

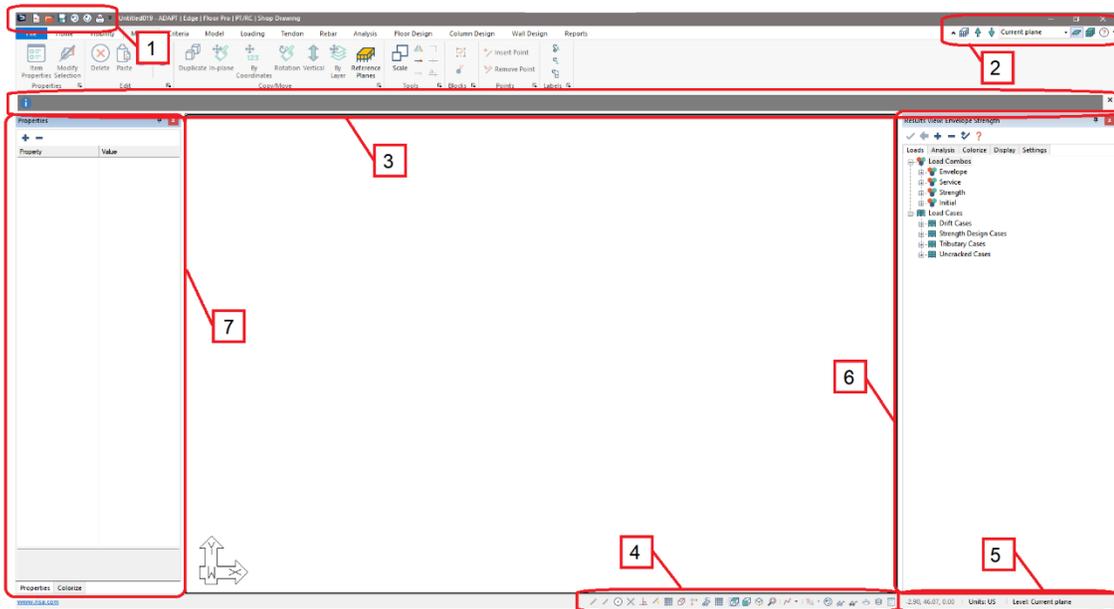
STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-Builder® 20
Graphical User Interface
Quick Reference Guide

Introduction

The purpose of this document is to act as a quick 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.

1 Main User Interface



1.1 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 previously undone action.



Opens printer selection and settings and prints active screen.



Allows the user to customize the Quick Access Toolbar.

1.2 Top-Right Level Tools (2)



This tool allows the user to display/hide the ribbon menu.



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



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



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



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



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.

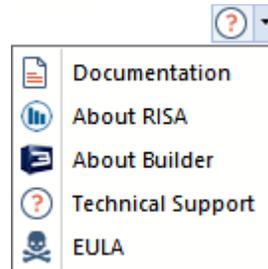
Multi-Level mode

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 at RISA.
- **EULA** 
 - Opens a window displaying the End-User License Agreement of the software.

1.3 Message Bar (3)



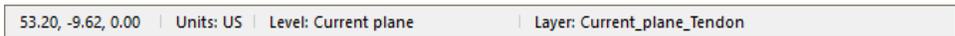
Display's model information or prompts user for input.

1.4 Bottom Quick Access Tools (4)



This is a compilation of commonly used icons from within other ribbons. Please see their description and use in the description of the main ribbon/panel they are located in.

1.5 Bottom-Right – Status Bar (5)



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.

