STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-PT/RC 20

USER MANUAL

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

LIST OF CONTENTS

1 Overview

ADAPT-PT/RC is the state-of-the-art industry standard computer program for the analysis and design of one-way or two-way conventional reinforced and post-tensioned floor systems and beams. It is based on a single-story frame analysis with upper and lower columns. For two-way floor systems, the equivalent frame modeling of ACI can be used as an option. ADAPT-PT/RC is a Windows-based program. ADAPT-PT/RC treats multi-span continuous slab/plate and beam frames with or without cantilevers.

ADAPT-PT/RC can be launched in a PT mode or RC mode for an applicable design. Saved files are interchangeable and can be opened in either mode once they are created. This dual functionality allows the user to easily navigate between reinforced concrete and post-tensioned design and determine the most suitable and economical design.

- In addition to drop caps, ADAPT-PT/RC allows drop panels of different sizes to be specified for different supports. Having a general frame analysis module for variable sections, ADAPT-PT/RC can accurately model a wide range of drop cap/panel geometries. Also, special modeling features implemented in the program facilitate the modeling of local thickening in the slab along the line of columns, generally referred to as slab bands.

- The slab/beam frame may be supported by walls, beams, or columns with different connection details such as clamped, rotational, free, and more.

- ADAPT-PT/RC fully incorporates the equivalent frame option as described in ACI-318 with no simplifications. In addition to the capability to handle the conventional configurations of column capitals and drop panels, the program allows the user to define a wide range of cross-sectional shapes. The software allows for the cross-section of the member to change along the length of a span, with abrupt steps, at the top, bottom, or both.

- Box girder bridge sections can be readily modeled as equivalent I-sections. For post-tensioned designs, ADAPT-PT/RC is well suited for a first design of box girder bridges, where an initial estimate of the amount and location of prestressing is sought to achieve given stress levels and design criteria.

- For flanged beams, you can either select the software’s built-in effective width computation, based on ACI-318 or EC2, or input a user-defined alternative effective width. The computations used are identical for RC and PT modes.

- The program recognizes and accounts for the difference between the effective width in bending and pure compression.

- Using the geometry of the structural model input by the user, the program calculates the self-weight loading of the structure for combination with other loads based on the selected concrete weight type. The user has the option to override the program defined material density. The calculated values of the self-weight are reported in the program's output.

- The program includes a slab recommendation feature for one- and two-way slab systems relative to recommended slab thicknesses without consideration of deflections per ACI318 for RC slab systems and recommendations given in "Post-Tensioned Buildings – Design and Construction," Bijan O. Aalami for PT slab systems.

- All the three systems of units—the American customary units (ft-lb), SI units (mm, Newton), and the MKS units (m, kg)—can be executed from the same copy of the software. Also, all the different codes are integrated into a single version.

- You can either edit the factory-set or define your own default values for input of data, while retaining the option to revert to factory-set default values if you so choose.

- Input to ADAPT-PT/RC is by means of a user-friendly, free-format, context-sensitive, full-screen input editor. The user may freely browse through, view, and modify the input prior to execution or saving of data.

- ADAPT-PT/RC includes an option for input of base (i.e. existing) reinforcement at any location along any span. The user can enter isolated or mesh flexural reinforcement and/or shear stirrups as base rebar. This functionality is ideal for investigative models. The program gives the option to report design capacity and demand/capacity ratios relative to base or calculated reinforcement. For PT mode only, prestressed reinforcement is considered as base reinforcement. When shear reinforcement is modeled, the one-way shear reinforcement report provides design checks considering base stirrups.

- The program contains a new Rebar Curtailment input module. This module can be invoked in post-processing of reinforcement output to customize and make real time modifications to longitudinal reinforcement results. User-defined or code-prescribed customization for longitudinal bar arrangements in beams and slabs for top and bottom reinforcement at cantilever, exterior and interior conditions can be input to reflect conditions on final drawings and documents. This tool can be active for top and bottom bar conditions or turned off. Curtailment templates allows the user to save specific bar arrangement configurations for crossover use among multiple users. The tool applies to both PT and RC modes.

- Reinforcement schedule output includes data transferred from the rebar curtailment input including the bar mark, bar size, bar diameter, quantity, bar length, shape codes, bar area and weight of bar. The schedule can be saved as .XLS format for ease of customization.

- The program includes an automatic rebar spacing check for minimum spacing between longitudinal bars. Design code minimum requirements are checked, and the program reports the number of layers required to fit the quantity of bars.

- For PT designs, a library of tendon profiles allows the user to select a suitable tendon layout for each case. Common profiles included are the simple parabola, reversed parabola, simple parabola with straight portions over the supports, harped, strait and extended reversed parabola tendons. Low relaxation and stress-relieved strands, as well as unbonded and grouted post-tensioning systems, are also supported by ADAPT-PT/RC.

- Uniform, line, partial, concentrated loads and moment, triangle, variable and trapezoidal loads may be specified in practically unlimited numbers and variations. Loading types are identical between PT and RC modes.

- ADAPT-PT/RC accounts for the shift in the location of the neutral axis of a member at changes in cross-section along the member length. Thus, the program can correctly handle steps along a member.

- ADAPT-PT/RC executes either automatically or interactively. This option is active when in PT mode and does not apply to RC mode. In its automatic mode, based on user's specifications, the software determines the required post-tensioning and the associated reinforcement. In its interactive mode, the program displays the calculated required post-tensioning on the screen to the user. You have the option to modify both the forces and the drapes during the program execution and recycle the selection until a satisfactory design is obtained, or you can input the conditions of an existing design. You can select the actual number and position of strands along a member. Also, in the interactive mode, you can graphically view the distribution of stresses, tendon profile and the required post-tensioning. This provides a good guide for you to achieve an optimum design.

- For PT mode only, stresses are computed and reported using the actual forces and drapes selected. This feature distinguishes ADAPT-PT/RC from simple programs, where a single-pass analysis is performed, in which the option of the user-initiated changes in post-tensioning are not reflected in the subsequent calculations. ADAPT-PT/RC has a multi-pass processor. It updates all the design values based on changes made in the tendon profile and force before it concludes its report of design values.

- For PT mode only, serviceability design of the slab/beam is achieved through detailed stress control. For both modes, the program carries out comprehensive deflection calculations for both instantaneous and long-term deflections. Where the cracking limits of concrete are exceeded, a cracked section deflection estimate is carried out using an equivalent moment of inertia.

- A thorough strength analysis and design is conducted to determine any reinforcement that may be necessary to meet the ultimate strength limit conditions. In PT mode, this calculated reinforcement is in addition to the prestressing steel. For RC mode, the calculated reinforcement is the total amount of reinforcement necessary to meet the demand actions for the entire strip width. The user specifies allocation percentages (%) for column and middle strips which result in reported reinforcement for each strip type.

- Other code requirements for reinforcement, such as the minimum requirements of the building codes, are also checked and a listing of the reinforcement based on different criteria is reported.

- The program recognizes mixed/hybrid models that include design sections that are not intersected by post-tensioning. The program can determine proper amounts of reinforcement relative to differentiating requirements of RC and PT design sections.

- Bar sizes and lengths are selected and reported both in a graphical and tabulator format, ready to be entered on the structural drawings.

- Base (user-defined) reinforcement can be modeled for flexural reinforcement in beams and slabs and for shear reinforcement for beams and one-way slabs. The program considers base reinforcement in determining required reinforcement necessary to meet minimum and strength level demand actions.

- The punching shear option checks the adequacy of the column caps as well as the immediate slab/drop panel region beyond the cap and provides punching shear reinforcements if required. For one-way slabs and beams, a one-way shear analysis is carried out. Shear reinforcement is calculated, and the stirrup requirements are given.

- For PT designs, ADAPT-PT/RC can handle both the effective force and the variable force methods.

- For PT mode the variable force method ADAPT-PT/RC calculates the change of tendon force along its length and can use the force at each location along the length of a member to perform a code check for that location. In addition to the immediate losses due to friction, and seating loss, and at user’s option, the software accounts for the long-term stress losses along the length of the structure. Since long-term losses for grouted tendons are functions of local strain, an iterative non-linear capability is built into the program. The non-linearity in the solution is with increments of load.

- For PT mode when the effective force method is used, the force along each tendon is assumed constant. In this case, the design is based on a non-iterative linear solution in terms of the effective force.

- For PT mode another execution option is the computation only of immediate losses resulting from the friction and seating of strands. The long-term losses in prestressing are then affected through a user defined lump sum stress loss.

- The geometry of the structural model can be viewed on the screen in a three-dimensional space. The capability to rotate, pan, zoom and view the model allows the user to examine the structure in detail prior to the execution of the program. Errors in data entry or modeling are readily detected when user's input is displayed on the computer screen. Hard copies of the graphical display of the structural model can be readily obtained.

- ADAPT-PT/RC uses the detailed scheme throughout its operation. This scheme is based on 1/20th point values along each span. However, to retain the simplicity of presentation of the report, in addition to the optional 1/20th point reports, a summary of the solution is compiled for the left, center and right of each span.

- In addition to graphical reports, the outcome of the analysis and design is composed into a clear text file that can be viewed, edited, and printed by you. The text file can be produced in Word (.RTF) or as an Excel (.XLS) file. The content and extent of the report can be controlled by you through a user-friendly menu.

- It is also possible to generate a one-page graphical summary report that extracts and incorporates all important design information in an easy-to-interpret format. The report may also be exported as a DXF file for incorporation into construction documents. The DXF feature includes options to customize the name and scale of the drawing file. For PT mode, up to three distinct tendon profiles can be shown, with respect to the datum line or the soffit of the structure if they differ. Tendon heights can be exported in the format of the tendon CGS heights or tendon support heights. Further details of exporting to DXF include the customization of visualization of the tendon profile, anchor points and control points, as well displaying information at 20th interval points or user-specified intervals. The visualization of longitudinal and one-way shear reinforcement can also be generated through the DXF import option. This applies to both RC and PT modes of operation. The user has the option to overlay the reinforcement graphical output in one drawing or create separate drawings for each.

- The graphical display option of ADAPT-PT/RC provides a vivid exposition of the distribution of calculated values along the entire structure or for its selected members. The displays relative to RC and PT mode include moments, shears, deflections, and reinforcement required/provided. The displays relative to PT mode only includes stresses, post-tensioning required, post-tensioning provided and tendon profile. Each graph may be printed or exported as a .bmp or a metafile.

- ADAPT-PT/RC input data is stored in a single file with the .ADB extension. However, the program is also backward compatible with input generated by earlier Windows versions of the program. The program best operates in a local machine environment, but the user can also execute models to network drive locations so long as the drive is mapped to the local machine with a pre-designated drive. The program is compatible with 32- and 64-bit versions of Windows 7, Windows 8 and Windows 10.

- ADAPT-PT/RC is integrated into the ADAPT-Builder software suite. Structural models generated using the Modeler module of the Builder suite can automatically be transferred to ADAPT-PT/RC for analysis and design. This capability provides a seamless link between the Finite Element Method of ADAPT-Builder and ADAPT-PT/RC.

2 Sign Convention

The following is the sign convention used in ADAPT-PT/RC.

Applied Loads
Downward loads and counterclockwise moments are considered positive (**Fig. 2.1 (a)**).

Span Actions
Counterclockwise shear is considered positive.

Bending moment is considered positive if it causes tension at the bottom (**Fig. 2.1 (b))**.

Column actions
Counterclockwise column moments are considered positive (**Fig. 2.1 (d))**.

Positive direction of frame as well as definition of right and left tributary region is defined as shown in **Figure 2.1 (c)**.

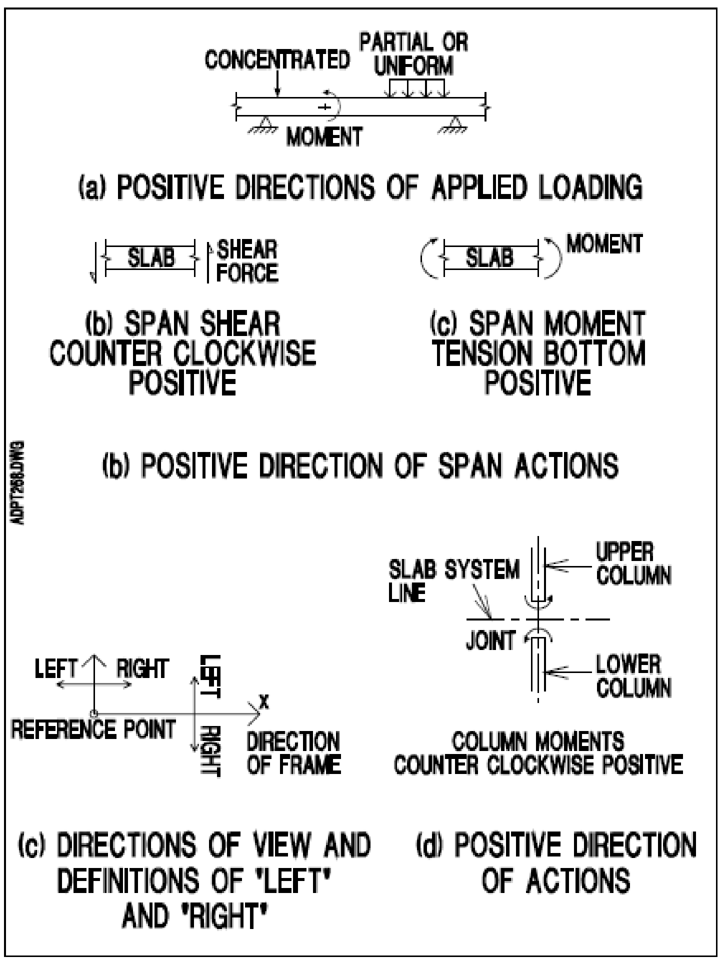


Figure 2-1: Sign Convention

3 Workspace Overview

This chapter describes Graphical User Interface (GUI) for the main program screen and data input module of ADAPT-PT/RC program.

All program functions, including data entry and program execution are accessed through the Main Program window. The data entry is done through separate program modules called ADAPT-PT or ADAPT-RC Input.

When ADAPT-PT/RC is opened, the user has the option of selecting the initial design type by selecting the ADAPT-RC or ADAPT-PT button.

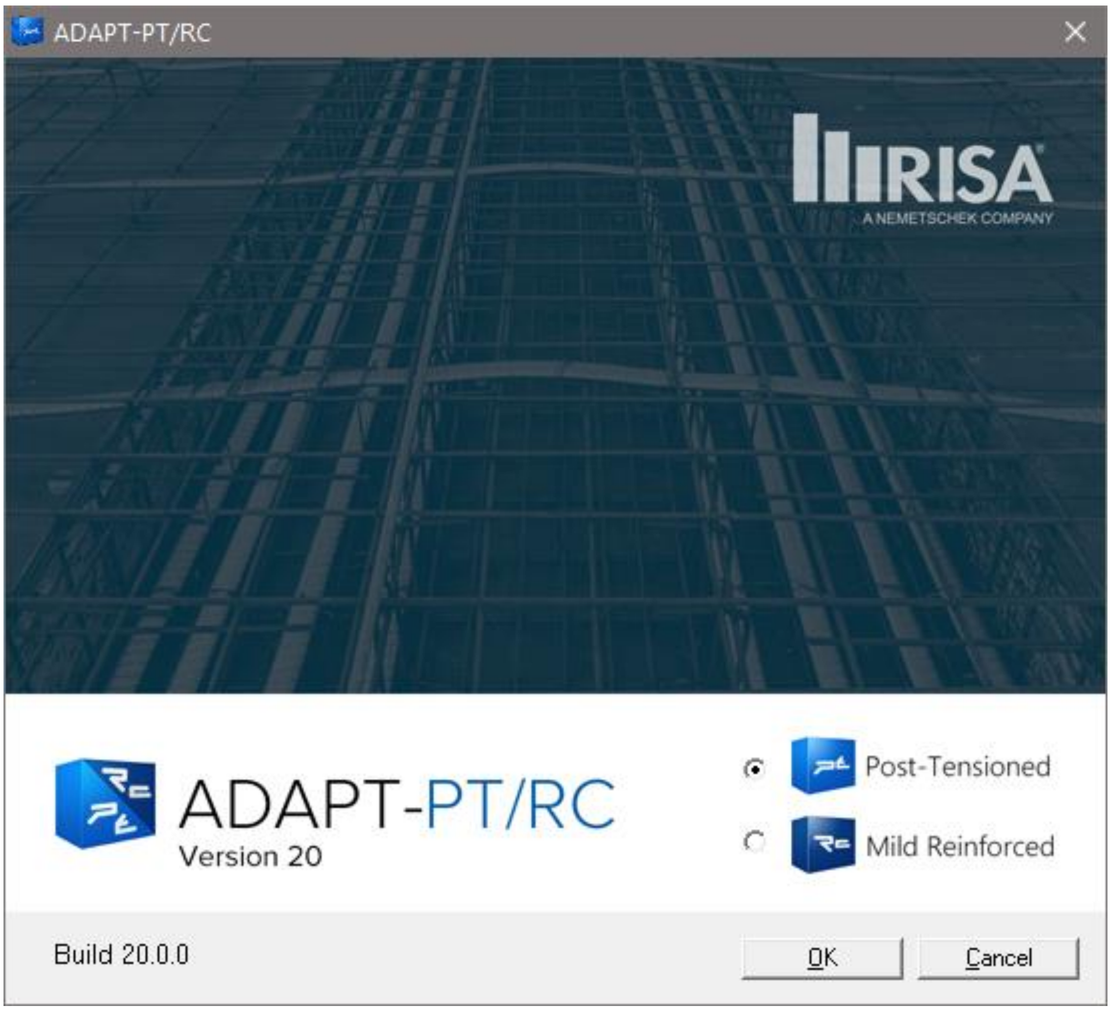


Figure 3.1

At any time before creating a model or after a design selection and model have been made, the user can switch to the other design mode by selecting Actions>Convert Project to ADAPT-RC or ADAPT-PT. If a model has not yet been generated or saved, the user can also go to Options>Program Mode and make a selection of ADAPT-RC or

ADAPT-PT. Once a model has been saved, this option is grayed out, inactive and shows the current mode of operation.

3.1 The Main Program Window

Figure 3.1-1 shows the main ADAPT-PT/RC program screen as it appears once a project has been opened.

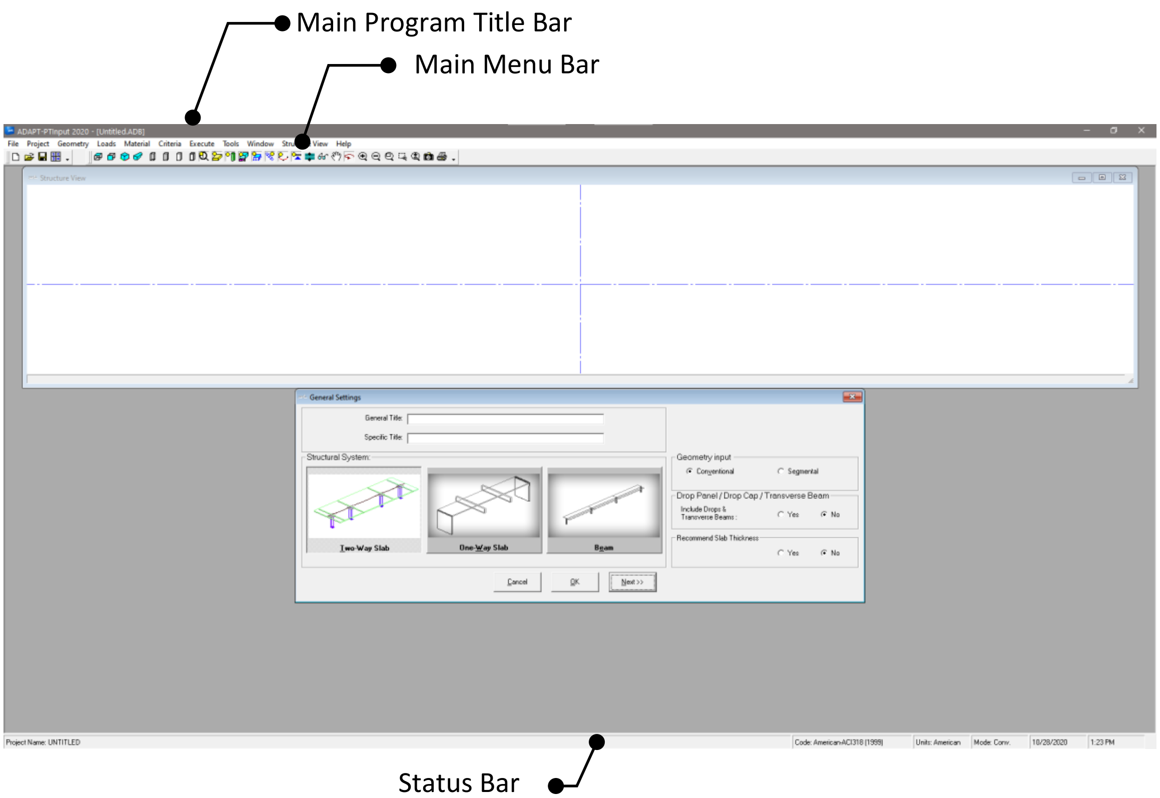


Figure 3.1-1 Main Program Window

Main Program Title Bar. Contains the program name and name and location of the opened file.

Main Menu Bar. Lists all available menus in main program window. Menu options will be grayed out when they are not applicable. For example, if there is no project open, the Save As, Close and Print options on the File menu will be grayed out.

Main Toolbar. Main Toolbar contains all available tools in the main program window. The tools will not be available when they are not applicable. For example:

- If you open the **ADAPT-PT/RC** program the available tools are:

- If you open an existing file, the main toolbar contains the following options:

Status Bar. Status bar gives you information about project name, selected design code, units, execution mode, key type, current date, and time. To turn Status Bar on or off go to **View→Status Bar**.

Hint Window. Hint window gives you information on how to proceed if you want to edit, execute, or view your structure. To close the Hint Window, right click anywhere on the screen.

3.2 Main Program Window Menu Items and Corresponding Tools

All options that can be accessed by the main program menus are listed below. For the commands that might be activated using the toolbar, the appropriate icon is displayed next to the feature.

3.2.1 File Menu

The File Menu operations are:

- **New** - Starts a new project. The project will be initiated in the active design type that was selected in the Start screen.
- **Open** - Opens an existing project. The project will be opened for the design type which it was last saved as.
- **Save As** - Saves both the input files and the results report file under a user-specified filename as the active design type (RC or PT).
- **Save As Default Values -** Once you save data using this option, the program will open all the new projects in the future using the values you saved. To change the default values, open a new file, modify the parameters of your choice, and re-save it using Save As Default Values.
- **Close -** Closes the currently open project.
- **Delete Intermediate Files** - Deletes all intermediate calculation files from the current project directory.

- **Export Graph** - Allows the user to export the currently active graph as either a bitmap (.BMP) or a Windows metafile (.WMF).
- **Print** - Prints the currently active report or graph window.
- **Page/ Print Setup** - Sets the paper size, report margins, paper orientation and printer.
- **Exit -** Closes all windows and exits the program.

3.2.2 Action Menu

The Action menu operations are:

- **Enter/Edit Data** - Opens the data input editor.
- **Execute Analysis -** Executes the program calculations.
- **Recycle Window -** Opens the recycling window. Used when re-running a project to adjust the post-tensioning force or profile.
- **Convert project to RC or PT -** Converts the active project type to the alternate design type of RC or PT. This activates necessary input windows required for the specific design type.

3.2.3 View Menu

The View menu operations are:

- **Status Bar -** Turns the status bar at the bottom of the main window on and off.
- **Graphs -** The Graphs menu item opens a submenu, which allows any or all the Results Graphs to be viewed. The Show Graphs button on the main toolbar displays all graphs.
- **RC or PT Summary -** Allows you to see result graphs for moments, forces, reinforcement, and stresses (PT mode only) for each load combination and envelope. Also, it displays the report summary sheet and DXF export option.

3.2.4 Options Menu

The Options menu operations are:

- **System of Units -** Allows the user to select the default units (American, SI, MKS).

- **Design Code -** Allows the user to select the default code.
- **Remember Printer Selection -** If this option is checked, the program uses the latest printer settings for all future runs, regardless of the default printer selected in the Windows settings.
- **Report Setup** - Opens a Report Generator window where the report contents may be set. 
- **Compress Report Bitmaps** - Compresses the size of bitmap files created after executing a model. This significantly reduces the .BMP file and program solution folder size.
- **Graph Properties** - Configures the graphs generated by the program. Options include whether to include X and Y gridlines, min/max data points and a legend.
- **Spreadsheet Options -** Configures the action of the ENTER key in the data entry spreadsheets. The key may be set to move the cursor right, down or stay in the same field.

3.2.5 Window Menu

This menu lists which of the windows are open. The windows may be stacked vertically, horizontally, or the windows may be cascaded.

3.2.6 Help Menu

The Help menu operations are:

- **Documentation** - Opens folder with product documentation.
- **About ADAPT -** Company address, phone, fax, and e-mail information.
- **About ADAPT-PT/RC -** Program information such as version.
- **Support Programs -** Information on how to obtain program support.
- **Disclaimer -** Defines responsibility of the software user.
- **Calculator -** Invokes the standard windows calculation.

3.3 ADAPT-PT/RC Input screen

Project input data is created/edited through a separate program module called ADAPT-PT or ADAPT-RC Input, depending on what the active mode is. The input editor is used both to enter new projects and edit existing files.

- To enter a new project either click **New** on the **File** menu or click the **New** icon, , on the Main Menu bar.
- To edit existing file either click on **Enter/Edit Data** on the **Actions** menu on the Input Menu bar, or click the **Edit Data** button, , on the Input Toolbar.

Figure 3.3-1 shows the ADAPT-PT/RC Input screen as it appears once the input editor is opened. Note that the main input screen is similar for both PT and RC modes.

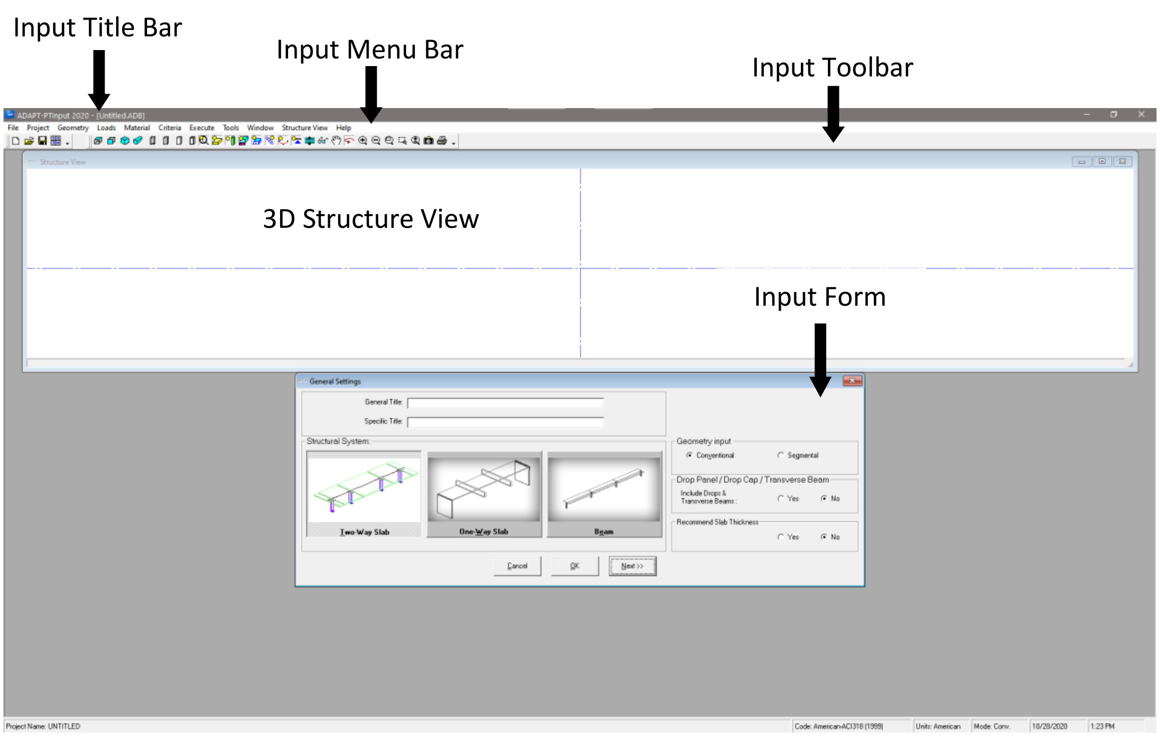


Figure 3.3-1 ADAPT-PT/RC Input Screen

PT/RC Input Title Bar. Contains program name, and name and location of the opened file.

PT/RC Input Menu Bar. Menu bar lists all available menus in ADAPT-PT/RC Input.

PT/RC Input Toolbar. This toolbar contains all available tools in the ADAPT-PT Input.

Input Form. Each input form is the dialog box that will prompt you to select options or enter required information. When first entering data for a project, you would typically go through the screens in order by clicking on the Next and Back buttons or pressing ALT-N and ALT-B. In subsequent editing, specific screens may be selected from the PT/RC Input menu.

If the input form contains a table (**Fig. 3.3-2**) the right mouse click will give you the following options:

- Insert a new line.
- Delete line.
- Copy selected lines.
- Paste lines.

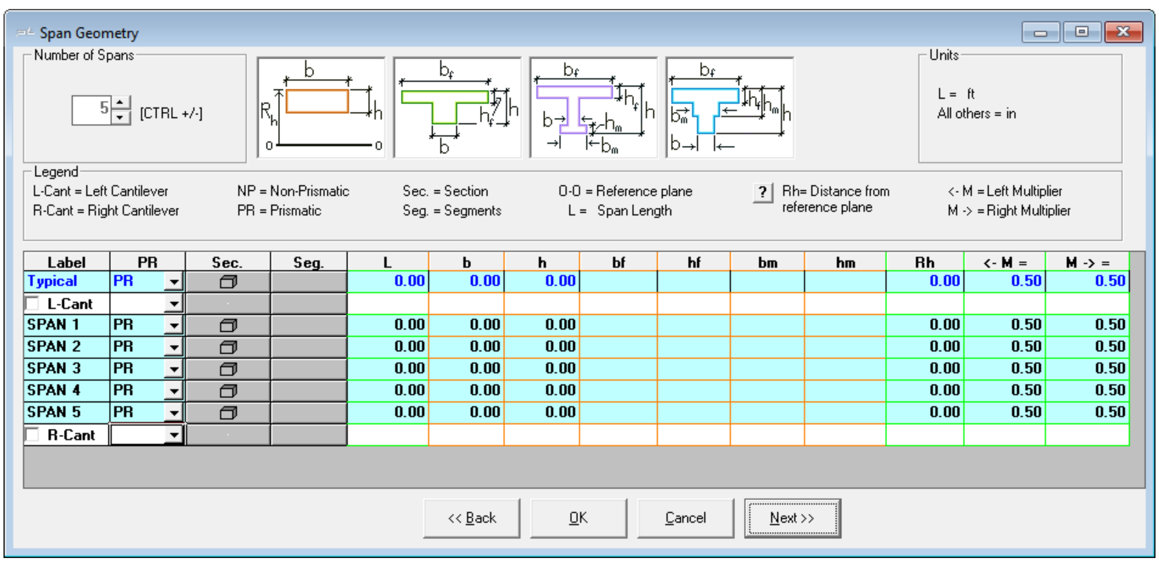


Figure 3.3-2 Span Geometry Input Screen

Each table contains a Typical row for fast input of data. The typical input row (top row) can be used if several rows in a column have same data. To enter typical values, type the value into the appropriate cell in the top row and then press ENTER. The typical value will be copied to all rows in the column. The value of any field initialized in this manner can be subsequently changed, as necessary. Data can be entered in the typical row at random; it is not necessary to enter values in all fields of the typical row.

