** 

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.

4.8.11 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.

4.8.12 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 Ramps 
Displays on/off ramps.

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.

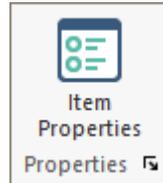
Display/Hide Line Springs  (N/A to SOG)
Displays on/off line springs. Double-click on spring to see sub-grade moduli settings for Kxx, Kyy and Kzz and to set compression or tension spring type.

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.

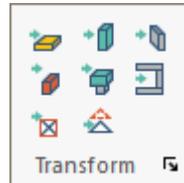
4.8.13 Properties Panel



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**.

4.8.14 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.

4.9 Loading Ribbon



4.9.1 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 compatibility 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.

4.9.2 LL Reduction Panel



Reduction Settings ^{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 **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**.

4.9.3 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.

4.9.4 Lateral/Building Panel



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.

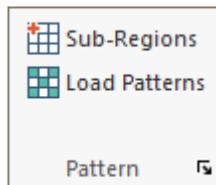
4.9.5 Tributary Panel



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.

4.9.6 Pattern Panel



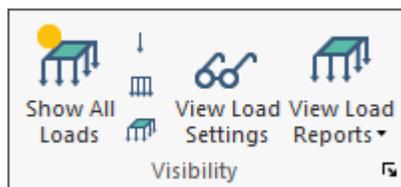
Sub-regions

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.

4.9.7 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.

Show Point Loads ↓

Displays on/off all point 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.

Show Line Loads 

Displays on/off all line 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.

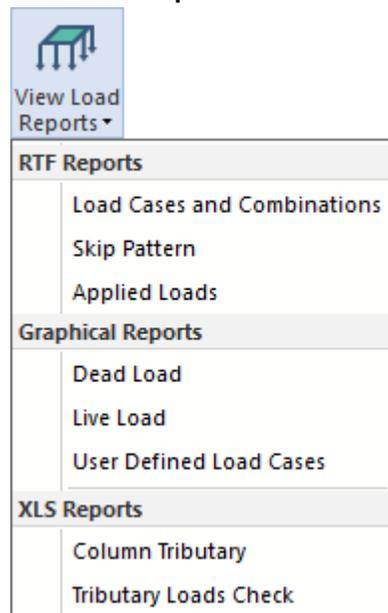
Show Area Loads 

Displays on/off all area 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.

View Load Settings 

Opens the graphical display settings for loading. Controls font and symbol sizes for loading

View Load Reports



	
View Load Reports ▾	
RTF Reports	
	Load Cases and Combinations
	Skip Pattern
	Applied Loads
Graphical Reports	
	Dead Load
	Live Load
	User Defined Load Cases
XLS Reports	
	Column Tributary
	Tributary Loads Check

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.

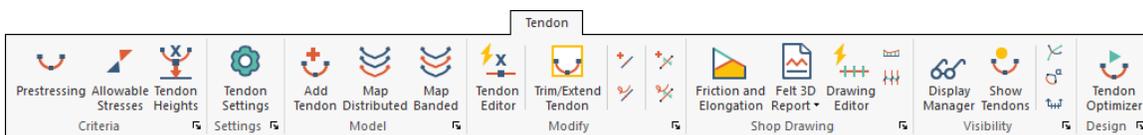
Column Tributary

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.

4.10 Tendon Ribbon



4.10.1 Criteria Panel



Prestressing 

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

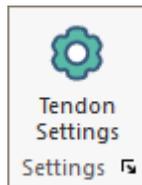
Allowable Stresses 

Opens the **Criteria – Allowalbe 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.

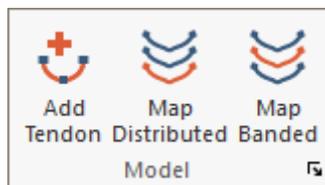
4.10.2 Settings Panel



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.

4.10.3 Model Panel



Add Tendon 

Adds 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**.

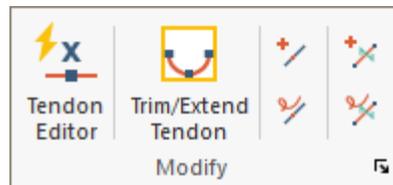
Map Distributed

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.

4.10.4 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.

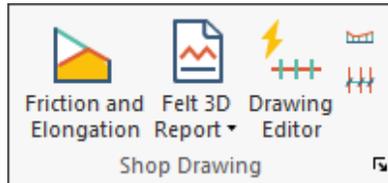
Insert Swerve Point

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 Swert Point 

Removes a swerve point on the selected tendon on the current plane. Use the swere 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.

4.10.5 Shop Drawing Panel (add-on Module)

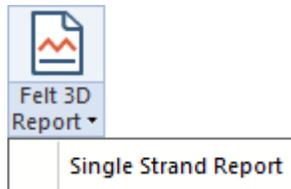


Friction and Elongation 

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.

Felt 3D Report 

Produces combined prestress short- and long-term loss and elongation report for tendons set to the “Calculate Force” method.



Single Strand

Option to report losses as a function of a single strand when the number of strands designated exceeds 1.

Drawing Editor 

Contains tools for modeling and drawing customization of tendons for shop drawing production including general tools, chair grouping, tendon color representation and construction joints.

Tools –

Options provided to edit modeled tendons. Including:

- Converting tendons from straight or spline representation to fillet-radius representation. The default radius for in-plane swerve locations is 20ft (6m)

as set in the the individual Tendon Properties menu and from Modify>Modify Selection>Tendons.

- Removal of all non-auto swerve points on spline or fillet-radius tendons. Auto swerve points are added to anchorage locations of tendons with a default offset.
- Update of tendon and construction joint intersections when a tendon and/or construction joint is removed or relocated.

Chair Groups

Options provided for support bar spacing and support bar extension each side of the control location. After entering the value, tendon locations that can be edited will be enabled with a yellow circle. Select the location and Apply the values set. The program will report the support bar height and number of spaces with spacing dimension.

Colors

Provides options to group tendons by customized colors for shop drawing production.

Construction Joints

Options provided for drawing construction joints as discontinuous and continuous (with varying anchor options) and pour sides and labels relative to the modeled joints. The joints are used to virtualize tendon loss and elongation calculations relative to the joint type and location.

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.

4.10.6 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.

4.10.7 Design Panel



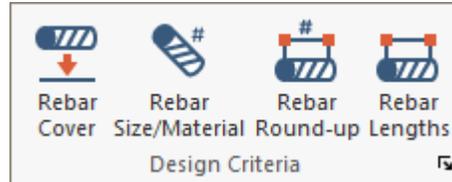
Tendon Optimizer

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.

4.11 Rebar Ribbon



4.11.1 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.

4.11.2 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.

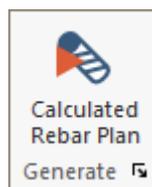
Banded

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.

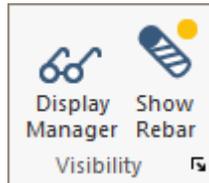
4.11.3 Generate Panel



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.

4.11.4 Visibility Panel



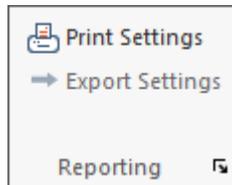
Display Manager

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.

4.11.5 Reporting Panel



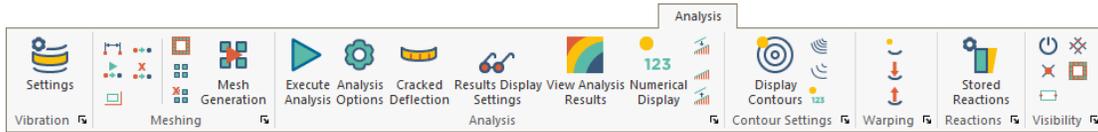
Print Settings

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

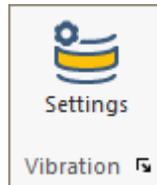
Export Settings

Opens the **DWG/DXF Export Settings For Reinforcement** dialog to configure the rebar appearance settings for export to AutoCAD files.

4.12 Analysis Ribbon



4.12.1 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.

4.12.2 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.

4.12.3 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  123

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.

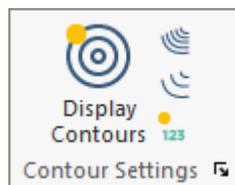
Default Results Scale 

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.

4.12.4 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 123

Displays on/off text values for contour lines of FEM slab and wall results.

4.12.5 Warping Panel



Display/Hide Contour Warping

Displays on/off a warped view of the selected FEM slab or wall graphical contour results.

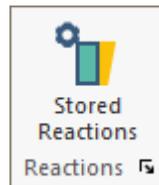
Increase Warping Scale

Increases the warping scale of the selected warped graphical result.

Decrease Warping Scale

Decreases the warping scale of the selected warped graphical result.

4.12.6 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**.

4.12.7 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**.

4.13 Floor Design Ribbon



4.13.1 Punching Shear Panel



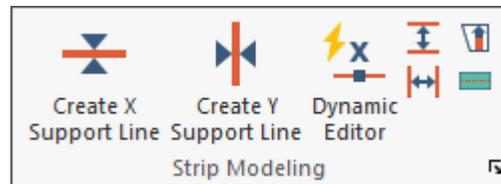
Shear Design

Opens the Shear Design Criteria tab where the user can setup the shead design properties and settings such as reinforcement type and size.

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. After a user executes a shear check the results are available in the **Results Viewer**. A strength combination must be selected in order to activate the Punching Shear results.

4.13.2 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

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.

Dynamic Editor

A new and improved Strip Modeling Dynamic Editor includes tools used to more rapidly model and modify support lines and design strips. These include:

Support Line Wizard – creates a support based on a construction line defined by snap points along the strip path.

Support Line Limits – Changes selected support line design criteria and tributary limits for the design strip.

Wall – allows the user to set constraints for how walls are considered for design strip generation.

Display - Sets the support line display for Direction, Criteria and Width Limit

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.

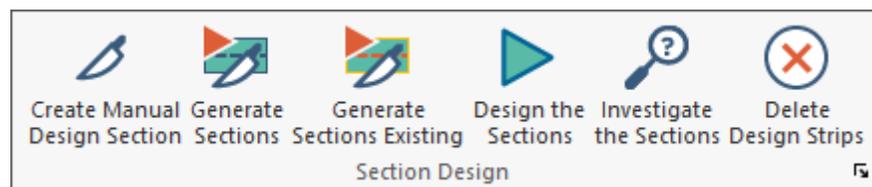
Extend Support Lines to Slab Boundaries 

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.

4.13.3 Section Design Panel



Create Manual Design Section 

Creates a manual design section by graphical or coordinate entry of the load start and end-point.

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.

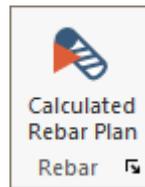
Investigate the Sections

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.

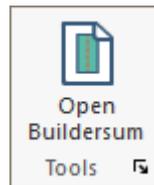
4.13.4 Rebar Panel



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.

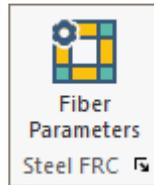
4.13.5 Tools Panel



Open BuilderSum

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

4.13.6 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.

4.13.7 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 123

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 Manual Design Sections

Toggles on/off the display of manual design sections.

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.

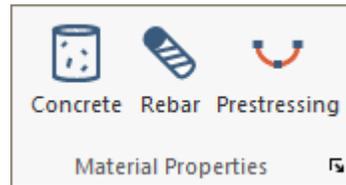
Display/Hide Support Line Tributaries Y-direction 

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.

4.14 PT/RC Export Ribbon



4.14.1 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.

4.14.2 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.

4.14.3 Load Factors Panel



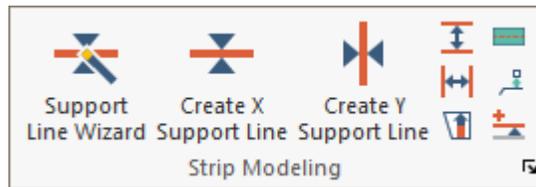
Load Combinations for Export to ADAPT-RC (RC Only Mode)

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.

4.14.4 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.

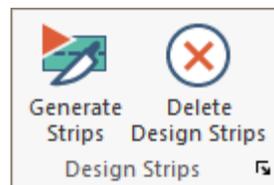
Connect Support Lines to Columns and Walls

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.

4.14.5 Design Strips Panel



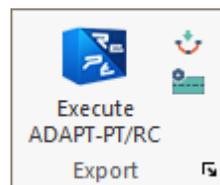
Generate Design Strips for Strip Method

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.

4.14.6 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.

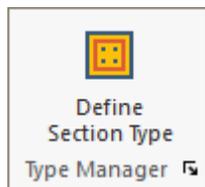
Generate Input Data for ADAPT-PT/RC 

Auto-generates and stores the .adb input files for all design strips to be executed within ADAPT-PT/RC.

4.15 Column Design Ribbon



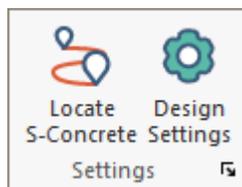
4.15.1 Type Manager Panel



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.

4.15.2 Settings Panel



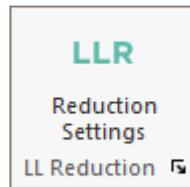
Set Path for S-Concrete 

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.

4.15.3 Live Load Reduction Ribbon

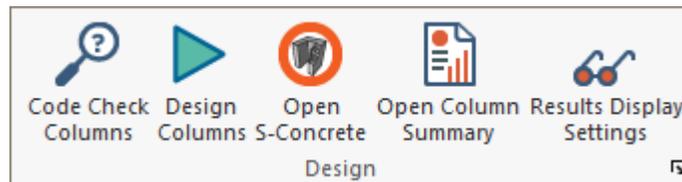


Live Load Reduction ^{LLR}

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 **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**.

4.15.4 Design Panel



Code Check

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**.

Design Columns

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.

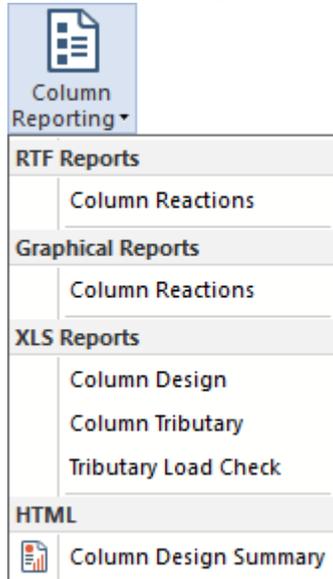
Result Display Settings 

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

4.15.5 Reports Panel



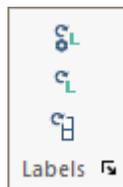
Column Reporting Expanded



- 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.

4.15.6 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.

4.15.7 Visibility Panel



Columns Only

Isolates the columns for on/off display.

4.16 Wall Design Ribbon



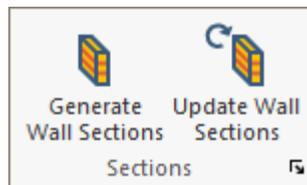
4.16.1 Settings Panel



Define Pier Labels

Creates pier labels for assignment to walls or wall groups.

4.16.2 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.

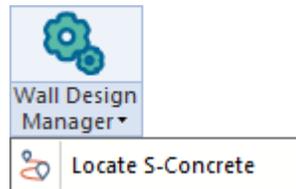
4.16.3 Design Panel



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.

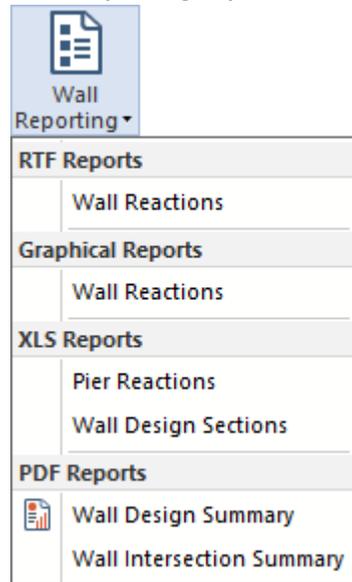
Result Display Settings

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

4.16.4 Reports

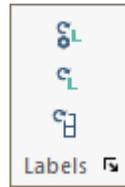


Wall Reporting Expanded



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

4.16.5 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.

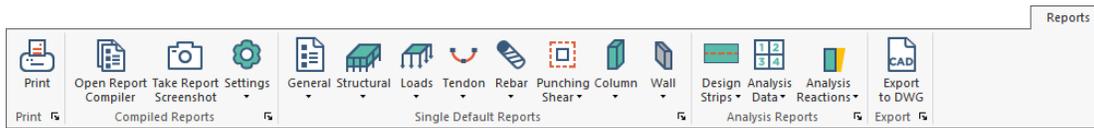
4.16.6 Visibility Panel



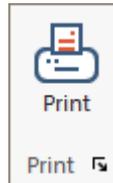
Walls Only

Isolates the walls for on/off display.

4.17 Reports Ribbon



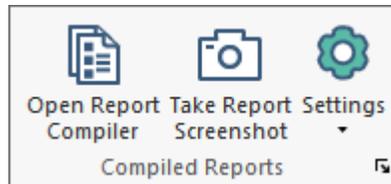
4.17.1 Print Panel



Print 

Opens printer selection and settings and prints active screen.

4.17.2 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**.

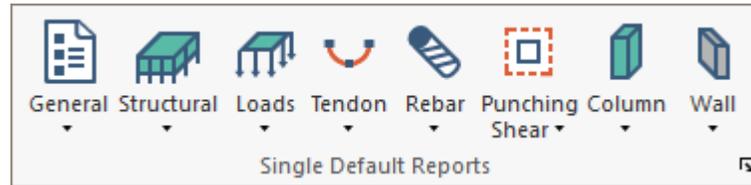
Take Report Screenshot 

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

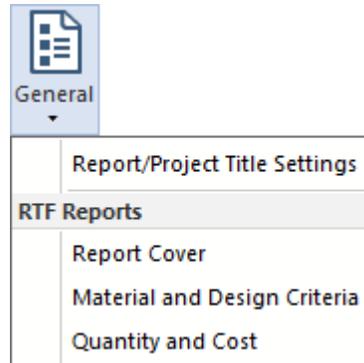
Settings 

Opens settings windows for various graphical reports that require user input before being created. For compiled reports the user can use the settings icon to set the inputs for these report pages prior to creating the compiled report.