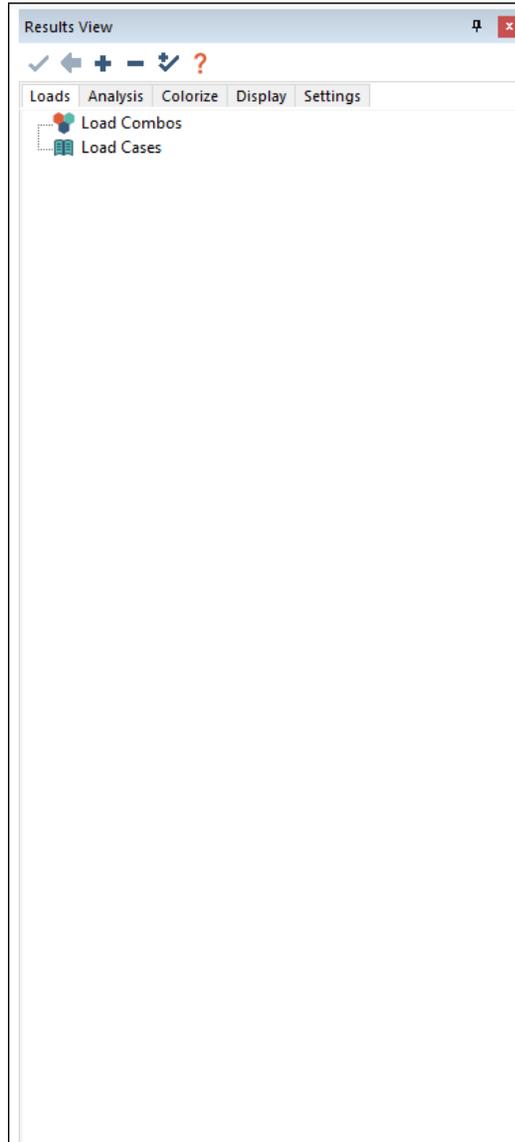
Level Level: Current plane

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

Layer Layer: Current_plane_Tendon

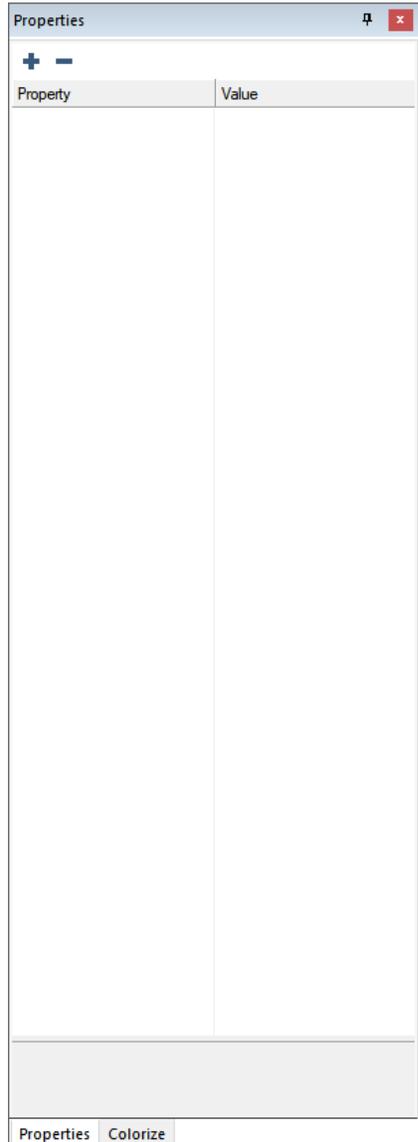
Reports the active layer. Select this text to change the active layer.

1.6 The Result Browser (6)



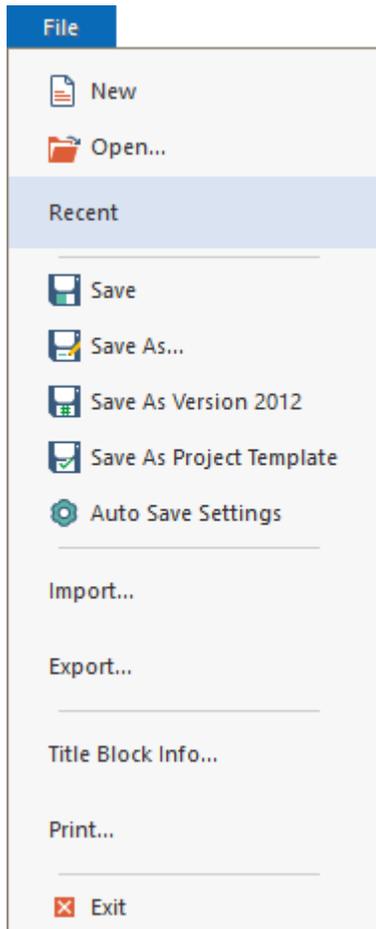
Graphical **Result Browser**. The user can view graphically by load combination/load case the results/actions for Slabs, Beams, Columns, Walls, Design Strips, Punching Shear results, and more. In addition the user can edit the code check limits as well as the display of graphical results within the **Result Browser**. Review the **Results Display Settings User Interface Guide** for detailed information on the result options provided by the Result Browser. This guide can be found by going to *Help* → *Documentation* within the software.

1.7 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**.

2 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.
- **Loads in XLS file**
Used to import point and line loads from a formatted Excel file

Export

Opens options to export:

- **Latest INP Version**
Used to update or create a Builder .adm from INP file. Additionally this option is used for import of INP files created from the latest version of the ADAPT-REVIT Link (ADAPT-Revit Link 2020 currently).
- **Builder 2019.0 Format**
Used to update or create a Builder .adm from INP file. Additionally this option is used for import of INP files created from older versions of the ADAPT-REVIT Link (ADAPT-Revit Link 2019 and prior versions).
- **DWG/DXF file**
Used to export drawing data from Builder to Autocad

- **Revit**
Used to create a new Revit .rvt file from legacy ADAPT-Revit Link versions (versions prior to ADAPT-Revit Link 2019).

- **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.

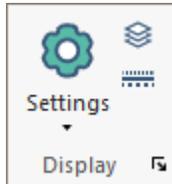
Exit 

Exits the application.

3 Home Ribbon

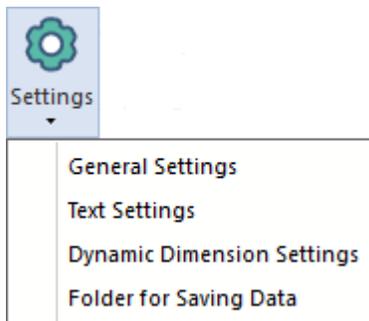


3.1 Display Panel



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.

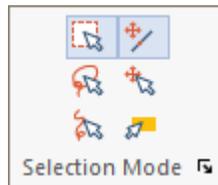
Layer Settings

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.

3.2 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.

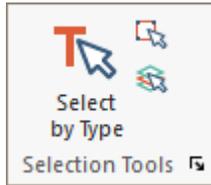
Move Selection

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.

3.3 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.

3.4 Draw Panel



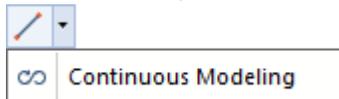
Create point

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** 

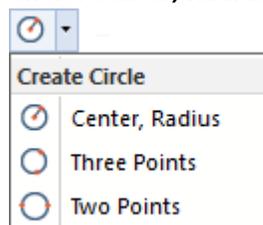
Creates a polygon by selecting number of sides, start point at mid-distance location of first edge, and end point at mid-distance location of second edge.
- **Polygon: Vertices, Start Edge, End Edge** 

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.