3D Structure View. Structure View window allows you to view the structure, loads, tendons, and rebar in 3D as you enter the data. You can also display the properties of each component if you bring the mouse pointer over it and right click. You can change the display in 3D window with View Tools toolbar.

3.4 ADAPT PT/RC-Input Menu Items and Tools

All options that can be accessed by the PT and RC Input menus are listed below. Only the set of input forms will appear to the user for the active design mode.

3.4.1 File Menu

The menu items have the same function as File menu in the main program screen.

3.4.2 Project Menu

This menu enables you to specify the project title, structural system and select analysis and design options through General Settings and Design settings input forms.

3.4.3 Geometry Menu

This menu enables you to access input forms that you use to define geometry of a model. The input forms included in this menu are:

- Span Geometry
- Effective Flange Width
- Drop Cap/Transverse Beam
- Drop Panel
- Support Geometry
- Boundary Conditions

3.4.4 Loads Menu

This menu opens the Loads input form where you can specify the loads.

3.4.5 Material Menu

This menu enables you to access input forms where you can specify material properties for:

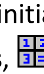
- Concrete
- Reinforcement
- Post-Tensioning (PT mode only)

3.4.6 Criteria Menu

Criteria menu contains all input screens that you use to specify project criteria. The Criteria menu input forms are:

- Base Non-Prestressed Reinforcement
- Allowable Stresses (PT mode only)
- Post-Tensioning (PT mode only)
- Calculation Options (PT mode only)
- Tendon Profile (PT mode only)
- Minimum Covers
- Minimum Bar Extensions
- Rebar Curtailment
- Load Combinations
- Design Code

3.4.7 Execute Menu

If you click on the Execute menu the program will initiate an analysis. The corresponding tool is **Save & Execute Analysis**, , in the Common toolbar.

3.4.8 Tools Menu

The Tools menu allows you to convert units and change color settings. The menu items are:

- Convert units.
- Color selection.

3.4.9 Window Menu

This menu lists the graphical windows that are currently active. The windows may be stacked vertically for scrolling, or the windows may be cascaded.

3.4.10 Structure View Menu

If you click on this menu it will open the 3D structure view.

3.4.11 View Tools Toolbar

This toolbar contains tools for selecting the entities that you want to be visible on the screen, as well as for creating rendered or shaded views of structural models. It is used to zoom in or out, pan, increase scale, or create a screen shot and print.

 **Plan View.** It will show structure in plan view.

 **Elevation View.** It will show structure in elevation view.

 **Isometric View.** It will show structure in 3D isometric view.

 **Free Rotate Perspective View.**

 **Wire Frame.**

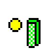
 **Transparent Shader.**

 **Solid Shader.**

 **Outline Shader.**

 **Show/Hide Gridlines**. Displays or hides gridlines and reference line.

 **Show/Hide Spans**. Displays or hides spans of the structural model.

 **Show/Hide Supports**. Displays or hides columns and walls of the structural model.

 **Show/Hide Drops/Beam**. Displays or hides drop caps and beams of the structural model.

Show/Hide Loads. Displays or hides all loads that have been identified as "visible" in the Select/Set View Items window.

Show/Hide Rebar. Displays or hides the entire user defined (base) reinforcement.

Show/Hide Tendons. Displays or hides the entire tendons of the structural model.

Show/Hide Fixity. Displays or hides all fixity symbols of the supports.

Increase Scale Factor. This tool provides you with the option of distorting the dimension of the model in one or more direction, in order to obtain a better view of its details, such as magnifying the profile of a tendon within a slab thickness.

Select/Set View Items (**Fig. 3.4-1**). This button is used to set the display of the project items on the screen. Depending on which boxes you select, you can display additional information about each entity.

Dynamic Pan.

Rotate.

Zoom In.

Zoom Out.

Zoom Extents.

Dynamic Zoom.

Screen Capture. Takes a screen shot of the Structure View window.

Print. Prints currently active window.

Add or Remove Buttons. This option is used to add or remove buttons from the Input Toolbar.

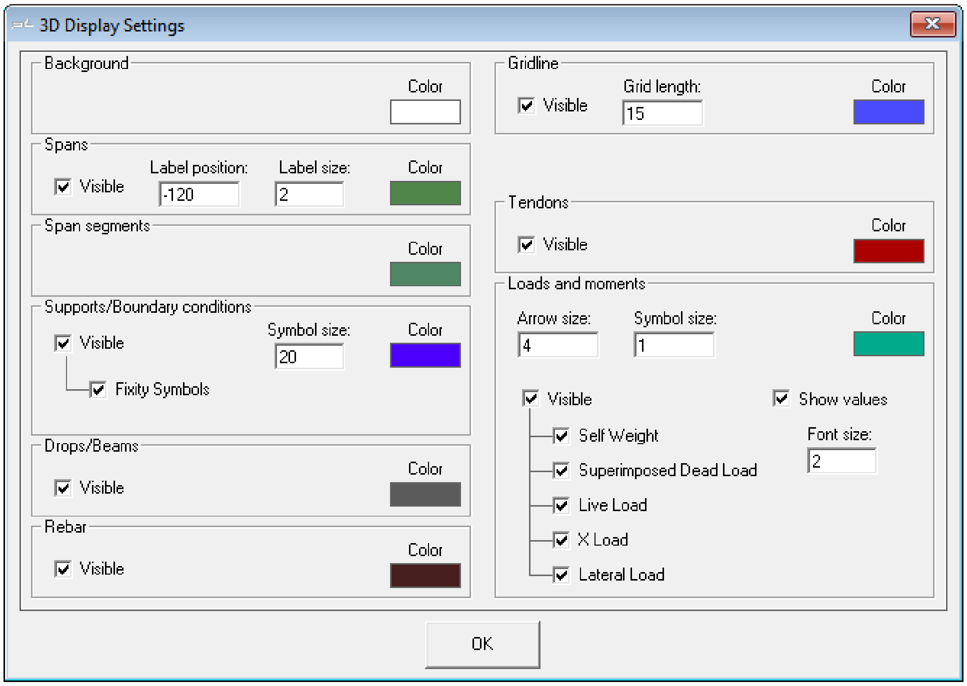


Figure 3.4-1 3D Display Settings Dialog Box

4 Basic Program Operations

This chapter explains the basic program operations.

4.1 Start a New project

When starting a new project, you should specify design code and system of units.

To start a new project:

- Double-click on the **ADAPT-PT/RC** icon of your desktop to open the program.
- Select **File→New**, or click on the **New Input Session** icon, .

4.2 Open an Existing Project

To open an existing project:

- In the Main program window select **File→Open**, or click on the **Open Project** button, .
- Select the .ADB file that you want to open. Note that the program will open in the design mode which was exited from when the project was last saved.

Note: The four most recently accessed files will be recorded in File menu for easy access.

4.3 Save Input Data

To save the input data and execute the analysis, either select Execute from the Input Editor menu or click on the Save & Execute Analysis button, .

- If you are entering a new project, you will be prompted for a file name and directory in which to save the file. Once the file is saved, the program will automatically execute the analysis.
- If you opened an existing project, it will be saved to the same directory, under the same filename. The program will then automatically execute the analysis.

To save the input data and return to the Main Program window, select either **Save** or **Save As** from the **Input Editor File** menu or select the **Save** icon, , on the **Input Editor Toolbar**.

- If you have opened an existing file, **Save** will save the file under the same name, in the same directory.
- **Save As** will allow you to change the file name and/or directory.

Once the file is saved, select **Exit** to return to the Main Program window.

4.4 Save Input Data as Default

Note that it is often not necessary to go through all the input screens, even when entering a new project. Much of the information on the Materials and Criteria input screens will be the same on many projects. The program is set up with ADAPT defaults for all screens. The program includes separate default settings depending on which design mode the program is opened in.

To change the default values installed with ADAPT-PTRC:

- Open the file with the data you would like to save as default and use in other files.
- Choose **File→Save as Default**. The program will save the file as "default" ADB file. Once you saved data using this option, the program will open all the future new projects using the values in the data you saved. To change the default values, open a new file, and re-save it using Save As Default Values.

4.5 Select System of Units

ADAPT PT/RC features three systems of units: SI, MKS and American (known as Imperial).

To select system of units for the new project:

- Double-click on the **ADAPT-PT/RC** icon on your desktop to open the main program window.
- Select **Options→ System of Units**.
- Check one of the options: SI, MKS or American. The program will automatically close the Options menu.

Note: If a file which has been previously created is opened, and the System of Units option is selected, the program will gray out the options and show a check mark by the system in use for that model.

4.6 Convert System of Units

To convert system of units:

- Open an existing project.
- Click on **Edit Data** icon, , to open **Input Editor**.
- Select **Tools→Convert Units**. The **Convert-Units** dialog box opens (**Fig. 4.6-1**).
- Select **New Unit** and click **Convert**.

Note: The **Convert Unit** option is available only if the design code is ACI.

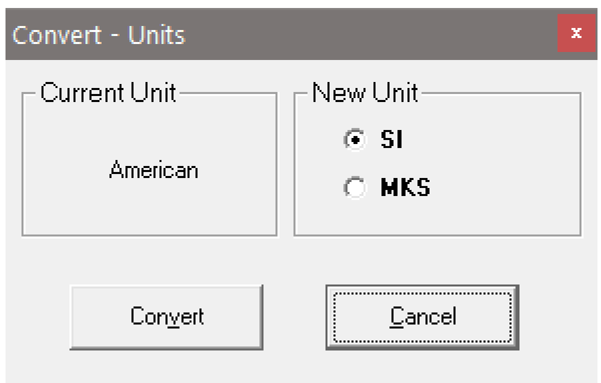


Figure 4.6-1 Convert Units Dialog Box

4.7 Program mode

To select the design mode to work in (PT or RC mode):

- Select **Options→ Program Mode**.
- Check one of the options: **ADAPT-PT** or **ADAPT-RC**.

Note: If a file which has been previously created is opened, and the Program Mode option is selected, the program will gray out the options and show a check mark by the active mode. To switch modes, the user must go to **Actions→Convert Project** to ADAPT-RC or ADAPT-PT.

5 Structural Modeling Overview

During the structural modeling step, the user defines the basic analysis and design parameters, i.e. the structural system (beam, one-way or two-way slab), the span lengths, cross-sectional geometries, tributary widths, supports and boundary conditions. The user also defines the loading, material properties, base (existing) reinforcement, allowable stresses (PT mode only), post-tensioning criteria (PT mode only), calculation method and tendon profile (PT mode only), reinforcement covers and load combinations. This is the most critical stage of the modeling process. The user's experience and engineering judgment play a major role in the selection of suitable design parameters. This stage of the modeling should be performed, or at least reviewed, by a senior engineer. A structure that is not modeled correctly is not likely to yield reasonable results using ADAPT-PT/RC or any other software.

Data entry in ADAPT-PT/RC is independent from the execution of the analysis. Data for a particular project may be entered at any time for later execution. Data is entered through ADAPT-PT and RC Input screens described in Section 4.2.

5.1 Project Information

Project information includes specification of general information and analysis and design options.

5.1.1 Specify General Project Information

The **General Settings** window automatically opens when a new project is started, or an existing project is opened. This screen is also available through menu option **Project→General Settings**.

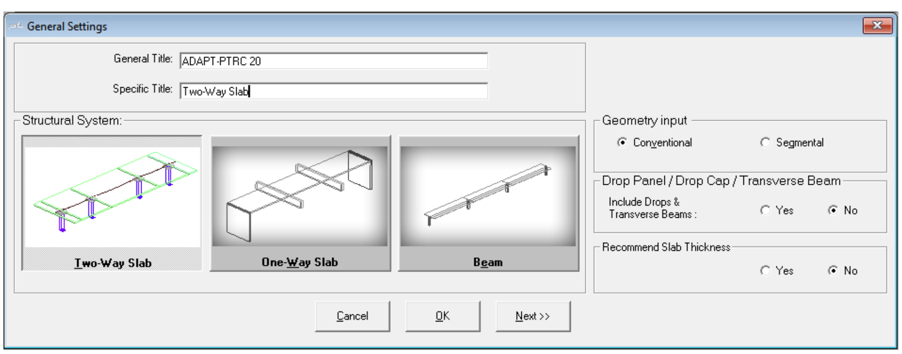


Figure 5.1-1 General Settings Input Screen

Input information as follows:

1. Type in General and Specific title. Note that the General title and Specific title of the project will appear at the cover page of a report and in the header of each page of a report.

2. Select Geometry input option.

   - If you model spans with uniform geometry where the tributary width, section type and/or section depth do not change within a span, select **Conventional** input. This will also give you an option to include drop caps, drop panels, and/or transverse beams.

   - If you model spans with non-uniform geometry within a span, select **Segmental** input.

3. Select a type of a Structural System.

   - If you select Two-Way slab you will have an option to include drop caps, drop panels and/or transverse beams.

   - If you select One-Way Slab you will have an option to include transverse beams only.

   - When a Two- or One-Way Slab is selected, you will have the option to have the program recommend a slab thickness in the Geometry input window. When the span length is entered, the program will auto-fill the thickness 'h' value. The value can be overwritten. Values given by the program are taken from ACI318 and PTI recommend slab thicknesses for geometrical considerations and span locations.

   - If you select Beam you will have an option to include an effective flange width in the calculations. Note that if effective flange with is considered, the options for 'I' and Extended 'T' sections in the geometry input are grayed out. See **Section 5.2.1.**

4. Click **Next**. This will save input data and open next input screen **Criteria – Design Code** where you can select design code.

5.1.2 Specify Analysis and Design Options

You can select various analysis and design options through the **Design Settings** dialog box (**Fig. 5.1-2**). Note that some settings are applicable only to PT mode.

To specify analysis and design options:

1. Click on **Project→Design Settings**. The **Design Settings** input screen opens.

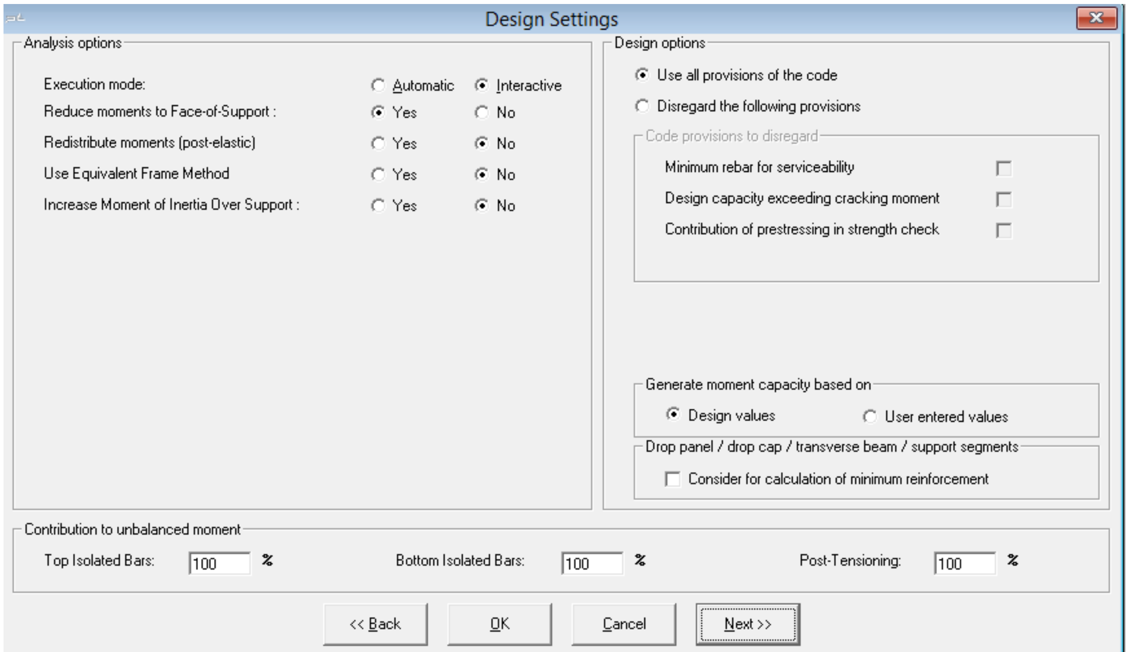


Figure 5.1-2 Design Setting Input Screen – PT Mode

2. Select Analysis options

- **Automatic (PT mode only)**. In the automatic mode, the program attempts to select a post-tensioning force and profile based on the parameters specified by the user. If a solution is possible, the program will complete all calculations and return to the Main Program window. The results can then be viewed and/or printed. If a satisfactory solution is not possible, the program will display a message box, which describes the problem and will switch to the interactive mode. The user can then decide whether it is possible to overwrite the original design criteria and continue with the design.
- **Interactive (PT mode only)**. The interactive mode gives the user an opportunity to optimize the design by adjusting the tendon forces and tendon drapes in each span.
- If you select "Yes" for **Moment reduced to face-of-support**, the calculated moment at the support centerline will be adjusted to face-of-support and used in design.
- If you select "Yes" for **Redistribute moments (post-elastic)**, the program will perform redistribution of moments and readjust

elastic moments based on the provisions of the selected design code.

- If you select “Yes” for **Use Equivalent Frame Method**, the program models the structure using the Equivalent Frame Method (EFM). This option is available only for two-way systems.
- If you select “Yes” for **Increase Moment of Inertia Over Supports,** the program will internally model the structure with increased moment of inertia over supports. This option affects the relative stiffness of the beam and column members. It also, in turn, affects the relative distribution of the moments and may affect the amount of post-tensioning required. The option is available for one-way systems and two-way systems where the Equivalent Frame Method is not used.

3. Select Design options

- If you select **Use all provisions of the code** the program will consider all provisions of the selected design code including calculation of minimum rebar for serviceability, check for cracking capacity (PT mode only) and add reinforcement if needed, considering the contribution of post-tensioning in strength check (PT mode only).
- If you select **Disregard the following provisions** you will have an option to choose which of the following provisions you would like to disregard in design**:**
  - If you choose to disregard **Minimum rebar for serviceability** the program will not report minimum rebar. This applies to PT and RC modes.
  - If you choose to disregard **Design capacity exceeding cracking moment** the program will not report the rebar due to design capacity exceeding cracking moment capacity. This applies to PT mode only.
  - If you choose to disregard **Contribution of prestressing in strength check** the program will not consider post-tensioning in strength check. This scenario applies when a user adds tendons to a project for deflection or crack control. This applies to PT mode only.
  - If you choose to **Auto Increase Allowable Tension Stress,** the program will increase the allowable tensile stress per code requirements for BS, Indian and Hong

Kong codes when the calculated tensile stress exceeds the user-defined allowable tensile stress. If this option is not selected, the program will optimize a design based on the user-defined allowable stress without the automatic increase.

- If data is being entered for a beam you will have an option to **Include the (DL + 25% LL) case of UBC.** This is a UBC (Uniform Building Code) requirement used to determine the amount of mild steel reinforcement required. If this is answered "Yes," the ratio of reduced live load to actual live load must be entered. This option allows a reduced live load to be used for the post-tensioning if so desired but provides the full live load for the 25% UBC design loading. Live load reduction is optional; if the live load entered on the Loading screen was not reduced, the ratio of reduced to actual live load would be 1. Note that the (DL +25%LL) provision is not required by ACI-318, nor is it included in the IBC-2000 (International Building Code). This applies to PT mode only.
- If you select the checkbox for **Consider for calculation of minimum moment**, the program will use the drop panel / drop cap / transverse beam / support segment for the calculation of the area of the section at support when calculating minimum reinforcement over supports for two-way slabs or one-way slabs (with Transverse Beams).
- The program calculates and reports the moment capacity in both graphical and tabular format, based on the user selection.
    - If you select **Design values**, the moment capacity will be calculated using the rebar and PT required (PT mode only) for the design, including user defined base reinforcement.
    - If you select **User entered values**, the moment capacity will be calculated only with user entered rebar (base rebar) and PT (PT mode only).

The moment capacity for the 1/20th points can be observed in the appropriate tabular report (Tabular Report-Detailed/ 34-Demand Moment and Moment capacity). The moment capacity graph is plotted together with the moment envelope (**Fig. 5.1-3**). To see the graph Open BuilderSum, select **Envelope** from the drop-down list and click on Moment Diagram. Click on the Design Moment Capacity frame and select positive or negative moment capacities.

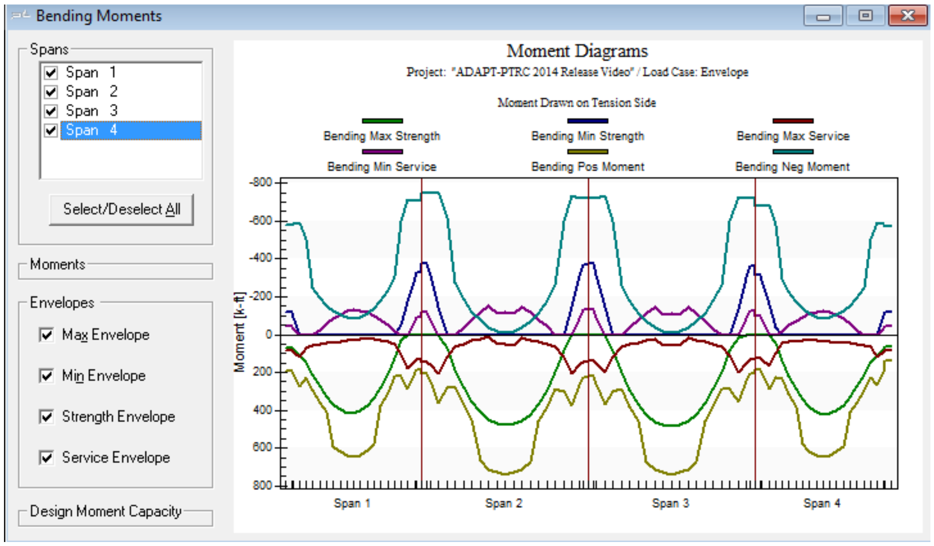


Figure 5.1-3

4. Specify the percentage of top and bottom reinforcement and post-tensioning (PT mode only) that will participate in resisting unbalanced moment. This option is available only for two-way systems.

5. Click **Next**. This will save input data and open a new input screen Span geometry where you can specify geometry of spans.

5.2 Geometry

The geometry of the problem is defined via a series of input screens that can be accessed through the Geometry menu. The screens will vary depending on which structural system has been specified. There are three basic screens: Span Geometry, Support-Geometry, and Support-Boundary conditions. Additional screens are used to enter effective flange widths, segmental data, drop caps, drop panels and transverse beams.

5.2.1 Specify Span Geometry

The span geometry can be modeled as prismatic (uniform) or non-prismatic (non-uniform).

- The geometry without changes in cross-section along the span excluding geometry of drop cap, drop panels or transverse beams is called prismatic (uniform) geometry.
- Geometry of a cross-section that changes along the span is called non-prismatic (non-uniform) geometry. You will have to

model span geometry as non-uniform if at least one span is not uniform.

5.2.1.1 Prismatic (Uniform) Spans

The **Span Geometry** screen is used to enter the cross-sectional geometry of the slab or beam at mid-span **(Fig. 5.2-1)**.

To input data for uniform spans, do the following:

1. Open Span Geometry.
2. To set the number of spans, use **CTRL+/-** or click on the up/down arrow at the left of the screen. If there are cantilevers on the right and/or left ends of the frame, add them by clicking on the appropriate check box. This will activate the input fields for the corresponding cantilever. Spans other than cantilevers can include overwritten with customized names.
3. Select section type by clicking on the button in the **Sec.** column. Section type can be set to Rectangular, T section, I or L section, and Extended T section.
4. Enter the dimensions of the span sections. All dimensions are defined in the legend at the top of the screen and/or illustrated in the appropriate section figure.
   - Span lengths are measured from support centerline to support centerline.

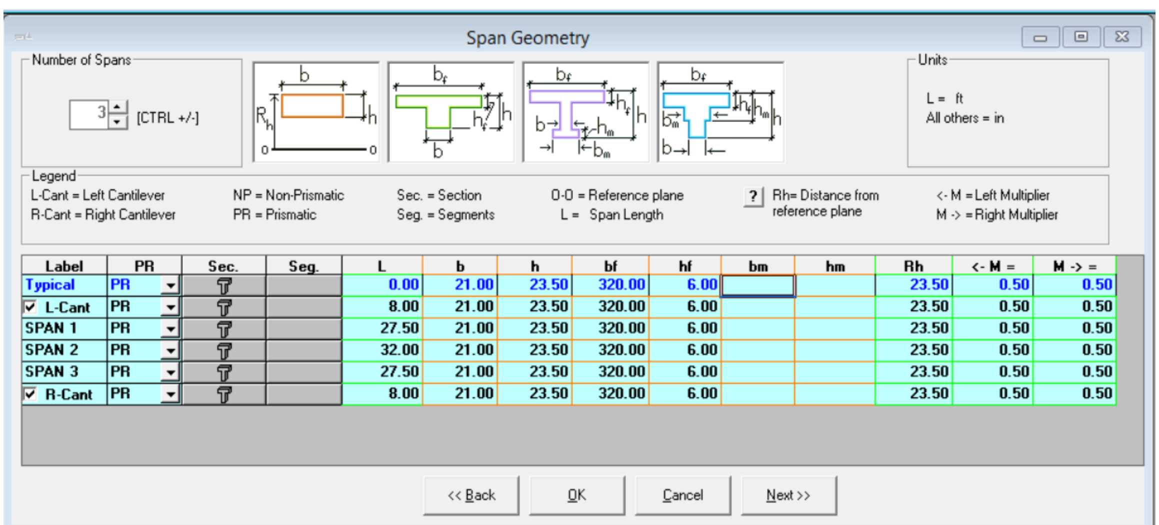


Figure 5.2-1 Span Geometry Input Screen

- The tributary width (dimension b) is composed of left tributary (the portion of the tributary width that falls to the left of the frame line) and the right tributary (the portion that falls to the right of the frame line). The tributary width can vary from span to span but is assumed to be constant within a single span unless segmental input is used.

  There are two methods of modeling tributary width: Unit Strip input and Tributary input. Both methods produce the same results, which method to use is a matter of user preference. Once a method is selected however, it should be used consistently throughout a given project to avoid confusion. Note that the calculations and results are always shown in terms of the total tributary width, regardless of the way the slab was modeled during data entry.

  Unit strip modeling (Fig 5.2-1A). It is typically easiest to model slabs with the unit strip method. A unit strip is a strip parallel to the span with a width equal to or less than the total tributary width. Although the unit strip width is typically 12 in. or 1000 mm, any reasonable value may be used. The unit width has no effect on the analysis if the total tributary width is modeled correctly. The tributary is modeled by specifying a unit strip width along with left and right multipliers. The left and right multipliers (<–M and M–>) indicate the number of times the unit strip needs to be multiplied to cover the left and right tributaries. The multipliers need not be whole numbers.

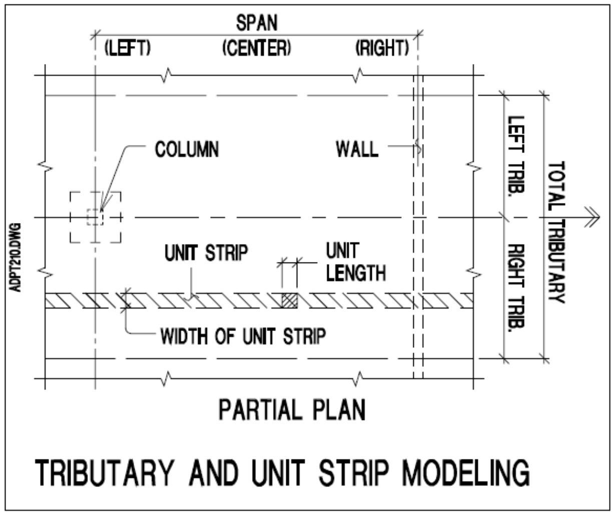


Figure 5.2-1A

Tributary Modeling . In tributary modeling, the total tributary width is entered as the 'b' dimension. The width multipliers (<-M and M->) are used to indicate how much of the tributary falls on either side of the frame line. The sum of the left and right multipliers should be one.

Example: Model the following tributary using A) unit strip method B) tributary method.

Total tributary = 300 in.

Left tributary = 180 in.

Right tributary = 120 in.

A) Unit strip method:

b = 12 in. <-M = 15 M-> = 10

Total width = Left tributary + Right tributary = 12*15 +12*10 = 180 + 120 = 300 in.

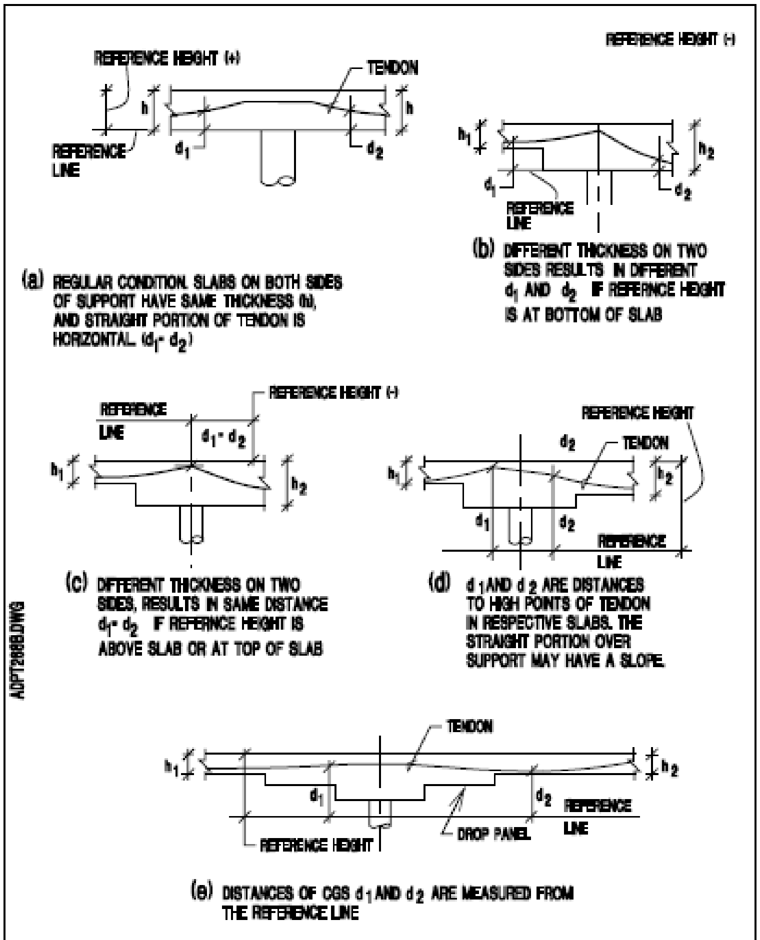
B) Tributary method:

b = 300 in. <-M = 0.60 M-> = 0.40

Total width = Left tributary + Right tributary = 300*0.60 +300*0.40 = 180 + 120 = 300 in

5. Enter reference height (**Rh**). The reference height identifies the position of a reference line that is used in determination how to display tendon heights. The **Rh** indicates the distance from the reference line to the top of the slab with positive being measured upwards. Typically, the reference height is set equal to the slab depth.