4.17.3 Single Default Reports Panel



4.17.3.1 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

Generates 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).

4.17.3.2 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.

4.17.3.3 Loads



RTF Reports	
	Load Cases and Combinations
	Skip Pattern
	Applied Loads
Graphical Reports	
	Dead Load
	Live Load
	User Defined Load Cases
XLS Reports	
	Column Tributary
	Tributary Loads Check

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.

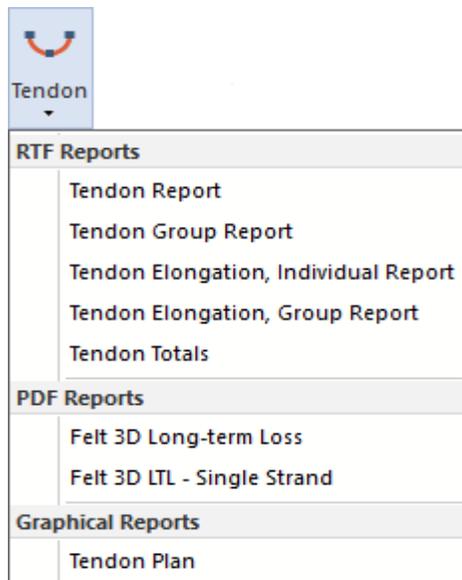
Column Tributary

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.

4.17.3.4 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.

Felt 3D Long-Term Loss

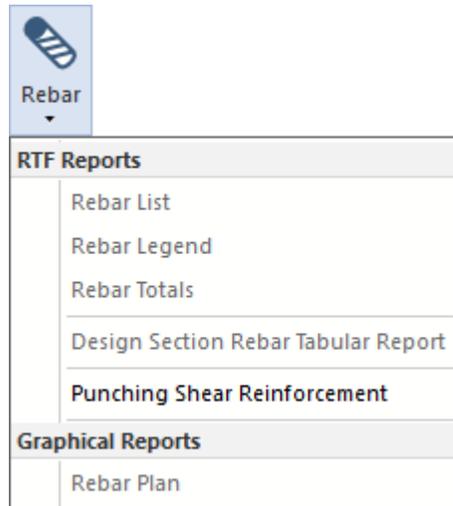
Produces combined prestress loss and elongation report in PDF format for the total number of strands assigned to the tendon.

Felt 3D LTL - Single Strand

Option to reports losses as a function of a single strand when the number of strands designated exceeds 1.

Tendon Plan

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

4.17.3.5 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.

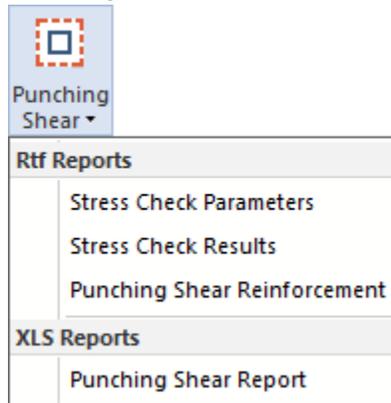
Punching Shear Reinforcement

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.

4.17.3.6 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

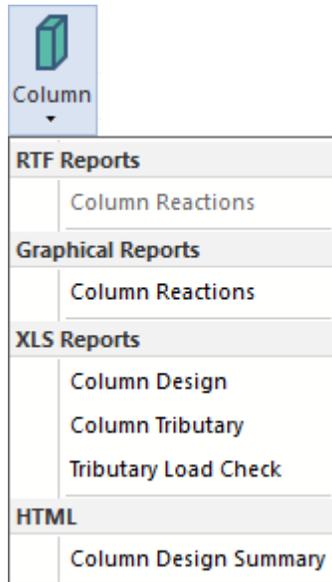
Generates a .RTF file with a summary of column punching shear results for the critical section producing the controlling stress ratio.

Punching Shear Reinforcement

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

Punching Shear Report (XLS)

Generates a .XLS file including shear calculation results for each critical layer checked along with schedules for stud rails or links at column locations requiring shear reinforcement.

4.17.3.7 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.

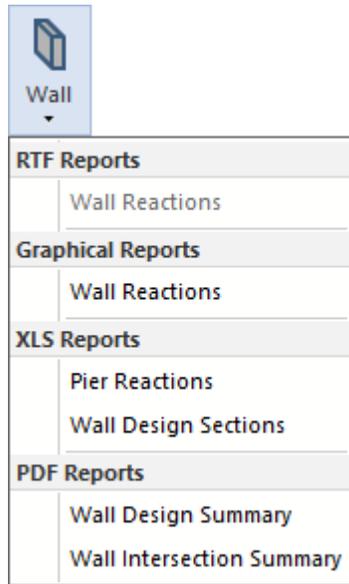
Tributary Load Check

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.

4.17.3.8 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.

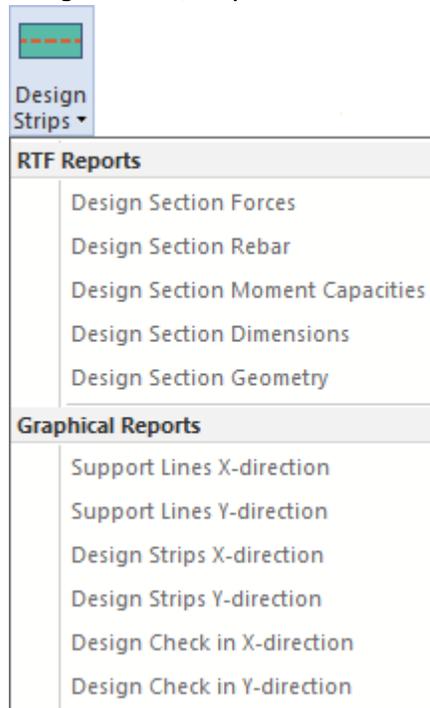
Wall Intersection Summary

Creates a combined PDF summary report for joint intersection details of designed wall sections.

4.17.4 Analysis Reports Panel



4.17.4.1 Design Section/Strip



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.

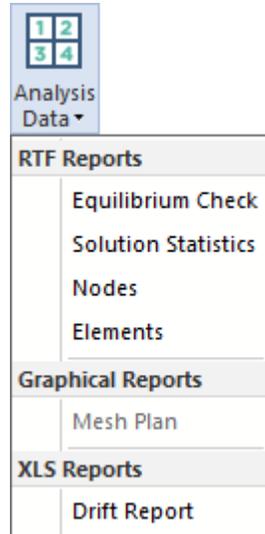
Design Check X-Direction Plan

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.

4.17.4.2 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.

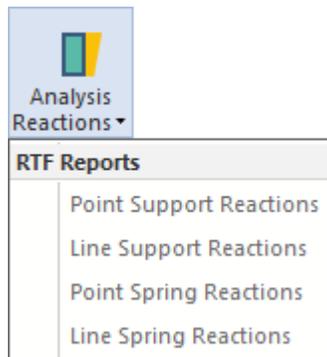
Analysis Mesh Plan

Displays a plan view of the current analysis finite element mesh.

Drift Report

Generates a .XLS file after selecting Report Options for combinations, Allowable Values and directions considered. The drift report contains column coordinate and elevation data, displacement and drift data for columns and story data for average drift at each level. Plot data is reported for graph production in Excel. .

4.17.4.3 Reactions 



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.

4.17.4.4 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.

5 Structural Modeling Tools

5.1 Overview

This chapter describes in detail how to generate a structural model. You will use the structural model for your analysis and design, either via ADAPT-PT or using the ADAPT-Floor Pro option. Your structural model will be the same, regardless of which method you use. The creation of a structural model is your first step in using the BUILDER platform for your design and drafting, regardless of whether the structure is conventionally reinforced or prestressed. The previous chapters described the environment of the program and the general-purpose tools of drafting, editing and viewing. The focus of this chapter is on components that form your structure, such as columns and slabs. These are called Structural Components.

In this manual we will go through the process of modeling a single-level model. Please refer to the program tutorials for the modeling and design of multi-level models. The structural model will be a faithful three-dimensional representation of the shape of the floor system you plan to design, along with one story of supporting structure (walls and columns) below it and, if any, one story of walls and columns above it. It will be a single-level floor system. However, it will include all the details, such as openings, steps on the slab, beams, drop caps and drop panels. The following figure shows a representation of a structural model displayed in wire frame format.

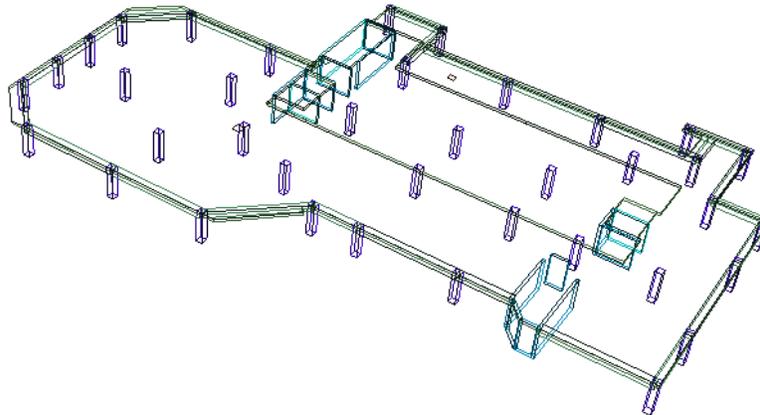


Figure 5.1-1 - Three-Dimensional View of a Structural Model

You can view the structural model in solid and other formats too. This is explained in the sections dealing with the viewers of the program.

In addition to the geometry, you also will define boundary conditions for the model. That is to say, the manner in which the structure is supported, or the

way parts of the structure are connected. For example, you may wish the connection between a slab region and its supporting column to be hinged. This is described in the section on *Release/Restraints*.

Presence of loads, such as self-weight, live load or other loads also is considered to be part of the structural model and will be discussed in this chapter. How the loads should be factored and combined to meet the requirements of various building codes, however, this will be covered later in the sections dealing with analysis and design options. Similarly, selection of design codes and design criteria are handled in the chapters covering analysis and design.

5.2 Structural Model Parts

5.2.1 Structural Components

The building structure you model consists of entities called Structural Components. You assemble the components to form a true representation of the floor system and its supporting structure.

The primary Structural Components of the current version of the program are:

- Slab regions
- Columns
- Openings
- Beams
- Drop cap/drop panel
- Ramps (see the ADAPT-Builder 2019 New Features Supplement manual)
- Springs (point spring, line spring or area springs)
- Supports (point support or line support)

If you use ADAPT-Floor Pro, with the *PT Module* active, an additional component is available:

- Prestressing tendons

If you use ADAPT-MAT for the mat foundation, you have:

- Soil area support

Further, the *Rebar Module* of the program includes:

- Bar reinforcement (rebar)

5.2.2 Common Properties of Structural Components

Each Structural Component has properties specific to itself, such as material and location. The properties of each Structural Component are listed in their component's property box. Some of these property items, such as specification of material, are common among all components. The following describes the most common properties.

There are four ways you can open the property box of an entity. These are:

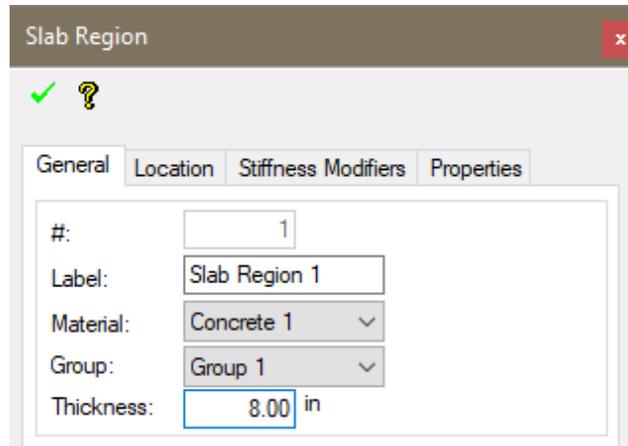
- Double-click on it.
- Select the entity, then click on the Item Properties  icon on the **Bottom Quick Access Toolbar**.
- Select the entity, go to Modify → Properties and click on the Item Properties  icon.
- Place the cursor over it, right-click and select *Properties* from the list that opens.

The property box of a slab region (**Fig. 5.2-1**) is used as a sample to describe the common properties.

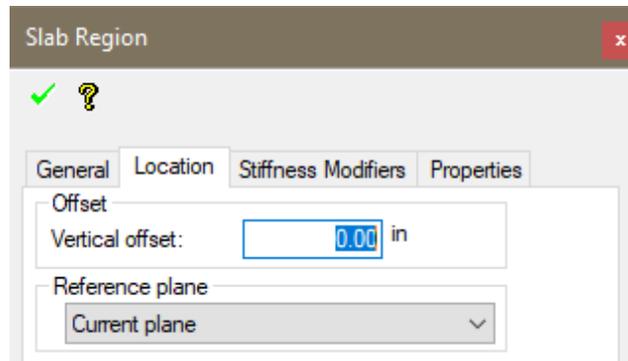
GENERAL TAB (Fig.5.2-1(a)):

#. This field displays the number of the entity. Modeler automatically assigns an ascending number (1, 2, 3...) to this field each time an entity of the same type, such as a column, is created. If you delete an entity, the program automatically re-numbers the entities of that group, before saving the data. The last number gives the total count of entities of any given group used in your model. You cannot change the information displayed in this field.

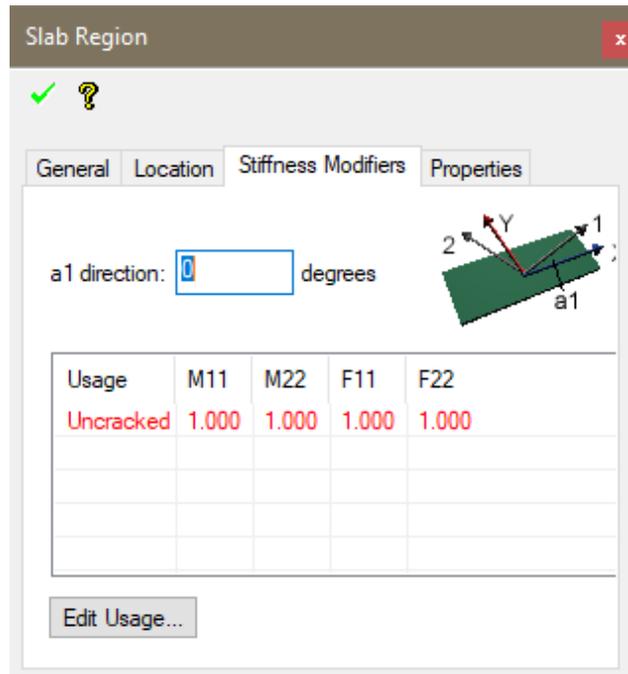
Label. In this field, the program assigns a name to each entity. For example, for the third column you generate, it will assign "Column_3." The important difference between this field and the one above is that in this field you can enter a name that you assign to the entity, such as "Rose." In its reports, the program will list the name you have assigned to the entity along with the number it selected in the previous field.



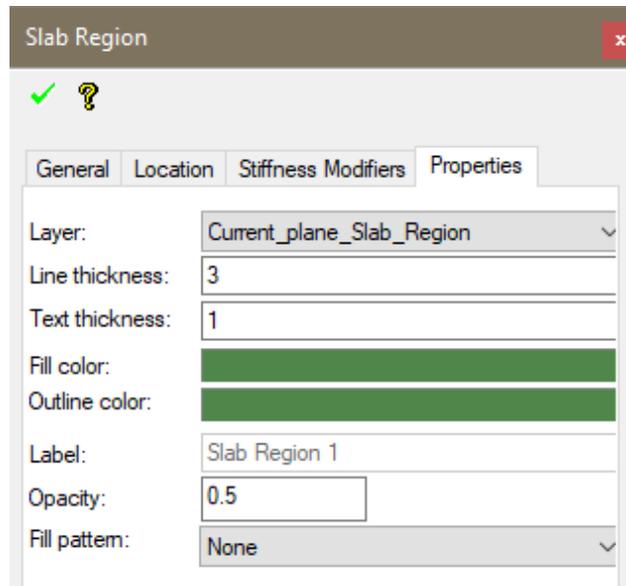
(a) General Tab



(b) Location Tab



(c) Stiffness Modifiers



(d) Properties Tab

Figure 5.2-1 - Sample Property Box

Material. This field identifies the material selected for the structural component. You can change the material of the structural model by selecting a different one listed in the combo box. If the material you intend to use is not listed, you must go to the *Material* pull-down menu and first define it. Once you do so, the new material will appear on the list of the combo box for you to select.

Group. Each entity, whether a Structural Component or an arc, is placed in a group. It was described in earlier chapters how to select specific groups for display or other operations. Initially, the program places every entity in its default group "Group 1." You can move the entity created into a different group by selecting the group of your choice from the list in the combo box. If you want to create a new group, you must go to the *Grouping* menu item in the *Settings* pull-down menu.

Other Properties. Depending on the type of the entity at hand, there will be other properties, such as dimensions, that will be listed for view and editing.

LOCATION TAB (Fig.5.2-1(b)): The value given as offset defines the location of the entity with respect to the reference plane to which it is associated. For example, an offset for slab will describe the distance of the top of slab to the current plane. An offset for the top of an upper column gives the distance of the column top end from the top reference line.

Coordinates. The coordinates of the vertices of each entity is listed in a table. You can select each vertex and edit its coordinates. This option enables you to place an entity at an exact location.

Stiffness Modifiers Tab (**Fig.5.2-1(c)**):

A1 direction. Angle between local element axes 1 and 2 and the global X and Y axes. Default is zero. This applies only to slab regions.

Modifiers table. Shows the default (uncracked) usage case and any user-defined usage cases with applicable modifiers (1.0 or less) for bending, shear and axial.