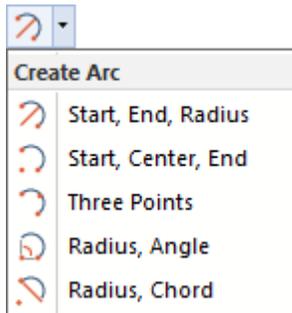
Circle: Center, Radius Expanded



- **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



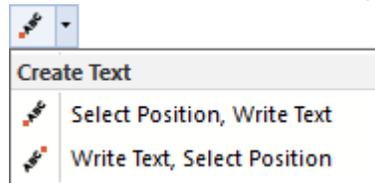
- **Arc: Start, End, Radius** 
Creates an arc by specifying starting, ending, and radius value.
- **Arc: Start, Center, End** 
Creates an arc by specifying start, center and end points.
- **Arc: Three Points** 
Creates an arc by specifying three points.

- **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.

3.5 Tools Panel



Measure Tool

Measure the distance between two points.

Coordinates 

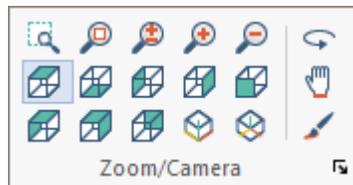
Reports the X, Y and Z coordinates of a user-selected point.

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.

3.6 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.

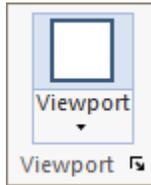
Pan View

Pans the model view from a user-defined selected point and mouse control.

Redraw

Redraws the current model view.

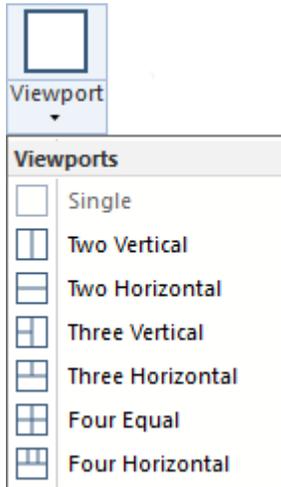
3.7 Viewport Panel



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.

3.8 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.

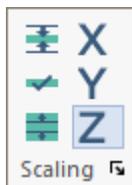
UCS: 5 points

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.

3.9 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 **X**

Enables scaling distortion in the X-Direction.

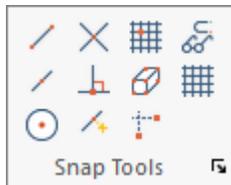
Modify the Y-Direction Scale **Y**

Enables Scaling distortion in the Y-Direction.

Modify the Z-Direction Scale **Z**

Enables Scaling distortion in the Z-Direction.

3.10 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.

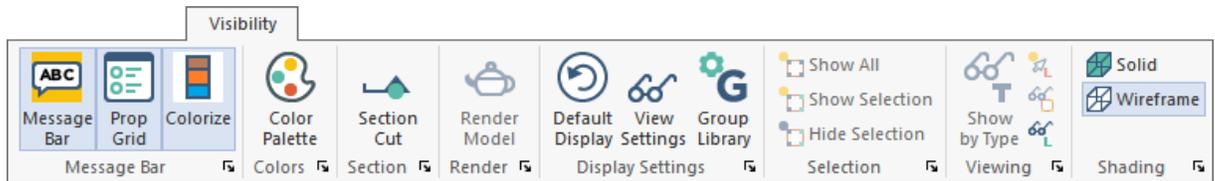
Snap Settings

Opens the Snap Settings dialog box.

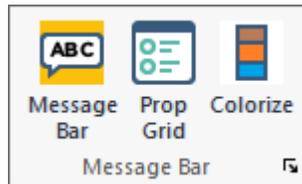
Grid Settings

Opens the Grid Settings dialog box

4 Visibility Ribbon



4.1 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.

4.2 Colors Panel



Color Palette

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

4.3 Section Panel



Create Section Cut

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

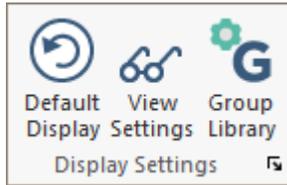
4.4 Render Panel



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 Display Settings Panel



Default Display

Restores the view to the user-defined default display

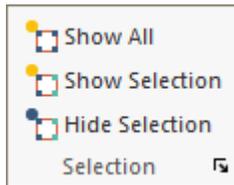
View Settings (Select/Set View Items)

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.

4.6 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.

4.7 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**.

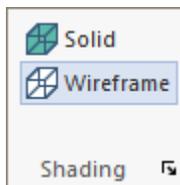
Display All Items

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.

4.8 Shading Panel



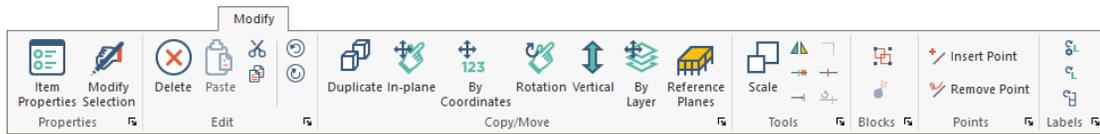
Solid Fill

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.

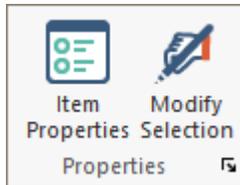
Wire Frame

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

5 Modify Ribbon



5.1 Properties Panel



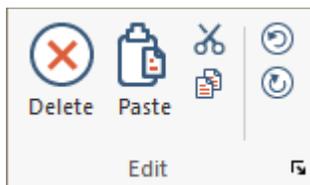
Items Properties

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.

5.2 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.

5.3 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

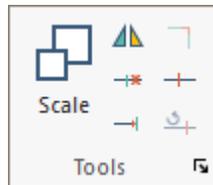
By Layer (Translation)

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**.

5.4 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 compents.

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.

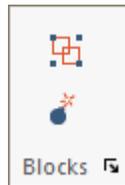
Split

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.