Figure 5.2-2 shows several different reference height configurations. Typically, the same reference height is used for all spans. The reference height can thus be set via the typical row. If the slab or beam depth changes, the same reference height can still be used if the resulting tendon heights are adjusted accordingly when transferred to the structural drawings for PT designs. Alternatively, the reference height can be entered as zero, which will set the reference line at the top of the slab. If the reference line is at the top of the slab, tendon heights will be shown as negative numbers indicating distance below the top of the slab.



DEFINITION OF DISTANCE OF CGS (CENTER OF GRAVITY OF STRAND) FROM USER DEFINED REFERENCE LINE

Figure 5.2-2

The reference height can be changed from span to span to model steps at the top of the slab. If this is done, however, it will be necessary to adjust the tendon profiles, so they match at the supports for designs performed using PT mode. In general, it is best to use the same reference height for all spans. Changes in the slab depth should be modeled accurately, however, to ensure that the calculations are done correctly.

5.2.1.2 Non-prismatic (non-uniform) spans

The following example illustrates data entry for a non-prismatic section using the segmental option. The example is a four-span beam, which is made up of three segments of varying cross-section for the two interior spans (**Fig. 5.2-3**).

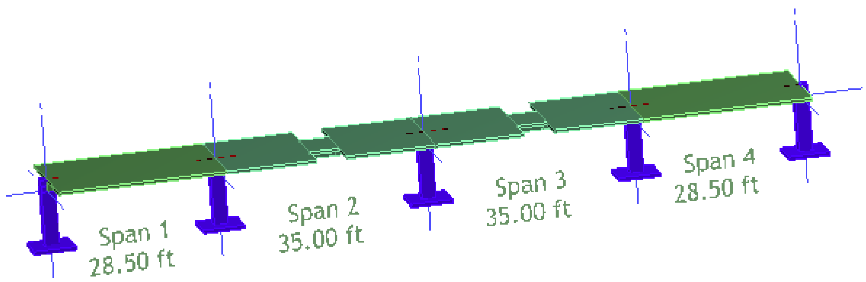


Figure 5.2-3

To model a non-prismatic span, do the following:

1. Select **Segmental** in the **General Settings** input form.

2. Open **Span Geometry (Fig. 5.2-4).**

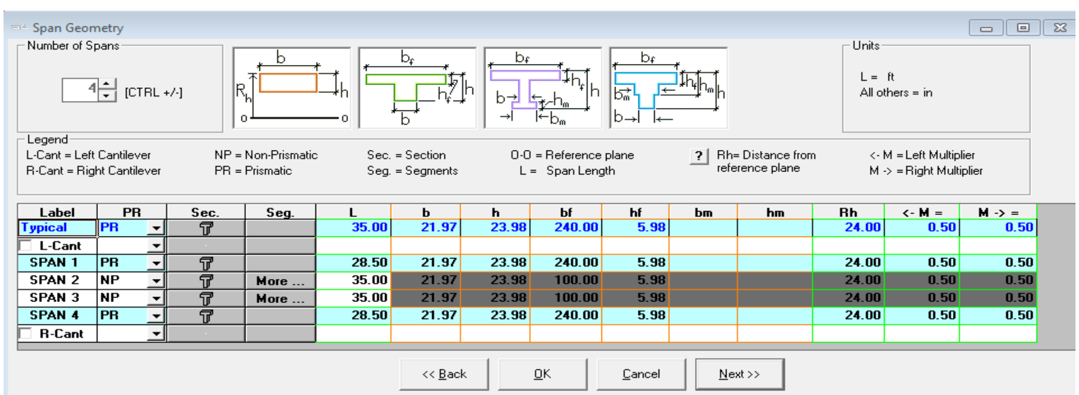


Figure 5.2-4 Span Geometry

3. To set the number of spans, use **CTRL+/-** or click on the up/down arrow at the left of the screen. If there are cantilevers on the right and/or left ends of the frame, add them by clicking on the appropriate check box. This will activate the input fields for the corresponding cantilever.

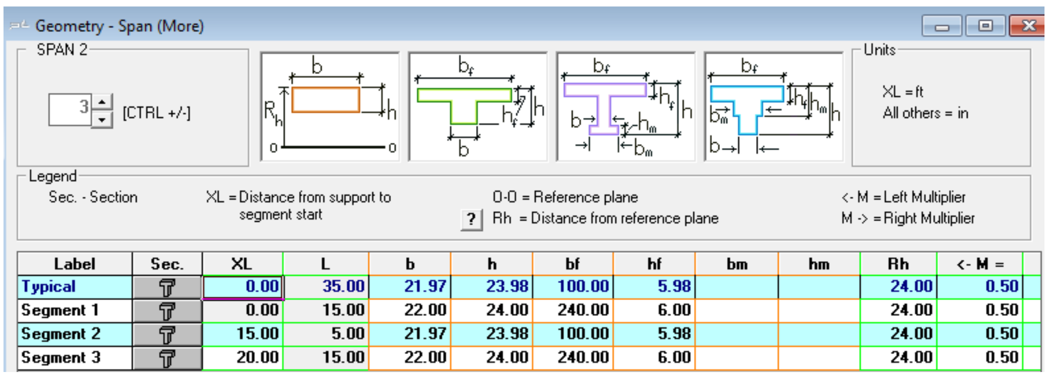
4. Select section type by clicking on the button in the **Sec.** column.

5. Enter the dimensions of the span sections. All dimensions are defined in the legend at the top of the screen and/or illustrated in the appropriate section figure. The dimensions specified in the **Span Geometry** screen including reference height and left and right multipliers define the geometry of a mid-segment of the span. All other segments of the span

are defined in the **Geometry-Span (More)** screen as explained in the following. Note that any span set to non-prismatic, **NP**, will be grayed out in the **Span Geometry** screen and can only be modified in the segmental input described below.

6. Change prismatic column **PR** to **NP**. Changing a span to **NP** activates the More ... button in the **Seg.** column.

7. Click on the More ... button in the **Seg.** column to open the **Geometry-Span (More)** window for that span (**Fig. 5.2-5**).



Label	Sec.	XL	L	b	h	bf	hf	bm	hm	Rh	<- M =
Typical		0.00	35.00	21.97	23.98	100.00	5.98			24.00	0.50
Segment 1		0.00	15.00	22.00	24.00	240.00	6.00			24.00	0.50
Segment 2		15.00	5.00	21.97	23.98	100.00	5.98			24.00	0.50
Segment 3		20.00	15.00	22.00	24.00	240.00	6.00			24.00	0.50

Figure 5.2-5 Geometry – Span (More) Screen

8. To set the number of segments in the spans, use **CTRL+/-** or click on the up/down arrow at the left of the screen. Up to seven segments may be entered per span. The parameters are input in the same manner as general span geometry data except the **XL** column is used to specify the distance from the left support centerline to the start of the segment. The length of each segment is calculated automatically based on the distance to the start of the next segment and reported in the **L** column. The start of the first segment is always zero.

Note that if either the **Use Equivalent Frame Method** or **Increase Moment of Inertia Over Support** option was selected, the program will automatically generate additional segments over each support using the geometry entered for the first and last segments. If the first and last segments are generated before the support dimensions are entered, their **XL** values will be initialized with values of zero and the span length, respectively. These values will be updated when the support dimensions are entered.

9. Click **OK** to save input data and close **Geometry-Span (More)** window.

10. When the user completes data input for all spans click **Next** to save the data and open next input screen **Support Geometry and Stiffness**.

5.2.2 Specify Effective Flange Width

If you enter a beam and you answer 'Yes' to **Consider Effective Flange Width** on the **General Settings** screen, the **Span Geometry** screen will be followed by the **Effective Flange Width** screen (**Fig. 5.2-6**). This screen is also available through menu **Geometry→Effective Flange Width**.

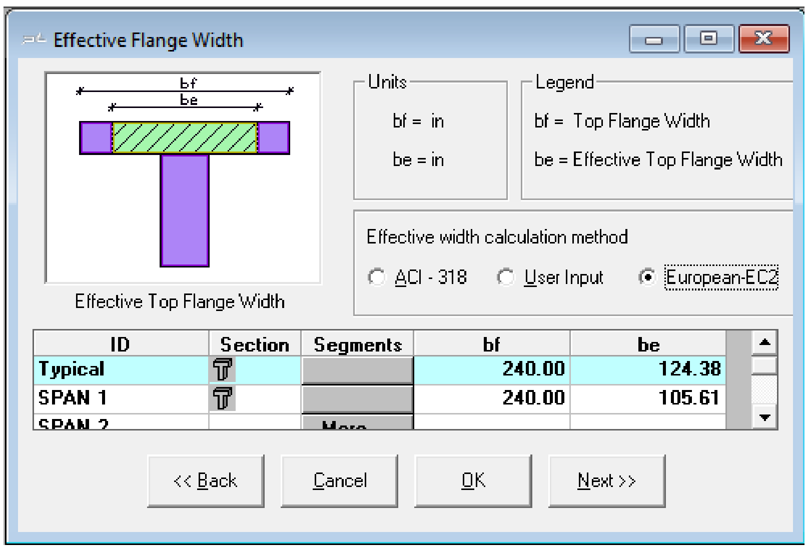


Figure 5.2-6 Effective Flange Width Input Screen

1. Open Effective Flange Width input screen.

2. Select Effective width calculation method.

   - If you choose to use the **ACI-318 or European-EC2** method of effective flange width calculation, the resulting flange widths will be displayed, but you will not be able to edit them.

   - If you select **User Input** calculation, the effective width column **b_e** will default to the ACI calculated values, but you will be able to change them.

3. Click **Next** to save data and go to the next input form.

Note: ACI does not require that effective flange widths be used for prestressed beams. The widths calculated by the program are in accordance with the ACI recommendations for non-prestressed beams.

5.2.3 Specify Geometry of Drop Cap and/or Transverse Beam

If you enter a two-way system and you answered “Yes” to the **Include Drops & Transverse Beams** question on the **General Settings** screen, the **Span Geometry** screen will be followed by the **Geometry-Drop Cap/Transverse Beam** screen **(Fig. 5.2-7)**. This screen is also available through menu **Geometry→Drop Cap/Transverse Beam**.

The input parameters are defined in the figures at the top of the screen. Note that **H**, the depth of the cap or beam, is the total depth of the section, not the depth below the slab.

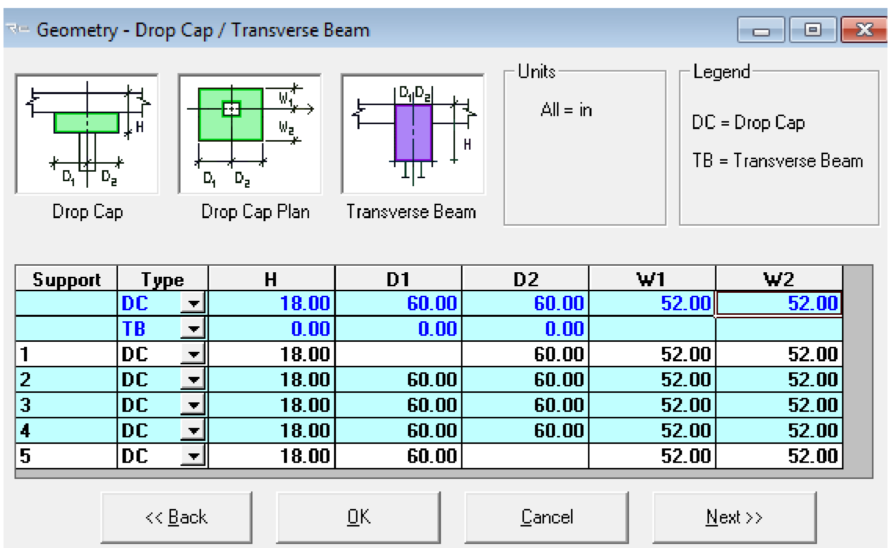


Figure 5.2-7 Drop Cap/Transverse Beam Input Screen

If there are drop caps or transverse beams with the same dimensions at several supports, their dimensions may be entered using the typical row. To enter typical values for drop caps, type the value into the typical row and press ENTER. The value will be copied to any supports that have been marked as having drop caps. Any supports which are subsequently marked as having drop caps will also be assigned this value as a default.

Transverse Beams dimensions are entered in the same manner.

5.2.4 Specify Geometry of a Drop Panel

If you enter a two-way system and you answered "Yes" to the **Include Drops & Transverse Beams** question on the **General Settings** screen, the **Drop Cap/Transverse Beam** screen will be followed by the **Geometry-Drop Panel** screen **(Fig. 5.2-8)**. This screen is also available through menu **Geometry→Drop Panel**.

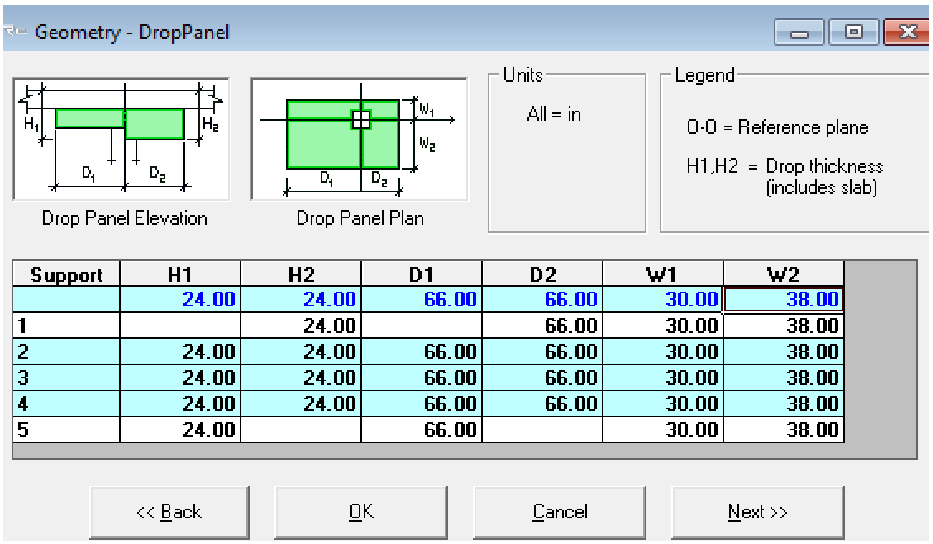


Figure 5.2-8 Drop Panel Input Screen

The data entries for drop panels are the same as for drop caps. Typical values can be entered with the typical row at the top of the table.

5.2.5 Specify Support Geometry and Stiffness

This screen is used to input support heights, widths, and depths of supports (**Fig. 5.2-9**). Support selection options will change depending on the structural system you selected.

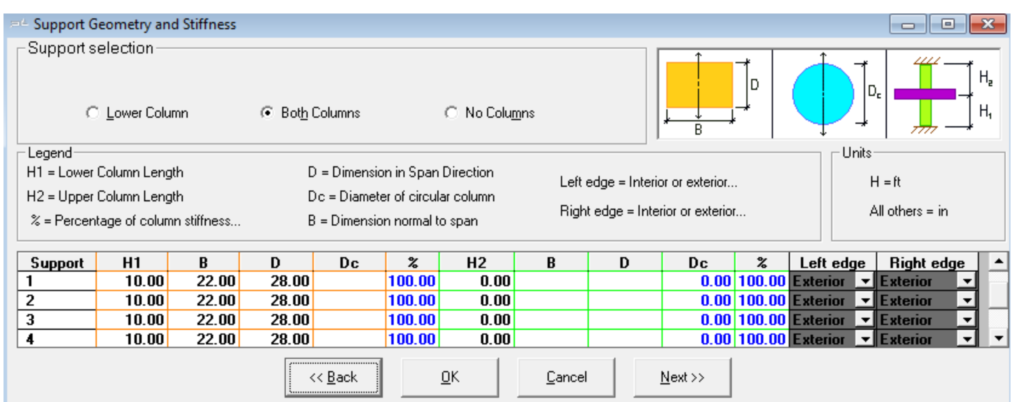


Figure 5.2-9 Support Geometry and Stiffness Input Screen

If you model a two-way system or a beam, the available support options will be:

- Lower column,
- Both columns,
- No columns

If you model a one-way system, the support options will be:

- Lower wall,
- Both walls,
- Point support or transverse beam.

To model supports do the following:

1. Select lower, both or no support option.
2. Enter the height of lower supports (**H1**) if any. **H1** is the distance from the mid-depth of the slab to the top of the slab below.
3. Specify cross section dimension for support:
    - If rectangular, enter data in column **D** (dimension in span direction) and **B** (dimension perpendicular to span direction).
    - If circular, enter data in column **Dc** (diameter of circular column).
4. Specify the percentage column stiffness that you would like to consider in analysis.
5. Repeat the procedure to define geometry of upper columns if any, or simply copy the data.
6. If you model a two-way system you will have an option to assign a **Left edge** and **Right edge** condition. This option is available only for two-way systems and it is used to determine column condition for punching shear check.
    - If you select **Exterior** the program will automatically check the left and right tributary width. If the tributary that falls to the left or to the right side of the column is less than code required for interior column, the program will automatically consider it as an exterior column.

- If you select **Interior** the program does not check left and right tributary width and treats the column as interior.

Note: The program always checks left and right distance to the slab edge. If the dimension **B** of a column is at least 80% of tributary width the program will treat this column as wall and will not check it for punching shear.

5.2.6 Specify Support Boundary Conditions

This screen is used to enter support widths and column boundary conditions (**Fig. 5.2-10**).

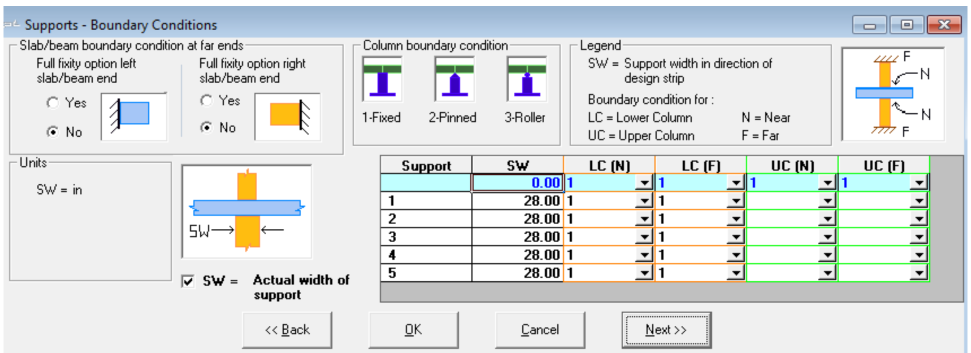


Figure 5.2-10 Supports-Boundary Conditions Input Screen

1. Select Slab/beam boundary conditions at far ends.

   - Choose "No" if the slab end is rotationally free. This occurs when a slab/beam terminates over wall, column, or beam.
   - Choose "Yes" if the slab end is rotationally fixed. This occurs if the span/beam end is tied to a structure that is rigid enough to prevent rotation. A typical example might be slab tied to a stiff shear wall. A rotationally fixed end condition can also be used to model half of a symmetrical, multi-span frame if there will be no rotation over the support at the line of symmetry.

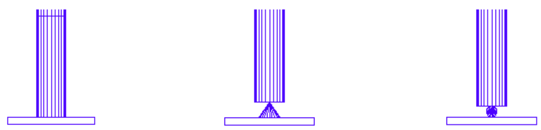
   Note: If there is a cantilever at the right or left end of span, the corresponding slab/beam end condition option will not be available.

2. Specify support width **SW**. This option is available if you choose "Yes" to the **Reduce Moments to face-of -support** in the **Design Settings** screen.

- To automatically set the values in **SW** column to actual support widths, **D**, check box in front **SW = Actual width of support**. This option will not be available if you select no columns or point support transverse beam option for supports.

- To manually enter support width, uncheck box in front **SW = Actual width of support** and input data.

3. Choose support boundary conditions by clicking on the arrow in the appropriate cell of the table. The following figure shows the available boundary conditions and symbols used in 3D view.



Condition 1	Condition 2	Condition 3
fixed	pinned	roller
(a)	(b)	(c)

Figure 5.2-11 Boundary Conditions

Note: If **No Columns** option was specified on the **Supports-Geometry** screen the boundary condition entries will be ignored. The support widths will be used to calculate reduced moments, however.

5.3 Loads

ADAPT PT/RC allows you to specify a variety of load types including dead, live, earthquake or wind loads (lateral loads).

5.3.1 Specify Dead, Live, and Other loads

Figure 5.3-1 shows the screen used to enter loading information.

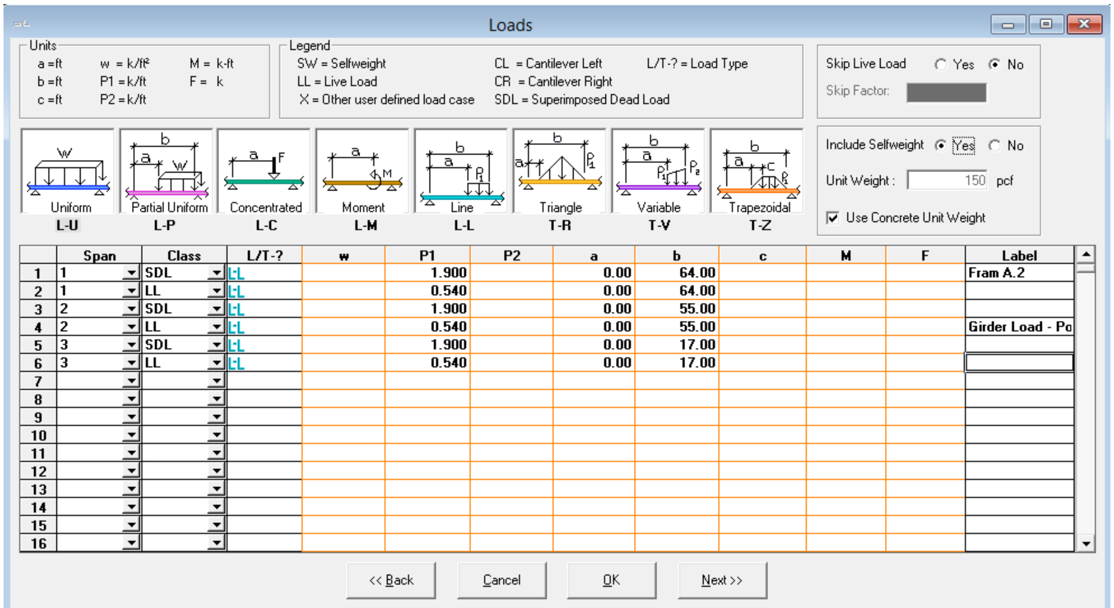


Figure 5.3-1 Load Input Screen

1. Specify loaded spans.

   - To enter load for an individual span, click on the arrow in cell of the **Span** column and select a span number from the list of all available spans, or just type in a span number.
   - To enter a load for all spans, enter 'all' or 'ALL' as the span number.
   - To enter loads on a left cantilever, enter either LC or 0 as the span number. To enter loads on a right cantilever, enter either RC or the number of spans +1 as the span number.

2. Specify the class of load by clicking on the arrow in the cell of a **Class** column. There are four available classes:

   - **SW** – self-weight. This load class will be available only if you select "No" for Include self-weight. In this case you have an option to enter self-weight of the structure manually instead of allowing the program to calculate it. If "Yes" is selected for self-weight, the program calculates the structure self-weight based on the density of the concrete material selected. The user has an option in this dialogue menu to override the concrete unit weight.
   - **SDL** – superimposed dead load

- **LL** – live load
- **X** – other load

3. Specify the type of loading by typing **U**, **P**, **C**, **M**, **L**, **R**, **V**, or **Z** in the **L/T-?** column, or by dragging the icon from the graphics of the loading that you intend to apply to the cell in the **L/T-?** column. There are eight load types:

   - U - **U**niform,
   - P - **P**artial uniform,
   - C- **C**oncentrated,
   - M - **M**oment,
   - L - **L**ine,
   - T - **T**riangle,
   - V - **V**ariable, and
   - T - **T**rapezoidal.

   Note: Uniform and partial loads are assumed to be uniformly distributed over the upper most surface of the member with a constant intensity per unit area. The user only needs to enter the loads intensity and ADAPT calculates the frame loadings. These frame loadings are reported in report table **3.2 Compiled loading**.

4. Enter load intensity and position. The schematics for each load type indicate the required input data. Note that on cantilevers, distances are always measured from the support **(Fig. 5.3-2)**. The distances for a left cantilever are thus entered contrary to those of the typical spans.

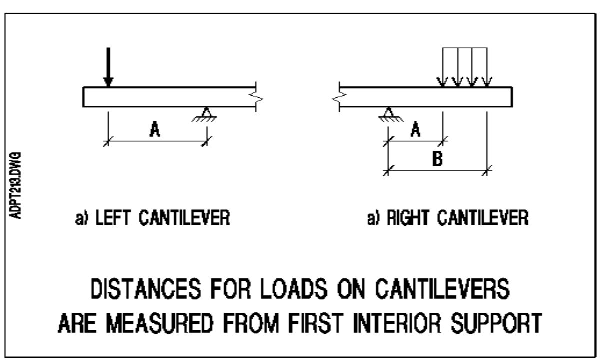


Figure 5.3-2

Any number of different loads and load types may be entered for a span. You may also specify whether to skip the live loading and whether to calculate self-weight automatically. If these features are selected, the skip factor and concrete unit weight must be entered. When the Skip Loading function is used, the program considers those 6 patterns shown in **Fig. 5.3-3.**

The program reports deflections at all 1/20th points along spans for all skip patterns and enveloped Max and Min results in report **Section 36**. For Sustained Load combinations, the program considers the user-specified portion of live load for creep/long-term computations. Creep Max and Creep Min values are also reported.

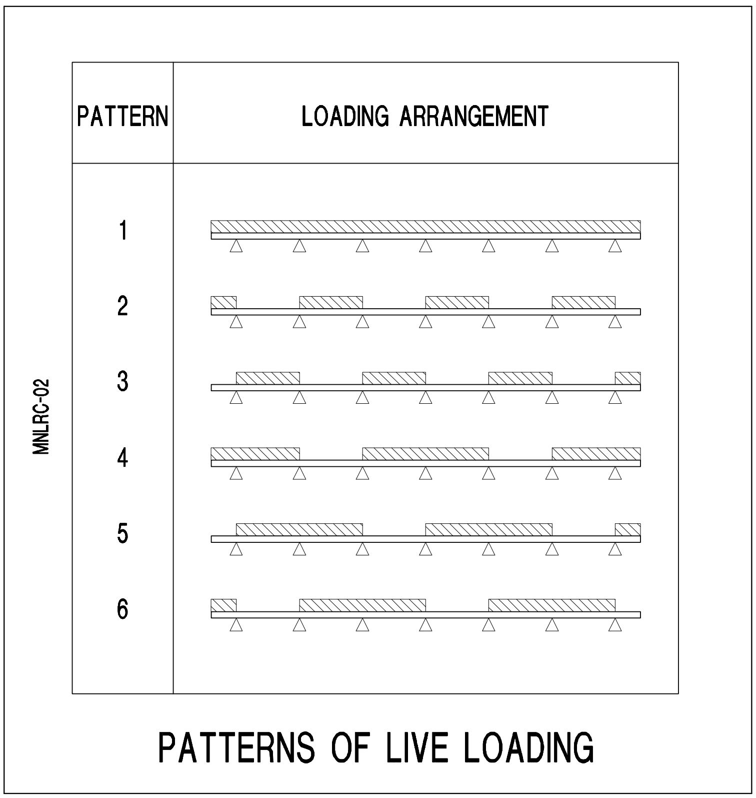


Figure 5.3-3

5. The loading input table as shown in **Figure 5.3-1** contains a **Label** column allowing the user to enter notes associated with each represented load. Note that loading notes are not reported in the tabular report output but are contained only with the input dialogue window.

The loading dialogue window contains sorting functions for all input columns. Sorting can be made by Ascending or Descending order or can be restored to the original input. At the title cell for each input column, click on the cell to sort by Ascend, click again to sort by Descend and click a third time to reset back to the original input.

Ascending Order

Descending Order

5.3.2 Specify Lateral Loads

ADAP- PT/RC allows you to specify lateral loads (wind or earthquake loads) as unbalanced concentrated moments acting at the face of supports. To define these loads:

1. Go to **Criteria→Load Combination**.
2. Check **Include Lateral Loads** and click **Set Values**.
3. Go to **Lateral moments** tab and input values. (**Fig. 5.3-4**)

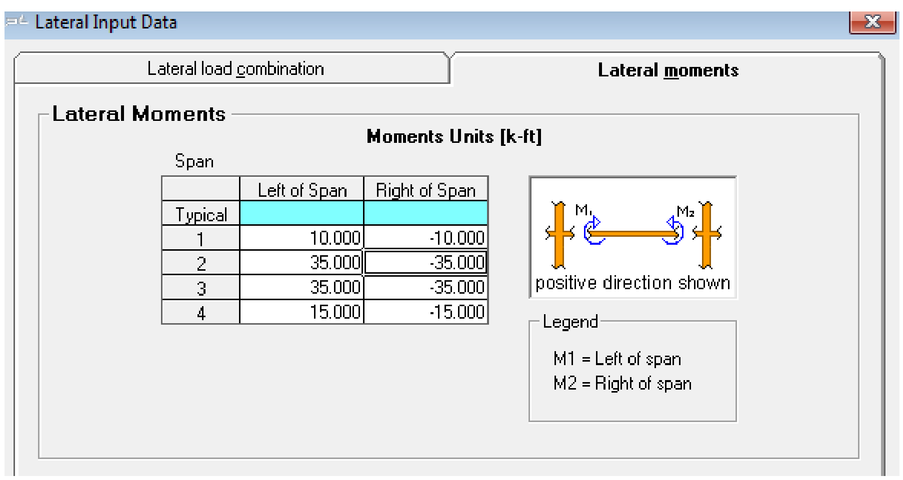


Figure 5.3-4 Lateral Input Data Screen

4. Click **OK**. The lateral moments will show on the screen.

5.4 Material

5.4.1 Specify Concrete Material

This screen is used to enter concrete properties **(Fig. 5.4-1)**.