Edit Usage. Opens the Combination Usages dialog to define a new usage case and stiffness modifiers.

Properties TAB (Fig.5.2-1(d)):

Layer. This field shows where the graphical display information of the entity is saved. You can change the layer to another one from the list in combo box.

Line Thickness. This field is used to edit the width of the lines used to display the entity. It applied to display on the screen and in printed hardcopies.

Color. This field is used to edit the color of the object on the screen.

Filling. This field is used to choose and display a fill pattern for the display of an entity, if it covers an area such as a slab.

5.3 Level Assignments

When modeling a single level, you model one floor level along with the supports immediately below and above it. You will be using three horizontal planes to describe the structure. The first is the level you have selected to design. It is referred to as the current plane. All the features of the floor system, such as the position of the slab regions, beams and steps on the slab will be expressed with respect to this plane (**Fig 5.3-1**). The walls and columns below the slab are assumed to extend from the current plane to the plane below it (bottom plane). Likewise, the walls and columns above the slab extend from the current plane to the next plane above (top plane). These are the default length assignments of the program. The user can modify the length of a support above or below to a value different from the distance between the two respective planes. This will be explained along with the modeling of walls and columns.

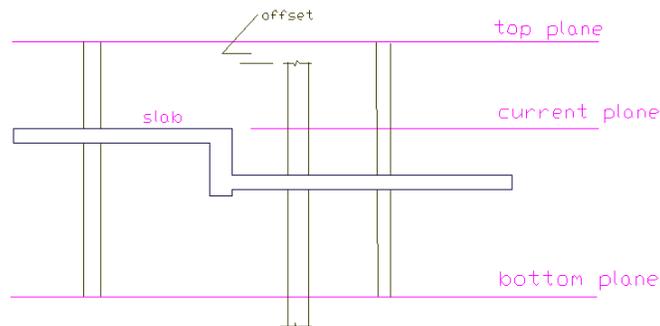


Figure 5.3-1 - Reference Planes (Level Assignment) in Elevation

Note that in **Fig. 5.3-1** one of the columns does not extend to the top plane. The top of that column is said to have an offset with respect to its natural position (top plane). Similarly, a section of the slab is stepped down below the current plane. Again, this slab region is viewed to have an offset with respect to its natural (default) position.

Unless you specify otherwise, the program initially inserts each structural component with respect of the reference planes as shown in **Fig. 5.3-2**.

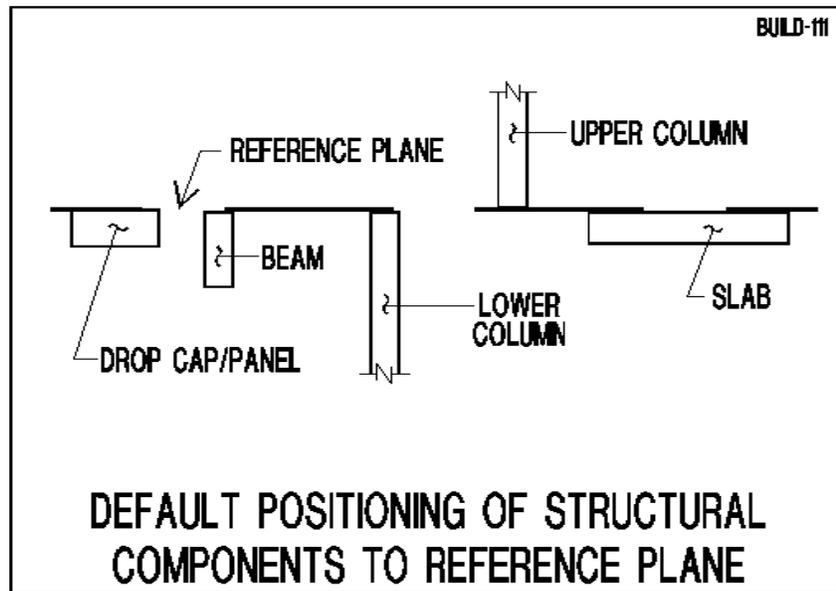


Figure 5.3-2

You can adjust the program's initial positioning of the structural components, by using the *Offset* feature of each component (Fig. 5.3-3). This is described later.

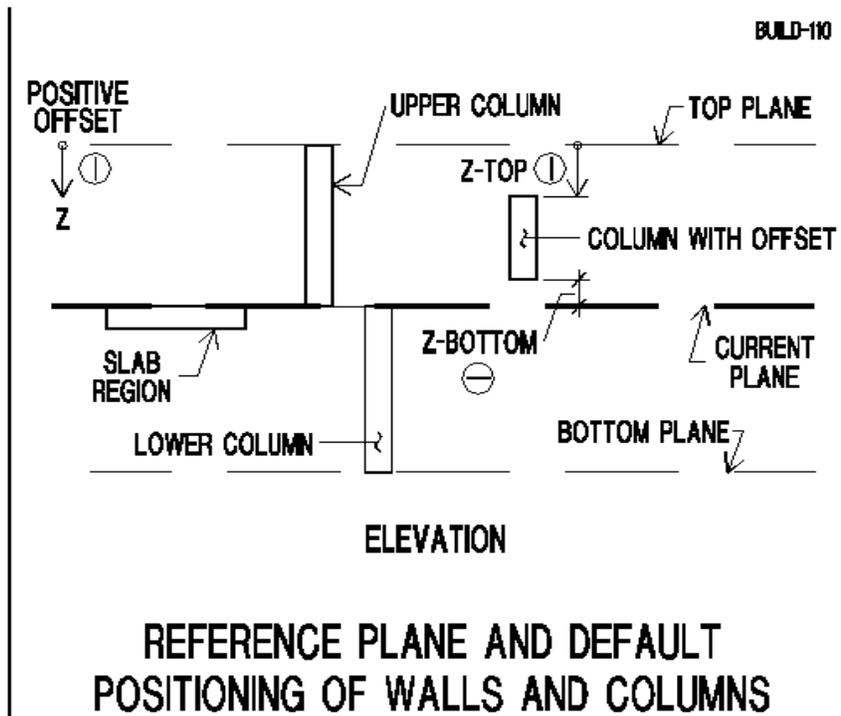


Figure 5.3-3

Figure 5.3-4 is an illustration of a complex floor system composed of the basic Structural Components and the offset features.

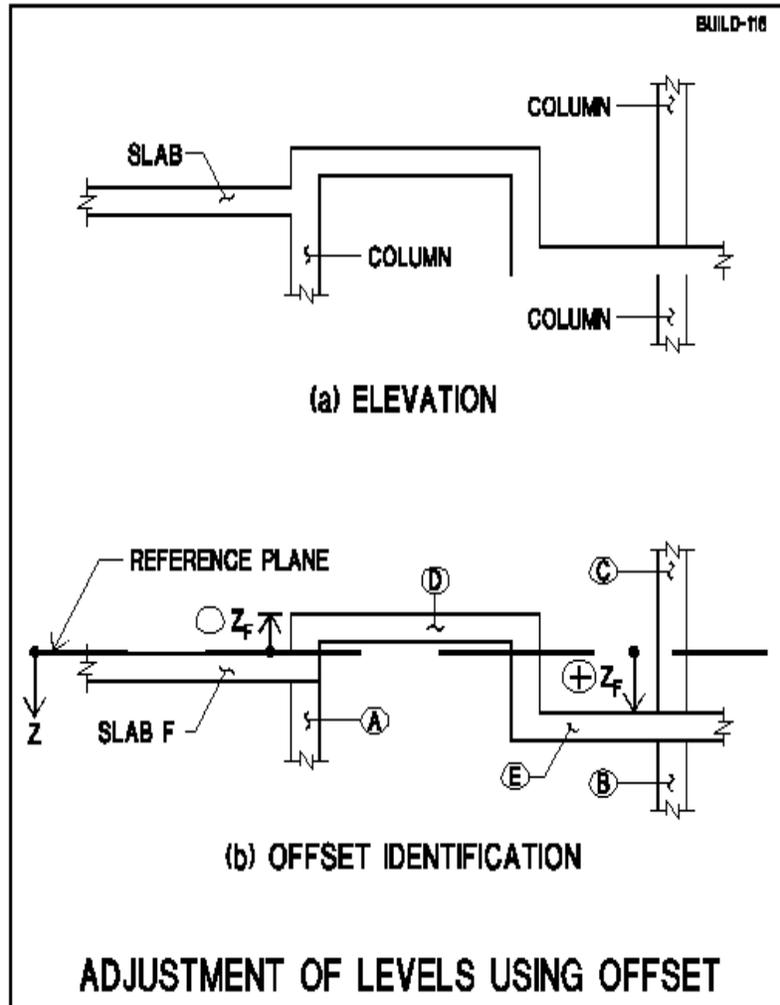


Figure 5.3-4

The program comes with default values for the distance between the current plane and the top and bottom planes. You may need to change this, however, to match the details of your project. This is explained next.

5.4 Level Assignment Tool

The **Level Assignment** tool specifies the distance between the top of the typical region of the floor slab you are designing to the top of the slab above (top plane) and the top of the slab below (bottom plane). When you create a column or wall, it will initially assume the distances you specify here, until you change it.



Level Assignment. This button is in the Upper-Right Level Toolbar. It opens the *Reference Plane* dialog box (Fig. 5.4-1) in which the distance

between the default three reference planes is described by their height above a datum level.

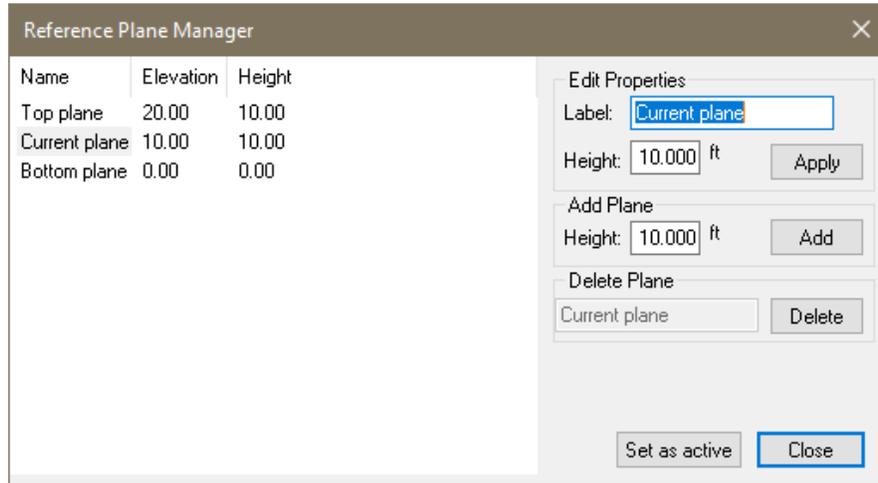


Figure 5.4-1 - Reference Plane Dialog Window

5.5 Organization of the Structural Components Data

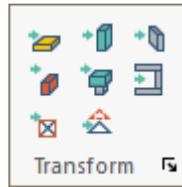
The program stores the graphical information of each of the Structural Components in a dedicated default layer. You can view these layers by clicking on the layer tool within the **Bottom Quick Access** toolbar of the screen and selecting the layer of your choice for viewing.

5.6 Modeling Options

There are two principal options for modeling real-life projects. One is to start with an available architect's drawing (DWG or DXF file). The other is to draft the model entirely within the environment of the BUILDER platform, using the Modeler's drafting capability. In the first case, you may need to use some of the Modeler's drafting capability along with the information obtained from a DWG/DXF file.

When you use a DWG/DXF file to generate a model, most of your work will be in the transformation or conversion of the drawing items into Structural Components. You are likely to start with the *Transform* panel of the **Model** ribbon. On the other hand, when you plan to draft your model from scratch, you will use the *Add Structural Components* panel of the Model ribbon.

5.6.1 Transform Panel



You will use these tools if you start with a DWG or DXF file, and plan to convert the items shown on the drawing to Structural Components for your model. In other words, if you are not importing a drawing for conversion to your model, you are not likely to use the tools on this panel. You will use the *Add Structural Components* panel of the *Model* ribbon.

Once you import a DWG or DXF drawing, your first choice is to transform the items on the imported drawing directly to structural model. The items shown on the imported drawing are simply lines (graphics). The process of conversion is to (1) pick an item on the drawing, such as a column, and (2) click on the associated structural component tool (*Transform to Column*), in order to convert it to a structural component.



Transform Polygon Only items that are in form of a closed polygon can be picked and converted directly into structural components. Not every column or opening in the imported structural drawing is drawn as a closed polygon.

This button is used to transform groups of lines that are drawn separately, but intersect into a polygon. Click on the tool. The program searches the drawing and creates polygons for the instances where conversion of intersecting lines into polygons is practical. The program retains the original items on the drawing, but writes the polygons created in a new layer called "Polygons_from_line."



Transform Slab Region. This button is used to transform a polygon (closed polyline) to a slab region. The tool operates in the same manner as the *Transform Column*  tool, which is described in greater detail below. The transformed slab region is always located at the *Active Reference Plane*.



Transform Column. This tool is used to transform a rectangle (polygon) or circle to a column.

To transform a (polygon) rectangle into a column do the following:

- Select one or more rectangles that are made up of polygons.
- Click on the *Transform Column*  tool. All selected entities will be transformed into columns with the same dimensions as the rectangle or circle.

The default of the program places the transformed columns below the slab. If they are intended to be placed above the slab or both below and above, you must make the correction either through the property box of the column, or by way of using the copy/move commands. The use of the property box is described in the section on “Add Structural Components Panel.”



Transform Wall. This button is used to transform a rectangle (closed polyline) into a wall. The tool operates in the same manner as the *Transform Column*  tool.



Transform into Several Walls. When a polygon represents two or more intersecting walls, this button is used to transform it into several individual walls, each having a rectangular cross-section. In the analysis, however, the program treats the walls integrated into one along their common vertical joints. The tool operates in the same manner as the *Transform Column*  tool.



Transform Drop Cap/Panel. This button is used to transform a rectangle (closed polyline) into a drop cap. This tool operates in the same manner as the *Transform Column*  tool.

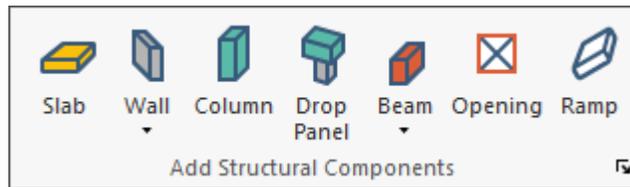


Transform Beam. This button is used to transform a polygon (closed polyline) into a beam. The tool operates in the same manner as the *Transform Column*  tool.



Transform Opening. This button is used to transform a polygon (closed polyline) into an opening. The tool operates in the same manner as the *Transform Column*  tool.

5.6.2 Add Structural Components Panel



The primary application of this panel is to create a structural model from scratch – as opposed to using an imported DWG/ DXF file. In addition, it works as a supplement to the *Transformation* tool described in the preceding, since not all Structural Components can be transformed directly from an imported file. You may have to create some of them manually, using the tools of this panel.

For a major real-life structural model, you are likely to start by setting out some gridlines, and/or use a grid on the background to help you in your drafting. For this purpose, you use the *Gridline* panel of the *Model* ribbon, described earlier in this manual.



Create Slab Region. This tool creates a new slab region with the corresponding properties input in the dialog box shown in **Fig.5.2-1**. To create a slab region, use the following procedure:

- Click on the *Create Slab Region*  icon. The dialog box shown in **Fig. 5.2-1** will open¹.
- Edit the input fields as required (each input field is described below). These values will be used in the future, until they are changed again².
- Draw the outline of the slab region. Click the mouse at each point required³.
- After the last point has been selected, Press *C* to close the area.

To edit the properties of an existing slab region, open its property box and edit as described earlier. If you want to

¹ The property dialog box window will automatically open if you selected option “Open Items Property Dialog Box Automatically” in the *Home* → *Settings* → *General Settings* menu, item General Settings (**Fig. 5.2-1**)

² To have the parameters, such as thickness, material and offset, of the slab region you have created to be used for the subsequent slabs you create, you should use the option of “*Use Last Properties as Default.*” This option can be turned on from *Home* → *Settings* → *General Settings*.

³ You are likely to make mistakes as you trace a slab boundary. During modeling of a slab region the user can use CTRL-Z to undo entered points of a slab region.

change the shape of a slab region on plan graphically, do the following:

- To change the location of an entire slab region, make sure the Move Selection  tool is highlighted, then just drag the slab region by a vertex to its new location. The Move Selection tool is located at Home → Selection Mode.
- If you would like to change the location of just one vertex, make sure the Move Selected Point  tool is highlighted, then just drag the vertex to its new location.
- If you would like to remove a vertex, use *Remove Point*  tool of the *Modify* ribbon *Points* panel. Alternatively, you can right-click on the slab region and choose *Remove Vertexes* from the slab's right-click menu.
- If you would like to add a vertex, use *Insert Point*  tool of the *Modify* ribbon *Points* panel. Alternatively, you can right-click on the slab region and choose *Insert Vertexes* from the slab's right-click menu.



Create Column. This tool creates a new Column with the properties input in the corresponding dialog box (**Fig.5.6-1**). To create a Column, use the following procedure:

- Click on the *Create Column*  tool. The dialog box shown below will open.
- Edit the input fields on each tab as required. These values will be used in the future, until they are changed again.
- Click the mouse at the column location (you may do this in conjunction with snapping tools).

Column x

✓ ?

FEM	Release	Stiffness Modifiers	Properties
General	Location	Boundary Condition (Strip Method)	

Section Type: None Edit...

#: Material: Concrete 1

Label: Group: Group 1

Cross-section

Shape: Rectangular A: in

Ang: ° B: in

COLUMN SECTION

Unbraced Length

Program calculated User defined Update

Individual Lu (s-s): ft (r-r): ft

Group Lu (s-s): ft (r-r): ft

Figure 5.6-1 - Column Property Box

The properties specific to a column are as follows:

A: This field is used to input dimension “A” of a rectangular or square column, and it is also used to input the diameter of circular columns. For a rectangular column, dimension “A” is the length of the side along X-axis, if no angle is specified (Ang field, See below).