5.5 Blocks Panel



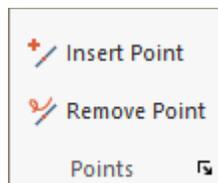
Create Block

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

Explode Block

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

5.6 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.

5.7 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.

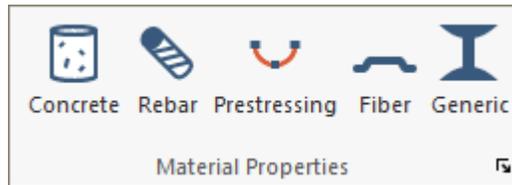
Reset Column/Wall Stack 

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.

6 Criteria Ribbon



6.1 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**.

6.2 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.

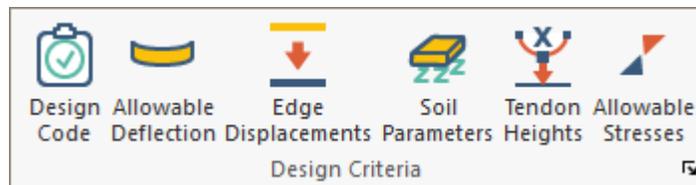
Rebar Round-Up 

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.

6.3 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.

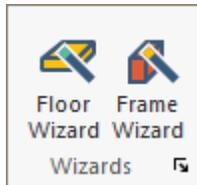
Allowable Stresses

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

7 Model Ribbon



7.1 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.

7.2 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.

7.3 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



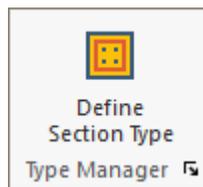
- **Gridline Wizard** 

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.

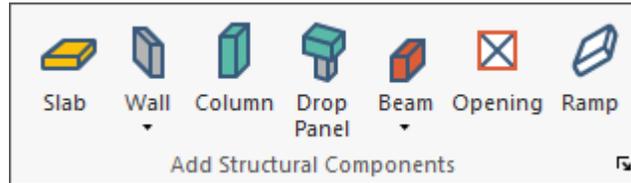
7.4 Type Manager Panel



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.

7.5 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.

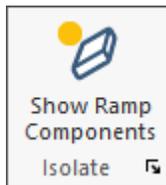
Opening

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.

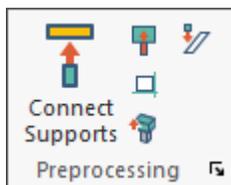
7.6 Isolate Panel



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.

7.7 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.

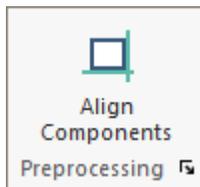
Connect Drop/Cap Panel to Column 

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.

7.8 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.

7.9 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

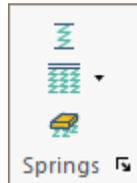
Creates a point support by graphical or coordinate entry at the support insertion point. Double-click on the support to open the **Properties** and change fixity conditions.

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.

7.10 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 (K_{xx} , K_{yy} , and K_{zz}) 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 (K_{xx} , K_{yy} and K_{zz}) 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.

7.11 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.

7.12 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 Gridlines 
Displays on/off gridlines.

Display/Hide Ramps 
Displays on/off ramps.

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.

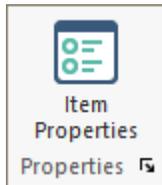
Display/Hide Area Springs 
Displays on/off area springs. Double-click on spring to see sub-grade moduli settings for Kxx, Kyy and Kzz and to set compression or tension spring type.

Point Displacement  (SOG only)

Displays on/off point displacements.

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

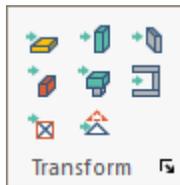
7.13 Properties Panel



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**.

7.14 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.

8 Loading Ribbon



8.1 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.

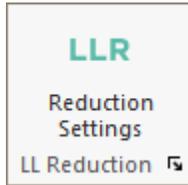
Load Combinations

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.

8.2 LL Reduction Panel

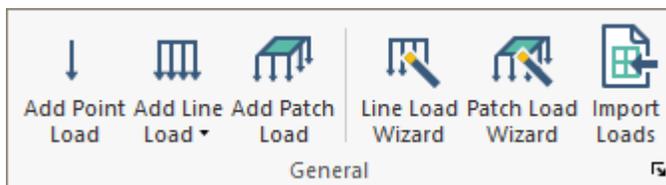


Reduction Settings ^{LLR}

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**.

8.3 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.

8.4 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.

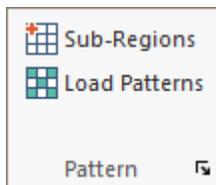
8.5 Tributary Panel



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.

8.6 Pattern Panel



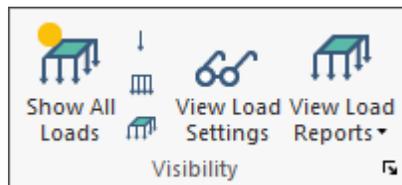
Sub-regions 

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.

8.7 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.

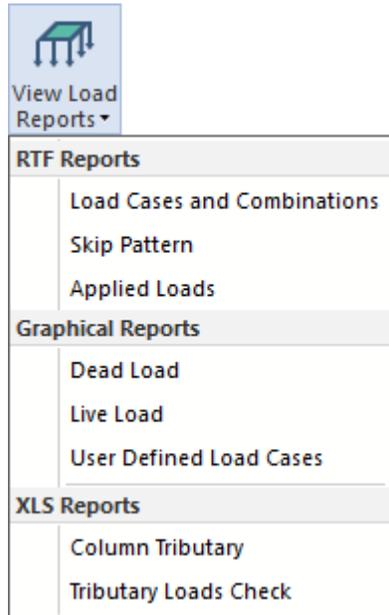
Show Area Loads

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



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.