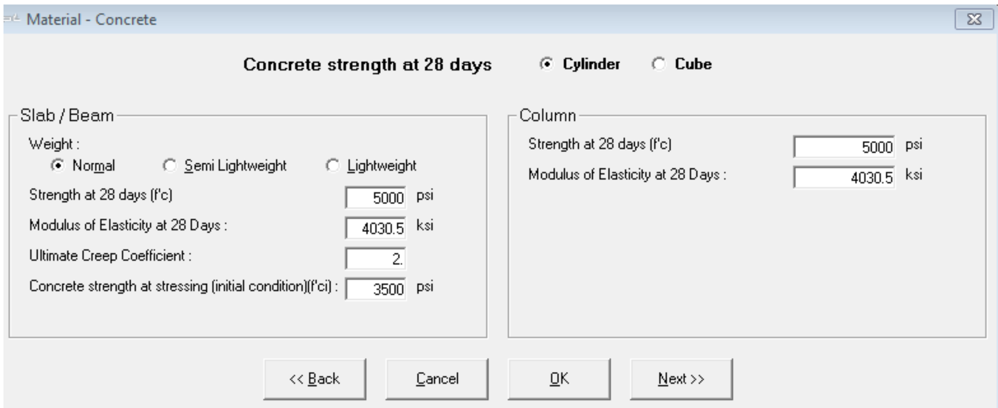
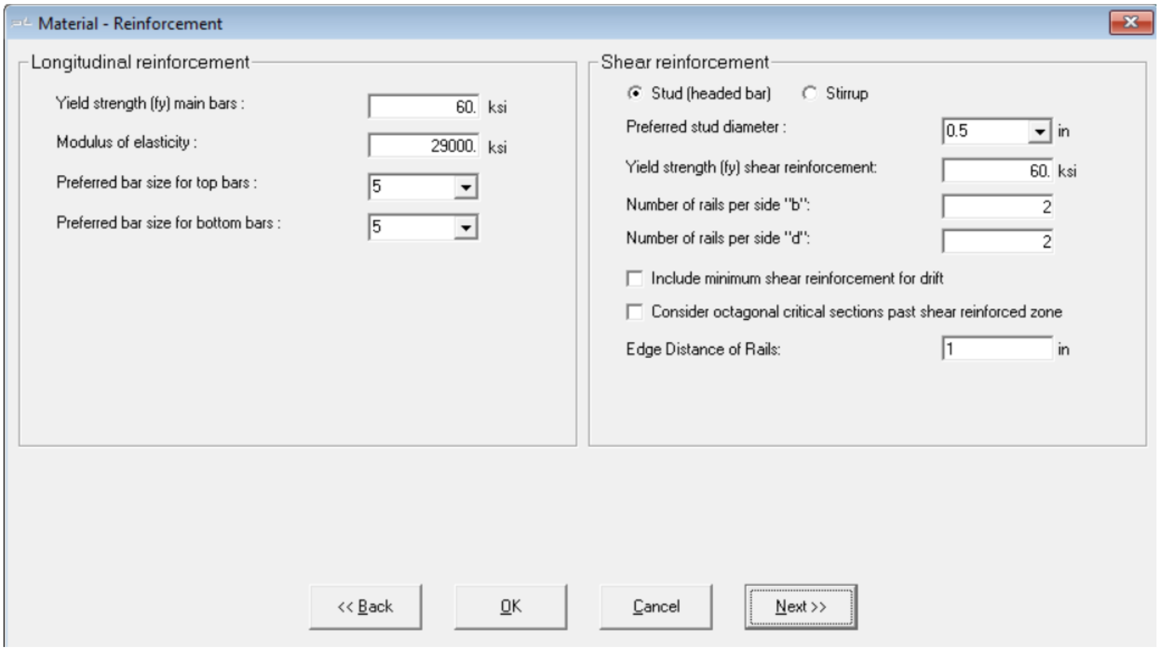


Figure 5.4-1 Concrete Material Input Screen

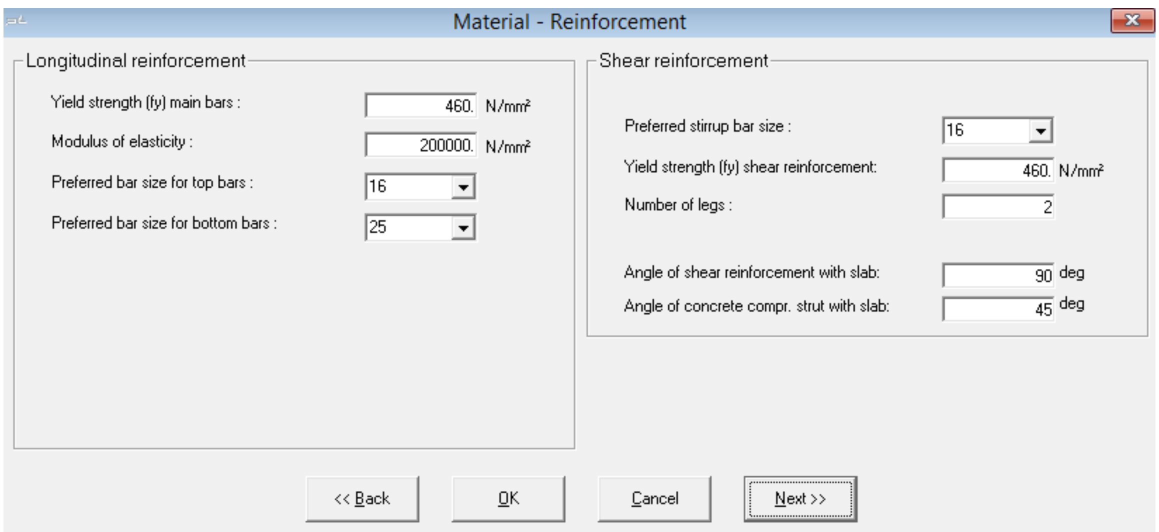
Depending on the code, the concrete weight classification is used in shear and/or flexure calculations. Default values of the modulus of elasticity are calculated based on the concrete strength, concrete type (Normal, Semi-Lightweight, and Lightweight) and appropriate code formula. The ultimate creep coefficient is used in the calculation of long-term deflections. The option for concrete strength at stressing is used only for PT mode and does not apply to RC designs.

5.4.2 Specify Reinforcement Material

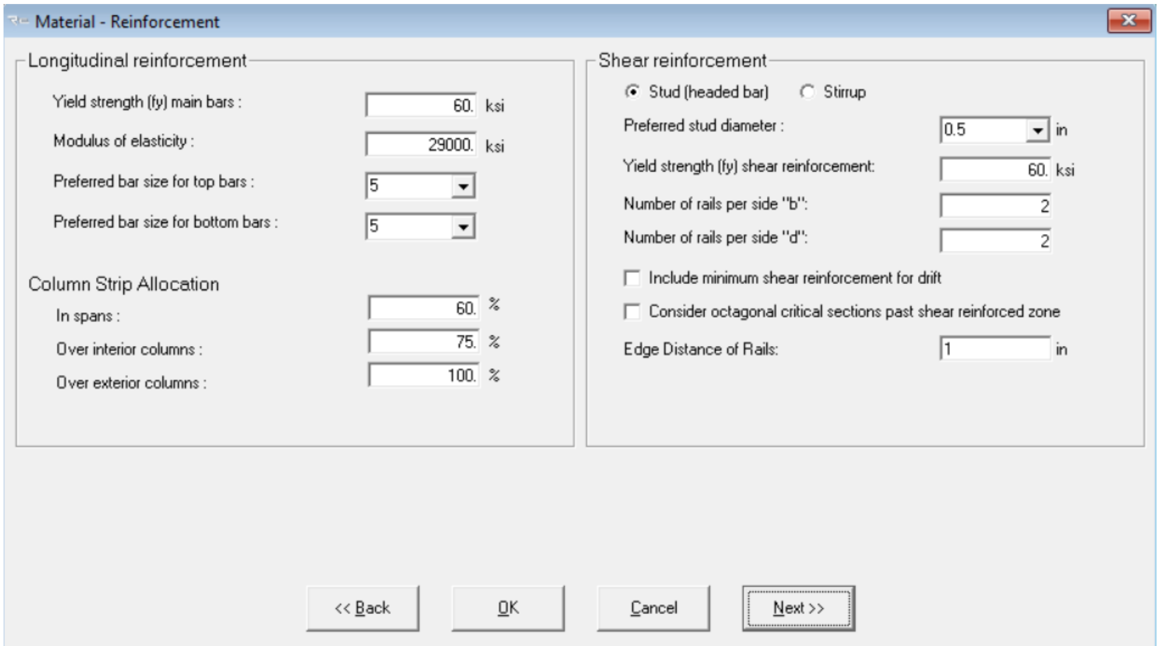
This screen is used to specify bar sizes and properties for longitudinal and shear reinforcement **(Fig. 5.4-2)**. For RC mode, the program includes entry for **Column Strip Allocation**. Here the user specifies the percentage of total strip reinforcement to be allocated to the column strip in spans, over interior columns, and over exterior columns. The balance of calculated reinforcement is allocated to the middle strips.



PT Mode – Two-Way System



One-Way/Beam System (EC2)



RC Mode – Two-Way System

Figure 5.4-2 Steel Material Input Screen

When entering data for a beam and one-way slab, entry for preferred stirrup bar size and number of legs is active. When entering data for two-way slab options to select between stirrups and studs is active.

- If you select **Studs** the program will ask you to specify stud diameter and number of rails per side of a column. ADAPT-PTRC includes separate input for number of rails per side for individual 'b' and 'd' column sides. The 'b' side is that transverse to the span direction.
- If you select **Stirrups** the program will ask you to specify stirrup bar size. For EC2 the program gives input for angle of beam or slab-to-stirrup and beam or slab-to-compression strut for use with inclined stirrups less than 90 deg.

The preferred bar sizes are used when calculating the number of bars required. The bar sizes may be changed on the Summary report, however.

For RC mode, when the European – EC2 code is selected as the design code, the program includes the option for the allowable crack width. The program adds reinforcement necessary to control the crack width to the limiting value. EC2 methodology is used in calculating crack widths and reinforcement.

There are 3 new input fields in this input window for two-way shear checks in ADAPT-PT/RC. These include:

- Include minimum shear reinforcement for drift.
- Consider octagonal critical sections outside shear reinforced zone.
- Edge distance of rails – This option is required for the 2nd option listed above.

New improvements have been made for punching shear checks in ADAPT-PT/RC according to ACI318-14. The first improvement is the option to consider the least critical section as an octagonally-shaped section at d/2 outside the last line of reinforcement. The applicable code sections being 22.6.4.2 and 22.6.6.1.

The second improvement is the option to consider minimum reinforcement up to a distance 4*slab thickness to satisfy a minimum strength of 3.5*f'c^1/2. This requirement is found is Section 18.14.5.1 and is applied to satisfy adequate ductility of members not part of a lateral-resisting system in seismic design categories D, E, and F.

The third improvement applies to the application of minimum reinforcement at critical sections as dictated in ACI318-14 Section 22.6.8.3. The critical section used to determine the minimum amount of reinforcement is that of the section located d/2 from face of support. In previous versions of the software, the critical section length as based on the section being checked. In other words, the amount of reinforcement necessary to meet minimum became larger each critical section (layer) that was checked. Upon a formal review of the program code and the interpretation of design code, this was modified and improved.

The following sections provide more detailed description of the first two features described above.

5.4.2.1 Critical Section Outside the Reinforced Zone

The option to consider critical sections outside the shear reinforced zone with either a rectilinear or octagonal-shaped critical section is given in the program. ACI318-14 Sections 22.6.4.2, 22.6.6.1 apply. The following images show the octagonal-shaped section for compliance with ACI318-14. Note this section shape can now also be used for designs performed using all ACI318 versions supported by the program.

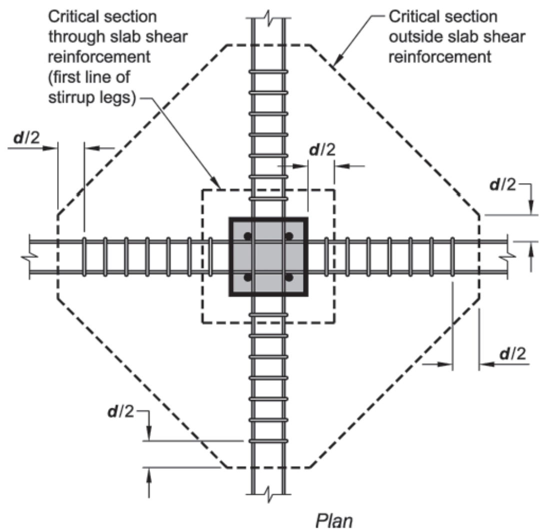


Fig. R22.6.4.2a—Critical sections for two-way shear in slab with shear reinforcement at interior column.

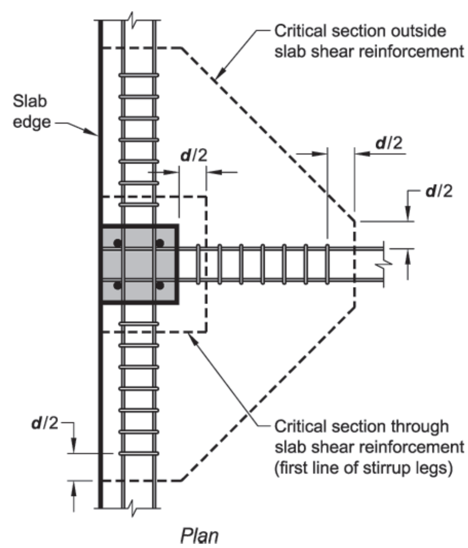


Fig. R22.6.4.2b—Critical sections for two-way shear in slab with shear reinforcement at edge column.

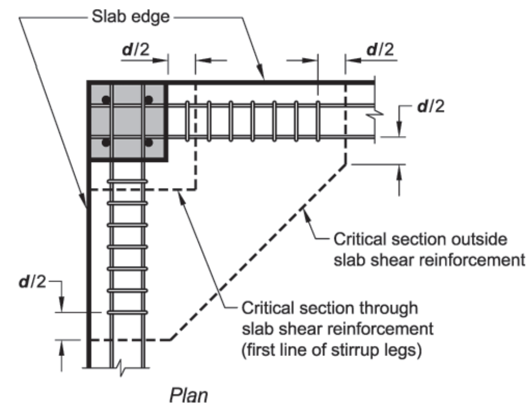


***Fig. R22.6.4.2c**—Critical sections for two-way shear in slab with shear reinforcement at corner column.*

When this option is selected the program determines the critical section, based on a rectilinear shape, where no reinforcement is required. Each critical section is located at d/2 from the previous. At the critical section where no reinforcement is required (As = 0), the program rechecks the same location but with a reduced critical length and area. This is calculated using an octagonally-shaped section and is checked until the stress is less than code specified. The user can specify the edge distance between the edge of column and stud rail or shear stirrups. This value affects the critical section length.

5.4.2.2 Minimum Reinforcement for Drift

The option to apply two-way shear provisions for minimum reinforcement for seismic drift as found in ACI318-14 Section 18.14.5.1 is now included in ADAPT-PT/RC. The referenced provision is found in the excerpt below. Note that the provision is conditional upon a check of the inter-story drift ratio at the support location. However, the program does not check this but gives the user the option to directly enforce the minimum shear strength as given as $3.5*f'c^{\wedge 1/2}$.

18.14.5 *Slab-column connections*

18.14.5.1 For slab-column connections of two-way slabs without beams, slab shear reinforcement satisfying the requirements of 8.7.6 or 8.7.7 shall be provided at any slab critical section defined in 22.6.4.1 if $\Delta_x/h_{sx} \geq 0.035 - (1/20)(v_{ug}/\phi v_c)$. Required slab shear reinforcement shall provide $v_s \geq 3.5\sqrt{f_c'}$ at the slab critical section and shall extend at least four times the slab thickness from the face of the support adjacent to the slab critical section. The shear reinforcement requirements of this provision shall not apply if $\Delta_x/h_{sx} \leq 0.005$.

The value of (Δ_x/h_{sx}) shall be taken as the greater of the values of the adjacent stories above and below the slab-column connection. v_c shall be calculated in accordance with 22.6.5. v_{ug} is the factored shear stress on the slab critical section for two-way action due to gravity loads without moment transfer.

The current expresssions used to calculate As for two-way shear is shown below as taken from ADAPT-Technical Note TN504. If the new minimum reinforcement for drift requirement is selected, the program checks that the quantity in the parentheses is ≥ $3.5*f'c^{1/2}$ for all critical sections with a distance 4*slab thickness from face of the support. Additional checks with respect to Av,min still apply as shown below.

$$A_v = \frac{(v_u - \varphi_v v_c)\ us}{\varphi_v f_y \sin(\alpha)}$$

For studs, $A_v \geq A_{vmin}$[24] where $A_{vmin} = \frac{2\sqrt{f'c}us}{f_y}$

Where,

v_c = $2*\lambda*\sqrt{f'c}$[25] [$0.17*\lambda*\sqrt{f'c}$ in SI] for stirrups

= $3*\lambda*\sqrt{f'c}$[26] [$0.25*\lambda*\sqrt{f'c}$ in Si] for studs

α is the angle of shear reinforcement with the plane of slab and u is the periphery of the critical section. s is the spacing between the critical sections [d/2].

If required, shear reinforcement will be extended to the section where v_u is not greater than $\Phi_v*2\lambda*\sqrt{f'c}$ [$\Phi_v*0.17\lambda*\sqrt{f'c}$ in SI].

5.4.3 Specify Post-Tensioning Material (PT mode only)

This screen is used to input the post-tensioning system parameters **(Fig. 5.4-3)**

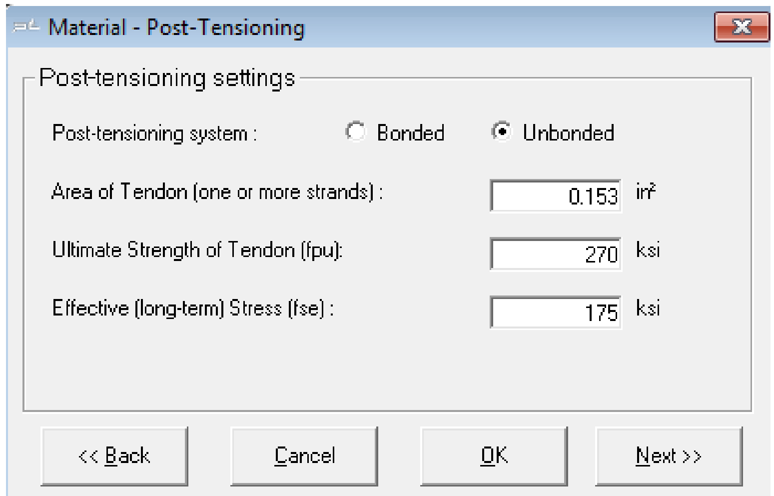


Figure 5.4-3 Prestressing Material Input Screen

The information entered here defines the post-tensioned system type and is: (a) used to calculate the ultimate moment capacity of the member when the "effective force" calculation method is used. When the "tendon selection" calculation method is used, the program calculates the effective stress. The stress in the tendon at nominal strength (f_{ps}) is calculated from the effective stress and the reinforcement ratio, and (b) determines the effective force per stand used to calculate the final force in the Recycler window when the "effective force" calculation method is used.

5.5 Criteria

5.5.1 Specify the Design Code

To select the code:

1. Click **Criteria→Design Code**. The **Criteria-Design Code** dialog window will open **(Fig. 5.5-1)**

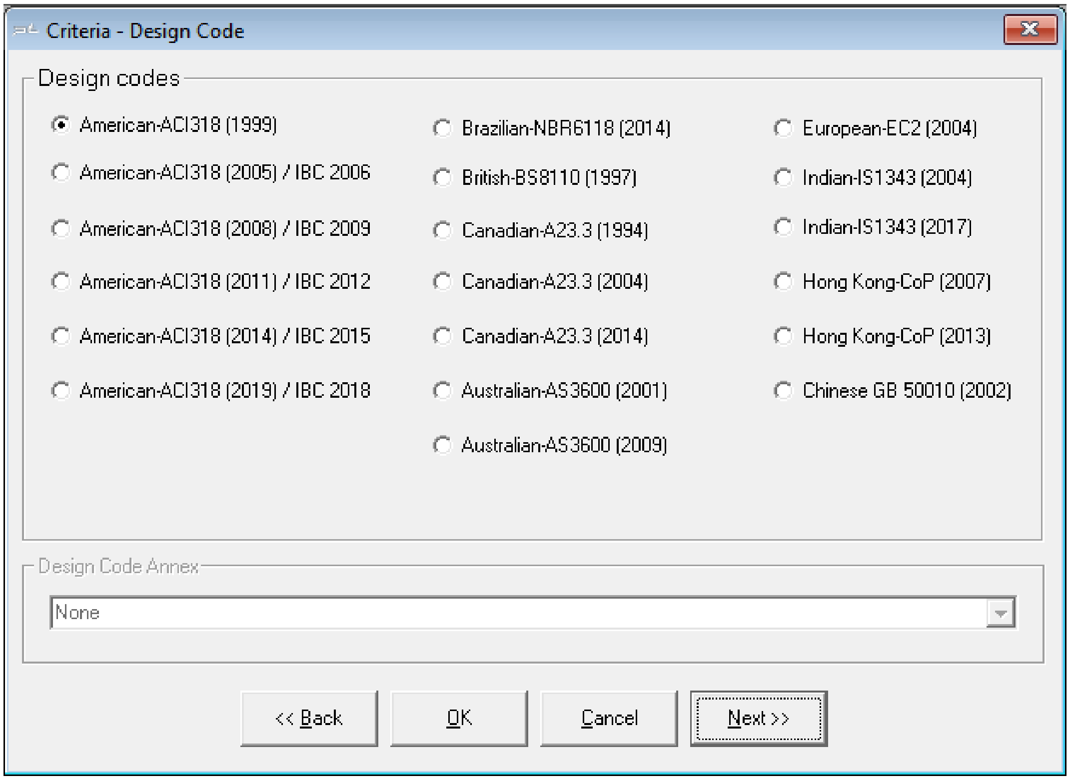


Figure 5.5-1 Criteria-Design Code Input Screen

2. Select the design code from the list.

3. Click **Next**. This will save input data and open a new input screen **Design Settings** where you can select your analysis and design options.

Note: Depending on the code chosen, material factors and other design parameters may need to be entered. These are entered on the Load Combinations screen (**Fig. 5.5-10**). If you model in American or MKS units only ACI318 codes are available. All design selectable design codes other than the ACI318 code are active when SI units are used.

5.5.2 Specify Base Non-Prestressed Reinforcement

This screen is used to specify base non-prestressed reinforcement (**Fig. 5.5-2**).

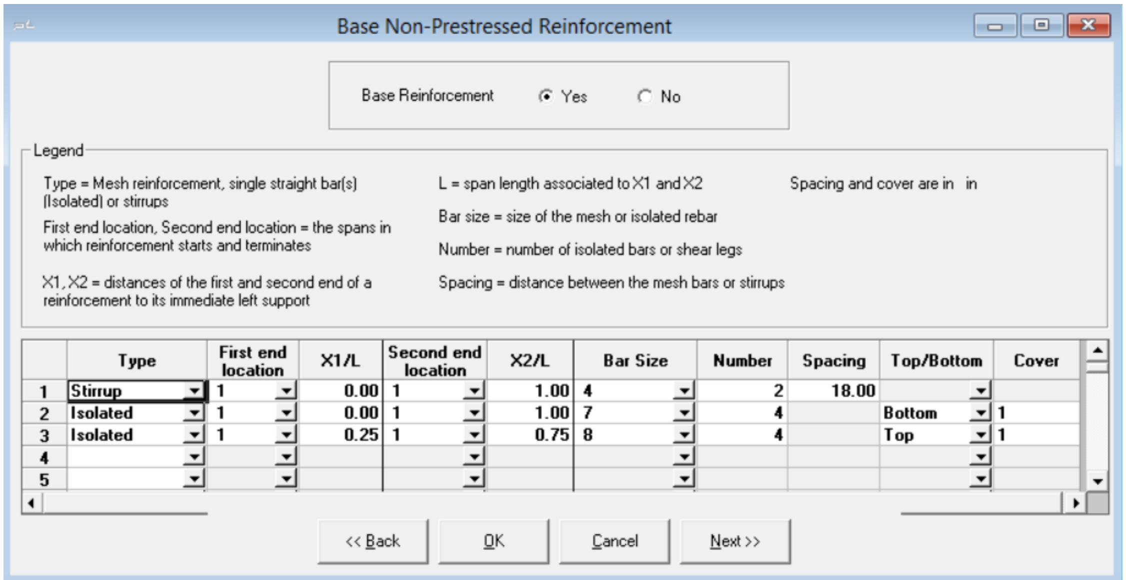
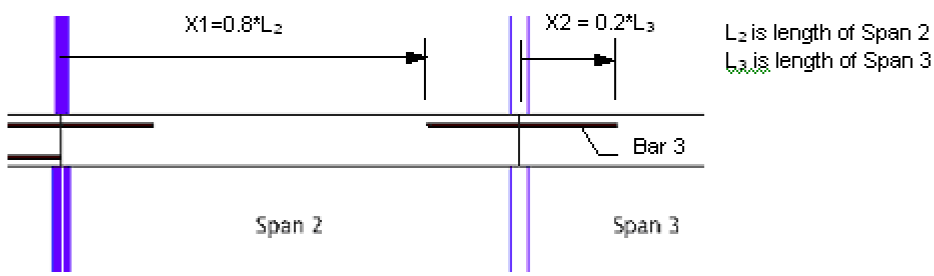


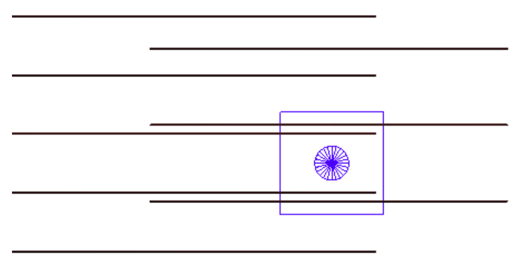
Figure 5.5-2 Base Non-Prestressed Reinforcement Input Screen

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. You have an option to define reinforcement as mesh or isolated longitudinal bars for all system types and stirrups for one-way slabs and beams. The position of a bar is defined with distances of the first X1 and second X2 bar end to its immediate left support.

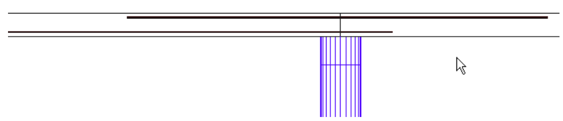
For example, Bar 3 starts at 0.8 of Span 2 and ends at 0.2 of Span 3.



The user can see the base reinforcement in the 3D Structure View window as shown in **Figure 5.5-3**.



Bars shown in plan.



Bars shown in elevation.

Figure 5.5-3 Base Reinforcement View

Multiple entries can be used for stirrups at different spacing increments, stirrup size and spacing and the range where the stirrups are applied. If the spacing of the base stirrup entry exceeds the maximum spacing for the code selected, the base stirrups are ignored in the calculation. Where base stirrups are applied, the program will report a status check in Report 12 – Shear Reinforcement as “OK” or “NS.”

If the base stirrups meet the required area and spacing by calculation, “OK” will be reported in the “Base” column of Report 12 and no additional reinforcement will be reported. If “NS” is reported, the base stirrups are not sufficient to meet the demand and the program will report the demand balance for required reinforcement along with the necessary spacing. This reported reinforcement in Report 12 is in addition to the base reinforcement entered by the user.

5.5.3 Specify Allowable Stresses (PT mode only)

This screen is used to enter initial and final allowable stresses **(Fig. 5.5-4)**.

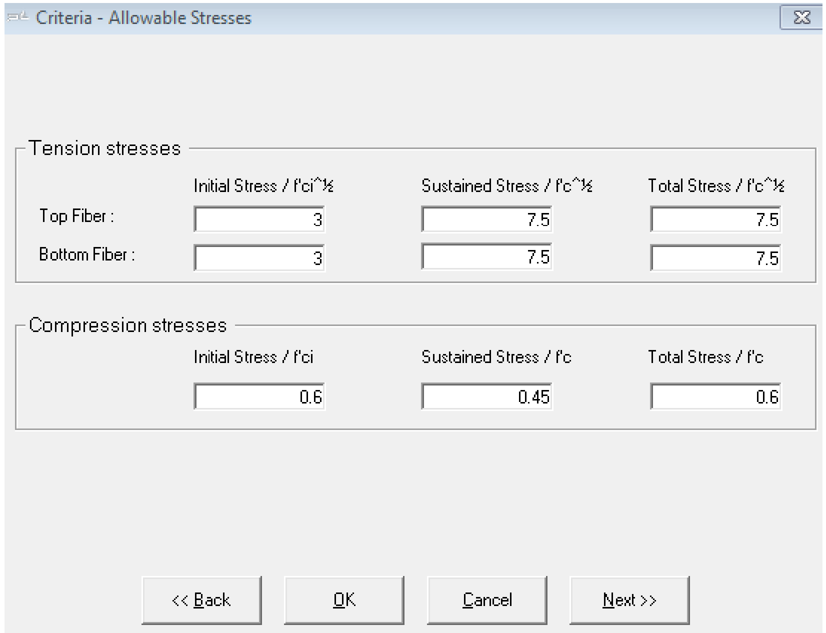


Figure 5.5-4 Criteria-Allowable Stresses Input Screen

Tension stresses are input as a multiple of the square root of f'_C, compression stresses are entered as a multiple of f'_C. The values entered for final allowable stresses will be shown on the Stresses Compression and Tension tab of the Recycle window.

5.5.3.1 PT Crack Design Mode/Allowable Crack Width

When the European – EC2 code is selected as the design code, the program includes an option for **PT Crack Design Mode/Allowable Crack Width** in the **Allowable Stress** screen **(Fig. 5.5-5).** The user can specify if the design is to be checked as an **Uncracked** or **Cracked** design. If the option for **Uncracked** is selected, the options for **Unbonded (Quasi)** and **Bonded (Frequent)** are grayed out and the **RC Sections (Quasi)** option is active.

The program treats the **Uncracked** condition similar to those designs performed using the ACI code where if the allowable stress is exceeded, the design is No Good (NG) in the Recycler Screen and corrective action must be taken to modify the tendon force and/or profile to achieve acceptable fiber stresses.

For an **Uncracked** design, any design section that does not include post-tensioning is considered a RC section. The program calculates and limits cracking to the user specified value for allowable crack width by adding reinforcement to the slab or beam.

If the option for **Cracked** is selected, the user can set allowable crack widths **for Unbonded (Quasi)** and **Bonded (Frequent)** conditions, in addition to the RC Sections option. If the allowable stress is exceeded for either condition, the program will calculate and limit cracking to the user specified value for allowable crack width by adding reinforcement to the slab or beam.

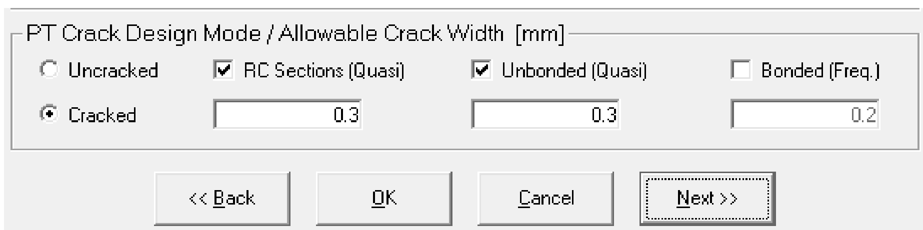


Figure 5.5-5 Criteria – PT Crack Design Mode

Crack widths and required reinforcement to limit crack widths are calculated using methodology outlined in the EC2-2004 code.

The program graphically reports allowable and calculated crack widths for those combinations selected to be checked by the user **(Fig. 5.5-6).**

Figure 5.5-6 Crack Width Diagram

5.5.4 Specify Recommended Post-Tensioning Values (PT mode only)

This screen allows the user to specify minimum and maximum values for average pre-compression (P/A; total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (W_{bal}) **(Fig. 5.5-7)**.