B: This field is used to input dimension “B” of a rectangular column. Dimension B is the side along the Y-axis if no angle is specified.

Ang: This field is used to input the angular orientation of the column on the XY-plane. The orientation is expressed with the counterclockwise angle from the positive direction of X-axis to side “A” of the column.

There are other properties listed in the Column property box that relate to the analysis and design features of the program. These are discussed either in the Strip Modeling or Finite Element modeling sections of the manual.



Create Wall. This tool creates a new wall with the properties input in the corresponding dialog. To create a new wall, do the following:

- Click on the *Create Wall*  tool. The wall's property box opens (if this is the option you have selected in the *General Settings* tool).
- Edit the input fields on each tab as required. These values will be used in the future, until they are changed again.
- Click the mouse at one endpoint of the wall and then click at the other end.

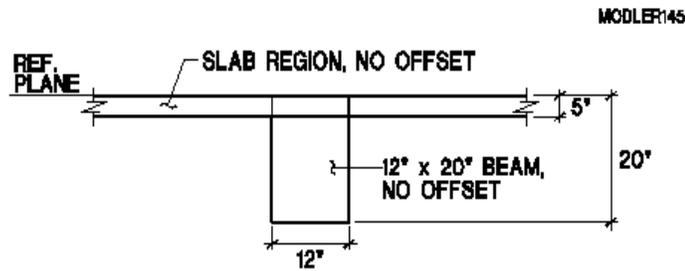


Create Beam. This tool creates a new beam with dimensions and properties that are reflected in its property box. To create a new beam, do the following:

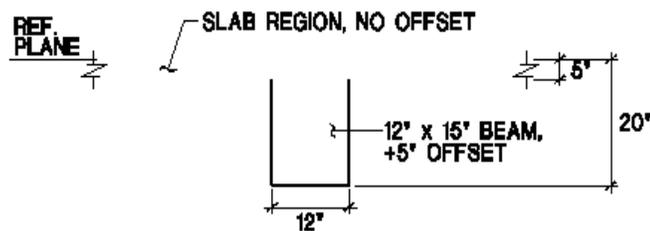
- Click on the *Create Beam*  tool.
- Draw the beam. Click the mouse at one endpoint of the beam and then click at the other end. You may have to use the snap tools, if you want to connect the beam to other structural components.

Offset. This field is used to input a vertical offset to the beam from the reference plane. A value of "0" places the top of the beam at the level of the reference plane. A positive offset moves the beam down and a negative offset moves the beam up. **Fig. 5.6-2** illustrates two options for inputting the T-beam cross-section. Both options shown are equivalent and will produce the same input data for ADAPT-PT. Option 2 uses a 5-inch offset. Option 1 will result in a larger local stiffness, due to the overlap of slab and beam stems. Both options, however, will give the same stress and same reinforcement for the same design values (moments and shears).

Beams may be drawn alone or in conjunction with a topping slab to create a T-section. Beams and slabs are drawn separately in modeling the structure.



(a) INPUT OF T-SECTION, OPTION #1



(b) INPUT OF T-SECTION, OPTION #2

INPUT OF EQUIVALENT T-SECTIONS IN MODELER

Figure 5.6-2



Create Opening. This tool creates a new opening. Click on the tool and follow the instructions on the screen.



Create Drop Cap/Panel. This tool creates a new drop cap or drop panel. Click on the *Drop Cap/Panel* tool and insert it on a slab region. If you want it centered on a column, use the *End Snap* tool. Change the default orientation and dimensions of the drop created by opening its property box.

Drop caps/panels generated using this tool will be rectangular on plan, but do not necessarily need to be oriented along the X and Y axes. You can rotate the drops using their property boxes.

If you have a drop with an irregular shape, you can use the slab region tool. With the slab region tool, draw the cap on plan and specify its offset and depth to match that of the drop. A newly drawn slab region will override the properties (including

thickness) of the slab region over which it has been drawn. The new slab region around a column will act as a drop cap/panel.



Create Ramp. This tool creates a new ramp. Click on the *Ramp* tool and insert it on a slab region or within an opening. Change the thickness of the ramp created by opening its property box.

5.6.3 Model → Visibility Panel



This toolbar contains tools that toggle the visibility of the entity they display. The following is the list.



Display Slab Region



Display Wall



Display Column



Display Drop Cap



Display Beam



Display Opening



Display Ramp



Display Gridline



Display Point Support



Display Line Support



Display Point Spring



Display Line Spring



Display Soil Support (MAT)/Area Spring (Floor Pro)



Display Point Displacement (SOG Only)



Display Line Displacement (SOG Only)Error! Bookmark not defined.

Support lines, splitters and point supports will be discussed later in the section on Strip Modeling.

5.6.4 Loads

You can apply an essentially unlimited number of loads on the structure. The loads can be applied at arbitrary locations, regardless of the method and details of the analysis that would follow. The types of load that you can specify are:

- Point Load (concentrated or moment)
- Line Load (linearly varying force or moment)
- Patch (area load), uniform intensity or varying in magnitude

Each load belongs to a load case, such as dead load, superimposed load or other load cases that you choose to have. There is essentially no limit on the number of load cases. The program comes with a number of pre-defined load cases. But you can add your own to them. The program's pre-defined load cases are:

- Selfweight
- Dead Load
- Live Load
- Prestressing (If you use ADAPT Floor Pro with Prestressing)
- Hyperstatic (If you use ADAPT Floor Pro with Prestressing)

Each load has its own property box and can be edited individually. Also, you can view each load case on its own. Later, we will discuss how load cases can be factored and combined for code check.

Loads can be applied at any point, line, or area over the slab region. Apart from having a shape, such as point load, line load, or patch load, each load has a magnitude and belongs to a load case. Loads that when projected along the vertical axis do not fall on a slab beam or wall will be disregarded by the program. For example, if you place a uniform load over an area that contains an opening, or slab cutouts, the program disregards the portion of the load that falls on the opening, or falls outside the boundary of a slab region.

Loads can overlap. In other words, you can specify two or more loads that in whole, or part, cover the same region. Where there is overlap,

the program adds the contribution of each to obtain a total load. Negative loads can be used to subtract load from the region as well.

5.6.4.1 Load Case Definition

If you want to have one or more load cases in addition to those already defined by the program and listed above, do the following:

From the Loading→Load Case/Combo panel, select the Load Case  icon. The dialog window shown in **Fig. 5.6-3** will display. The Load Case Library is made up of three parts. The General Load (Gravity/Lateral) section deals with gravity and all other loads applied on the current slab. The Building Load section deals with lateral loads, such as wind and earthquake applied through the Wind Load and Seismic Load wizards in a multi-story analysis. The Lateral Load Solution Sets part will be displayed only if the user checks the box at the top of the Load Case window titled Lateral Load Solution Sets includes the lateral module⁴.

The load cases self-weight, prestressing and hyperstatic are not listed in the library displayed, since these are generated automatically by the program. You cannot edit them. For example, the self-weight is automatically calculated from the volume of the structural components and their unit weight. Likewise, prestressing, if available, is automatically calculated from the parameters of tendon profiles and the stressing of tendons.

To create a new load case, click on the *Add* button. A new load case will appear on the list. Click on it if you want to change its name (label).

⁴ The *Lateral Load Solution Sets* option of the program is described in the manual “Integration with Multistory Programs.” Loads assigned to this option will be treated differently.

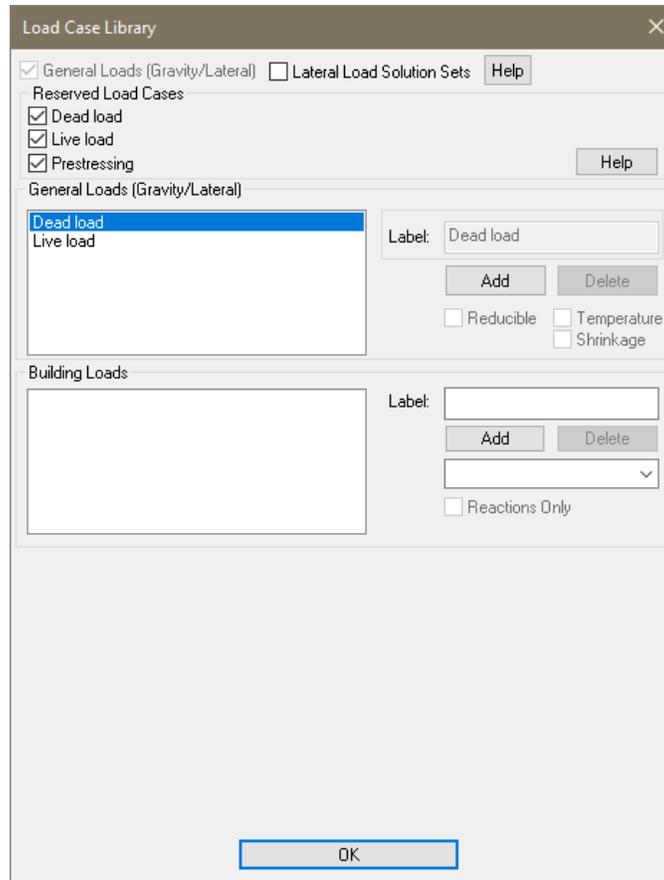


Figure 5.6-3 - Load Case Library

5.6.4.2 General Loading Panel



↓ **Add Point Load.** This tool inserts a new point load or moment with the properties shown in Fig. 5.6-4. To create a point load, do the following.

Click on the Add Point Load ↓ tool.

- Insert the point load.
- Open the property box of the point load (if it did not open prior to its insertion) and edit the input fields on each tab as required.

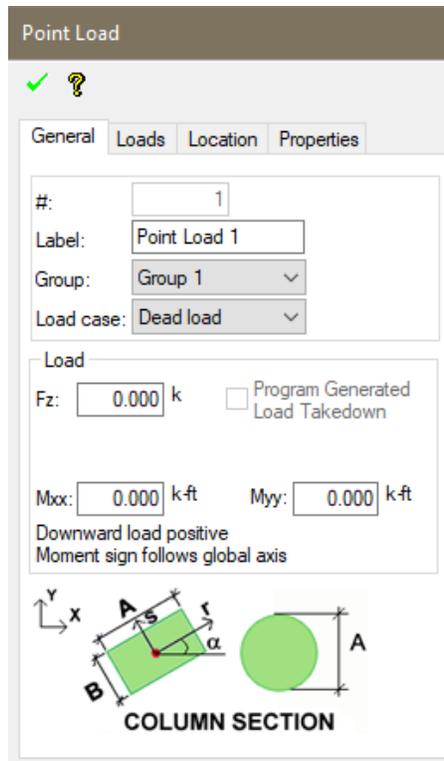


FIGURE 5.6-4 POINT LOAD DIALOG BOX

Each point load can be specified with the following load components:

- Forces: F_x , F_y , F_z
- Moments: M_{xx} , M_{yy}

The default point of application of a point load is on the reference plane. By changing its offset in the property box, you can move the point load in the vertical direction too. This becomes important to note when you deal with forces F_x and F_y . A force in the horizontal direction applied at the reference plane will have a moment about the slab that does not have its centroid on the reference line. The program automatically recognizes these conditions and accounts for them in its computations.



Add Line Load. This tool creates a new line load with the properties shown in Fig. 5.6-5. Line loads can consist of a constant or variable force along the length of the line with or without constant or variable moments M_{xx} and M_{yy} along the same line. The moments are specified about the global axis.

To create a line load, do the following:

- Click on the *Add Line Load*  tool.
- To draw the line load just click at the two endpoints of the load. The load may be drawn outside the slab or over openings. The program automatically recognizes this condition and ignores those portions that do not fall on a slab region.
- Edit the input fields on each tab as required.

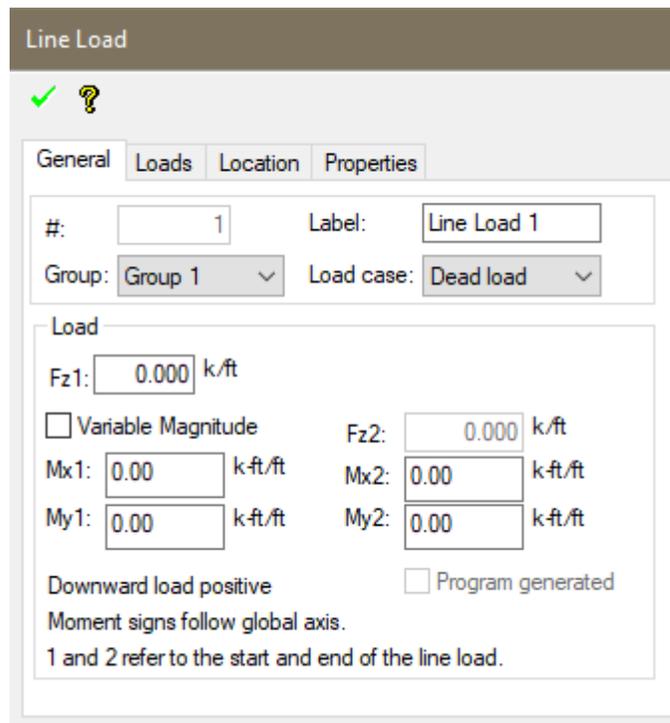


Figure 5.6-5 - Line Load Dialog Box



Add Patch Load. This tool creates a new patch load. A patch load covers an area within a boundary drawn or selected by you. If the area covered by the boundary you draw/identify falls in part outside a slab region, the program considers only the loads that fall on slab regions, beams, and walls.

A patch load is defined by the area within its boundary and a plane that specifies the distribution of load over it. The distribution of the load intensity over the patch is defined by the ordinates of three of its vertices, identified as vertices 1, 2 and 3. You can enter the value at these points, and view the distribution of the load

on the screen. If the vertices selected are not the right ones, use the arrow tool to move your selection along the boundary of the patch, until you reach the right location (↶↷).

You can specify the following load distributions over a patch load:

- Forces: F_x , F_y , F_z
- Moments: M_x , M_y

To create a patch load, do the following.

- Click on the *Add Patch Load*  tool.
- Edit the input fields on each tab as required (Fig.5.6-6).
- Click at the vertex points on the perimeter of the load area. The load may be drawn outside the slab, or over slab openings. The program automatically recognizes this condition and ignores those portions that do not fall directly on a slab region.
- After clicking on the last vertex, input C to close the input.
- To edit a patch load, open its property box (shown below).



Figure 5.6-6 - Patch Load Dialog Box



Patch Load Wizard. This tool creates an area load over an enclosed region that you select. To automatically create a set of patch loads, do the following.

- Select a polygon or slab region that you would like to apply a patch load over.
- Click on the *Patch Load Wizard*  tool.
- Enter the value and load case. The program will automatically generate a patch load over the area selected.
- To edit the load you selected, open the property box of the load.



Line Load Wizard. This tool automatically creates a set of line loads on a line, beam or wall that you select. It also works for a load over a polyline.

To generate a set of line loads on selected line do the following.

- Select the line or polyline over which you want to generate a set of line loads.
- Click on the *Line Load Wizard*  tool.
- Edit the input fields of the dialog box that opens as required.
- Click on the *Create* button to create the loads. A line load for each load case specified will be generated automatically over the lengths of the line(s) chosen.



Import Loads. This tool allows the user to import loads to the model through the use of an Excel file.

6 Strip Modeling Tools

6.1 Overview

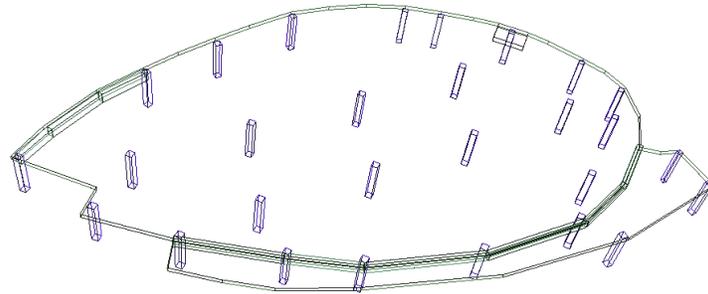
This Chapter describes the tools you will be using for the analysis and design of the structural model you have created, regardless of the method of analysis. The following tools may be found in the *Floor Design* and *PT/RC Export* ribbons. The tools that apply to the Equivalent Frame Method of design (EFM) that uses ADAPT-RC or ADAPT-PT can be found in the *PT/RC Export* ribbon, while the tools that apply to the Finite Element Method (FEM) of analysis and design that uses ADAPT Floor Pro, can be found in the *Floor Design* ribbon. The application of tools within that are similar in both ribbons will be the same in each ribbon.

Briefly, you identify a region of a floor system as a design strip, determine the actions (moments, shears, etc) that are associated with the design strip selected, find out whether the actions determined for the design strip can be resisted within the requirements of the building code you select (code check), and then calculate the reinforcement that must be placed within the design strip.

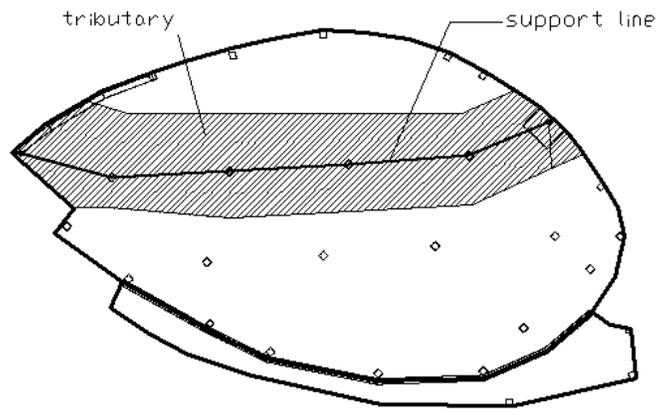
Figure 6.1-1(a) shows a floor system and a single design strip. For a complete design, you must cover the entire floor plan with several design strips. The elements of a design strip generally include:

- Support Line
- Supports
- Tributary
- Splitter

The tributary determines the region of consideration. The support line identifies the direction along which the reinforcement is calculated and is likely to be placed. Selection of supports, such as walls/columns, and beams, either by you or by the program automatically, will result in the appropriate selection of building code provisions in the code check part of the design process.