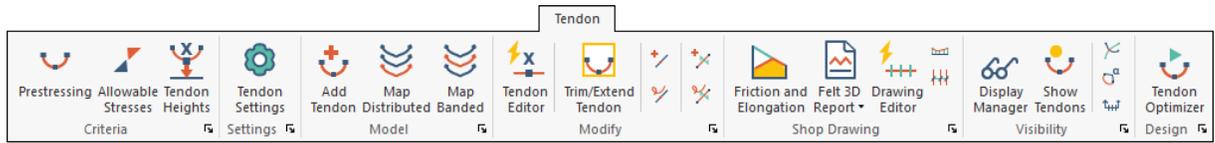
Column Tributary

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.

9 Tendon Ribbon



9.1 Criteria Panel



Prestressing

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

Allowable Stresses

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

Tendon Height

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

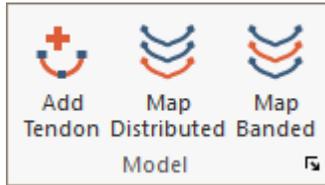
9.2 Settings Panel



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.

9.3 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**.

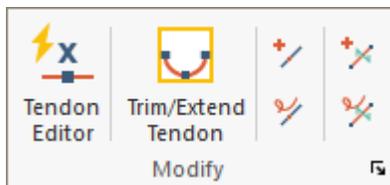
Map Distributed

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.

9.4 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.

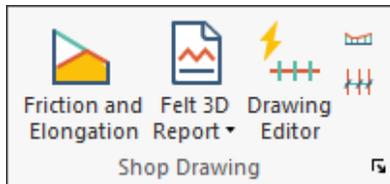
Insert Swerve Point 

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 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.

9.5 Shop Drawing Panel (add-on Module)

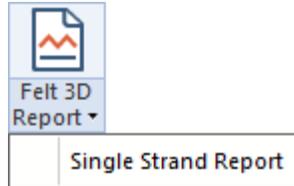


Friction and Elongation 

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.

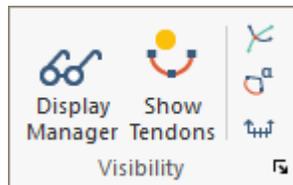
Display Tendon Heights

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.

9.6 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.

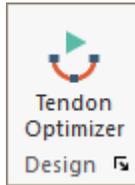
Display/Hide Radius of Curvature

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.

9.7 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.

10 Rebar Ribbon



10.1 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.

10.2 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.

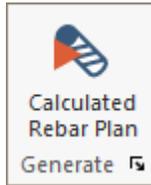
Banded

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.

10.3 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.

10.4 Visibility Panel



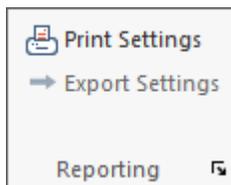
Display Manager

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.

10.5 Reporting Panel



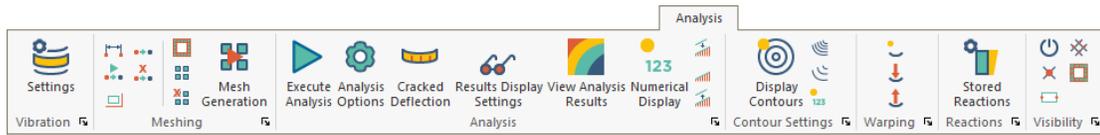
Print Settings

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

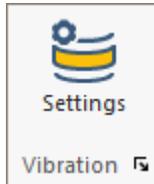
Export Settings

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

11 Analysis Ribbon



11.1 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.

11.2 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.

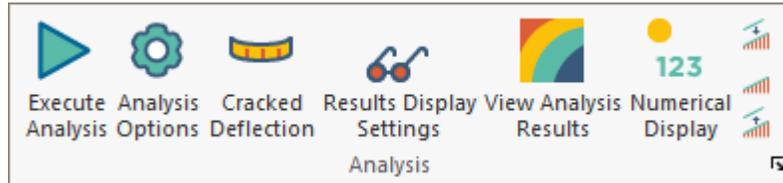
Erase Mesh

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.

11.3 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.

11.4 Contour Settings Panel



Display Contours

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

Increase Number of Contour Lines

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

Decrease Number of Contour Lines

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

Display/Hide Text on Line Contours

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

11.5 Warping Panel



Display/Hide Contour Warping

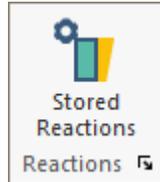
Displays on/off a warped view of the selected FEM slab or wall graphical contour results.

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.

11.6 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**.

11.7 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.

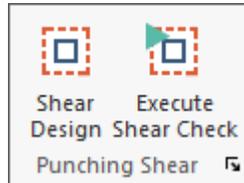
Display/Hide Excluder

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**.

12 Floor Design Ribbon



12.1 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.

12.2 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

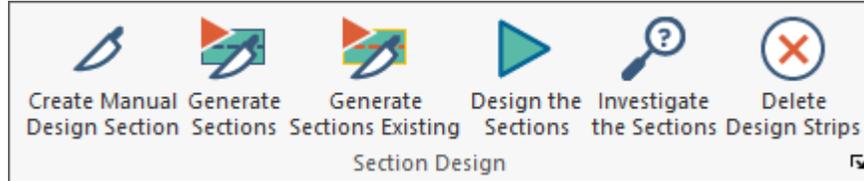
Auto-adjusts the selected support line/s end-points to the nearest slab edge.

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.

12.3 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.

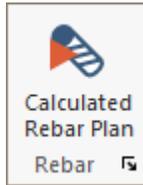
Investigate the Sections

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.