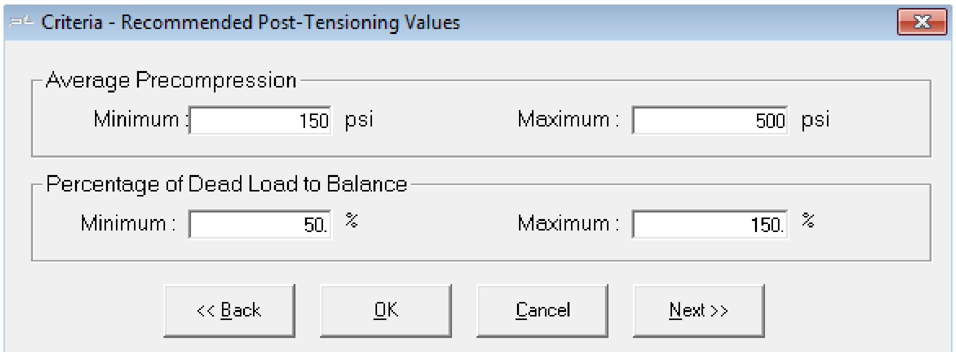


Figure 5.5-7 Criteria-Recommended Post-Tensioning Values

These values are used by the program to determine the post-tensioning requirements shown on the Tendon Forces and Heights tab of the Recycle window. They are also used to determine the status of the P_{min}/P_{max} and $W_{bal\ Min/Max}$ indicators on the Recycle window.

If data is being entered for a one-way or two-way slab, the bottom of the screen will ask for the maximum spacing between tendons. This is entered as a multiple of the slab thickness (i.e. 8 x slab thickness). The program does not check tendon spacing. However, this is something that must be checked on the shop drawings. Tendon spacing is typically more of an issue for detailing than design but on very thin, very lightly loaded slabs, it may govern the design.

5.5.5 Specify Calculation Options (PT mode only)

This screen is used to select the post-tensioning design option **(Fig. 5.5-8)**.

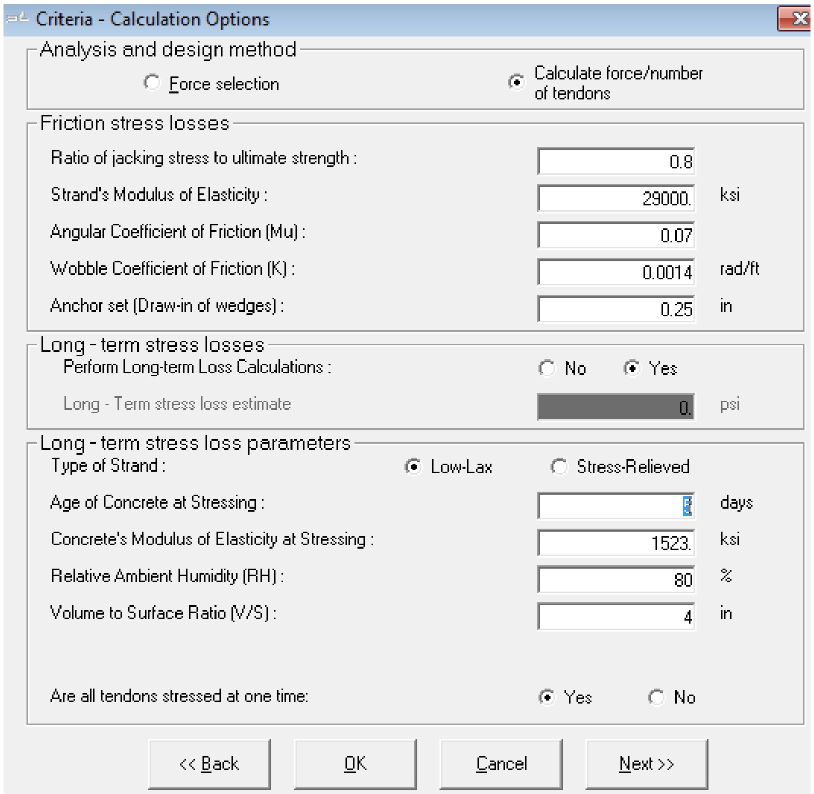


Figure 5.5-8 Criteria-Calculation Options Input Screen

The two options are Force Selection and Force/Tendon Selection. Force Selection is the default option. To use Tendon Selection, the Force/Tendon Selection option must be specified.

If Force/Tendon Selection is specified, the screen will prompt for the information required to calculate the prestress losses. The values given as defaults are typical in the industry and should be used unless more accurate information is available. Long-term losses may either be entered as a lump sum value, or the information required to calculate them may be entered.

5.5.6 Specify Tendon Profile (PT mode only)

This screen allows the user to specify the tendon profiles **(Fig. 5.5-9)**.

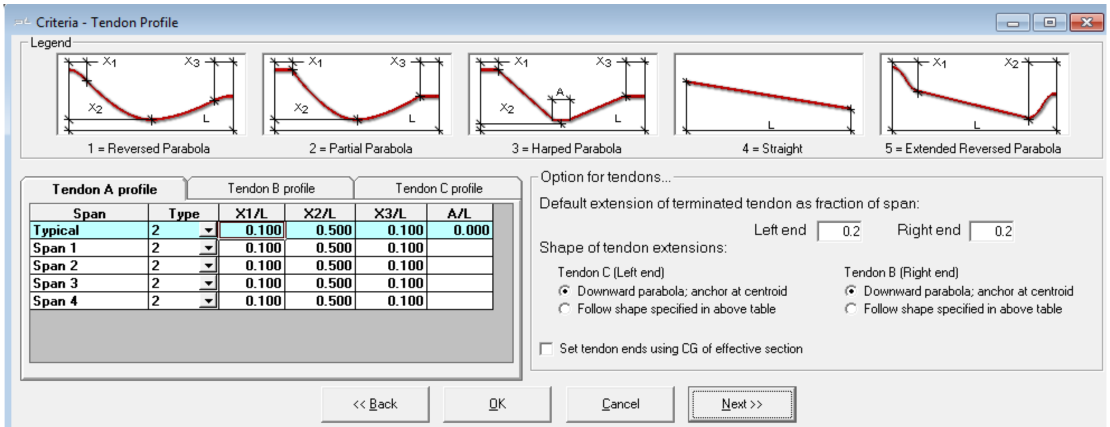


Figure 5.5-9 Criteria-Tendon Profile Input Screen

The parameters used to define the tendon are shown in the schematics at the top of the screen. The profile and values shown, a reversed parabola with the low point at mid-span and inflection points at span length/10, are the defaults. These are typical industry defaults; they will be appropriate for most designs with essentially uniform loading. Note that if a non-standard profile, i.e. a low point at somewhere other than mid-span is used, this must be clearly called out on the structural drawings. Transfer girders and slabs with heavy concentrated loads may require a harped profile. The low point is usually specified to coincide with the column being transferred or the concentrated load.

Tendons in the model can have up to three different profiles. To define all three profiles, fill in data in the tables of tab Tendon A, Tendon B, and Tendon C.

You have an option to define length and shape of tendon extension that terminates at the interior of a member.

You have the option to select the option to set tendon ends using CG of effective section. If left unchecked, the tendon ends will be set using the CG of the gross cross-section.

5.5.7 Specify Minimum Covers

This screen is used to specify minimum covers for both the post-tensioning tendons and mild-steel reinforcement **(Fig. 5.5-10)**.

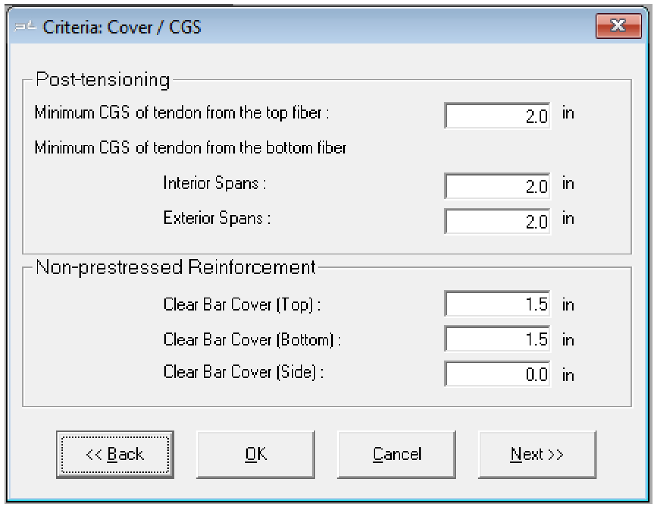


Figure 5.5-10 Criteria – Cover/CGS Input Screen

Note that the cover for the pre-stressing steel is specified to the center of gravity of the strand (cgs) whereas, for mild steel, it is clear cover. For ½ in. strand, the clear cover on the tendon will be ¼ in. less than the distance to the cgs. Only the Non-prestressed Reinforcement cover input applies to designs performed in RC mode.

5.5.8 Specify Minimum Bar Length

This screen is used to define how mild steel reinforcement bar lengths are calculated (**Fig. 5.5-11**).

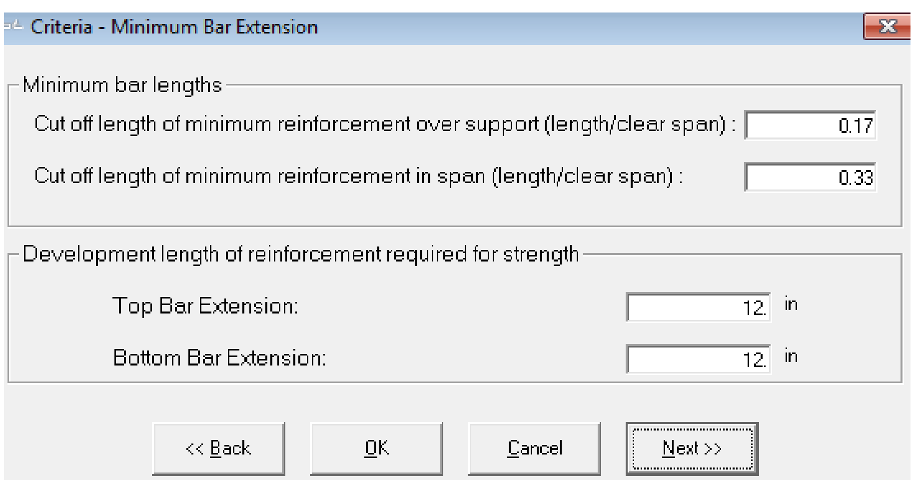


Figure 5.5-11 Criteria-Minimum Bar Extension Input Screen

The values entered for cut-off lengths apply only to PT designs and are used to calculate top and bottom bar lengths when minimum reinforcement requirements govern. The lengths of bars required for ultimate strength are calculated from the reinforcement necessary to supplement post-tensioning at 1/20th points along each span. Bar lengths for steel required for ultimate strength will include the specified extension lengths. When performing designs in RC mode, only the development length option for strength reinforcement is active and applicable.

5.5.9 Specify Reinforcement Curtailment

The program determines minimum bar lengths for calculated reinforcement from location of zero moment and with respect to the input for **Minimum Bar Extensions** and **Development Lengths** as described in the previous section. In addition to these settings, the program includes customizable **Rebar Curtailment** settings for top and bottom longitudinal reinforcement. Figure 5.5.-12 and 13 show the dialogue windows for the input settings related to rebar curtailment. When used, these settings supersede the default reinforcement output. Where the calculated lengths related to curtailment are less than the stored default lengths, the program ensures a safe design by using the default lengths.

Entries can be made for Cantilevers, Exterior spans, and Interior Span conditions for top and bottom rebar placement. A fraction of the total enveloped area of steel required can be designated for **Longer Bars** and **Shorter Bars**. Bars can be input as any fraction of the span length to which they belong. Note that the total fraction of span length for longer bars must be greater than that for shorter bars. For example, the program would not accept a value of 0.25*L for a Longer Bar and 0.33*L for a Shorter Bar. A minimum number of bars can be entered as part of the input. If the enveloped bar requirement is calculated as 1#8 bar and a user has made an entry for 3 bars minimum, the program uses 3#8 bars and applies the curtailment input to the equivalent area of steel calculated from 3#8 bars. Note that the curtailment rules do not apply to user-defined base reinforcement but only that reinforcement calculated by the program.

The program provides default options for curtailment rules based on **User-defined settings, ADAPT settings, ACI318 and EC2**. Curtailment rules can be turned off and reinforcement arrangement output defaults to rules based on the Bar Extension input as shown in **Figure 5.5-11**. Settings made to the Reinforcement Curtailment dialogue window can be saved as a template file and stored for broad user across multiple users. To save settings select the **File Save** tab under the **Curtailment File** settings.

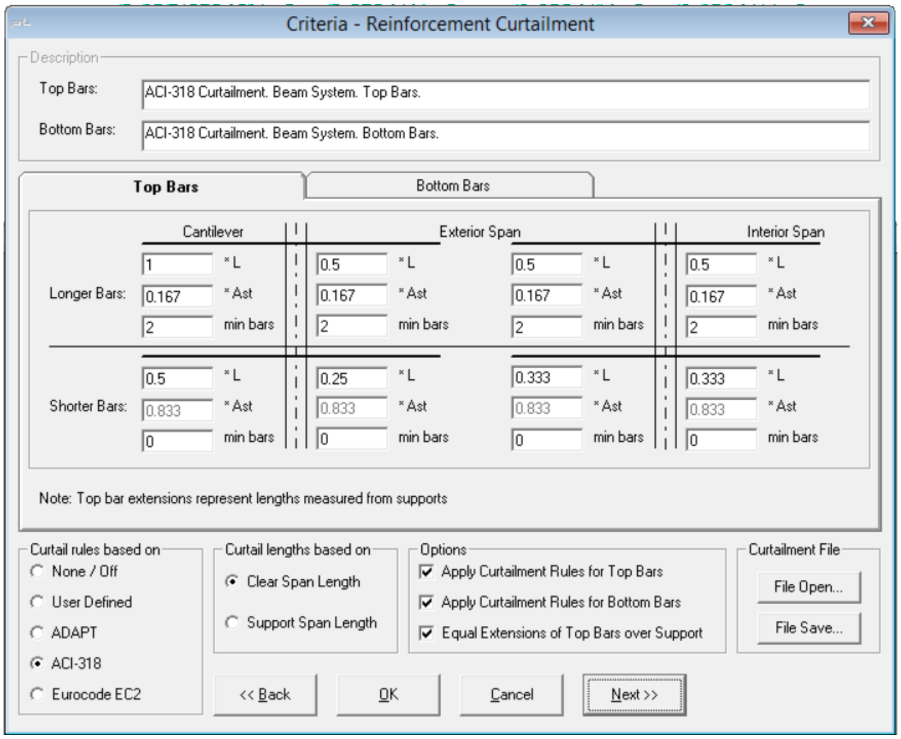


Figure 5.5-12 Criteria-Reinforcement Curtailment – Top Bars

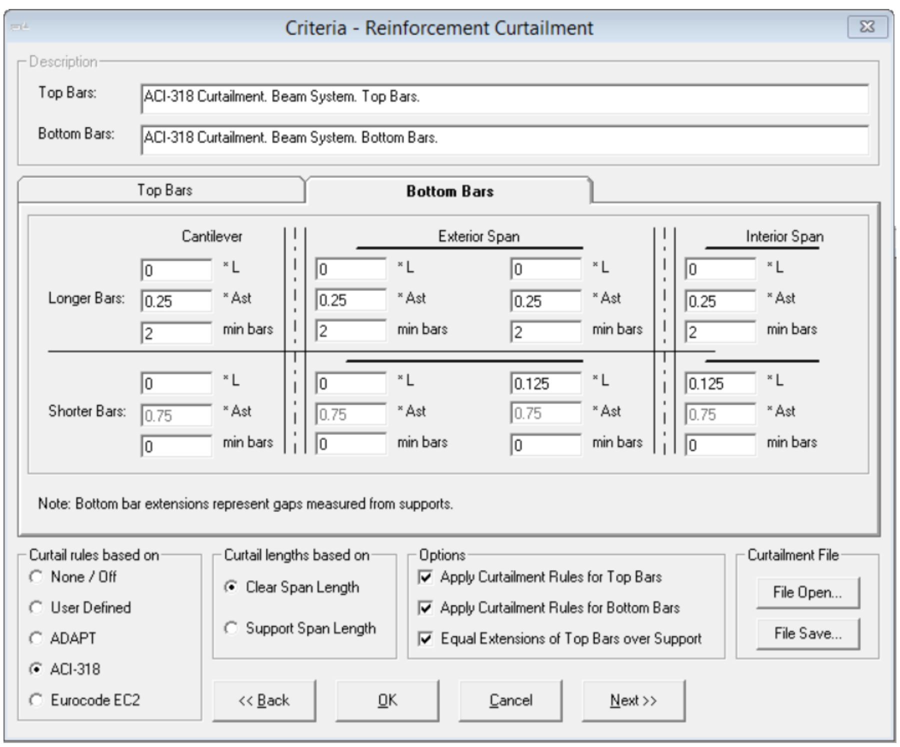
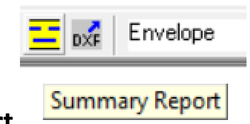


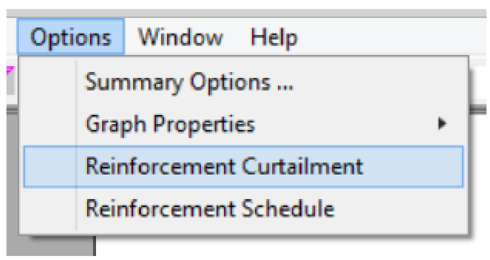
Figure 5.5-13 Criteria-Reinforcement Curtailment – Bottom Bars

The reinforcement output relative to settings made in the **Reinforcement Curtailment** input is realized in two areas:



1. **BuilderSum – Summary Report**

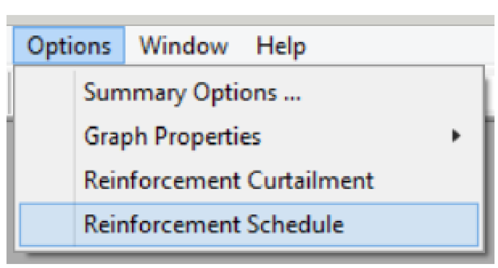
   When the **Summary Report** is generated, the reinforcement output show for Top and Bottom bars is calculated from the curtailment settings. The curtailment rules are **ONLY REFLECTED** when the **Envelope** of reinforcement is selected. If a **Service, Strength or Initial** load combination is selected, the reported reinforcement is based on the default values relative to the minimum bar extention and development length input. To make modifications within the Summary Report, select the **Options>Reinforcement Curtailment** selection to open the curtailment dialogue window. Modifications can be made and applied for real-time adjustments.

2. **Tabular Report Block 10.2 – Provided Rebar Report**

   The Provided Rebar report generated by use of the Report Generator outputs spans and specific locations, quantities and lengths of the provided reinforcement calculated by the program. Where curtailment is used in a design, the lengths are reflected in this report.

In **BuilderSum**, from the **Options>Reinforcement Schedule** selection, the reinforcement output can be generated in a schedule format as shown in **Figure 5.5-14**.



The schedule can be saved in .XLS format for collataing results from other ADAPT-PT or RC runs. The schedule contains: Project Information, Bar Mark, Bar Size, Bar Diameter, Bar Length, Shape Code, Bar Bend Tags, Area of steel for bars and Bar Weight.

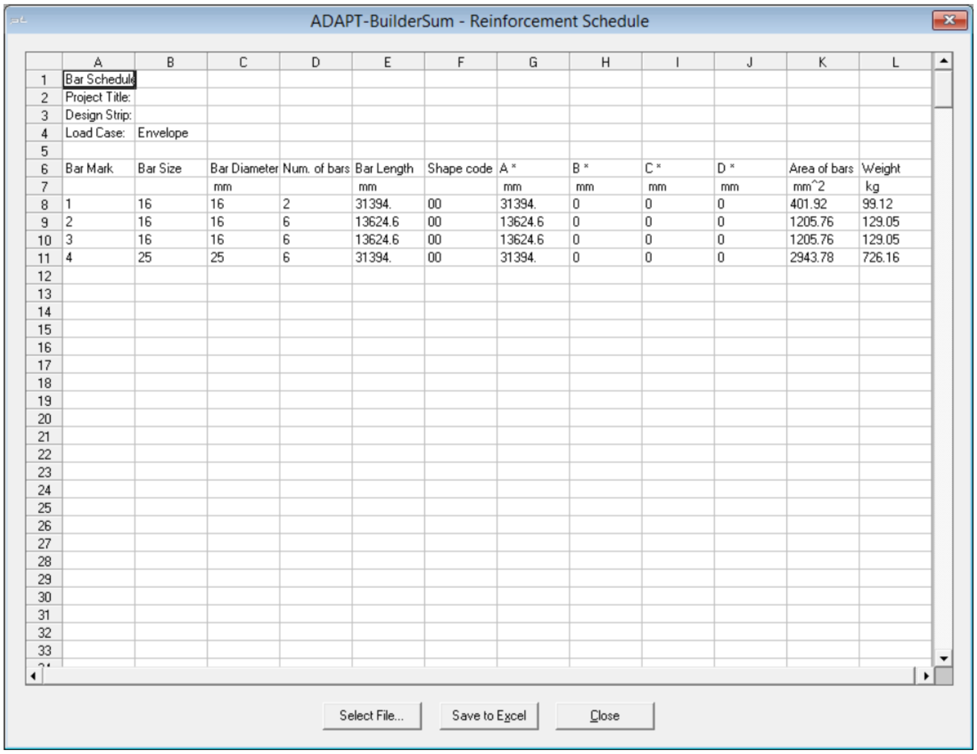


Figure 5.5-14 ADAPT-BuilderSum – Reinforcement Schedule

5.5.10 Specify Load Combinations

This screen is used to define the load combination factors for service, strength (ultimate), and initial load (PT mode only) conditions. It also gives access to the input screens for lateral loads and lateral loads combinations described in **Section 5.3.2** and is used to enter any applicable material factors or strength-reduction factors. The default values depend on selected design code.

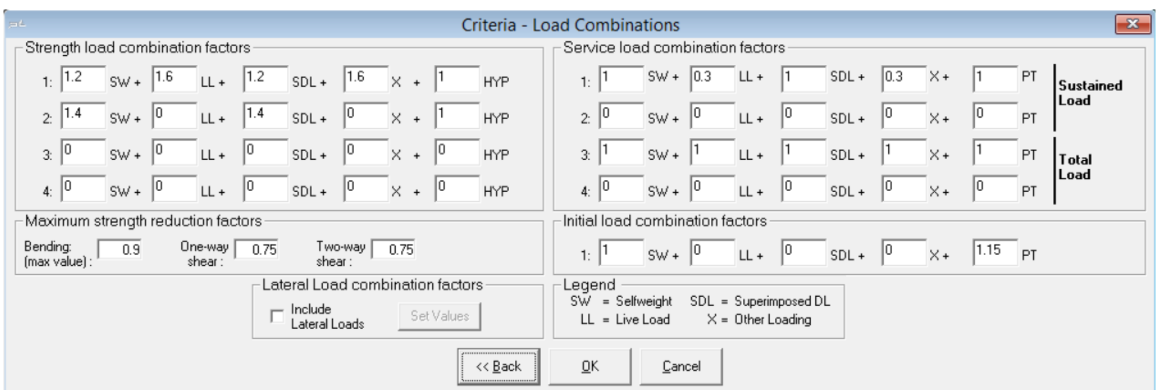
Note for RC mode, service combinations replicate those in PT mode. The program defaults to the same combinations shown below except for the **Initial load combination**. This is active only for PT mode.

The program automatically calculates minimum reinforcement for RC systems based on the design code selected and the system type for Sustained and Total Service conditions. Oftentimes, the minimum reinforcement requirements for these conditions is identical for RC systems. PT systems must be checked for serviceability requirements of pre-compression, tensile and compressive stress, and balanced loading for each service combination, hence, load combinations for Service conditions are active.

In the instance where a PT model contains spans or portions of spans that are RC (no post-tensioning force intersecting the design sections), the program is able to determine necessary Strength and Service reinforcement for the hybrid model and envelope the reinforcement accordingly.

For example, a two-way post-tensioned slab over supports requires 0.00075*Ac checked in each direction at the support. A two-way reinforced concrete slab is subject to minimum temperature and shrinkage steel that requires 0.0018*Ag. If two adjacent sections in a model are “PT” and “RC” sections, based on the positioning and termination of the tendons, the program reconciles the controlling reinforcement over the support and is able to envelope the correct amount.

For the calculations of ultimate (Strength) reinforcement at an RC section included in a PT model, the program’s design algorithm assumes zero force/stress and area for Fps and Aps and calculates the necessary reinforcement as it would in ADAPT-RC mode.



PT and RC mode

Figure 5.5-15 Criteria-Load Combination Input Screen

To define load combinations that include lateral loads check **Include lateral load** option and click **Set Values**. The Lateral Input Data window opens (**Fig. 5.5-16**).

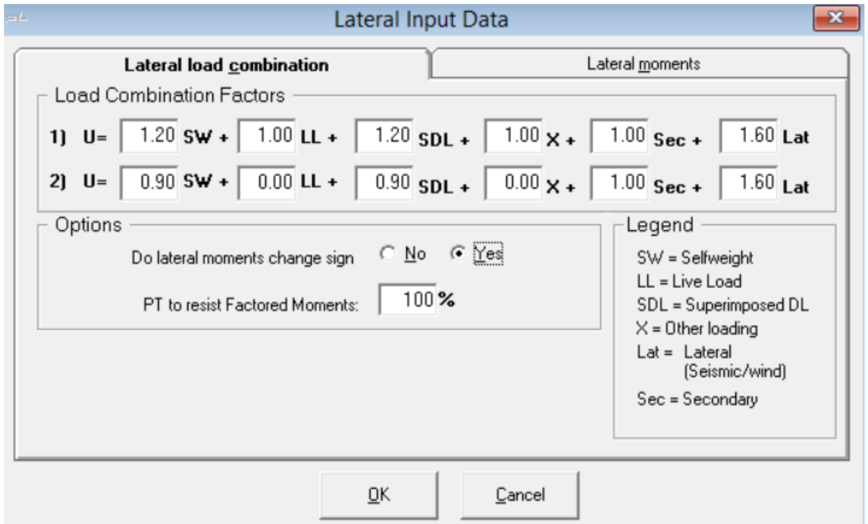


Figure 5.5-16 Lateral Input Data Input Screen

If you answer "Yes" to the **Do lateral loads change sign?** the program will internally consider a new load combination with the modified sign of lateral load and report results for it. **PT to resist Factored Moment** option is used to specify the percentage of contribution of post-tensioning to unbalanced moment due to lateral loads. Note that this option will not be available if in the **Design Settings** window you select to disregard contribution of prestressing in strength check.

6 Program execution Overview

The program can be executed either by selecting the **Execute** item on the Main Program window or clicking the **Execute Analysis** button, on the **Main Toolbar**.

The program begins by reading the data files and performing several preliminary data checks. If an error is detected, the program will stop and display a message box indicating the most likely source of the error. The data consistency checks are not exhaustive, however, which means that the user is ultimately responsible for ensuring that the data is entered correctly.

In RC mode, when the model analysis is run, the program will directly run through multiple calculations for the general frame analysis, deflections and reinforcement and return to the Main Program window. The results can then be viewed and/or printed.

In PT mode, the program can be executed in Automatic or Interactive mode. In the automatic mode, the program attempts to select a post-tensioning force and profile within the design bounds specified by the user. If a solution is possible, the program will complete the calculations and return to the Main Program window. The results can then be viewed and/or printed. If a satisfactory solution is not possible, the program will display a message box, which describes the problem and will switch to the interactive mode. The user can then decide whether it is possible to change the original design criteria and continue with the design.

The automatic mode begins by assuming the maximum drape for each span and determining the minimum force, which satisfies the maximum allowable tensile stresses. The same force is used for all spans. The force is then adjusted to meet the following requirements as specified by the user:

- Minimum percentage of dead loading to balance for each span.
- Minimum average precompression for each span.
- Maximum spacing of tendons (applies only to slabs).

After these initial adjustments, each span is checked for compliance with the following:

- **Maximum percentage of dead loading to balance:** if the balanced loading in any span exceeds the maximum percentage specified by the user, the program adjusts the tendon drape in that span to lower the balanced loading. It then recalculates the balanced loading and the related moments.
- **Average precompression and compressive stresses:** if either the average precompression or the compressive stresses exceed the maximum permissible values, the program will stop and display a message box. It then switches to the Interactive mode and displays the Recycle window.

The interactive mode gives the user an opportunity to optimize the design by adjusting the tendon forces and tendon drapes in each span. It can be executed using either the Force Selection or Force/Tendon Selection mode.

The program begins by going through the same calculations that it goes through for the Automatic mode. After it has determined an initial tendon force and profile however, it displays the **PT Recycling** window shown in **Fig. 6-1**

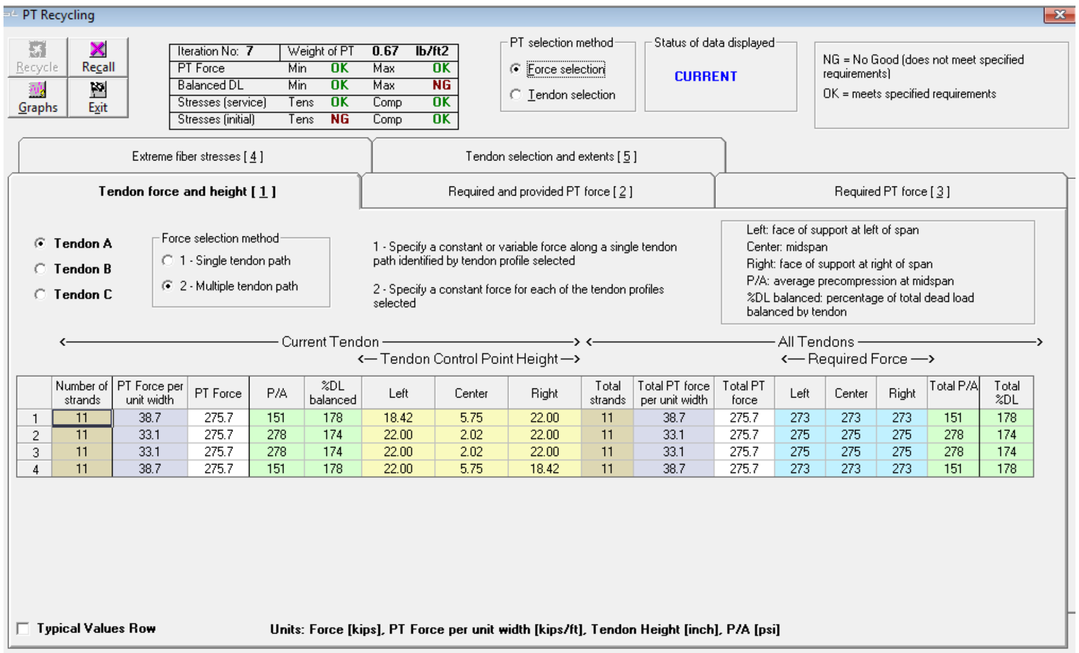


Figure 6-1 PT Recycling Window

The **PT Recycling** window is comprised of five tabs which display information about the post-tensioning design, a Design Indicator box which summarizes the status of the current design, a Status indicator and four control buttons: Recycle, Recall, Graphs and Exit.