(a) 3D Wire Outline View of the Structure



(b) Identification of a Support Line, Tributary and Design Strip

Figure 6.1-1 - Support Line, Tributary and Design Strip

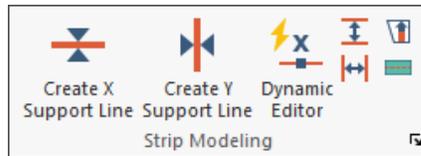
Design strips are generated automatically by the program. The automatic generation follows a number of rules detailed below. Some of the considerations are:

- **Width of a Design Strip.** The program creates a uniform tributary over the entire length of each span with a width equal to the largest width of the non-idealized tributary. This is generally conservative. It can be improved if you select a Splitter and break each span into more segments. Splitter is a tool that is described later in this Chapter.
- **Separation of Adjacent Design Strips.** The separation between two adjacent design strips is done by the program automatically, along the line that breaks the distance between two adjacent support lines into two equal parts. If this automatic separation is not where you want it, you can override it by drawing a Splitter along the tributary line of your choice.
- **Modeling of Openings.** Openings are not considered in the generation of tributaries or design cut. Section cuts ending within an opening will

only include the portion of the design section that cuts a concrete component, the area within the opening will be disregarded for design. Sections generated by the program that cut through and opening and into concrete on the other side of the opening will be idealized into one monolithic piece of concrete. Depending on the size of the opening, using *Splitters*, you can exercise full control of how the region around it should be accounted for in load transfer.

As mentioned earlier, strip modeling tools generally are applicable to both ADAPT-PT and Floor Pro. Where a tool is specific to ADAPT-PT it is so identified. By disregarding these, you will be concentrating on the tools that are applicable to ADAPT Floor Pro. As a guide, the tools that are specific to ADAPT-RC/PT are shown with a brown background. Skip these, if you do not intend to export your data to ADAPT-RC or ADAPT-PT. These specific tools also will not appear on your computer screen if you do not select the *PT/RC Strip Mode* option in the splash window (first screen) that appears when you open the program.

6.2 Strip Modeling Tools



Floor Design Ribbon



PT/RC Export Ribbon

The *Create X Support Line*, *Create Y Support Line*, *Dynamic Editor* (Floor Pro Only) and *Support Line Wizard* (PT/RC Strip Method Only) of these toolbars deal specifically with the creation of design strips. The remainder helps you to improve or correct your work.

  **Create X/Y Support Line.** You use these tools to create a new support line manually. Clicking on the *Create X Support Line*  will create a support line that has been designated in the X-direction. Clicking on the *Create Y Support Line*  will create a support line that has been designated in the Y-direction. In most instances the *Dynamic Editor* or *Support Line Wizard*  will be simpler and faster to use. Generally, it is recommended to use the *Dynamic Editor* or the *Support Line Wizard* and edit the support line they create, if needed. Use of the Support Line Wizard is described later in this section. Use of the Dynamic Editor is outline in the **ADAPT-Builder 2019 New Features Supplement Manual**. Note that in ADAPT-Builder 20 new features for creating middle strips have been added to the support line *Dynamic Editor*. For more

information on the new middle strip features please refer to the **ADAPT-Builder 20 New Features Supplement Manual**.

The program no longer requires a support line to end at a slab edge or splitter. The support line can end at any arbitrary location and the program will use the end point as the end tributary boundary for the design strip. The support line can start or terminate at another support line. The use of splitters is discussed later in this section.

You draw the support line manually using the mouse. Each support line is drawn by clicking from slab edge to each support along a chosen line to the other slab edge, using the *Snapping* tools to facilitate correct placement. The program uses each support line to define the following design parameters:

The number of spans in the design strip. A span is the support line segment between two insertion points. If an end insertion point does not correspond to a support location, then the span defines a cantilever.

The lengths of all spans and cantilevers. The length of each segment defines the length of each span in the design strip. In conjunction with adjacent design strips, the width of the design strip in each span is based upon midpoints between two adjacent support lines, unless you use splitters to delineate a tributary.

When generating support lines, it is important to keep in mind that all support line insertion points must correspond with one of the following points:

A slab edge, real or apparent: Generally, the first and last insertion point should be snapped to the outside edge of the slab (the real slab edge). These points may or may not also correspond to a support location. If not, the end segment defines a cantilever. Note that a cantilever may be defined by a very short support line segment if the support's insertion point lies just inside the slab edge.

For special situations, you may extend a support line beyond the boundary of a slab, if you wish an overhang that is not directly along the support line you are creating to be accounted for in your design.

Column, wall, beam, or point supports: Each interior insertion point must correspond to the insertion point (end point) of a column, wall, beam or point support. If the support line passes

over a wall or beam, the insertion point must correspond to the wall/beam centerline. If the support line runs along the top of a wall or beam, then one insertion point is placed at one end of the wall/beam and another insertion point at the other end of the wall/beam.

Another support line: Specifically, for ADAPT-PT modeling, if one support line is intended to support an intersecting support line, you will use a point support at the intersection of the two support lines, below the support line that is shedding load.²

To generate a support line:

Click on the *Create X or Y Support Line* () icon.

Set up object snapping tools so the cursor will snap to the desired points (endpoints for columns, nearest point for slab edges, nearest point for wall center line, end point for wall end).

Draw the support line. Left click the mouse at each point required.

After the last point has been inserted, Press *C* to close the input of the support line or press the *End* key.

To edit a previously drawn support line:

Select the desired support line.

To open the dialog box just double-click on the support line or right-click and select *Properties* from the object menu that opens.

Edit the input fields on each tab of the dialog box as required (each input field is described below).

Click *OK* to accept the new input values.

If you would like to change the support line location, make sure the *Move Selection* tool is highlighted, then just drag the support line by an insertion point to its new location.

² To insert support use *Create Design Strip Support* tool as described later in this Chapter.

If you would like to change the location of just one insertion point, make sure the *Move Selected Point* tool is highlighted, then just drag the insertion point to its new location.

If you would like to remove an insertion point, select the point and click on the *Delete Point* tool.

If you would like to add an insertion point, select the insertion point next to the one you plan to insert and click on the *Insert Point* tool. Then just drag the point to its final location.

Specific input controls are described below.

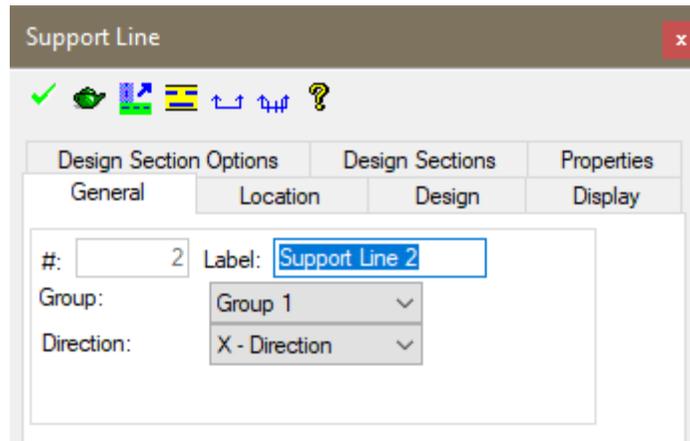


Figure 6.2-1 - Support Line General Tab

GENERAL TAB (Fig.6.2-1):

Direction, X-Direction; Y-Direction. This field is used to select the direction of the Support Line (X- or Y-direction). X and Y refer to the orientation of the principal reinforcement in two directions. They do not refer to the global axes that use the same symbol. For each support line, the direction is determined by the line you draw on plan for that support line. This field will be automatically set to X-direction when you use the *Create X-direction Support Line* tool, and Y-direction when you use the *Create Y-direction Support Line* tool. Note: Two support lines with the same defined direction cannot intersect/cross over one another. The user may end a support line at another support line with the same defined direction but the program will give errors if the support line crosses over another support line of the same defined direction.

DESIGN TAB (Fig.6.2-2):

Criteria. This combo box describes the code criteria you wish to be used for the design of the design strip selected. Depending on the building code you will select later, the program will give priority to the requirements of the building code and override the selection made here, should there be a conflict. For example, if there is a beam in the model, the program will treat it as a beam and will calculate the stirrup requirement, even if it is specified otherwise in this field.

Tendon Profile (Specific to ADAPT-PT). This field is used to select a predefined tendon shape for the support line. Before the support line is drawn, only one profile type may be assigned. After the support line is drawn, however, the dialog box may be reopened and a different tendon profile assigned to each span of the support line.

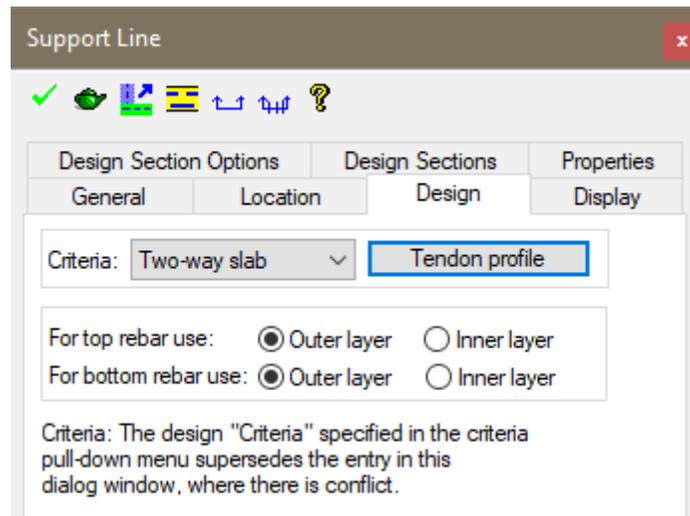


Figure 6.2-2 - Design Tab of Support Line

Outer Layer/Inner Layer. The rebar cover you specify in the criteria of the program is from the surface of concrete to the first layer of primary reinforcement. For reinforcement in two directions, one of the layers will be the outer layer (near the surface) and the other layer the inner layer (nearer to the center of the slab). Here, you specify the position of the primary reinforcement for and inner or outer layer.

DISPLAY TAB, Fig 6.2-3(specific to ADAPT-PT):

When a design strip is to be analyzed and designed separate from the rest of the structure, as is the case in strip methods, its geometry will be idealized. Likewise, the loading that falls on the tributary will be transferred to the support line of the design strip. The tools of the

Display Tab enable you to view the idealization done by the program. You will have the opportunity to review and edit the program's idealization before exporting the design strip information to ADAPT-PT for analysis and design.

Load region separator. Displays the load separators of the support line. Load separators are used to specify how a load is divided among adjacent spans. The initial orientation of a load separator, as calculated by the program, may be edited manually by dragging the end to any desired point on the slab. In most cases, however, load separators will not have to be moved.

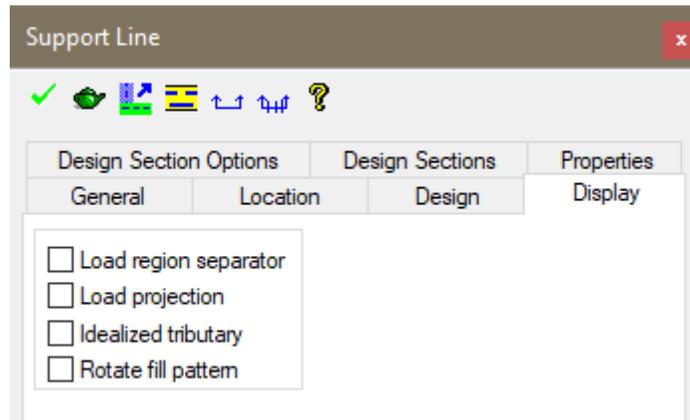


Figure 6.2-3 - Support Line Idealization Display Options

Load projection. Displays the manner in which an applied load over the tributary of a design strip is represented on the support line. Loads that are located on a design strip, but do not fall on its support line are brought to the support line prior to analysis using ADAPT-RC or ADAPT-PT.

Idealized tributary. Displays the idealized tributaries of the support line.

Rotate fill pattern. Changes the orientation of a tributary's hatching to avoid conflict with adjacent hatching. This tool is used when you generate a printed report of your modeling. If in print preview, adjacent design strips happen to have the same fill pattern, you may change the pattern using this option.

The following icons can be used only after you have used the *Generated Strips* tool of the *PT/RC Export* ribbon or *Generate Sections New* tool of the *Floor Design* ribbon. They are not used to create a support line but are an essential part of the data transferring process in the ADAPT-PT or ADAPT-RC programs. The buttons will not be visible until the design strips have been created.



Viewing the Design Strip. This button is used to create a 3-D rendering of the idealized design strip. The idealization is done for the purpose of exporting the design strip for analysis and design to either ADAPT-PT or ADAPT-PC. You can rotate and view the idealized model in detail and modify its geometry or loading before exporting it to ADAPT-PT or ADAPT-RC.

Each of the buttons displayed on the right margin of the viewing screen will open a text file that reports a specific aspect of the idealized model. These include information on span geometry, supports and loading. Click on each to view the contents. Some of the values of the report are linked to the displayed model. If you change the text value in the report, that change will be reflected in the display of your model.



Export to Strip Modeling. This button is used to write the support line input data to a subdirectory, ready to be analyzed with ADAPT-PT or ADAPT-RC. This tool writes the input data only; it does not open either program. The subdirectory to which the data is exported is given the same name as the model file.



BuilderSum. This button opens a summary report of the designed design strip. The summary report includes strip results including bending moment, shear, stresses, and reinforcement diagrams. The user must design the sections within ADAPT-Builder for this option to initiate.



Display Support Line Elevation. This button will generate a cut along the length of the support line and display it at an insertion point of your choice. You can enlarge the thickness of the slab, by increasing the Z-scale from the *Home* ribbon *Scaling* panel.



Display Design Section Elevation. This button will generate a elevation cut for each design section along the support line and display it at an insertion point of your choice.



Support Line Wizard. This tool creates a support line automatically for export to ADAPT-PT/RC. The Dynamic Editor is used to create a support line automatically for use with ADAPT-Builder. The use of the Dynamic Editor is described in the **ADAPT-Builder 2019 New Features Supplement** manual. Clicking on the *Support Line Wizard* tool opens the dialog box shown in **Fig. 6.2-4**. The support line wizard automatically

generates a support line in the direction that you specify. The wizard searches for possible supports over a strip specified by the band width you define. The wizard detects slab edges, column ends, wall ends and wall center lines that are located within the band you define. Once it creates a support line and displays it on the screen, you will be able to edit it, if needed. In most cases, it is simpler to use the wizard and edit its support line automatically than to create one manually.

To generate a support line with the wizard:

- Click on the *Support Line Wizard* button on the *Strip Modeling* panel of the *PT/RC Export* ribbon.
- Select the span direction in the dialog box shown in figure. Then select the scanning area in which supports may be regarded in creation of the support line.
- The program will automatically catch every support in this selected area.
- Click on the *OK* button to create the support line.

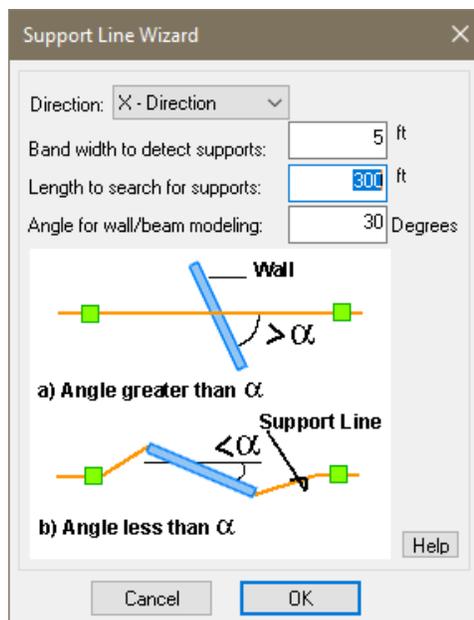


Figure 6.2-4 - Support Line Wizard Dialog Window



Splitter. These tools create a new splitter. Splitters are used to delineate the width of a design strip. They are now used solely to identify the width of a design strip tributary area in a location. Each splitter is associated with the design intended for one of the orthogonal directions, referred to as X- or Y-directions. It is defined according to the strip direction the splitter is meant to affect. Each splitter affects strips in one specified direction at a time. Therefore, separate splitters should

be drawn for each direction. For example, if a splitter is intended to affect the strips in the X-direction, draw the splitter in the X-direction.

To generate a splitter:

- Click on either the *Create X-Direction Splitter* or *Y-Direction Splitter* tool depending on the direction of the support line the splitter will be associated with.
- Set up the *Snapping* tools so the cursor will snap to the desired points (nearest point for slab edges, end point for column ends etc.).
- Draw the splitter. Left-click the mouse at each point required.
- After the last point has been inserted, Press *C* to close the input of the splitter, or press the *End* key.

To edit an existing splitter:

- Select the desired splitter.
- To open the dialog box just double-click on the splitter outline or right-click and select *Properties* from the menu that opens.
- Edit the input fields on each tab of the dialog box as required.
- Click *OK* to accept the new input values.



Strip Method Load Transfer (Specific to ADAPT-PT and ADAPT-RC). When a support line is designed to rest on another support line in the orthogonal direction and transfer its load to the supporting line, there is no physical support such as a column or wall at the intersection. The reinforcement in the slab or post-tensioning is designed to carry the load of the support recipient support line. In modeling for strip method, you must mark the location, where a support line is shedding load without the presence of a supporting wall or column.



Connect Drop Caps to Columns. This tool can be found in *Model* → *Preprocessing* panel. This tool is used to connect all existing drop cap endpoint with the endpoint of the column. The center point of the drop cap is moved to the center point of the column. The connection of column and cap makes sure that the complete cross-sectional area is considered at the support. The resulting offset is automatically calculated.