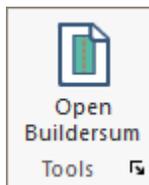
12.4 Rebar Panel



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.

12.5 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.

12.6 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.

12.7 Strip Results/Visibility Panel



Display Strip Results

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

Results Scale Down

Reduces the graphical results scale ordinate.

Default Results Scale

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

Results Scale Up

Increases the graphical results scale ordinate.

Numerical Display

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

Display Max/Min Values

Shows the maximum and minimum values of the selected results.

Perpendicular Projection

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

Display 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.

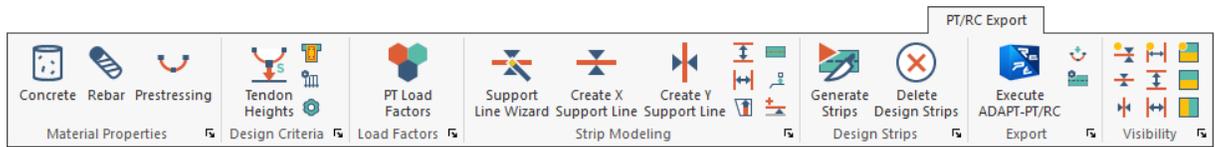
Display/Hide Support Line Tributaries X-direction 

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.

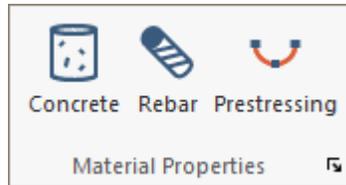
Display/Hide Support Line Tributaries Y-direction 

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.

13 PT/RC Export Ribbon



13.1 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.

13.2 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.

13.3 Load Factors Panel



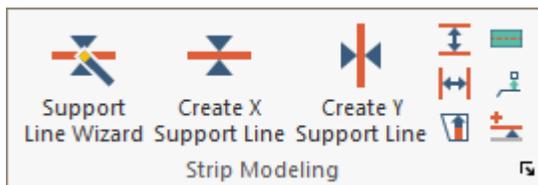
Load Combinations for Export to ADAPT-RC (RC Only Mode)

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.

13.4 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.

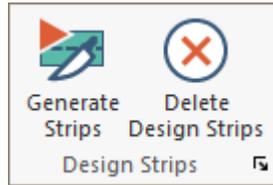
Connect Support Lines to Columns and Walls

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.

13.5 Design Strips Panel



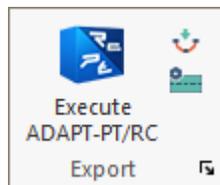
Generate Design Strips for Strip Method

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.

13.6 Export Panel



Execute ADAPT-PT/RC

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

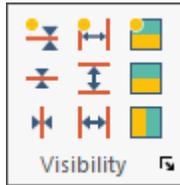
Import Tendons from ADAPT-PT

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.

13.7 Visibility Panel



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

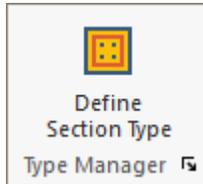


Displays on/off tributary regions assigned to Y direction. If Generate Design Sections from Floor Design-Section Design has been performed, tributary regions will not be shown by default.

14 Column Design Ribbon



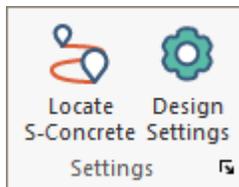
14.1 Type Manager Panel



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.

14.2 Settings Panel



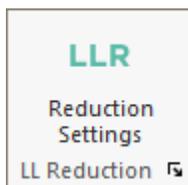
Set Path for S-Concrete

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.

14.3 Live Load Reduction Ribbon



ADAPT

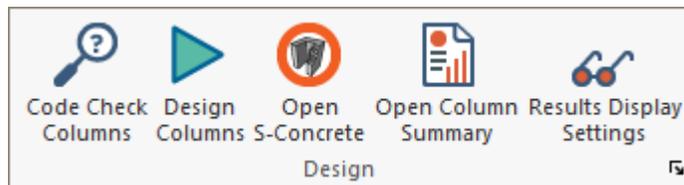
Live Load Reduction ^{LLR}

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**.

14.4 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.

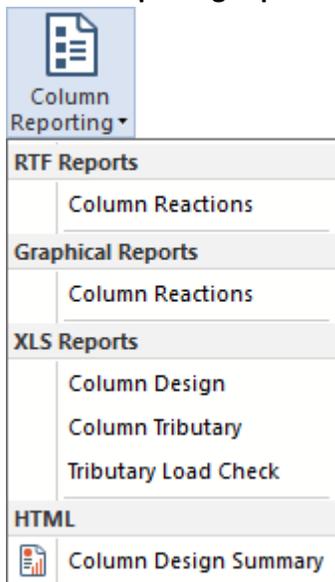
Result Display Settings

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

14.5 Reports Panel



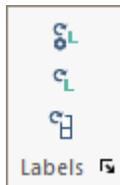
Column Reporting Expanded



- **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.

14.6 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.

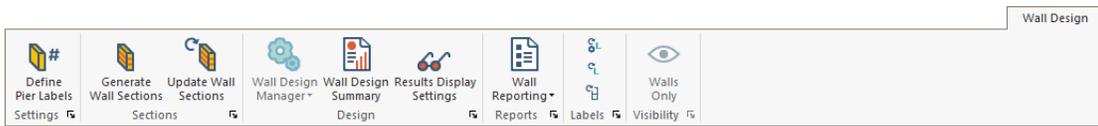
14.7 Visibility Panel



Columns Only

Isolates the columns for on/off display.