6.1 PT Recycling Window Tabs

The PT Recycling Window tabs are:

- **Tendon Force & Heights Tab.** This screen allows the user to select the tendon profile, adjust the tendon heights and post-tensioning forces (**Fig. 6.1-1**).

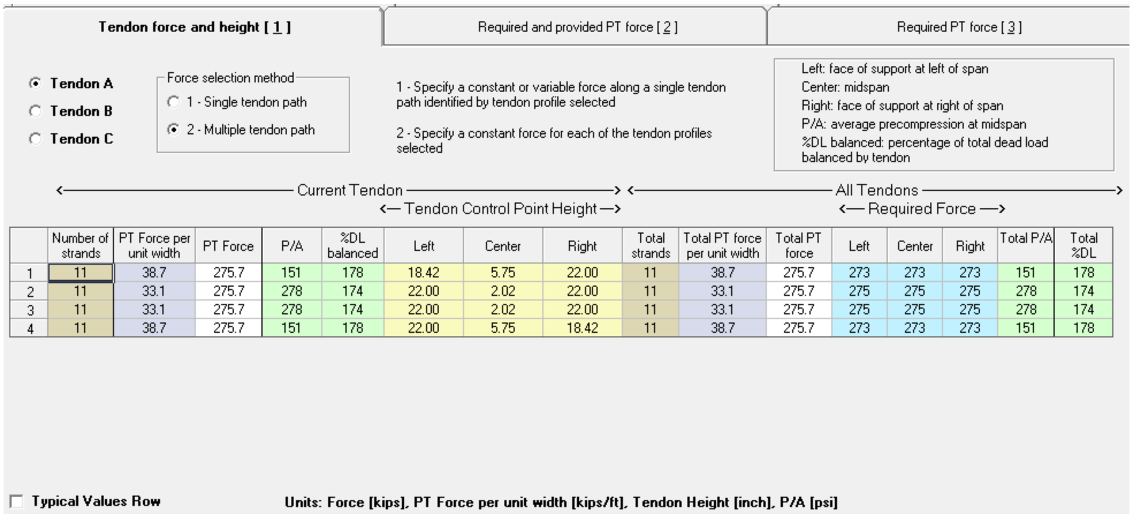


Figure 6.1-1

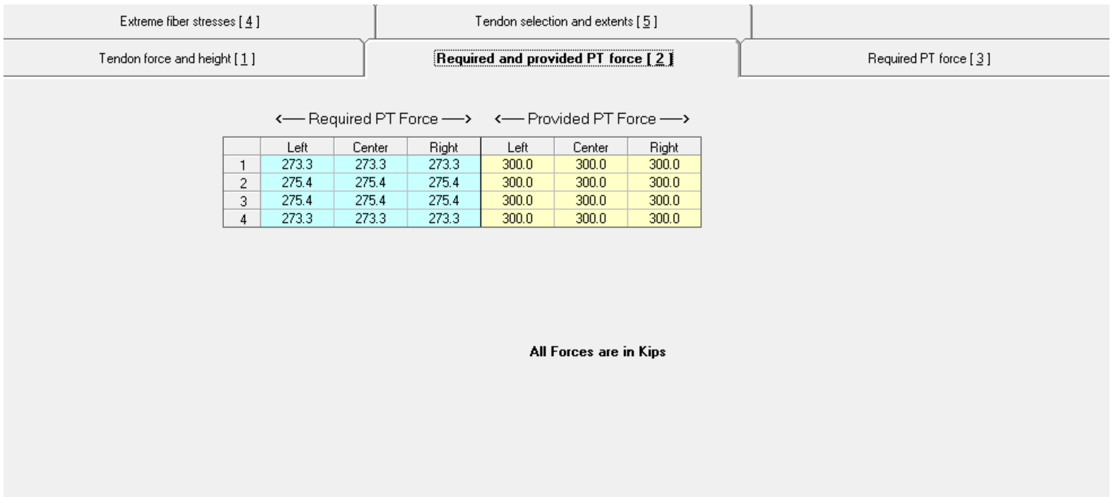
The table lists the post-tensioning forces at the midpoint of each span, the tendon heights at the left, center and right of each span, the average precompression at mid-span (P/A mid) and the percentage of dead loading balanced in each span (W_{bal} % DL) for current tendons and all tendons.

You can select either single tendon or multiple tendon paths for the force selection method. You can select the tendon (A, B or C) to evaluate when the Multiple Tendon Path is selected. If Single Tendon Path is selected, then only Tendon A will remain as an option for the analysis. For Multiple Tendon Path, the PT force must be the same for each tendon group. The user can modify the calculated force reported in **PT Force per unit width** and **PT Force** cells. For Single Tendon Path, the PT force can vary along the length of the structure by entering unique values in the cells.

Adjustments in tendon force, and/or tendon height may change the average precompression and the percentage of dead load balanced. These changes are reflected in the P/A and W_{bal} columns as the changes are made. To see how the changes, affect the stresses and average precompression at locations other than mid-span, however, it is necessary to recycle the window.

The Typical Values Row option allows for the inclusion of a typical values row in the Tendon force and height table. By activating this option, values entered in the TYP row, followed by hitting the Enter key, are populated into all the cells in that column.

- **Required and Provided PT Force Tab**. The PT forces tab shows the PT forces provided in the left, center, and right region of each span as well as the forces required in each region for minimum P/A and allowable tensile stresses (**Fig. 6.1-2**).

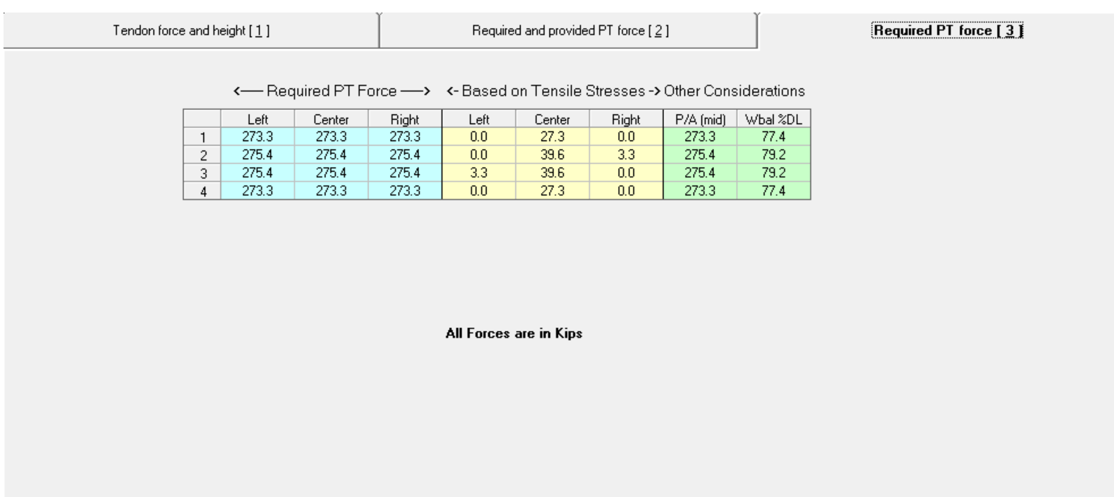


	Required PT Force			Provided PT Force		
	Left	Center	Right	Left	Center	Right
1	273.3	273.3	273.3	300.0	300.0	300.0
2	275.4	275.4	275.4	300.0	300.0	300.0
3	275.4	275.4	275.4	300.0	300.0	300.0
4	273.3	273.3	273.3	300.0	300.0	300.0

Figure 6.1-2

The post-tensioning force provided in each region is compared with the governing minimum force in that region as shown on the Required Forces tab. If the provided force does not envelop the required values, FORCE NG (No Good) is displayed in the indicator box at the top of the screen. Forces that are less than what is required will be highlighted in red in the Provided PT Force columns.

- **Required PT Force Tab**. This tab shows the required post-tensioning forces for only the most recently calculated profile (**Fig. 6.1-3**).



	Required PT Force			Based on Tensile Stresses			Other Considerations	
	Left	Center	Right	Left	Center	Right	P/A (mid)	Wbal %DL
1	273.3	273.3	273.3	0.0	27.3	0.0	273.3	77.4
2	275.4	275.4	275.4	0.0	39.6	3.3	275.4	79.2
3	275.4	275.4	275.4	3.3	39.6	0.0	275.4	79.2
4	273.3	273.3	273.3	0.0	27.3	0.0	273.3	77.4

Figure 6.1-3

Note that all values in the tables are forces and that these forces refer to the entire tributary width entered in the geometry input.

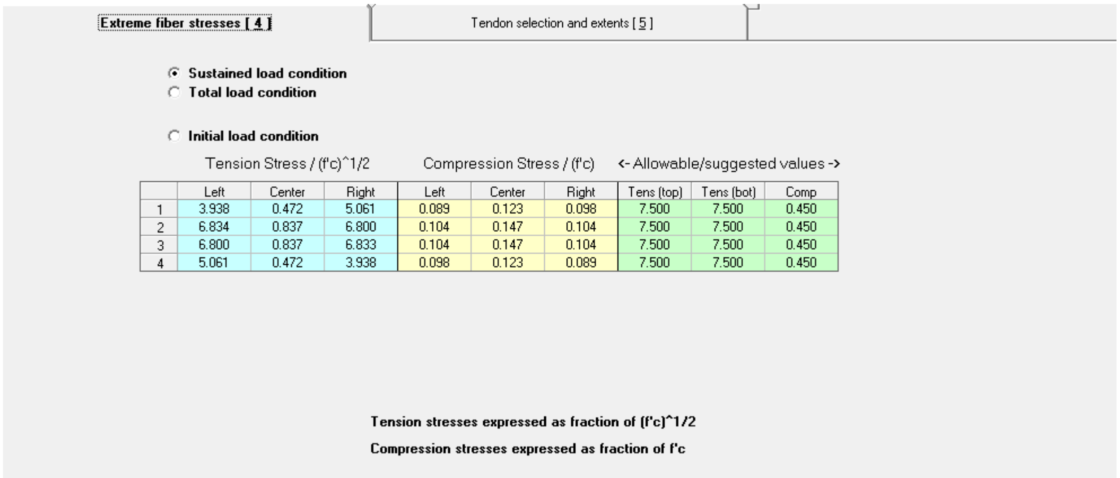
The window consists of three sections: The left (light blue) boxes display the governing forces for the left, center, and right region of each span. The force selected for each region is the largest required force based on tensile stresses in that region, minimum P/A, and minimum percentage of dead load to balance.

The middle (light yellow) boxes display the forces required for tensile stresses. If the moments in a particular region are such that no post-tensioning is required, a zero (0) is shown.

The first column of the right (light green) section is the post-tensioning force required to satisfy the minimum average precompression specified by the user, based on the member's cross-sectional area at mid-span (P/A mid). Average precompression P/A is not a function of the applied loading or tendon profile.

The second column of the right section (Wbal %DL) is the force required to provide an uplift equal to the minimum percentage of the total dead load specified by the user. The force required for each span depends on the tendon geometry and loading of that span. All the dead loads, including superimposed dead load, are summed for each span, regardless of whether they are self-weight, uniform or concentrated. Note that when calculating Wbal for display on this screen, the downward tendon forces are not included. This approximation is made only for the purposes of obtaining a rapid screen display. The actual computations of moments and stresses include all equivalent loads from each tendon.

- **Extreme Fiber Stresses Tab.** This tab shows the maximum tensile and compressive stresses in the left, center, and right regions of each span **(Fig. 6.1-4)**.



	Tension Stress / (f'c)^1/2			Compression Stress / (f'c)			<- Allowable/suggested values ->		
	Left	Center	Right	Left	Center	Right	Tens (top)	Tens (bot)	Comp
1	3.938	0.472	5.061	0.089	0.123	0.098	7.500	7.500	0.450
2	6.834	0.837	6.800	0.104	0.147	0.104	7.500	7.500	0.450
3	6.800	0.837	6.833	0.104	0.147	0.104	7.500	7.500	0.450
4	5.061	0.472	3.938	0.098	0.123	0.089	7.500	7.500	0.450

FIGURE 6.1-4

The stresses are calculated at 1/20th points and the highest stress in each region is displayed. If any of the stresses displayed are more than the allowable value, they will be highlighted in red. If the stress at any of the 1/20th points exceeds the allowable value, an NG warning is displayed in the indicator box. The location of the critical stress values can be determined by looking at the Stresses Recycle graph.

Tensile stresses are shown as a ratio of the square root of the concrete compressive strength at 28 days ($f'_c{}^{1/2}$). Compressive stresses are shown as a ratio of f'_c. The allowable stress values are shown for reference.

- **Tendon Selection and Extents Tab**. This screen is used to edit tendons. The options in this window will change depending on the PT selection method you chose in recycling screen.

  If you select Force selection, the screen will allow you to change the stressing ends of all the tendons and tendon extents of additional tendons B and C (**Fig. 6.1-5**). To change tendon extents, position the mouse cursor over the tendon end and drag the end to its new location. To change tendon ends (Stressing/Fixed), position the mouse cursor over the tendon end, left click while holding down Shift key. You can also use the table to the side of the tendon layout to enter/change the locations of tendon types B and C. These tendons must be at least 1 span length long but can be located anywhere along the length of the member. The values of the Left End location start at 0.0, at the far-left end of the modeled structure, and the Right End Location will vary, up to the number of spans modeled. The table values will update as the mouse is used to change the tendon extents graphically, and the graphical view will update if the values are entered into the table.

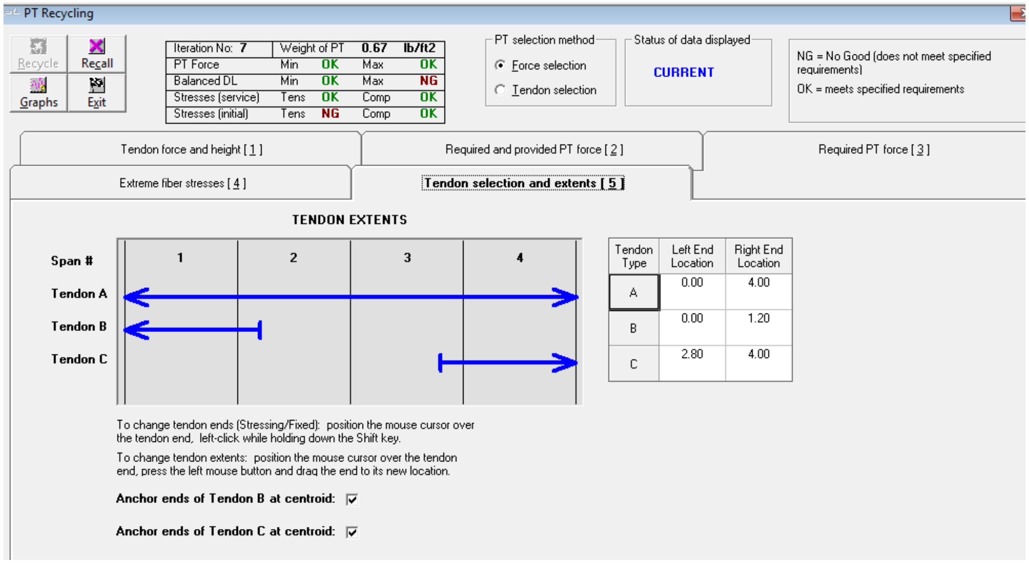


Figure 6.1-5

If you select Tendon Selection, you will be able to edit the number of tendons, change the length of short tendons and stressing ends (**Fig. 6.1-6**).

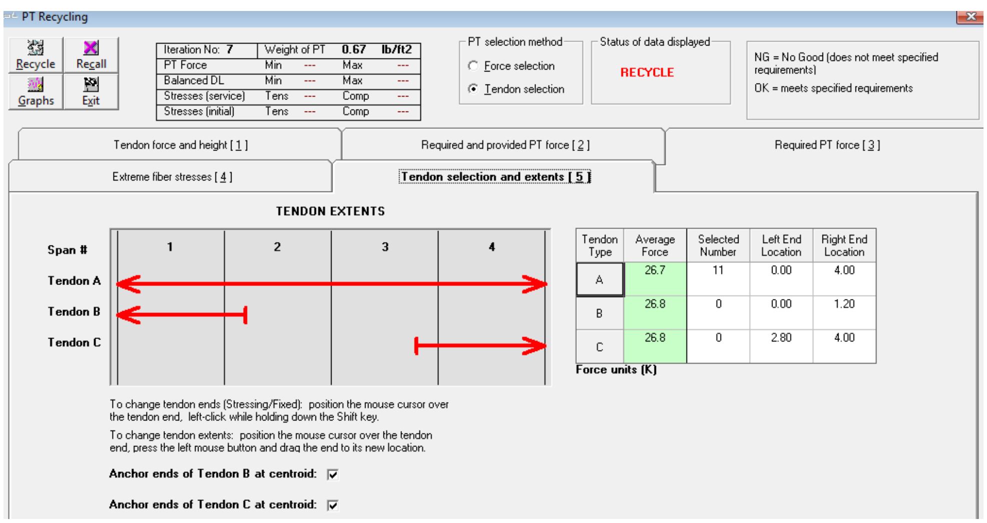


Figure 6.1-6

The tendon ends and extents can be modified as described above. In this case, the table also shows the Average Force and Selected Number of tendons for each tendon group. The average force in each strand is the force after all losses. Note, however, that the average forces are not actually used in the calculations. They are displayed to provide the user with a measure of the relative efficiency of each strand type. You can edit the number of strands per each tendon type, and the extents of the short tendons. You can choose to anchor ends of short tendon types B and C at the centroid of the concrete section. Once all changes have been made, Recycle the screen to make sure all changes are included in the calculations.

6.2 Design Indicator Box

Iteration No: 9	Weight of PT		0.73	lb/ft2
PT Force	Min	OK	Max	OK
Balanced DL	Min	OK	Max	NG
Stresses (service)	Tens	NG	Comp	OK
Stresses (initial)	Tens	NG	Comp	OK

Figure 6.2-1

The status of the current design is summarized and displayed in the Design Indicator box at the top center of the Recycle window. Each design check is identified as either OK or NG (No Good). For cracked designs performed using the European code – EC2, the program will also indicate if the calculated stress exceeds the allowable tensile or compressive stress by CR (Cracked).

The items displayed in the Design Indicator Box are as follows:

- **Iteration No: #** . Each time a force or tendon height is adjusted and recycled, the program recalculates the related balanced loadings, moments, stresses, average precompression and percent of dead load balanced. Each set of calculations is referred to as a cycle. The number of cycles executed for a particular design is shown in the Iteration block. In most cases two to three cycles are adequate to arrive at an acceptable solution. It is rarely necessary to exceed five cycles.

- **Weight of PT: ### lb/ft^2 or ### kg/m^2.** The weight of post-tensioning strand required to provide the selected forces is estimated and displayed in either pounds per square feet or kilograms per square meter.

  The weight is estimated as follows: The force supplied by each strand is calculated based on its cross-sectional area and final effective stress, both of which are values input during data entry. The number of strands required to provide the forces shown on the Tendon Forces and Heights tab is then determined. The actual length of each strand is assumed to be its calculated length plus 3 feet (1 meter) to allow for a stressing tail. If the force changes between successive spans, it is assumed that the larger force extends over the common support and the tendons are anchored at 1/5 of the next span if otherwise specified by user. If the forces are modified, the weight is recalculated and displayed after the window is recycled.

- **PT Force min OK   PT Force max OK**. This block compares the average precompression at mid-span with the minimum and maximum values entered by the user. If the average precompression is above or below the specified limits, an NG is displayed.

  Note that although the PT Force indicator considers the P/A all along the span, this block only considers the P/A at mid-span. If the P/A is above or below the specified limits in a support region, the P_{min} and P_{max} indicators will show OK, however the PT Force indicator will show NG.

  In two-way slabs with drops or transverse beams the cross-sectional area at the supports will be much larger than the cross-section at mid-span. Providing the minimum P/A at the supports may result in a much higher PT force than necessary. The program determines the required force for the min/max range based on the cross-sectional area at mid-span. This may result in a precompressive stress at supports being lower than the specified minimums.

  The PT Forces tab **(Fig. 6.1-2)** shows the post-tensioning force required in each of the three regions of each span.

- **Balanced DL Min OK, Balanced DL Max NG**. The total upward force of the tendon (Wbal) in each span is computed from the post-tensioning force in span and the tendon geometry in the span. This upward force is compared with the total dead loading on the respective span. An OK for both Wbal Min and Wbal Max means that the ratio of balanced loading to the total dead loading is within the limits specified by the user in all spans. The percentage of dead load balanced in each span is shown on the Tendon Force & Heights tab **(6.1-1).** The force required to balance the specified minimum percentage of dead loading is shown on the Required Forces tab **(Fig. 6.1-3)**.

- **Stresses (service) or (initial): Tens OK, Comp NG**. This block compares the tensile and compressive stresses with the allowable values specified by the user. The maximum stresses in each span are shown on the **Extreme Fiber Stresses** tab **(Fig. 6.1-4)**. If the compressive stress or tensile stress exceeds the allowable limits, the values will show in red in that tab, as appropriate, under Sustained, Total, or Initial load conditions. In addition to this the program will prompt warning message (**Fig. 6.2-2**) if the compressive stress exceeds allowable stress for the initial condition.

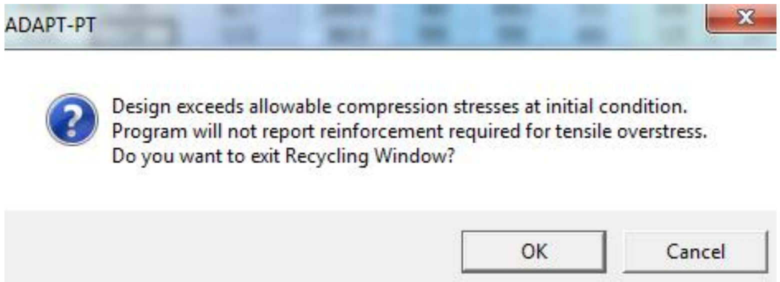


Figure 6.2-2

6.3 Recycle Window Control Buttons

The Recycle button causes the stresses and required forces along the member to be recalculated based on the current tendon profile and forces.

If changes are made to either the tendon profile or force in any span, the status indicator at the top right of the Recycle window will begin to flash. Once all the changes are made, click on the Recycle button to update all the tabs, the Design Indicator box, and the Recycle Graphs.

The Recall button allows the user to undo editing changes by recalling the tendon forces and profile from the previous recycle. After selecting Recall, the window must be recycled again to update the tabs, the Design Indicator box, and the graphs. If the Single Tendon Path was originally selected, with non-constant PT force, and then Multiple Tendon path is selected, the program will give you the warning shown in **Fig 6.3-1**. You must click cancel and select single tendon path to keep the non-constant PT force, if you click "OK" force will be made constant due to multiple tendon path selection.

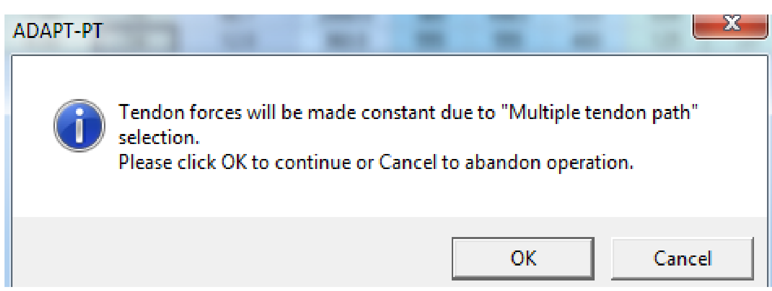


Figure 6.3-1

The Graphs button displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces. The Recycle graphs are shown in **Fig. 6.3-2.**

The graphs are as follows:

- **Tendon Height**. The Tendon Height graph can be used as a means of verifying that the tendon profile is at least reasonable. This graph allows the user to see the tendon profile either by itself or as it relates to the member elevation (concrete outline). This can be helpful for finding input errors such as a tendon profile that extends outside the member, or a profile that is not continuous. The concrete outline shows all steps, drop caps/panels, transverse beams, and changes in thickness.

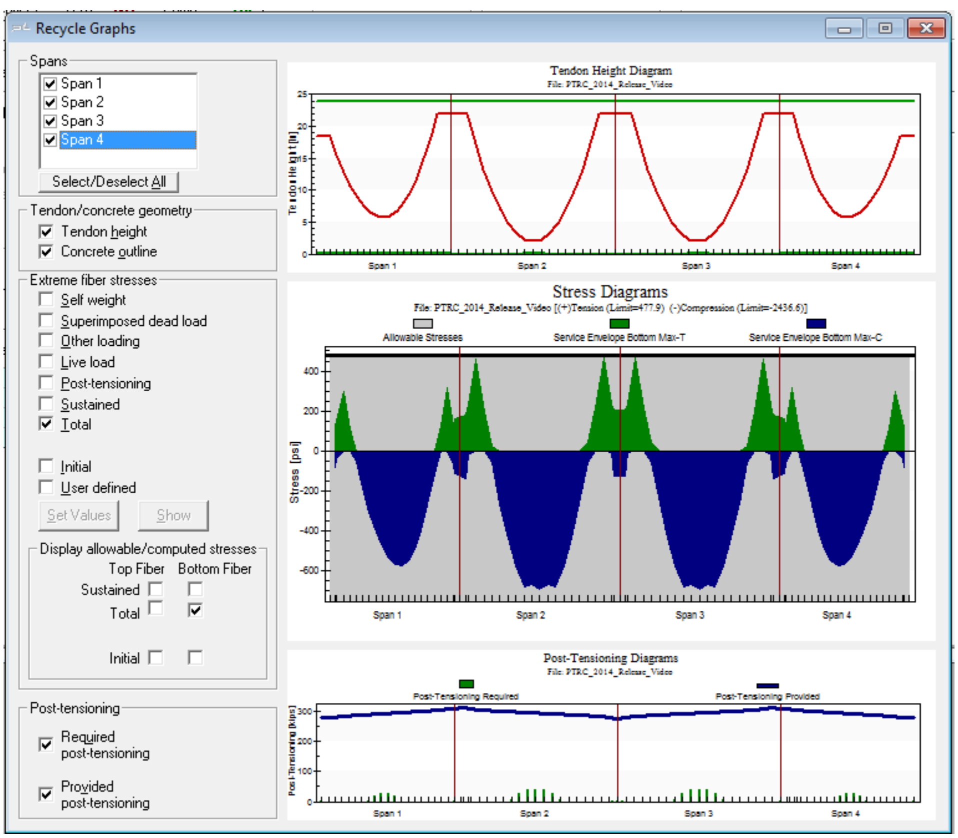
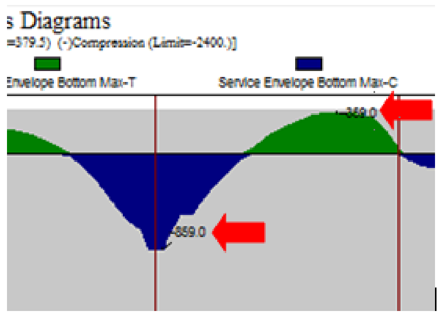


Figure 6.3-2

- **Stresses**. This graph plots the maximum compressive and tensile stresses at the top and bottom face of the member. All types of loadings can either be shown separately or combined. The **Display allowable/computed stresses** options show the combined stresses along with an envelope of the allowable stresses. The graph provides easy interpretation of stress results and clearly shows if stress limits are exceeded. All stress graphs in Recycler and BuilderSum report the max and min stresses for the system.



- **Post-tensioning**. This graph shows the required and provided post-tensioning force at 1/20th points along each span.

  The graphs may be configured to show only certain spans and values by clicking on the check boxes at the left of the window. To maximize a graph for detailed viewing or change the display options, right-click on the desired graph and use the editing menu that opens.

Selecting the **Exit** button closes the **PT Recycling** window and starts calculations of internal forces, deflection and reinforcement based on the most recent tendon force and profile selection. At the conclusion of the calculations, the user is returned to the Main Program window. The Results Report, the PT Summary Report and the Results Graphs may then be viewed and/or printed.

Note: If force or profile adjustments are made and you did not click on **Recycle** button before exiting, the program will automatically do a **Recycle**.

6.4 PT Selection Method

If you select **Force/Tendon Selection** option in the **Criteria - Calculation Options** screen you may choose between the **Force selection** and **Tendon selection** modes in the **PT Recycling** window (**Fig. 6.4-1**)

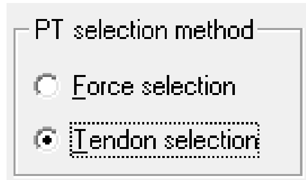


Figure 6.4-1

6.4.1 Force Selection Method

It assumes that a tendon will be assigned a final and constant effective force equal to the jacking force minus all stress losses expressed as a single representative value.

6.4.2 Tendon Selection Method

The Tendon Selection method is a newer, more accurate procedure than force selection method. In the Tendon Selection method, the post-tensioning force is assumed to vary along the length of the tendon. The variation accounts for stress losses in the tendon due to both immediate and long-term effects. It also includes consideration of the interaction between the various sources of loss. It is thus more accurate than procedures which account for losses as a lump sum approximation.

In Tendon Selection mode, the actual number of strands, as opposed to effective forces, may be specified. The user can see what the final stresses will be and can adjust the number of strands, short tendon locations, the tendon profiles, and the stressing ends, as necessary. At each design section along a span, the program performs an analysis based on the post-tensioning force at that section. Consideration is given to both short-term (friction, seating loss) and long-term (elastic shortening, creep, shrinkage, and relaxation of the prestressing steel) stress losses.

If the tendon profile is altered, friction and long-term losses are recalculated, and the revised tendon forces are used for the computations. If the tendon forces have changed significantly, however, the selected profile may not be satisfactory. The solution thus becomes iterative since subsequent changes to the profile will also result in changes to the tendon forces. The iteration is automatically continued until an acceptable solution is reached.

6.4.2.1 Description of Features

Tendon types. For each member, up to three tendon types, A, B, and C, may be specified. Each type can be configured to have a different length and different stressing/fixed ends. A given tendon type may include one or more strands.

Figure 6.4-2(a) shows a five-span beam with three different tendon arrangements. Tendon A extends the entire length of the beam and is stressed at both ends. It is shown in **Fig. 6.4-2(b)** as a straight line with two arrowheads representing the stressing ends.

The other two tendon types, B and C, start at either end of the beam and extend only part way through the member. The short vertical lines signify a fixed (non-stressing) end.

Figure 6.4-2(c) illustrates the shapes that the different tendon types can assume. Tendon type A must extend from one end of the member to the other. It can be stressed at one or both ends. Tendons types B and C can be configured the same as A, the same as one another, or completely different. They can be stressed at one or both ends. So long as Tendon types B and C are in at least one span, the tendon ends can be anchored internally and are not required to be fixed at one end.

Under normal conditions, the three tendon types will be configured differently. A post-tensioned member may not need all the three tendon types, however. Many members have only a Type A tendon. Type B and C tendons are typically configured to provide additional post-tensioning in end spans if necessary.