The following tools improve the quality and accuracy of modeling. For a support line to fully account for the existence of a column or wall support, when it comes to code check or export to ADAPT-PT, it should snap to the column or wall at the time of its creation. Once a support line is snapped to a column, it would automatically detect the properties of the column and report to you the

moment and stresses at the face of the column. It also will give you the reinforcement at the face of column, as well as along other locations within the span. The support line wizard will automatically detect supports and snap to them. If you generate the support lines manually, however, or move a support after you created a support line, the snapping feature may be breached. Two of the tools given below are intended to correct this situation.



Connect Support Lines to Columns and Walls. Use this tool to connect the existing support lines to walls and columns. Support lines have to be connected to the endpoint of a column or both endpoints of a wall/beam to account for them fully in the design stage of your work.



Connect Beams to Columns and Walls. This tool can be found in the *Model* → *Preprocessing* panel. This tool establishes the snapping connection between all existing beams and adjacent walls or columns, where beam ends are adjacent to a support but are not snapped to it. While the program can handle a beam that terminates short of a column, for proper treatment of beams that are connected to columns you should use the *Snap* option. This tool will search for such instances in your model and will establish the connection.



Extend Support Lines to Slab Boundaries. This tool extends all existing support lines to the edges of the slab. Use this tool if you created a support line manually, as it is likely you missed snapping its ends to the slab edges. If the distance of the support line end to a slab edge is more than the program's tolerance, you must do the connection manually.



Align Structural Components. This tool can be found in *Model* → *Preprocessing* panel. This tool improves the quality of your modeling. You can automatically adjust the location of a wall or column that you have drawn to line up with the face of slab. Left click on the tool and follow the instructions at the bottom of the screen. **Fig. 6.2-5** illustrates the function of the tool in shifting the position of a wall and a corner column to the slab edges. The tool shifts the position of the wall and column but retains the edges of slab at their original position.

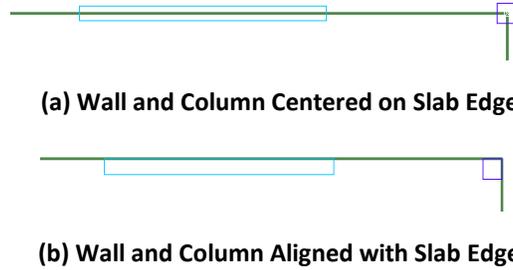
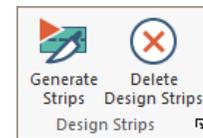


Figure 6.2-5 - Repositioning of Beam and Column to Slab Edge

6.3 Strip Generation Tools



Floor Design Ribbon



PT/RC Export Ribbon



Generate Sections (Floor Design) & Generate Strips (PT/RC Export).

This button is used to create the design strips automatically. It concludes by generating as many design strips as there were support lines created by you, taking into account the splitters that you may have used, in order to impose your preferences on strip width. The design strip calculation can be followed up by requesting that the data generated be sent to ADAPT-PT or ADAPT-RC by clicking on the *Execute ADAPT-PT/RC* icon of the *Export* panel of the *PT/RC Export* ribbon



Delete Design Strips. This tool erases the data of the last automatically generated design strip calculations, but retains all the information that you entered manually, such as support lines and splitters. This tool is generally used when you decide to modify design strips calculated by the program.

6.4 Strip visibility Tools



The above tools control the display of support lines, splitters, tributary regions, and design sections. Please refer to the description of each individual tool in section 4.13.7 of this manual.

6.5 Other Data Specific to ADAPT-PT and ADAPT-RC

In addition to the general modeling data and tools described earlier, there are additional data that you provide for export of data to ADAPT-PT and ADAPT-RC. These are grouped into data specific to the design strip you are currently dealing with, and more general data, such as allowable stresses, that apply to an entire project.

6.5.1 Data Specific to Current Design Strip

Support Boundary Conditions. Prior to exporting a design strip to ADAPT-PT or ADAPT-RC, you specify the boundary conditions of each of its supports. **Fig. 6.5-1** shows the dialog window for specification of column supports. Open the property box of a column to access the tab for its boundary conditions. A similar dialog window is available in the property box of walls.

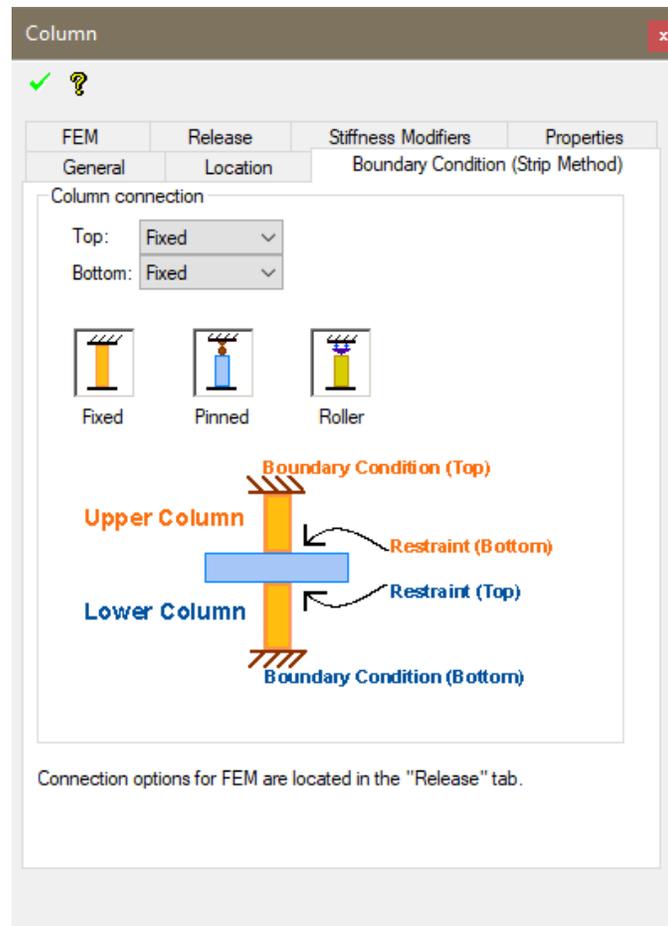


Figure 6.5-1 - Dialog Window for Specifying the Boundary Conditions of Columns for Adapt-RC/PT Data

6.5.2 Data Applicable to the Entire Project

Enter the data that is applicable to your entire project in the dialog windows listed in this section, knowing that in each instance, you have the option to go and edit them individually in ADAPT-PT and ADAPT-RC. The dialog windows and data fields used in the following are essentially the same as those you have come to know in ADAPT-PT and ADAPT-RC. A detailed explanation of each is provided in the respective manuals. In the following they are presented with a brief description.

Concrete Material. Access this dialog window from the *PT/RC Export* → *Material Properties* → *Concrete* tool.

Concrete (Strip Method EFM only)

Slab / Beam

Strength at 28 days (f'c) psi

Weight: Normal Semi lightweight Lightweight

Ultimate creep coefficient:

Modulus of elasticity at 28 days (Ec): ksi

Concrete strength at stressing (initial condition)(f'ci) psi

Column

Cylinder strength at 28 days (f'c) psi

Modulus of elasticity at 28 days (Ec): ksi

Unit weight pcf

Figure 6.5-2 - Materials Dialog Window for Adapt-RC/PT

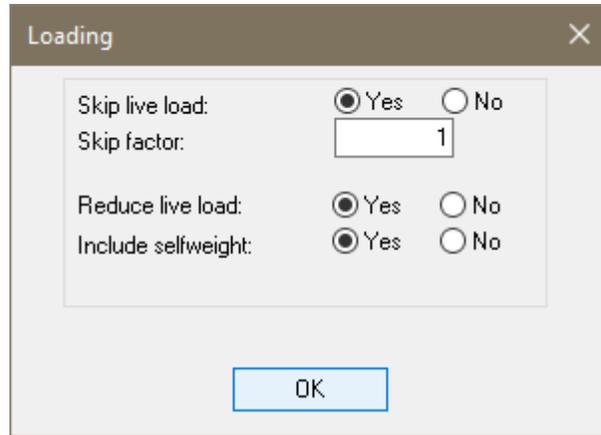
The following dialog windows (**Fig.6.5-3**) are accessed from the *PT/RC Export* → *Design Criteria* panel.

Effective Flange ✕

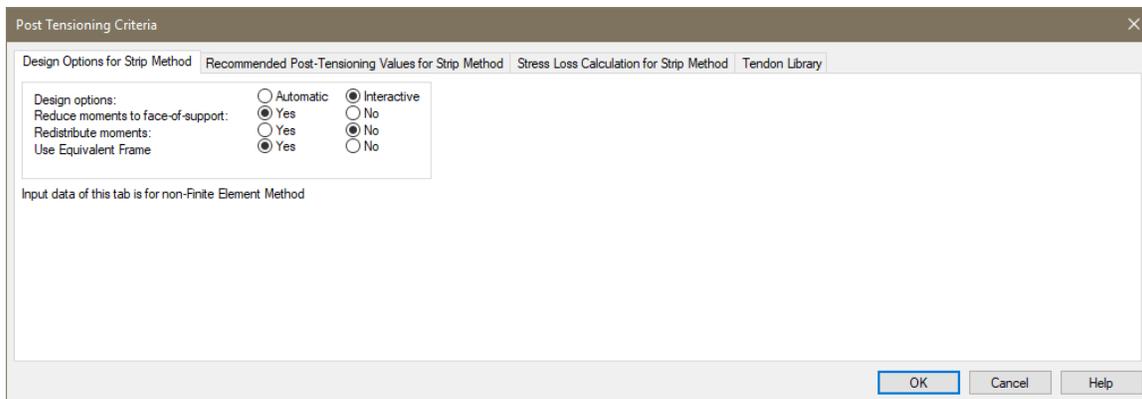
Effective width is used by some engineers for flanged beams when using the strip method, and when flange extends more than eight times the slab thickness on each side of the stem.

Consider effective flange width: Yes No

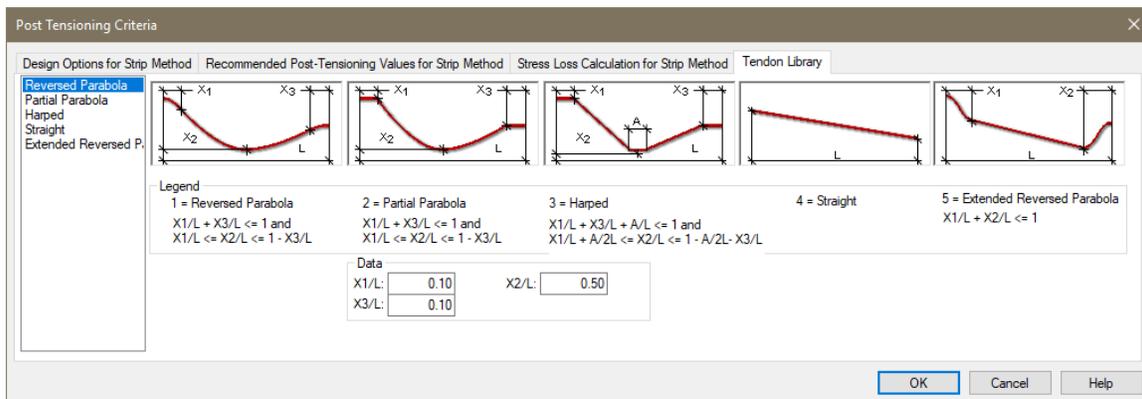
(a) Effective Width Input Window



(b) Handling of Load Input Window



(c) Tabs Each Covering a Section of Input Data



(d) Tendon Shape Library

Figure 6.5-3 - Input Data Windows Specific to Adapt-RC/PT

7 Finite Elements (Floor Design) Modeling Tools

7.1 Overview

In this chapter you will learn how to obtain a finite element solution for your structural model. The building code check and determination of reinforcement is covered in a separate manual¹. We start by assuming that you have created your structural model using the information in previous chapters. You have viewed your model in three dimensions (solid form) and are satisfied that it is correct. This chapter introduces the tools and steps that conclude with a finite element solution of the structure. The introduction of post-tensioning is covered in the ADAPT-Floor Pro manual.

To obtain a finite element solution you:

- Mesh the structure
- Select the load combination
- Solve the structure
- View the solution
- Examine the validity of the solution
- Edit/apply boundary conditions

Items related to building codes and calculation of reinforcement are handled in the **ADAPT-Floor Pro User Manual** and other manuals.

In the normal course of analysis and design, you would specify the boundary conditions of your structure prior to its analysis. The proven procedure recommended here, however, is that you first obtain a FEM using the defaults of the program to validate your structural model. Then customize your boundary conditions and releases. The program automatically applies a set of boundary conditions for the structure you have created, and automatically stabilizes it during the solution.

In *Single-Level* mode, the program assumes that the floor you are dealing with is a typical story of a multilevel structure. It is resting on a similar floor below and supports a similar floor above. The far ends of the walls and columns of your structural model are assumed to be fixed against rotation, but free to slide in the horizontal direction. Also, the far ends of the lower supports are assumed to have been placed on firm (non-sinking) supports. The columns and walls themselves, however, will shorten based on the load they receive and their material properties. The program places adequate supports at selected points

¹ ADAPT-Floor Pro User Manual

to stop your structure from floating in the horizontal direction. The fixity provided by the program against floating horizontally would not impact your solution. Note: If you have separate slabs that are not connected to each other in the same level of a model, in single-level mode, the program will only automatically support one slab in the horizontal direction. The user would have to manually add stabilizing point supports to the other slab(s) in order for all slabs to be stable in the horizontal direction.

The program offers a number of other features, such as including or disregarding one or more structural components in your analysis. It is good to review these features and take advantage of them after you have obtained your first finite element solution of your structural model. The descriptions of these features are included in the **ADAPT-Floor Pro 2019 Basic Manual**.

7.2 Meshing

7.2.1 Overview

The meshing stage of your analysis consists of subdividing the slab regions into well-proportioned, quadrilateral finite elements. Using the mesh of the slab regions, the program automatically generates the finite elements of the beams, walls and columns.² Hence, only slab regions need be meshed by you.

You have two options for meshing – manual and automatic. It is recommended that you use the automatic meshing option of the program and edit the mesh generated. In general, you do not need to edit the mesh generated by the program. In its automatic meshing option, the program uses an adaptive meshing capability that generates well-proportioned quadrilateral finite elements.

7.2.2 Maximum Mesh Size

In most cases, six to eight divisions for the typical span of a floor system gives design values (moments, shears) that are within 2 percent of the

² For the slab regions, walls and drop caps/panels, the program uses a flat shell element with both membrane and flexural degrees of freedom. For the beam and columns, the program uses a stick element with six degrees of freedom at each node. The subdivision of the columns and walls in the horizontal direction is controlled by a parameter that is in the initialization file of the program. Generally, two divisions are used. Vertically, the program automatically matches the division to that of the slab regions. Full connectivity between the walls, beams, columns and adjacent slab regions is automatically achieved by the program.

value obtained, if a very fine mesh is used.³ This accuracy applies to design stresses too. In your first attempt, start with a suggested cell size of three feet (1.00 meters). Most structural models can be analyzed and designed well with 3,000 to 4,000 cells. A complex structure may require up to 6,000 cells. It is very rare that you need to solve a single-level structure needing more than 10,000 cells. If you do, chances are that the structural modeling is not efficient.

7.2.3 Meshing Tools

The tools provided for meshing can be reached from the *Analysis* ribbon in the *Meshing* panel. They are:

General Tools:

- Mesh Generation (Automatic)
- Manual Mesh Generation
- Erase Mesh

Advanced Tools:

- Node Proximity Detection
- Shift Node Automatically
- Shift Node Manually
- Cancel Node Shift
- Exclude Meshing
- Display/Hide Component Representatives.

In this manual we will be dealing with the general tools, including *Node Proximity Detection* from the advanced tools. Once you become more familiar with the operation of the software, you may wish to explore the advanced tools.⁴

Automatic Mesh Generation. Click on the *Mesh Generation*  tool from the *Floor Design* → *Meshing* panel, the following dialog window opens.

³. The Technical Note TN-184: “*Mesh Density and Accuracy of Design Values Using ADAPT-Floor Pro*” gives a detailed evaluation of the mesh density and the accuracy of the solution. This Technical Note can be downloaded from the ADAPT website, www.adaptsoft.com.

⁴ The Advanced Tools of Meshing are described in the ADAPT-Floor Pro user manual.

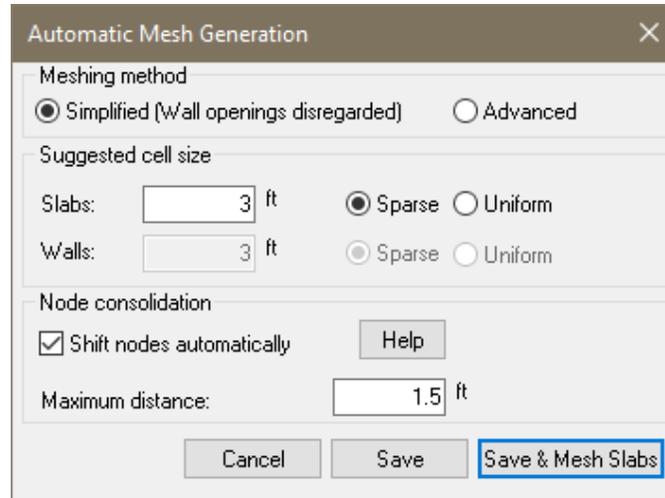


Figure 7.2-1 - Automatic Meshing Dialog Window

Select three feet (1.0 meters) for *Suggested Cell Size*.

For *Node Consolidation* select a value between 1.5 to 2.0 times the average slab thickness. Again, this feature is discussed in detail in ADAPT-Floor Pro User Manual.

The program will scan your structural model in order to determine whether the details of structure are such that a cell size will be less than the size suggested by you. If such a location is detected, the program will encircle and display it on the screen (**Fig. 7.2-2**). In most instances, it is best to ask the program to continue. Should it be impractical to mesh the structure with the mesh size you have specified, the program will let you know and request that you select a smaller mesh size. Follow the instructions on the screen.