15 Wall Design Ribbon



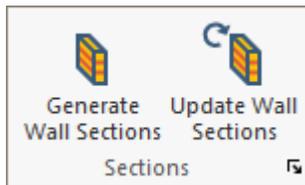
15.1 Settings Panel



Define Pier Labels

Creates pier labels for assignment to walls or wall groups.

15.2 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.

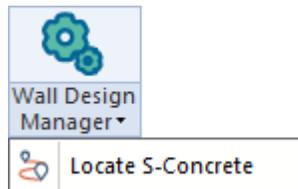
15.3 Design Panel



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.

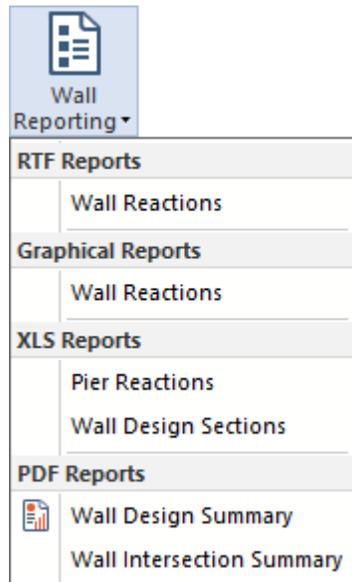
Result Display Settings

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

15.4 Reports

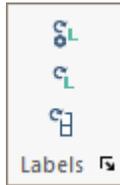


Wall Reporting Expanded



- **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.

15.5 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.

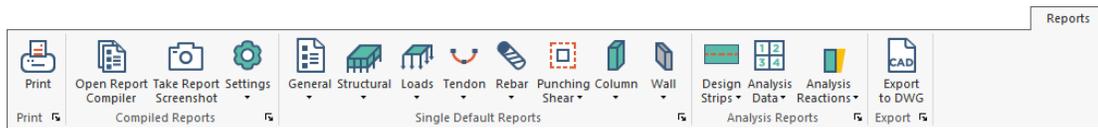
15.6 Visibility Panel



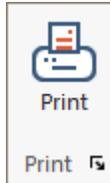
Walls Only

Isolates the walls for on/off display.

16 Reports Ribbon



16.1 Print Panel



Print 

Opens printer selection and settings and prints active screen.

16.2 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.

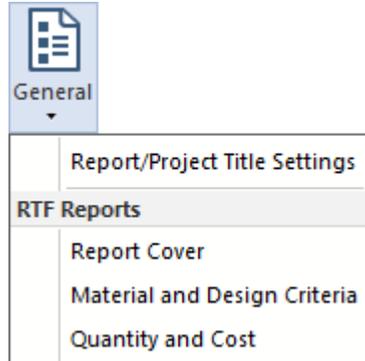
Settings 

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.

16.3 Single Default Reports Panel



16.3.1 General



Report/Project Title Settings

Set the General and Specific title and information lines.

Report Cover

Generates a .RTF file with report title settings.

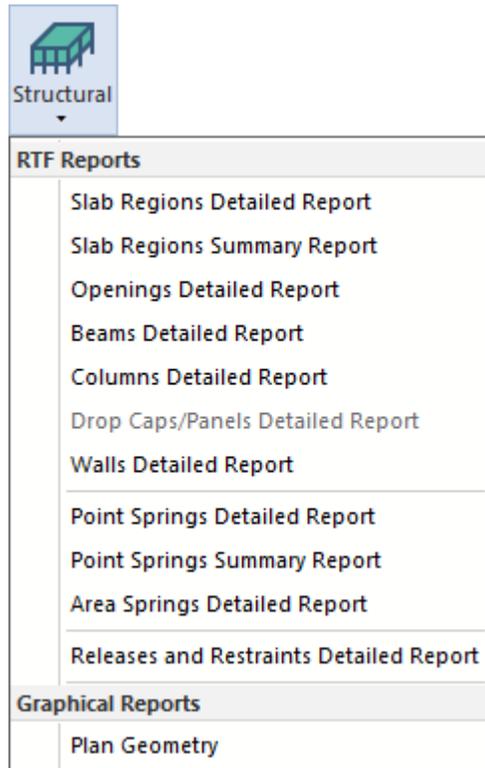
Material and Design Criteria

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).

16.3.2 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.

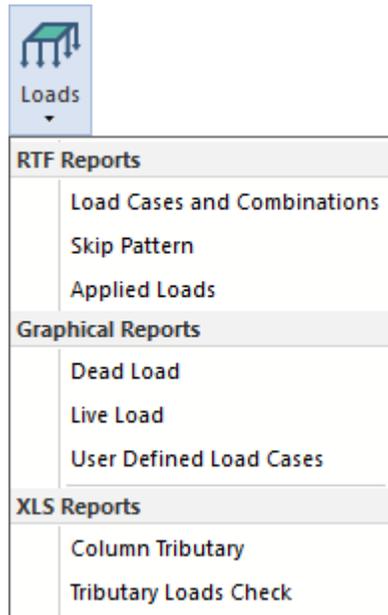
Releases and Restraints Detailed Report

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.

16.3.3 Loads



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.

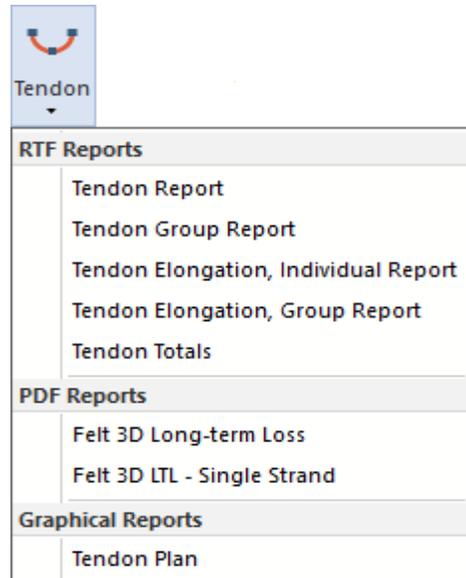
Column Tributary

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.