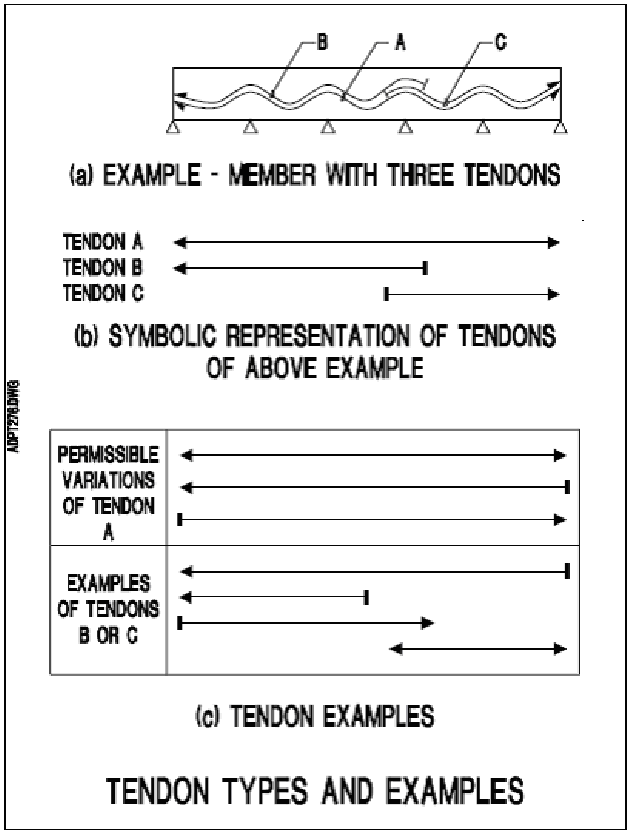


Figure 6.4-2

The number of strands in each type of tendon, and consequently the force in each tendon will usually be different. Tendons can have different profiles.

Stress Loss Calculations. There are two types of prestress losses:

- Immediate losses which occur at the time the tendon is stressed, and
- Long-term losses which may continue for several years.

The final effective force in the tendon is the jacking force minus all losses.

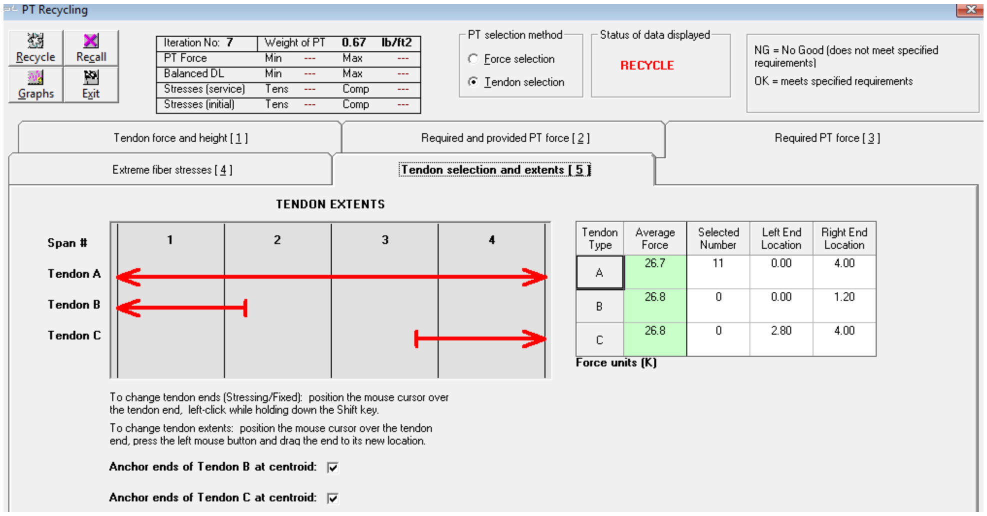
The immediate losses, friction, and anchorage seating are calculated based on the user-input friction parameters together with the tendon's profile and stressing configuration. The stress in the tendon immediately after it is seated, with due allowance for friction and seating loss, is referred to as the initial or lock-off stress. Although friction coefficients are different for grouted (bonded) and unbonded systems, the friction loss computations are essentially the same.

There are three options for long-term stress loss calculations:

- **Lump sum entry**. A lump sum value may be calculated by the user and entered during data input. The effective stresses in the tendon are calculated by subtracting this value from the initial stresses. Since the friction and seating losses cause the initial stresses to vary along the tendon, the effective stresses will also vary.
- **Long-Term Loss calculations for unbonded tendons**. For unbonded tendons, the strain in the tendon at any given point is not directly related to the local strain in the concrete. The program can calculate an average long-term loss value for the entire tendon based on the average precompression in the member and expected losses due to shrinkage, creep, elastic shortening and relaxation of the prestressing steel. The effective stresses in the tendon are calculated by subtracting the average long-term loss value from the initial stresses.
- **Long-Term Loss computation for grouted tendons**. Long-term stress losses in grouted tendons are a function of the local strain in the concrete. Long-term losses are thus computed at 1/20th points along the tendon. The effective stress at each point is the jacking stress minus the friction, seating loss and long-term stress losses at that point. The long-term losses are stored in the file LTLOSS.DAT. This is a text file and can be viewed with any text editor or word processor.

6.4.2.2 Description of Execution

The Tendon Selection & Extents tab (**Fig. 6.4-3**) becomes active when the Tendon Selection mode is chosen.



Tendon Type	Average Force	Selected Number	Left End Location	Right End Location
A	26.7	11	0.00	4.00
B	26.8	0	0.00	1.20
C	26.8	0	2.80	4.00

Figure 6.4-3

The right side of the tab shows the average force in each strand and the number of strands selected for each tendon type. The average force in each strand is the force after all losses. Note however that the average forces are not actually used in the calculations. They are displayed to provide the user with a measure of the relative efficiency of each strand type.

The left side of this screen shows a symbolic representation of the spans and the tendon layout. The default layout is a Type A continuous tendon stressed at both ends of the member, a Type B tendon stressed from the left and extending over the leftmost span and a Type C tendon stressed from the right and extending over the rightmost span.

You can edit the post-tensioning layout by:

- **Adjusting the tendon profiles**. Tendon heights are edited on the Tendon Force & Heights tab. Note that when the Tendon Selection option is active, you cannot access the Force column on this tab. In the Tendon Selection option, forces are calculated based on the number of strands and the final stresses in the strand.
- **Editing the number of strands in a tendon type**. The number of strands to use for each tendon type is shown in the Selected Number column. These numbers may be changed independently of one another. To delete a tendon type, set the number of strands to zero. To add

a tendon type, enter the number of strands to use for that type.

- **Changing the stressing ends and/or extent of the tendons**. To change a tendon end from dead to stressing or stressing to dead, hold down the Shift key and left click once at the end of the tendon. Clicking a second time will change the tendon back to its original configuration. Note that the tendon must have at least one stressing end. To change the extent of a Type B or C tendon, position the cursor over the tendon end, hold down the left mouse button and drag the end to the desired location. The table to the side of the tendon layout can also be used to change the location of tendon types B or C. These shorter tendons must be at least 1 span length long but can be located anywhere along the length of the member. The table will automatically update as the mouse is used to update the tendon extents graphically, and the graphical view will update if the values are entered into the table.

If any changes are made to the tendon profiles or number of strands, the window must be recycled to recalculate the force provided. There is no limit on the number of changes that can be made or the number of times the window can be recycled. Once an acceptable post-tensioning layout has been determined, select Exit to continue with the calculations.

Clicking on the Force Selection button at the top of the Recycle Window will toggle the program back to the Force Selection mode. Any changes that have been made while in the Tendon Selection mode will be reflected in the forces shown on the Tendon Force & Heights tab.

7 View/Validate Results

After the analysis is executed a graphical report can be generated. The ADAPT BuilderSum module opens and enables you to generate comprehensive graphical reports for each ADAPT-PT/RC run. You have an option to display and print results graphs for each load combination and generate a report that summarizes all post-tensioning parameters, rebar requirements and shear checks on a single page of output.

7.1 ADAPT-BuilderSum Screen

To invoke ADAPT BuilderSum, click on the Open the BuilderSum button, , or select the PT or RC Sumary menu item from the View menu in the Main program window. The window will open as shown in **Figure 7.1-1**.

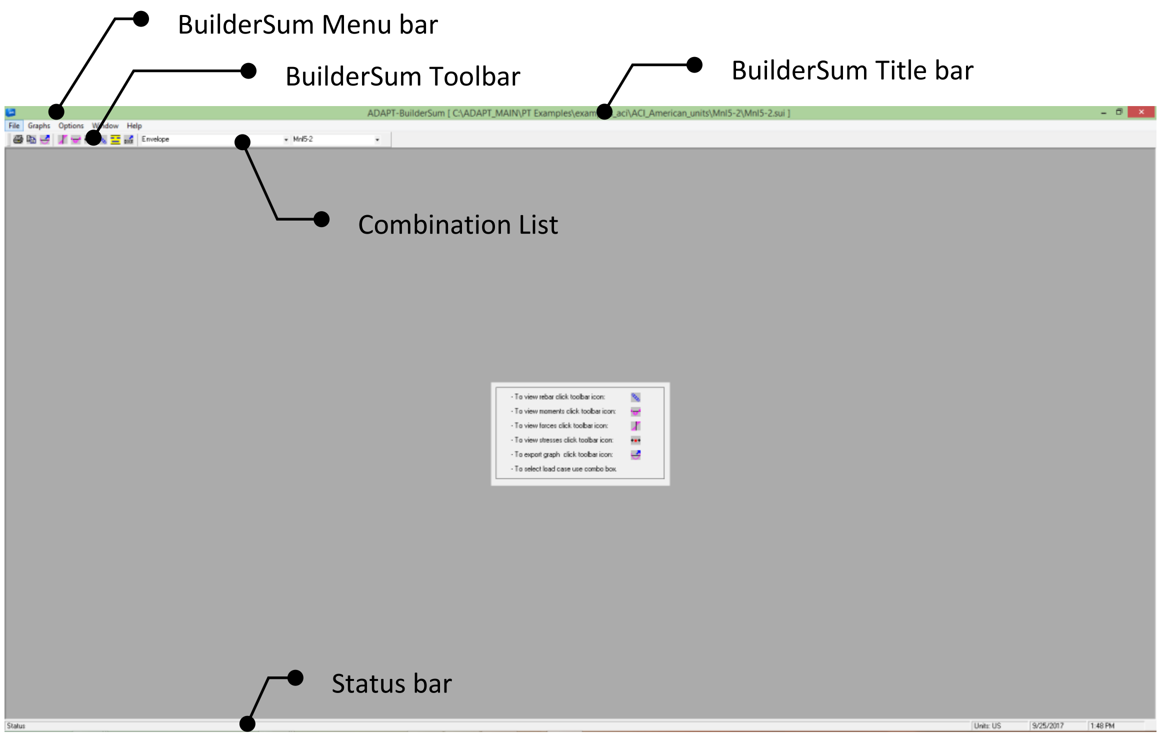


Figure 7.1-1

BuilderSum Title Bar. Contains program name, and name and location of the opened file.

BuilderSum Menu bar. Menu bar lists all available menus in ADAPT-BuilderSum module.

BuilderSum Toolbar. This toolbar contains all available tools in the ADAPT-BuilderSum screen.

Status Bar. Status bar gives you information about units, current date, and time.

Combination List. This is a drop-down list that contains all available load combinations.

7.2 ADAPT-BuilderSum Menu Items and Corresponding Tools

All options that can be accessed by the BuilderSum program menus are listed below. For the commands that might be activated using the toolbar, the appropriate icon is displayed next to the feature.

7.2.1 File Menu

Export Graph. Allows you to export the currently active result graph or summary report as either a bitmap (.BMP) file or a Windows metafile (.WMF). The graph or report must first be set up with the desired information and in the desired format.

Export to DXF File. Allows you to export the tendon profiles to a Drawing Exchange Format (DXF) file. If installed on your computer, AutoCAD will launch, and the file will automatically open after the file is created.

Print. This tool prints active result graph with frame containing project information or active Summary Report.

When you print a graph the program will display the Print Graphs Option screen (**Fig. 7.2-1**) where you have an option to select sheet orientation and add additional comments that will appear at the bottom of the graph (**Fig. 7.2-2).**

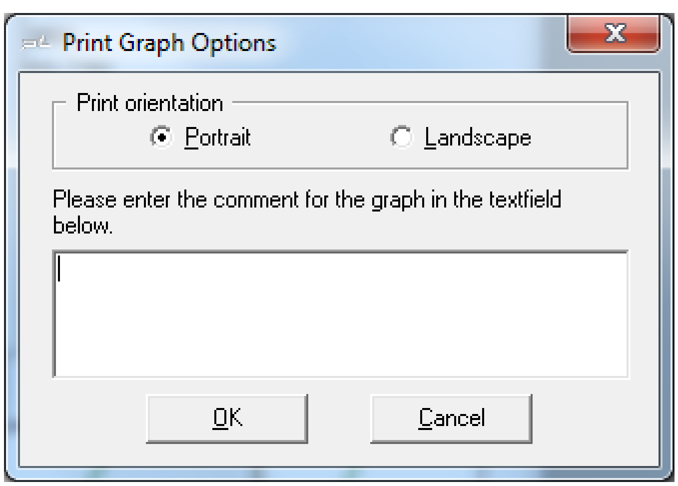


Figure 7.2-1

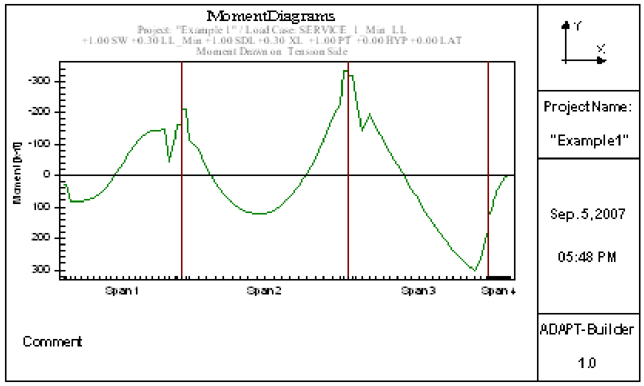


Figure 7.2-2

Page/Print Setup. This option allows you to specify the printer, set the margins or the orientation of the reports.

Exit. Exits the BuilderSum Module.

7.2.2 Graphs Menu

Summary. When you select this option the Summary Report window will open with a default format for the Summary report as shown in **Fig. 7.3-1**.

View All Graphs. This option will show all available graphs for selected load combination or envelope.

Forces Diagram. This tool displays forces diagram for selected load combination or envelope (**Fig. 7.2-3**).

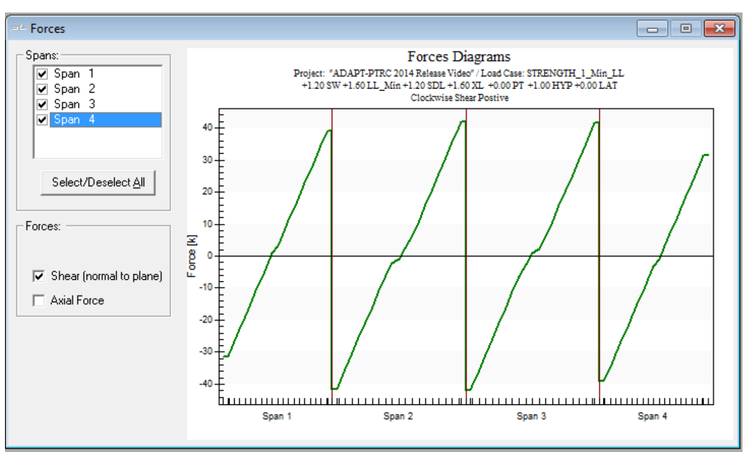


Figure 7.2-3

Moment Diagram. This tool displays bending moment diagram for selected load combination or envelope (**Fig. 7.2-4**).

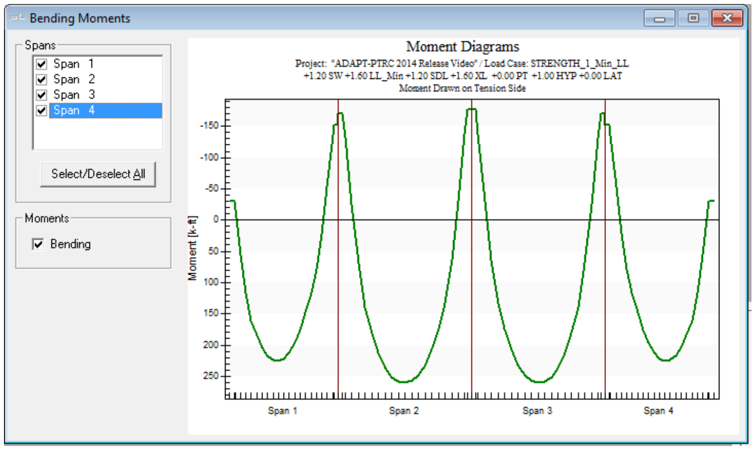


Figure 7.2-4

Stresses Diagram (PT mode only). Displays calculated stresses for selected initial or service load combinations (**Fig. 7.2-5**).

Note: this tool will not be active if you select strength load combinations.

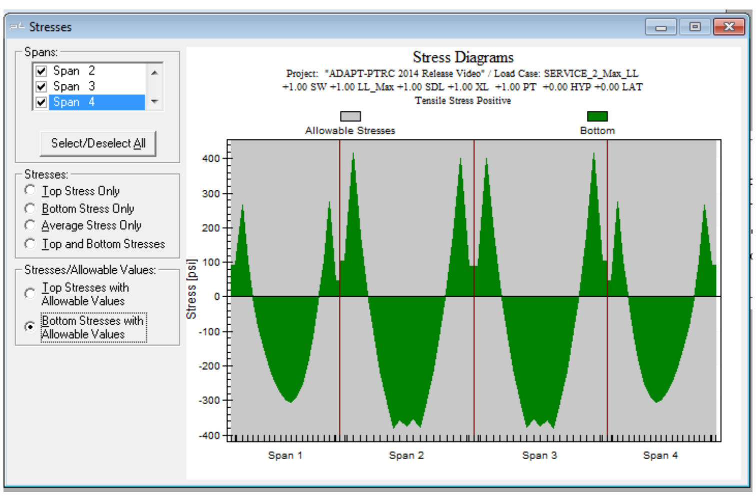


Figure 7.2-5

Rebar Diagram. Displays calculated rebar for the selected load combination or envelope. The graph shows required rebar calculated at 1/20th points and provided rebar (**Fig. 7.2-6**).

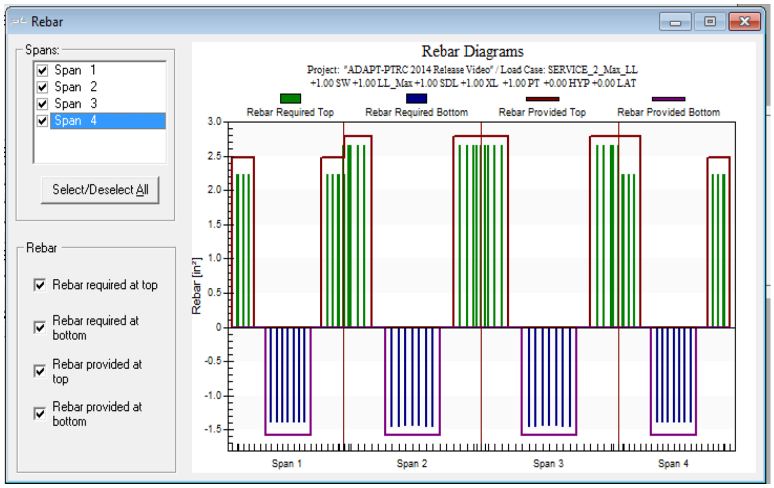


Figure 7.2-6

Crack Width. Displays allowable and actual crack width diagrams for Unbonded (Quasi) and/or Bonded (Frequent) load combinations when user defined for prestressed designs performed with the European code – EC2. For RC designs using the same code, the allowable and actual crack width diagrams are generated for the Quasi-permanent load combination. (**Fig. 7.2-7**).

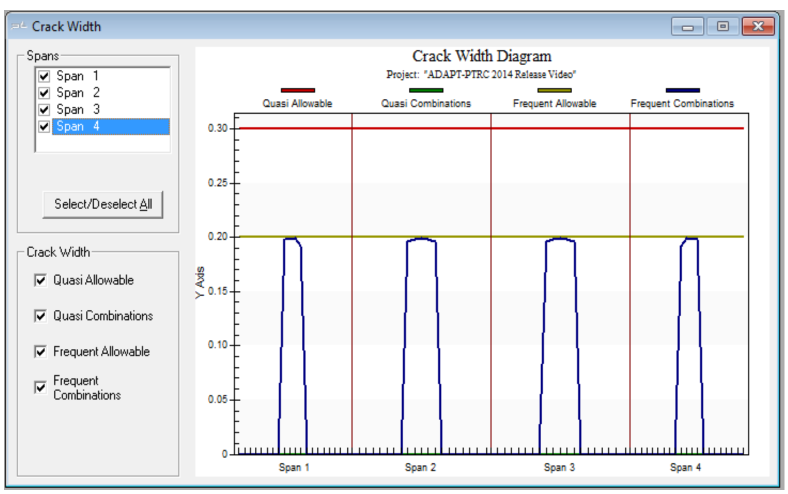


Figure 7.2-7

7.2.3 Options Menu

Summary options. Displays summary report setup window (**Fig. 7.3-2** (a)-(c)). It has the same function as Report Setup button, , on the Span Selection toolbar.

Graph properties. Configures the graphs generated by the program. Options include whether to include X and Y gridlines, min/max data points and a legend.

7.2.4 Window Menu

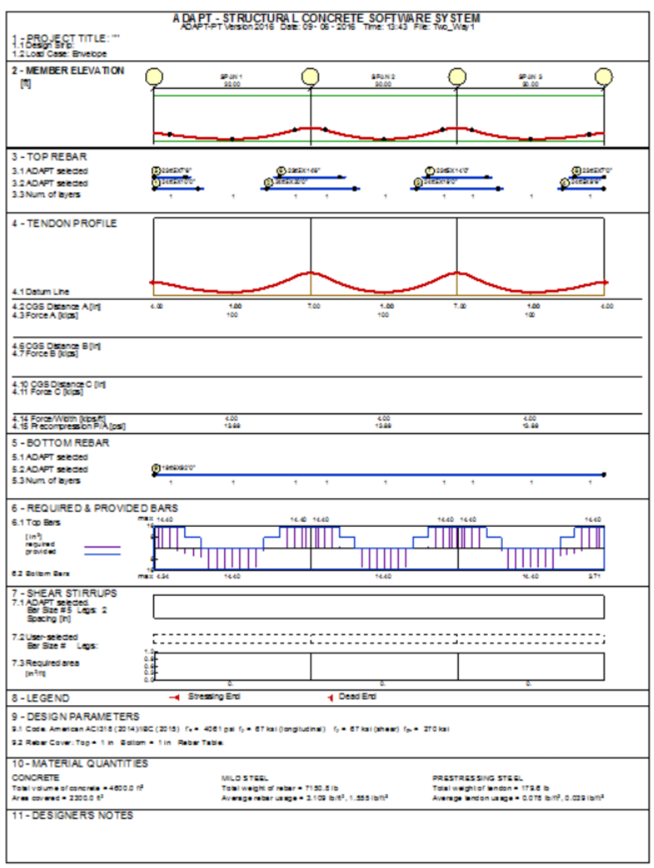
This menu lists which of the graph windows are open. The graphs may be stacked vertically for scrolling, or the windows may be cascaded.

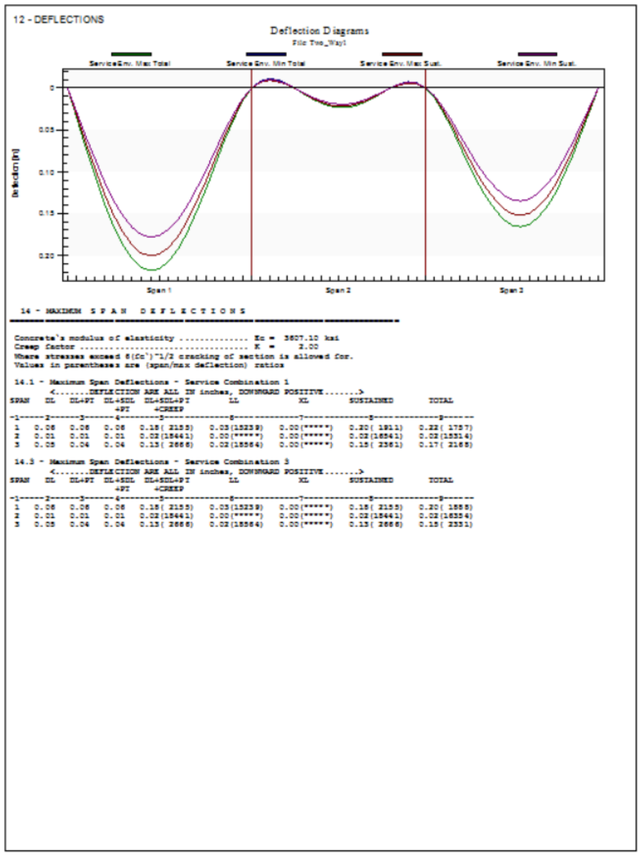
7.2.5 Help Menu

Documentation. Opens the folder where documentation is stored on the local machine.

7.3 PT Summary Report

The **Figure 7.3-1** shows Summary report as it opens once you select Summary from Graphs menu, or click on Summary report button, , in the main toolbar. Note the second page of the Summary reports graphical and tabular enveloped deflection results.





14 - MAXIMUM SPAN DEFLECTIONS

Concrete's modulus of elasticity .... Ec = 3607.10 ksi
Creep factor .... K = 2.00
Where stresses exceed 6(fc')^1/2 cracking of section is allowed for.
Values in parentheses are (span/max deflection) ratios

14.1 - Maximum Span Deflections - Service Combination 1

<.......DEFLECTION ARE ALL IN inches, DOWNWARD POSITIVE.......>

SPAN	DL	DL+PT	DL+SDL +PT	DL+SDL+PT +CREEP	LL	XL	SUSTAINED	TOTAL
1	0.06	0.06	0.06	0.18(2155)	0.03(15239)	0.00(*****)	0.20(1911)	0.22(1757)
2	0.01	0.01	0.01	0.02(18461)	0.00(*****)	0.00(*****)	0.02(16561)	0.02(15314)
3	0.05	0.04	0.04	0.13(2666)	0.02(18564)	0.00(*****)	0.15(2361)	0.17(2168)

14.3 - Maximum Span Deflections - Service Combination 3

<.......DEFLECTION ARE ALL IN inches, DOWNWARD POSITIVE.......>

SPAN	DL	DL+PT	DL+SDL +PT	DL+SDL+PT +CREEP	LL	XL	SUSTAINED	TOTAL
1	0.06	0.06	0.06	0.18(2155)	0.03(15239)	0.00(*****)	0.18(2155)	0.20(1888)
2	0.01	0.01	0.01	0.02(18461)	0.00(*****)	0.00(*****)	0.02(18461)	0.02(16354)
3	0.05	0.04	0.04	0.13(2666)	0.02(18564)	0.00(*****)	0.13(2666)	0.15(2331)

Figure 7.3-1

The format of Summary report can be modified using the Span Selection Toolbar. With this toolbar you can select which of the data blocks to print, or you can recalculate the mild steel requirements using a bar size which is different from what was initially specified in the ADAPT run. In addition, you can select to print the report in color or black-and-white, portrait or landscape, and on a variety of paper sizes. After the data blocks are selected, the report is automatically rescaled to fit the specified paper size. The following is the description of the Span Selection Toolbar.

7.3.1 Span Selection Toolbar

 **Report Setup**. To specify what information to print, select the Report Setup. A window with three tabs will appear. **Figure 7.3-2 (A-C)** shows these three tabs:

- Use the check boxes on the 'Sections to be printed' tab to select which data blocks to print.

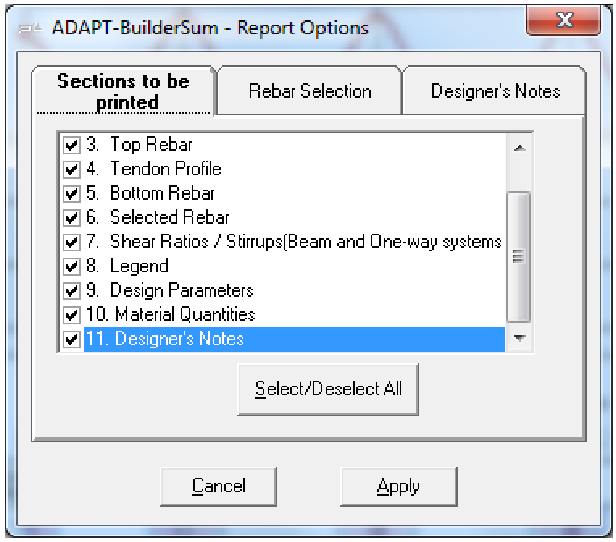


Figure 7.3-2a

- Use the 'Rebar Selection' tab to change the bar sizes or bar system used for top and bottom reinforcing steel.

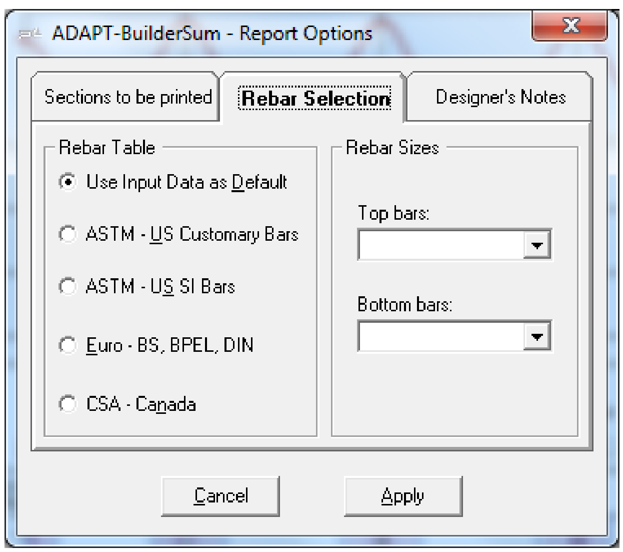


Figure 7.3-2b

The bar system used for the ADAPT-PT/RC analysis is determined according to the design code selected during data input. The preferred bar size is also specified during data input. Although these will be used as defaults for the Summary Report, both the bar system and bar size can be changed.

All the bar systems shown on the Rebar Selection tab (ASTM - US Customary, ASTM - US SI, Euro or CSA) are available, no matter what design code was used for the ADAPT-PT/RC run. First, select the desired bar system. Then, specify the top and bottom bar size from the pull-down list of bar sizes available for that bar system. Click on Apply to recalculate the mild steel reinforcing requirements with the new bar sizes.

To go back to the bar system and sizes in the original ADAPT-PT/RC run, select the 'Use Input Data as Default' option. Click on Apply to recalculate the number of bars required.

- Use the 'Designer's Notes' tab to input notes that will be printed at the bottom of the report.