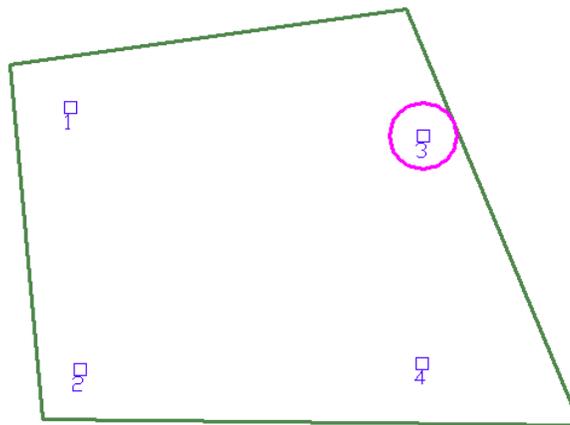


Figure 7.2-2 - Circle Identifies Where Distance is Less Than User Defined Size

Figure 7.2-3 shows an example of a structural model ready for mesh. Once successful, the program will display the mesh, as is shown in **Fig. 7.2-4**.

Manual Mesh Generation. You can generate the meshing of your structural plan manually. You can also use the manual meshing option of the program to edit automatically generated mesh. The process of manual meshing is illustrated through the following example (**Fig. 7.2-5**).

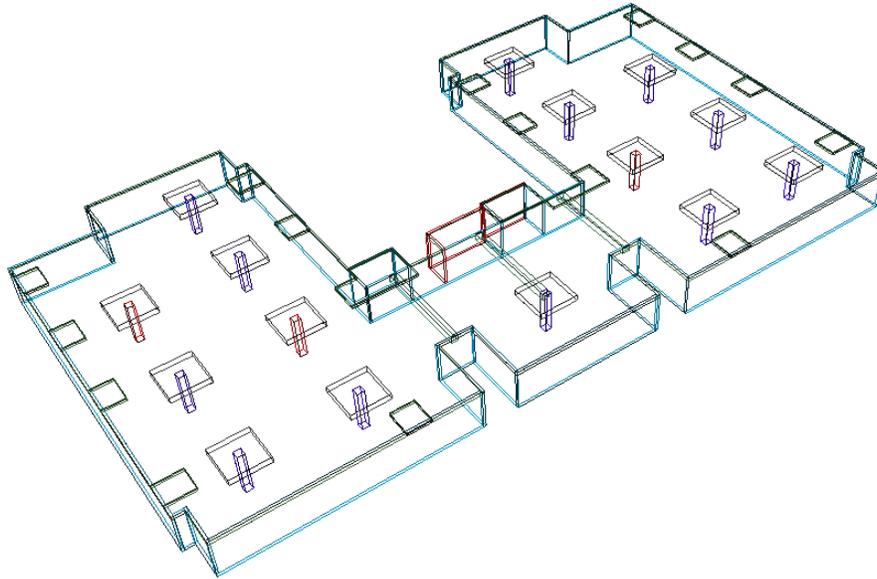


Figure 7.2-3 - Structural Model, Ready for Meshing

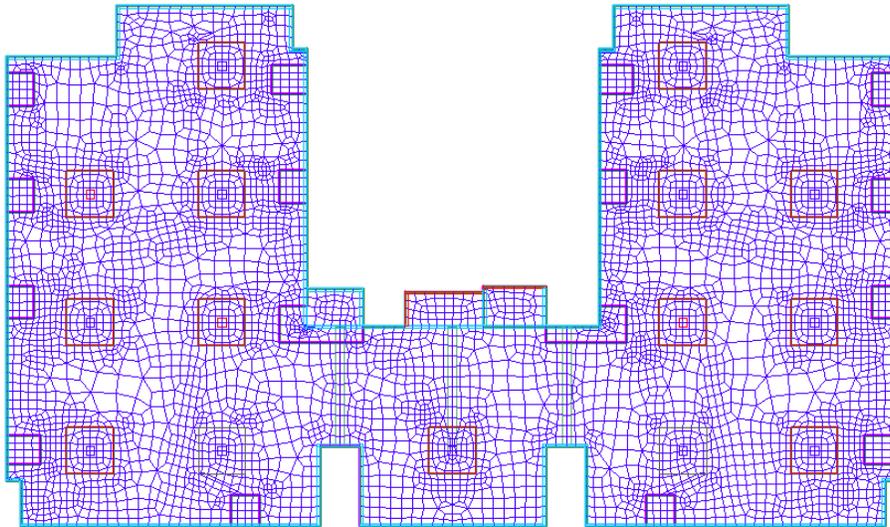
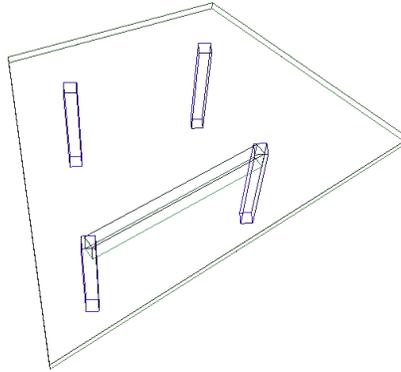
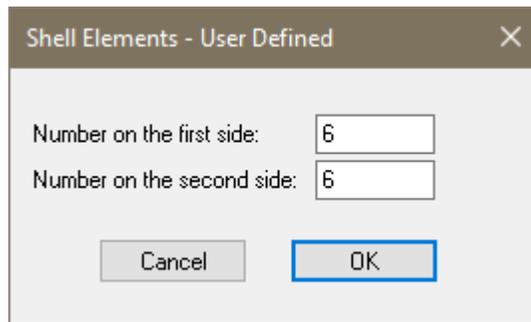


Figure 7.2-4 - Automatic Adaptive Meshing of the Structural Model



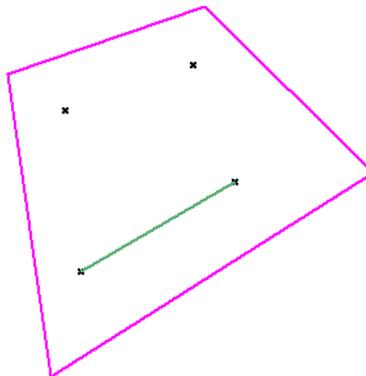
(a) 3D View of Structural Model

Click on the Manual Mesh Generation  tool, to get the dialog box shown in (b).

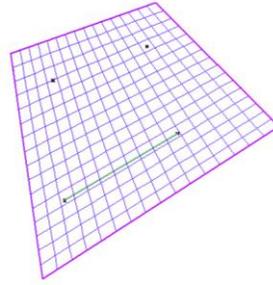


(b) This Dialog Box Opens, When You Click on *Manual Mesh Generation*.

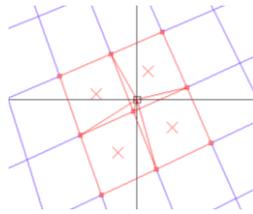
Select the number of divisions in each direction.



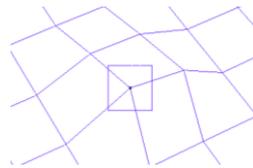
(c) Once the *Manual Meshing* Tool is Activated, the Program Converts Your Model to its Component Representation, Consisting Of Lines And Points. The Nodes of Your Manual Mesh Should Coincide With the Points and Lines Shown on Plan.



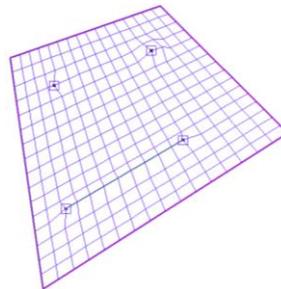
(D) Click on the Four Corners of the Region You Wish to Mesh. The Program Maps a Mesh Over The Region.



(E) Pick Each Node as Shown Above, Making Sure That it Includes All the Cells Common to That Node (Generally Four) And Pull it to Snap to the Center Of a Column (Point) as Shown in (F) or the Centerline of a Beam (Line).



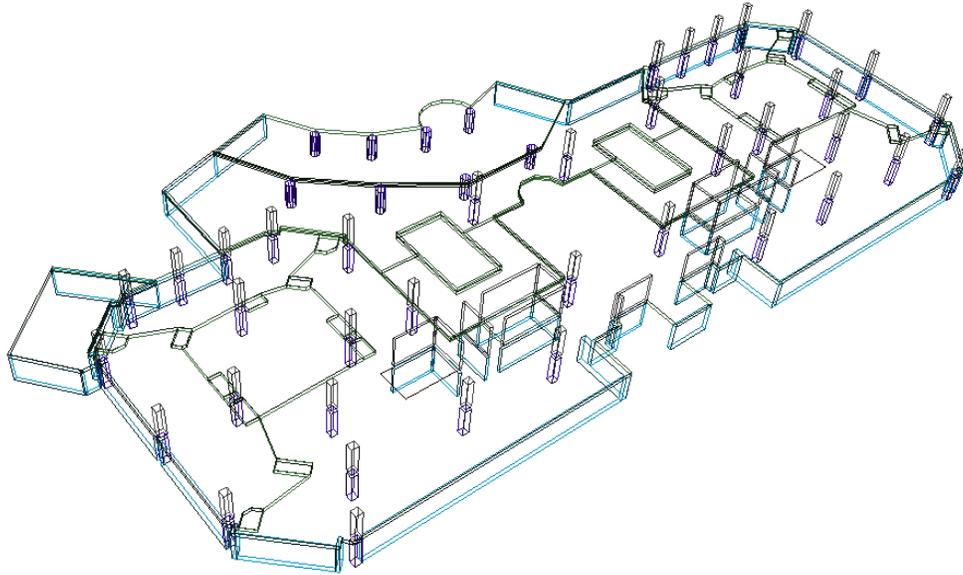
(F) Common Node of Four Cells is Pulled and Snapped to the Center of Column.



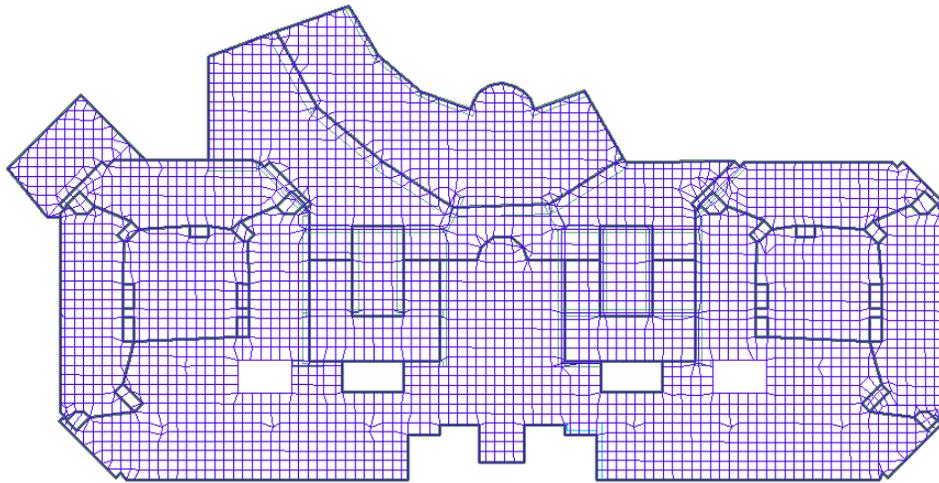
(G) Completed Model in Which the Cells Nodes are Pulled to Coincide with the Component Representatives of Columns and the Beam.

Figure 7.2-5 - Illustration of Manual Meshing Process

Figure 7.2-6(a) show a structural model that is meshed manually (Fig. 7.2-6(b))



(a) Structural Model



(b) Display of Manually Generated Mesh

Figure 7.2-6 - Manual Mesh Generation of a Structural Model

Clear Mesh.  This tool will erase the mesh, whether generated automatically or manually.

7.3 Execute Analysis

Execute Analysis.  Click on this button in the *Analysis* panel of the *Analysis* ribbon to start the analysis. The program goes through several steps and concludes with the analysis results. If you did not enter any loads prior to the analysis, the results will be for the selfweight of the structure under the Service

load combinations. The program will prompt you on the screen to resolve any errors.

Analysis Options.  Located within the *Analysis* panel of the *Analysis* ribbon. This tool opens the analysis options dialog window (shown below).

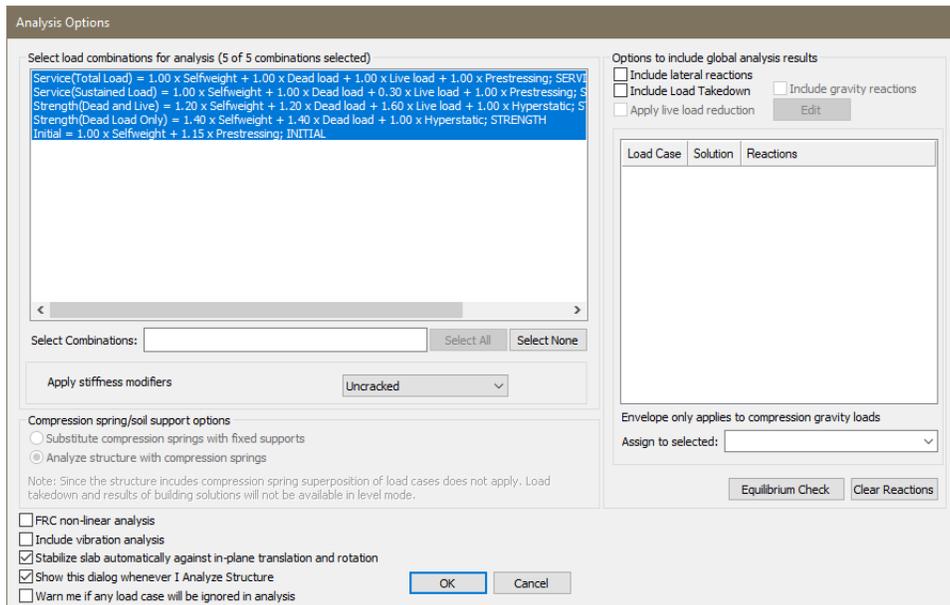


Figure 7.2-7 - Analysis Options Dialog Window

In your first finite element run, you need not disable the *Stabilize slab automatically against in-plane translation and rotation* option. If your input loading is limited to the vertical direction, and you do not provide fixity against horizontal body movement of the entire structure, the structural model is free to float (roll) in the horizontal plane. The program will automatically stabilize the structure. When there are horizontal loads and you have provided restrains against horizontal movement, however, you will need to disable the program’s automatic stabilization feature by unchecking the box.

7.4 View Analysis Results

The program has two locations where one can view Analysis Results. The user can view results through the View Analysis Results tool (legacy results viewer) or using the Results Browser that opens upon the completion of the Analysis.



View Analysis Results Tool. Click on this tool to open the three-dimensional viewer of the program and examine the solution obtained. Once the 3D viewer opens, do the following:

- On the left margin of the screen, click on the *Load Cases/Combinations* button.
- From the list that opens, select *Service (Total Load)*.
- Click on the *Results* button on the top of the screen.
- From the list that opens, select *Z-Translation*.
- Finally, click on the *Display Results* tool. 

This will display the deflected shape contour of your structure. If the

deflected contour does not display, click on this button: 

- Use the *Warp Display*  and *Rotate*  tools to examine the deformation of the structure. Make sure the slab does not displace excessively where it is supported on walls and columns.
- Use the other tools on the screen for closer examination of the solution.

Results Browser. The results browser opens on the left side of the *ADAPT-Builder User Interface* upon the completion of *Analysis*. Once the results browser opens, do the following:

- In the *Loads* tab of the *Results Browser*, check the box next to *Load Combos*→*Service*→*Service (Total Load)*.
- Click on the *Analysis* tab of the *Results Browser*, check the box next to *Slab*→*Deformation*→*Z-Translation*.

This will display the deflection contour of your structure. If your deflected contour does not display, click on the blue check mark at the upper left of the *Results Browser*.

- Use the *Zoom* and *Rotate*  tools to examine the deformation of the structure. Make sure the slab does not displace excessively where it is supported on walls and columns.
- Go to the *Settings* tab of the *Results Browser* to change the display of results for closer examination of the solution.

7.5 Examine Design Values

Close the 3D viewer and return to the main screen. You can examine the design values by generating manual cuts at locations of your choice. The program has the option of automatically obtaining the design values for the entire floor system and performing a code compliance check. This option is explained in the

ADAPT-Floor Pro user manual. In this section, we limit ourselves to the manual examination of selected locations.

Consider the structural model shown in **Fig. 7.5-1**. We already have obtained a solution for this structure, viewed it in three dimensions and are satisfied with the outcome.

Next, let us examine the design values, stresses and the reinforcement required at a given location in the slab. From *Floor Design* → *Section Design* panel, click on *Create Manual Design Section*  icon. Follow the instructions in the *Message Bar* of the screen and draw a line over the plan to identify the cut (location) for which you seek the values. **Figure 7.5-2** displays an enlargement of the interior of the floor plan with two manual cuts.

- From the Floor Design → Section Design panel, click on the Design the Design Sections  icon.
- Once the operation is complete, double-click on the cut to open up its property box (**Fig. 7.5-3**). This table lists the results of the forces acting on the cut, reduced to six actions (three forces and three moments) and expressed at the centroid of the cut, reinforcement required, and the deflection ratio (N/A to strength combinations for the load combination selected at the top of the *Actions* section. The sections moment capacity and demand are given in the window as well.
- Additionally, the user can view the results for a manual design section cut graphically through the *Results Browser*. In the *Analysis* tab expand the *Manual Design Section* tree. Select an *Action* or *Code Check* item you want to review the results for. The program will then show the result graphically on plan along the manual design section cut.