16.3.4 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.

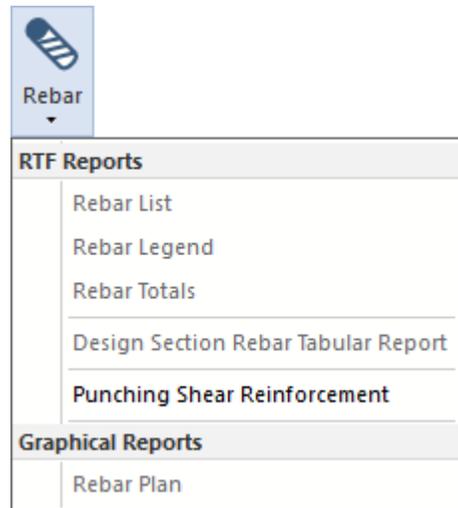
Felt 3D LTL - Single Strand

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.

16.3.5 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.

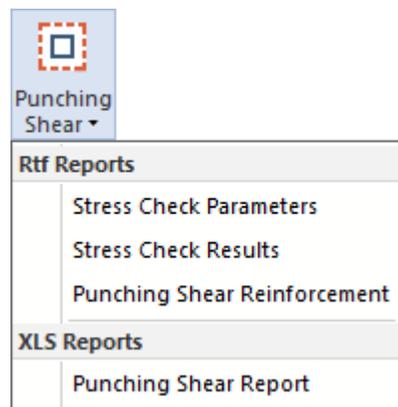
Punching Shear Reinforcement

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.

16.3.6 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.

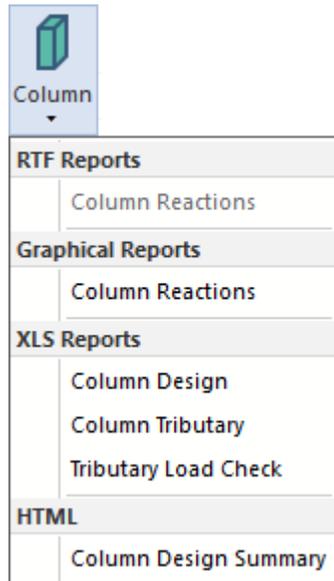
Punching Shear Reinforcement

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.

16.3.7 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.

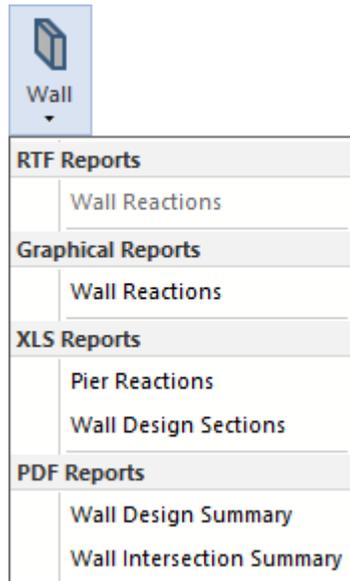
Tributary Load Check

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.

16.3.8 Wall



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.

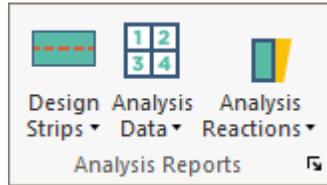
Wall Design Summary

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

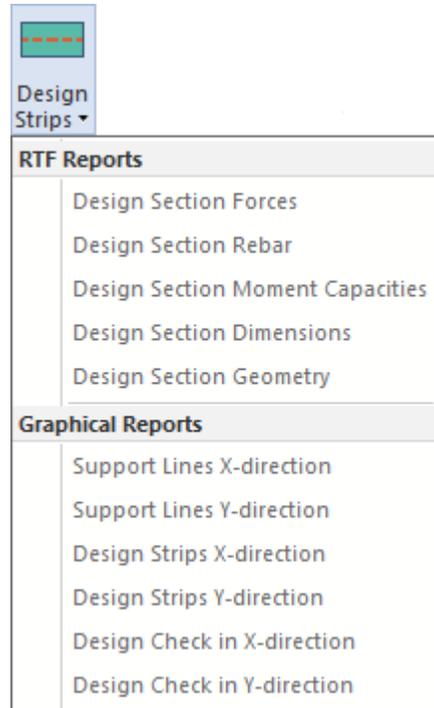
Wall Intersection Summary

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

16.4 Analysis Reports Panel



16.4.1 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.

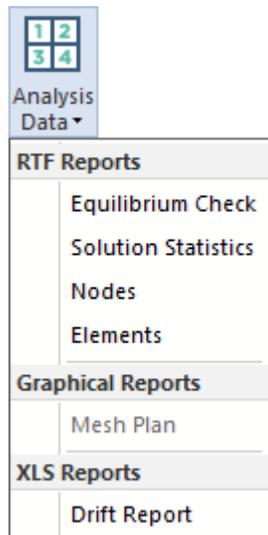
Design Check X-Direction Plan

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.

16.4.2 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.

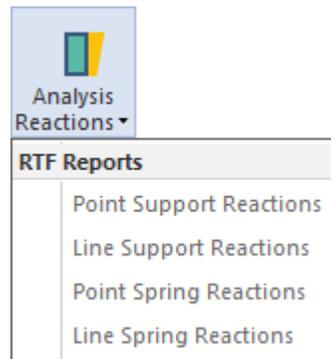
Analysis Mesh Plan

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. .

16.4.3 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.

16.5 Export DWG Panel



Export DWG/DXF

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