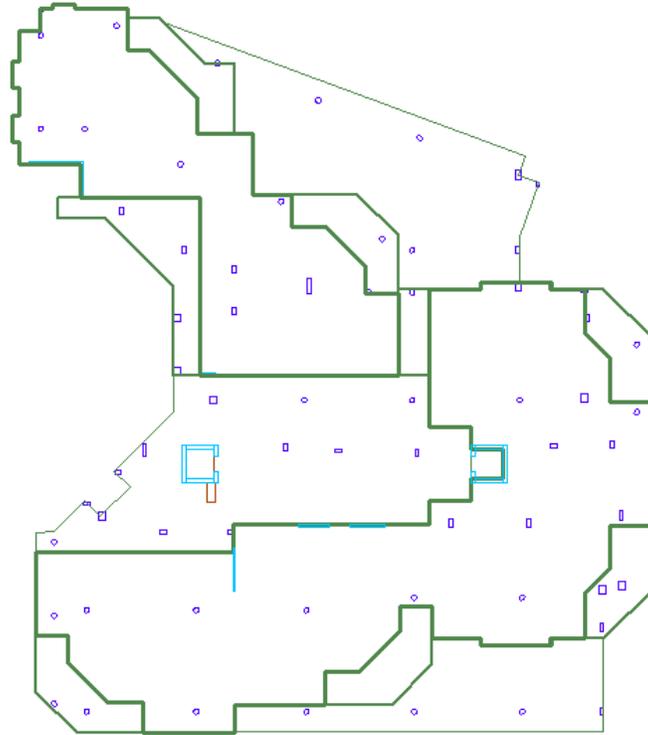


Figure 7.5-1 - Plan of the Structural Model

In addition to the window of design values, the program shows the envelope of reinforcement area on plan in the direction, where the reinforcement should be placed (**Fig. 7.5-4**).

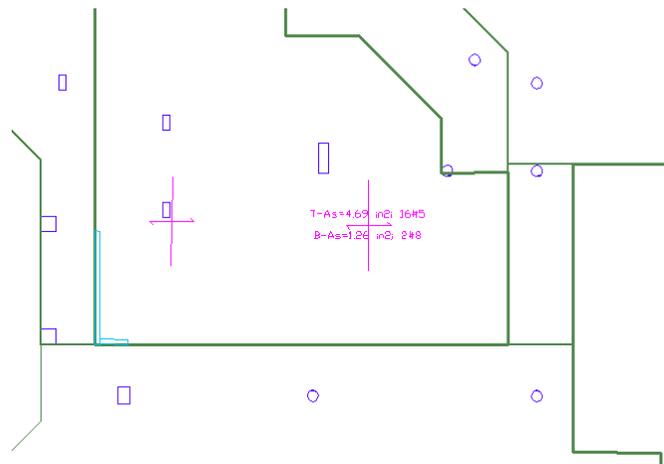


Figure 7.5-2 - Manually Generated Cuts on Plan

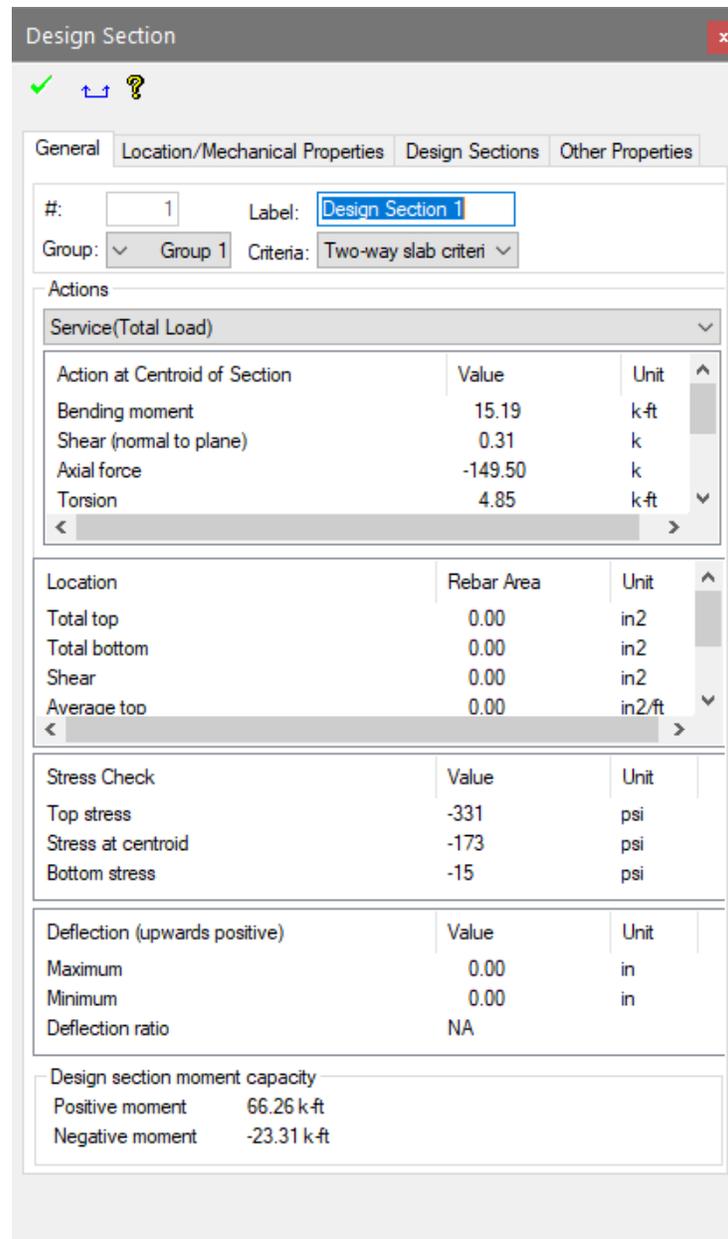


Figure 7.5-3 - Window of Design Values for Manual Cut

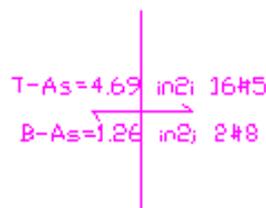


Figure 7.5-4 - Display of Rebar on Plan

7.6 Edit/Apply Boundary Conditions

7.6.1 Overview

As explained earlier, the program uses default support conditions and stabilizes your structural model automatically to obtain a solution. There will be situations, however, when you may want to specify support conditions other than the program defaults. This section guides you through the process of editing or applying boundary conditions.

The boundary condition describes the way your structural model is attached to its supports. All the components of your structural model are assumed to be rigidly connected to one another where they meet. For example, a column shown below a slab is assumed by the program to be rigidly connected to the slab. You may wish, however, to have a hinge connection to eliminate transfer of moment from the slab to the column. The specification of a condition other than full fixity among the structural components of your model is called a release. For example, you can release a slab to slide over a wall. The specification for releases can be directly accessed from the property box of each structural component.

The following is a quote from an earlier section of this manual and describes the default boundary conditions of the program.

In Single-Level mode, the program assumes that the floor you are dealing with is a typical story of a multilevel structure. It is resting on a similar floor below and supports a similar floor above. The far ends of the walls and columns of your structural model are assumed to be fixed against rotation, but free to slide in the horizontal direction. Also, the far ends of the lower supports are assumed to have been placed on firm (non-sinking) supports. The columns and walls themselves will shorten based on the load they receive and their material properties. The program places adequate supports at selected points to stop your structure to float in the horizontal direction. The fixity provided by the program against floating of the structure in the horizontal direction would not impact your solution.

7.6.2 How to View and Edit the Boundary Conditions

The program's default boundary conditions consist of point supports below the columns and line supports below the walls. There might be other support conditions in your model that you may have imposed, such as a line spring, or a point support below a slab.

From the *Select/Set View Items* (🔗) go to the *FEM* tab and select the following check boxes:

- Point Support
- Line Support

Click *OK* to close the dialog window. The icon of the boundary conditions should appear on the screen. For better viewing, click on the *Top-Front-Right View* button (👁️) or a similar one, in order to display a perspective view of the structure. If the symbol sizes of the boundary conditions displayed are not large enough for easy viewing and selection, go back to the *Select/Set View Items* window and change the symbol size of the point and line supports to a larger value.

Select the boundary condition symbol you want to view or edit. Depending on which type of boundary symbol you select, a dialog window such as the one shown below will open.

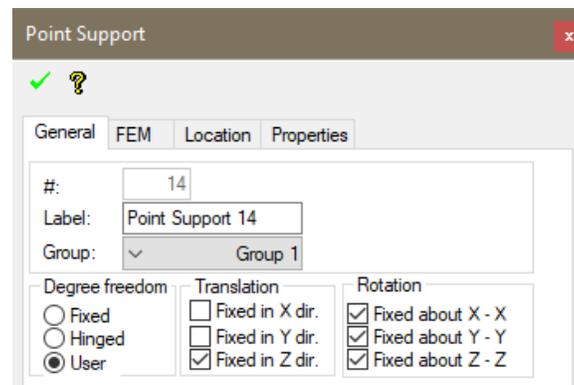


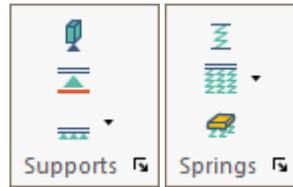
Figure 7.6-1 - Point Support Dialog Window

Among other information, the dialog window displays the restraints of the support in or about various directions. Edit as required. Click to tick mark button (✅) to accept your modification and close the window.

7.6.3 How to Apply Boundary Conditions

To specify a support condition at a point, or along a line in your structural model, use the tools in the *Model* ribbon *Supports* and *Springs* Panels.

7.6.4 Model Ribbon, Supports and Springs Panels



Add Point Support. This tool is used to specify a point anywhere on your structural model where you want to set the displacements (translation or rotation) in or about any of the three axes to zero. You can use this tool to specify fixity at a point in a slab region, or at the bottom or top of a column. To insert a point support:

- Click on the *Add Point Support* icon.
- Insert the point support by snapping it to the location/structural component. For example, for a column snap to column’s center. The point support will be inserted at the active plane.⁶
- If the point support was not inserted in the location you intended, edit the location by going through the view/edit steps described above. For example, if you want the point support to be at the bottom of a column, open the property box of the point support. From the *General* tab, select *Bottom Plane* for the location (**Fig. 7.6-2**).

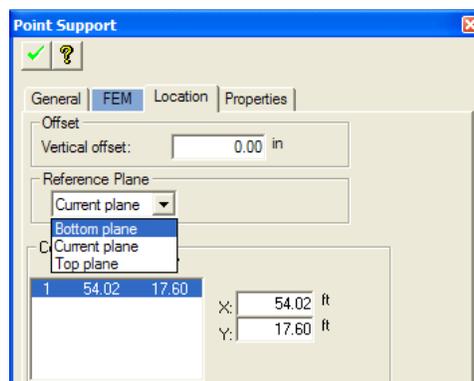


Figure 7.6-2 - Point Support Dialog Window

⁶ The active plane is the default for insertion of the supports. You have the option of defining the particulars of a support, such as its point of insertion and its fixity beforehand, then inserting similar supports at locations of your choice. Go to *Home* → *Settings* → *General Settings* and check the “*Open Item’s Property Box Automatically*” option.



Add Line Support. Many of the comments made for specification of point supports apply equally to line supports. Line supports can be placed below slabs, walls or beams. To insert or edit the specifics of a line support, follow the same type of instructions as described for point supports.



Add Point Spring. This tool is used to specify the point spring at a location on a slab. The insertion and view/editing are similar to a point support.



Add Line Spring. This tool is used to specify the lines in which the slab is supported on springs.



Add Area Spring. To insert an area spring below a slab:

- Click on the *Add Area Spring* icon.
- Follow the instruction on the screen to draw the region where area spring applies. If any part of the region you draw is not covered by a slab, the program will disregard that part.

8 Appendix A

8.1 Treatment of Compound (Interconnected) Wall Assemblies

Walls are not always isolated, simple, and rectangular in shape. In many cases walls are assemblies of two or more wall segments, such as (a) in **Fig. 8.1-1**. Both **ADAPT-Modeler** and **Floor Pro** can faithfully model, and account for the structural features of an entire wall assembly.

8.2 Structural Modeling

In your structural model, a wall assembly must be broken down into rectangular wall segments. There are two options as illustrated in **Fig. 8.1-1**. In option 1, each wall segment butts against the side of an adjoining wall, with no overlap of wall segments. In option 2, the wall segments are connected to one another at their centerlines. In both modeling options, the program treats the segments working together as one whole. Option 2 takes less computer resources for analysis and design and is the recommended modeling method. If you follow the modeling procedure outlined in this manual, the structural drawing you will generate from either of the two options will show the wall in its true form (**Fig. 8.1-1(a)**).

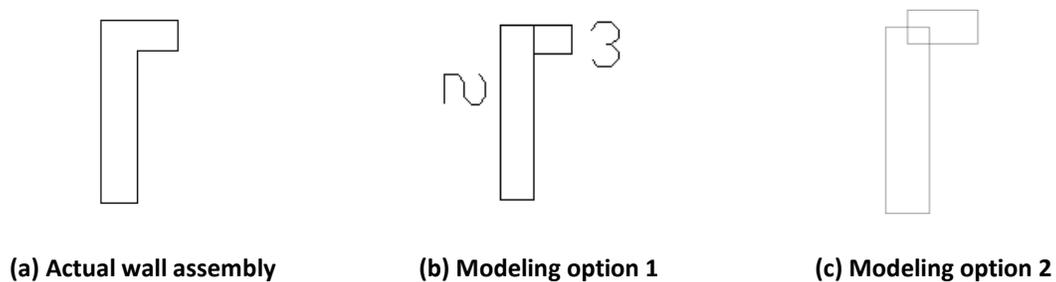


FIGURE 8.1-1 MODELING OPTIONS OF WALL ASSEMBLIES

8.3 Support Lines and Design Strips

As much as practical, you will use the *Support Line Wizard* () for export to **ADAPT-PTRC** or the *Dynamic Editor* () for design within **ADAPT-Floor Pro**, to generate support lines. For compound wall assemblies, it would not be practical to consider each wall segment as a separate support. Ideally, you would like to identify a point that is the beginning of wall support, and another one at the end of the wall support. Again, there are two options for you to achieve this, when you use the *Support Line Wizard* or the *Dynamic Editor*.

Option 1

In this option, you can change the property of a wall segment, such as wall segment marked “3” in **Fig. 8.2-1(b)** to “disregard.” You do so, by opening the property window of the wall marked “3,” and under the FEM tab change its property from “consider” to “disregard.” This change in property will result in the wall segment not to be accounted for when the support line wizard or dynamic editor automatically creates a support line. The outcome will be like **Fig. 8.2-1(e)**. If the wall segment 3 is not disregarded, the support line created automatically will be as shown in **Fig. 8.2-1(d)**.

Option 2

In this option you do not “disregard” a wall segment. After the support line is generated, you will eliminate the support line segments that zigzag over the interconnected wall segments. Do the following.

- Turn the support line symbol on. The small circles are the “support” locations of the support line.
- Retain the first and the last support points and delete the support points (circles) in between. Do so by using the *Remove Point* tool (). The outcome will be a view, like **Fig. 8.2-2**.

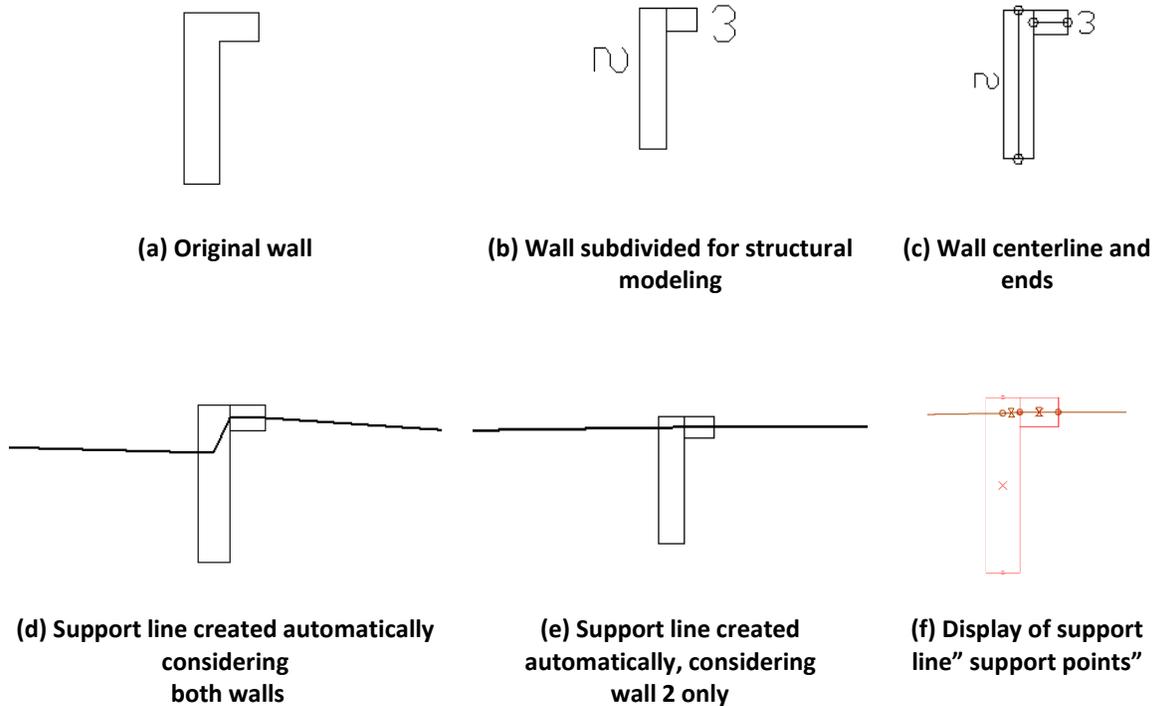


Figure 8.2-1 - Treatment of Support Lines for Compound Wall Segments

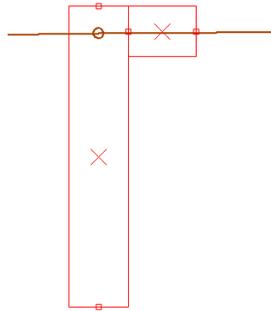


Figure 8.2-2 - Support Line with Two Support Points Over Compound Wall

If you disregard a wall segment prior to the analysis, its stiffness will not be included in the analysis of the floor system. But, at design time, its presence will be recognized. Walls segments that are primarily for architectural reasons and do not provide a significant support to the structure are best disregarded in the analysis. Otherwise, in creation of support lines use Option 2, whereby you would edit the automatically generated support lines.