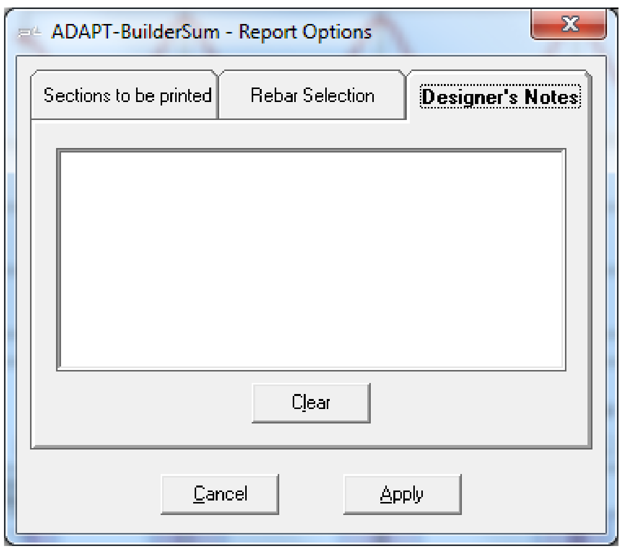
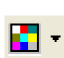


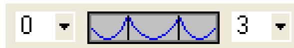
Figure 7.3-2c

Click on the Apply button to apply the selected options to the report.

 **Page Setup**. This tool gives you an option to print your report in portrait or landscape.

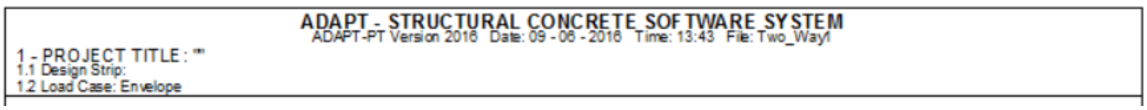
 **Color Settings**. The tool gives you an option to print your report in color or black-and-white. his will change the color setup on both the screen and the printout.

Zoom options. The Zoom buttons can be used to adjust the size of the report on the screen.



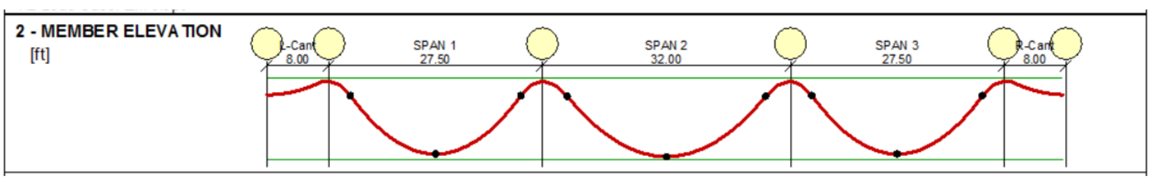
Spans selection. Allows you to format the report to include certain spans. From the left drop down list select first span and from the right drop down list specify last span that you want to include in the Summary Report.

The following is a description of the data blocks that can be included in the Summary Report:



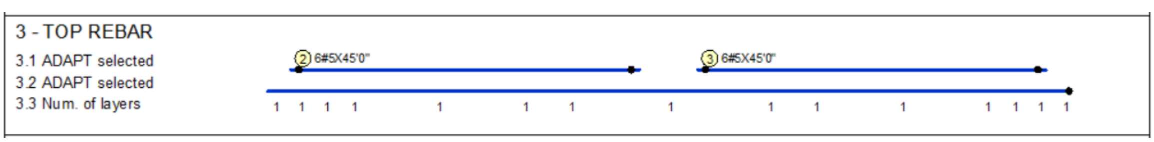
Data Block 1 - General Project Information

Data block 1 contains the General and Specific titles entered during data input.



Data Block 2 – Member Elevation

Data block 2 contains an elevation view of the member with span dimensions and labels. It also includes a graphical representation of the tendon profile that shows inflection points and low points. An elevation view of the member, including all drops and steps, with span lengths and the post-tensioning tendon profile, including inflection points and low points.



Data Block 3 – Top Rebar

Data block 3 reports the amount and length of rebar required at the top of the member. The rebar shown is the larger of the steel required to withstand the negative moment demand and code-specified minima. Where rebar curtailment rules are used in the input, the program arranges longitudinal reinforcement relative to those rules set by the user. See **Section 5** for additional information. Section 3.3 of this block reports the number of layers required to meet minimum reinforcement spacing requirements based on the design code selected. This is a longitudinal reinforcement spacing check for the provided reinforcement in the section. For example, when the ACI318 code is selected, the program calculates the spacing between bars as:

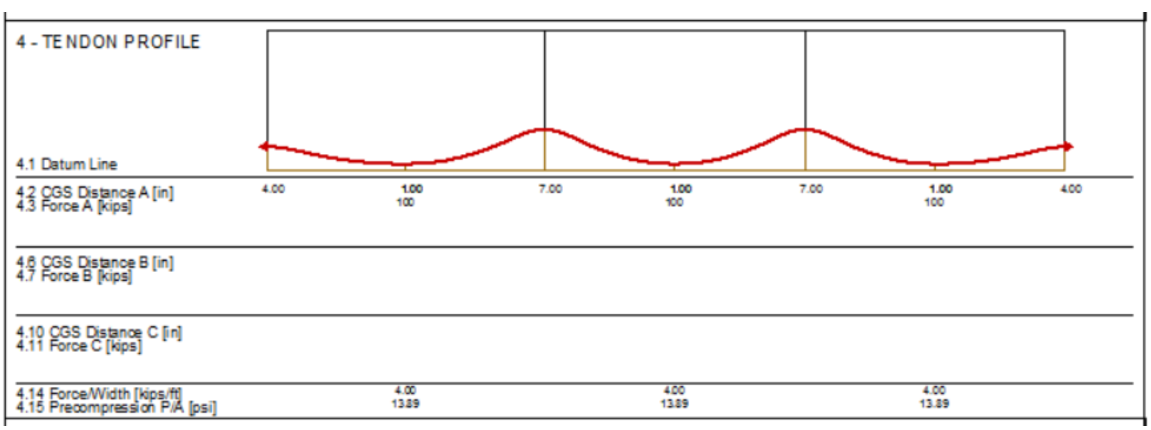
N-1 (where 'N' equals the number of bars provided) / Total section width (B) – 2*cover input (to longitudinal bar)

If the required number of layers is reported greater than 1, the user should modify the input for cover requirements to maintain the proper effective depth in design of the section.

If the steel required is controlled by the negative-moment demand, the bar lengths are based on the required rebar quantities at 1/20th points. By default, the selected rebar is calculated as two lengths to minimize material requirements, unless reinforcement curtailment rules are applied in which case these rules supersede the program defaulted calculated lengths.

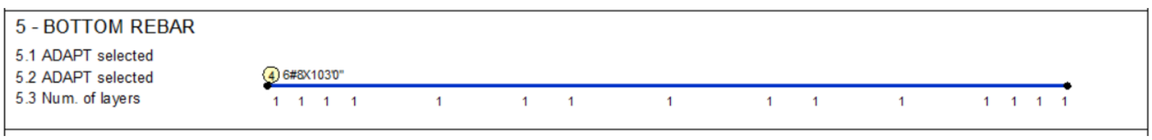
Note that the steel selected by the program is only one of several acceptable design solutions. Space has been provided in this data block for the designer to provide alternate information on rebar quantity, size, and length. The designer may also use this space to write in any additional notes or remarks pertaining to the rebar.

For two-way slabs designed to the ACI318 design code, the program adjusts the required reinforcement when the minimum flexural requirement exceeds the requirement for Strength by one-third. The adjustment is made to the graphical reinforcement layout and Block 6 for individual Service combinations when they are selected in the combination pull-down menu. The modification is also reflective in the Envelope selection.



Data Block 4 – Tendon Profile

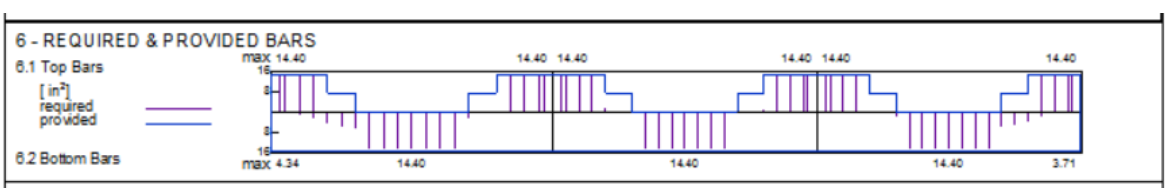
Data block 4 shows an elevation view of the tendon profile. Tendon control points are marked and their heights with respect to the reference line are given. If the computer run was done in the Force Selection mode, the program shows the total post-tensioning force in each span. If the Tendon Selection option was used, the elevation view also includes the total number of tendons, the location of all dead and live stressing ends, and any added tendons. Heights of tendon control points with respect to the reference line and the total post-tensioning force or total number of strands specified for each span. If Tendon Selection is used, this data block will report elongations at each end of Tendons A, B and C. This block also reports the total force/width for each span and precompression at the mid-span location of each span. If a span is segmented, the program uses the mid-span width for the calculation of data for items 4.14 and 4.15.



Data Block 5 – Bottom Rebar

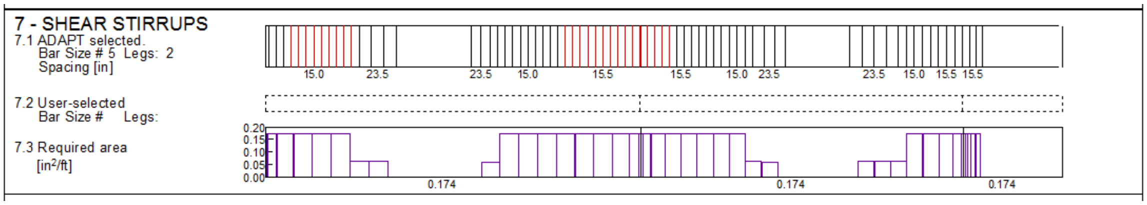
Data block 5 reports the amount and length of rebar required at the bottom of the member. The format is the same as data block 3 – Top Rebar.

For two-way slabs designed to the ACI318 design code, the program adjusts the required reinforcement when the minimum flexural requirement exceeds the requirement for Strength by one-third. The adjustment is made to the graphical reinforcement layout and Block 6 for individual Service combinations when they are selected in the combination pull-down menu. The modification is also reflective in the Envelope selection.



Data Block 6 – Required & Provided Bars

Data block 6 plots the rebar required and provided for the top and bottom of the section at each 1/20th point. The maximum required areas of steel required at each joint for the top and bottom of each span are also shown.

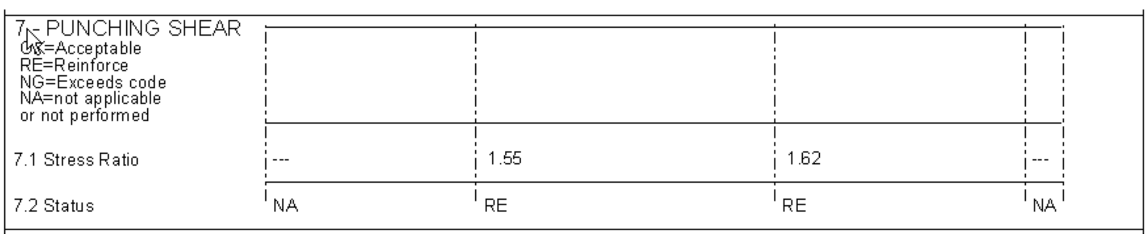


Data Block 7 – Shear Stirrups / Punching Shear (One-Way Shear)

For beams, data block 7 reports the stirrup size and spacing based on user input during data entry. The spacing shown is the maximum spacing along the different segments of the span. The data block also includes a bar graph of the area of shear reinforcement required along each span. The graphical diagram displays design sections as red where the concrete shear capacity is exceeded by shear demand (Vu/Phi,Vc > 1.0).

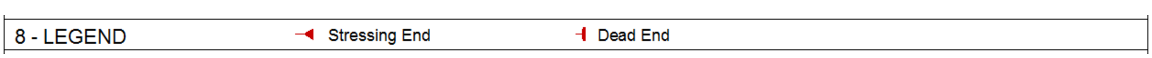
This block is typically not included on reports for one-way slabs since shear reinforcement is seldom required. Although this block may indicate that shear reinforcement is required at the supports for a one-way slab, a review of the Results Report will show that this is for beams only.

Note: the shear diagram is only available for strength and envelope load combinations.



Data Block 7 – Shear Stirrups / Punching Shear(Two-way Shear)

For two-way slabs, data block 7 plots an elevation view of the model, which indicates the punching shear stress ratio at each support and states whether the stress ratio is acceptable per the specified code. Note: This block is available only if you select Envelope from the drop-down list of load combinations on the Main toolbar.



Data Block 8 – Legend

Data block 8 identifies the symbols used to indicate stressing and dead ends. Note, however, that the stressing and dead ends are only shown when the Tendon Selection option has been used for the analysis. The legend is not applicable if Force Selection was used.

9 - DESIGN PARAMETERS
9.1 Code: American ACI318 (2011)/IBC (2012) f'_c = 4061 psi f_y = 67 ksi (longitudinal) f_y = 67 ksi (shear) f_{pu} = 269.77 ksi
9.2 Rebar Cover: Top = 1.5 in Bottom = 1.5 in Rebar Table:

Data Block 9 – Design Parameters

Data block 9 reports the following design parameters used in the ADAPT-PT/RC run:

- Design Code
- Concrete strength, f'_c
- Mild steel yield strength, f_y for longitudinal and shear reinforcement
- Ultimate tendon strength, f_{pu} (PT mode only)
- Minimum Top and Bottom rebar cover
- Rebar Table

10 - MATERIAL QUANTITIES		
CONCRETE	MILD STEEL	PRESTRESSING STEEL
Total volume of concrete = 0.0 ft^3	Total weight of rebar = 961.9 lb	Total weight of tendon = 680.9 lb
Area covered = 1723.3 ft^2		

Data Block 10 –Material Quantities

Data block 10 reports the following design parameters used in the ADAPT-PT/RC run:

- Concrete: Total volume and area covered
- Mild Steel: Total weight of rebar
- Prestressing Steel: Total weight of tendons (PT mode only)

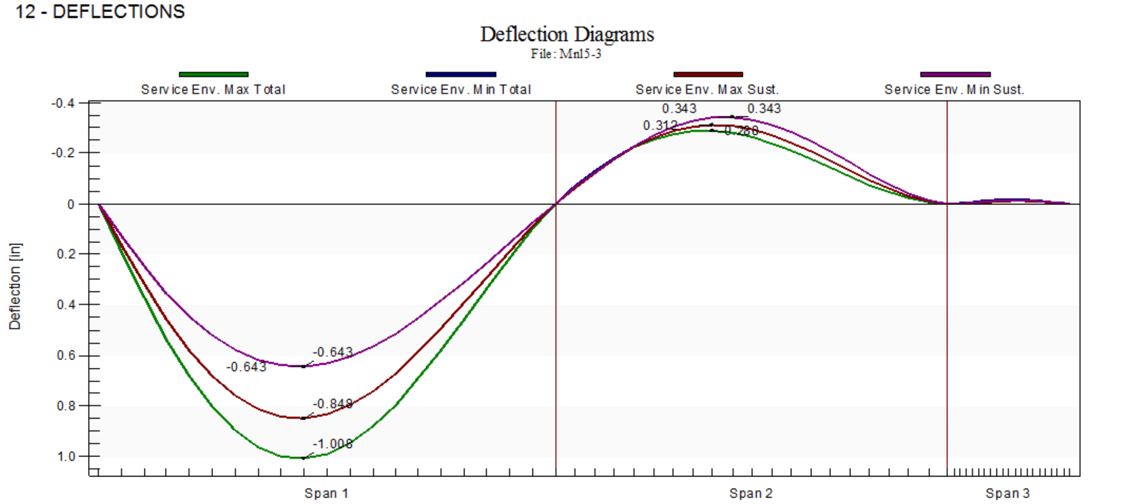
11 - DESIGNER'S NOTES

Data Block 11 – Designer's Notes

Data block 11 contains notes added by the designer. The entry in this box will be used on future runs and future projects until it is cleared. To clear the notes, select Clear and then click on Apply in the 'Designer's Notes' tab of Report Setup.

The Summary Report can be viewed in final form on the screen. It can then be printed or saved as either a .WMF or .BMP or copied and pasted to a word processor. If it is saved as a file, it can be inserted into contract documents, calculation packages or structural drawings.

12 - DEFLECTIONS



14 - MAXIMUM S P A N D E F L E C T I O N S

Concrete`s modulus of elasticity .... Ec = 3605.00 ksi
Creep factor .... K = 2.00
Where stresses exceed 6(fc`)^1/2 cracking of section is allowed for.
Values in parentheses are (span/max deflection) ratios

14.1 - Maximum Span Deflections - Service Combination 1

<.......DEFLECTION ARE ALL IN inches, DOWNWARD POSITIVE.......>

SPAN	DL	DL+PT	DL+SDL +PT	DL+SDL+PT +CREEP	LL	XL	SUSTAINED	TOTAL
-1-	-2-	-3-	-4-	-5-	-6-	-7-	-8-	-9-
1	0.00	-0.56	0.21	0.64(1194)	0.23(3367)	0.00(*****)	0.85(905)	1.01(761)
2	0.00	-0.25	-0.11	-0.34(1924)	0.05(14377)	0.00(*****)	-0.31(2117)	-0.29(2289)
3	0.00	0.01	0.00	-0.01(17695)	0.00(52736)	0.00(*****)	-0.01(14082)	-0.02(12068)

14.3 - Maximum Span Deflections - Service Combination 3

<.......DEFLECTION ARE ALL IN inches, DOWNWARD POSITIVE.......>

SPAN	DL	DL+PT	DL+SDL +PT	DL+SDL+PT +CREEP	LL	XL	SUSTAINED	TOTAL
-1-	-2-	-3-	-4-	-5-	-6-	-7-	-8-	-9-
1	0.00	-0.56	0.21	0.64(1194)	0.23(3367)	0.00(*****)	0.64(1194)	0.87(881)
2	0.00	-0.25	-0.11	-0.34(1924)	0.05(14377)	0.00(*****)	-0.34(1924)	-0.31(2140)
3	0.00	0.01	0.00	-0.01(17695)	0.00(52736)	0.00(*****)	-0.01(17695)	-0.01(13769)

Data Block 12 – Deflections

Data block 12 contains graphical and tabular deflections for Sustained and Total conditions. The graphical output displays maximum and minimum deflections for the Envelope of Service Sustained and Service Total conditions. The tabular display produces a summary of deflections for each span and service load condition and is like Block 14 of the tabular report produced by the Report Generator.

7.4 Export to DXF File

The DXF export feature will allow users to graphically extract relevant tendon (PT mode only), longitudinal reinforcement and shear reinforcement information from their ADAPT-PT/RC models.

The DXF feature includes options to customize drawing parameters, tendon properties and attributes, tendon placement parameters, reinforcement type selection, reinforcement drawing arrangement and reinforcement/tendon visualization parameters. The drawings for tendons, longitudinal reinforcement and shear reinforcement can be arranged as an overlaid or stacked profile.

The **Figure 7.4-1** shows the export to DXF input dialogue. This window will open once you select Export to DXF file from the File menu, or click on Export to DXF file button, 

, in the main toolbar.

The Drawing Title will be displayed on the top of the drawing file. The default name of the file will be the name of the ADAPT-PT/RC model.dxf. To change the name of the file, click the Change File button to define the new name of the drawing you are creating, then click Create DXF button to create the drawing file and launch AutoCAD.

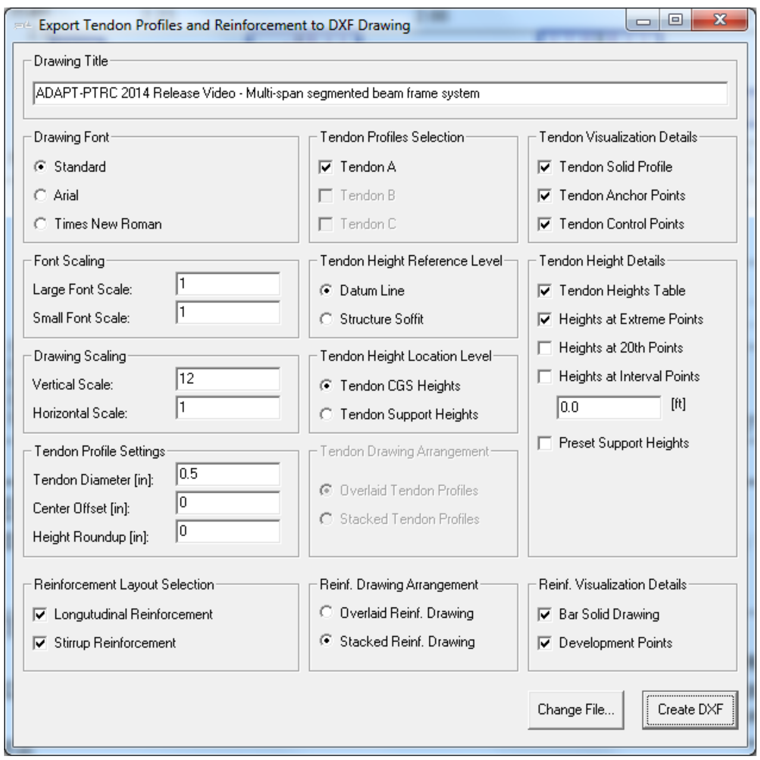


Figure 7.4-1

8 Reports Overview

This section describes in detail the reports generated by the ADAPT-PT/RC Report Generator for one-way slabs, two-way slabs, and beams. It is a useful tool in compiling customized, standard reports for your designs or those who review structural designs performed using ADAPT-PT/RC. The report can be produced as a standalone Microsoft Word® .RTF file or produced as the .RTF report and a Microsoft Excel® .XLS file.

The material presented identifies the program input parameters by the user, the parameters which the user may edit during the execution of the program, and results computed by ADAPT-PT/RC. The reports differentiate between those values input by the user from those calculated by the program.

Each report option is subdivided into sections. Each section is given a unique identification number. The report consists of those sections that are selected by the user. The user has control over which report sections can be included in the production of a single report. Customized reports can be saved as templates and recalled for future use each time the Report Generator is opened.

8.1 Report Generator Screen

To create a report, click on the Report Setup button, on the Main toolbar. The Report Generator window opens (**Fig. 8.1-1**).

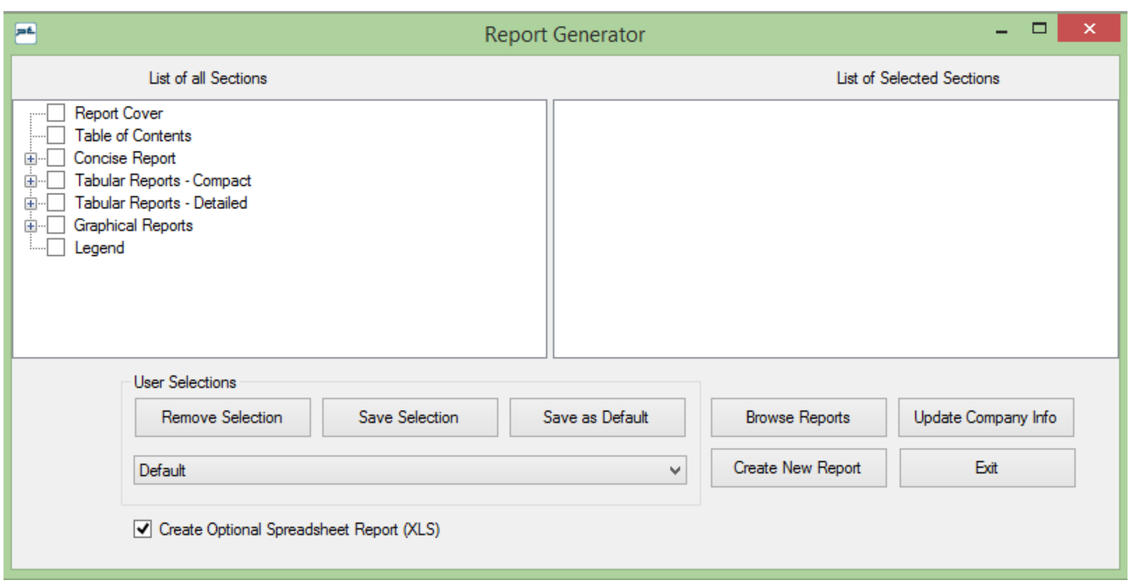
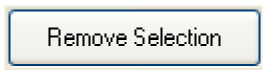


Figure 8.1-1 Report Generator Screen

The following is the description of items included in the Report Generator:

- **List of All Sections**. Includes a collapsible tree that lists the sections available to be included in a report. To expand the report section, click on the + tab. To select a section, check the box in front the section. Note that if a main report section is checked, all sub-sections will be included in the report. To exclude the section, deselect the box. If a report section does not apply to the results for the specific model type, the section will not be available in the Report Generator. For example, if your structural system is a beam, the punching shear report (Section 13) will not be available.

- **List of Selected Sections**. Lists all sections that were selected to be included in a report.



Removes the highlighted selection from the drop-down list.

Saves the current set of sections to a custom report in the drop-down list.

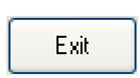
Saves the current set of sections as the default report.

Generates a report and displays them in rich text format (.RTF) when completed.

Opens a list of reports that have previously been saved to the last file location.

Allows you to customize report footer and cover page with specific company information.


Create Optional Spreadsheet Report (.XLS) – Allows the user to replicate the report information in a Microsoft Excel® spreadsheet file.



Exits Report Generator and goes back to the ADAPT Main program window.

8.2 How to Create A Report

To create reports, go through the following steps:

1. First set the report to show your company information when you create the report documents. To do so click on 

button. The **Company Information** dialog box opens.

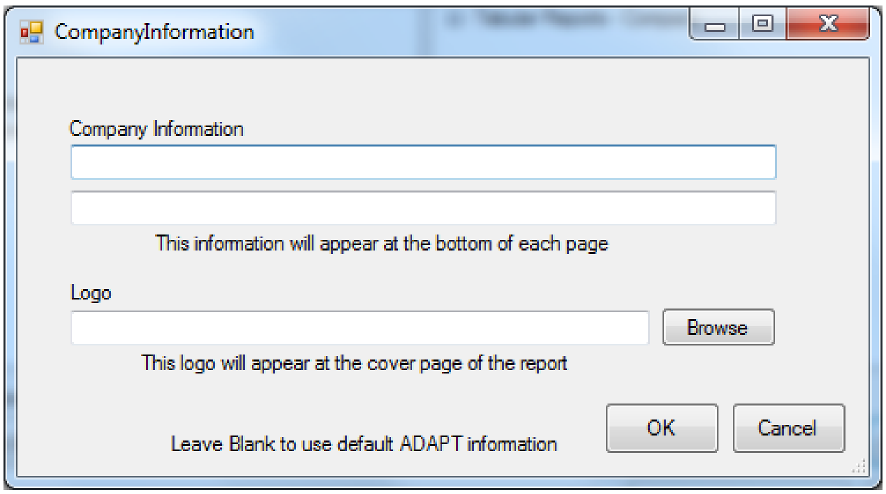


Figure 8.2-1

2. In the **Company Information** edit box, specify the text that you would like to include in the footer of each page of your report. To upload the company's logo, click on the **Browse** button and upload Bitmap or JPEG file of your logo. The logo will show on the report cover page. Once you set up your company information the program will use it whenever you create new reports. If left blank, the program will use default ADAPT information.

3. Click **OK** to close **Company Information** edit box.

4. In the **Report Generator** tree, select the sections that you would like to include in your report. The selection will appear at the right side of the window in the List of Selected Sections.

5. Click on 

button. The program will ask you to specify name and location where you would like to save your report. The default location is the .ADB file folder where your project is saved.

6. Click **OK**. The program will start generating the report. Once completed, the program will open the report in rich text format. The report content will

include sections you selected, and they will be shown in default program settings. You will be able to modify it as you wish.

8.3 Description of Report Sections

The main report sections available are:

- Report cover page.
- Table of contents
- Concise report
- Tabular report-compact
- Tabular report-detailed
- Graphical reports
- Legend

The following explains each of these sections:

8.3.1 Report Cover Page

The program-generated cover page will contain a company logo, title, bitmap of a 3D structure view, and date (**Fig. 8.3-1**). The cover page will be created only if you select the Report Cover option from the list of sections.

Your company logo will appear at the top of the report cover page and will show the bitmap or JPEG file that you uploaded (**Fig 8.2-1**). The default cover page title will be the Generic title and Specific title of your project that you specified in the General Settings window of PT or RC Input. While in PT or RC Input you can set the structure view as you want it to appear on the cover page of your report. When you exit PT or RC Input (click on Close button, or click Execute), the program will take a screen shot of the 3D structure and show it on the cover page. At the bottom of the cover page the program shows the date when you created report.

ADAPT

Your Partner in Concrete Design

Beam Design - 3 spans
ADAPT-PTRC 2018

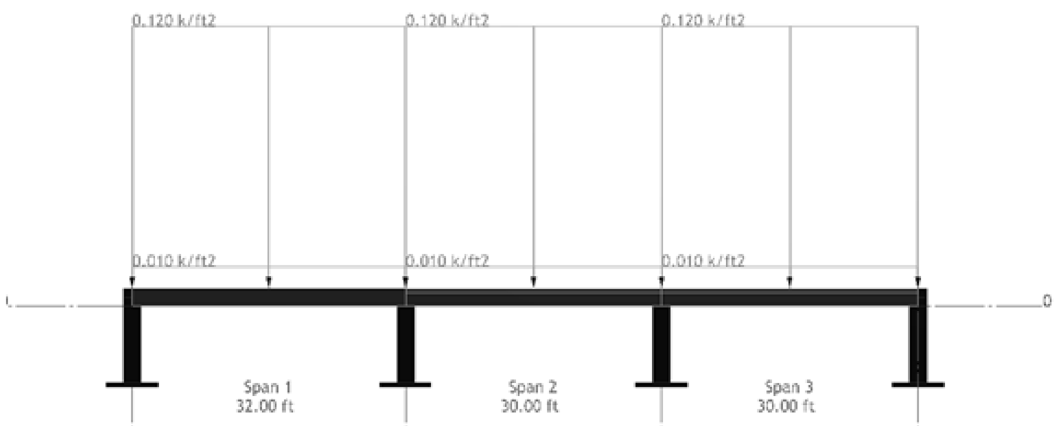


Figure 8.3-1 Report Cover Page

8.3.2 Table of Contents

To include a table of contents with your report, select the Table of Contents section in the Report Generator tree. The table will contain only sections that were selected by user and listed in the Selected Sections of the Report Generator window. The following table includes all sections available in ADAPT-PT/RC report. Note that some sections may not apply to RC designs using RC mode.

TABLE OF CONTENT:

36 - Detailed Deflections Live Load
36.1 - Detailed Deflections Live Load
36.3 - Detailed Deflections Live Load

Graphical Reports

PT-Force

PT-Profile

Deflection

Load Cases
Self-weight
Super Imposed Dead
Live
Other
Prestressing
Hyper-Static

Load Combinations
SERVICE_1_Min_LL
SERVICE_1_Max_LL
SERVICE_2_Min_LL
SERVICE_2_Max_LL
STRENGTH_1_Min_LL
STRENGTH_1_Max_LL
INITIAL_MIN_LL
INITIAL_MAX_LL
Cracking_Moment
Envelope

Legend

8.3.3 Concise Report

The concise report is the short version of the report that includes all information necessary to describe project input and results.

Sections of concise report are:

- Project Design Parameters and Load Combinations
- Design Strip Report – System type

Project Design Parameters and Load Combinations includes information common to the entire project and can be printed as verification that data in design criteria was properly entered into the program. This option includes:

- Material properties

- Covers
- Design Code
- Design Settings
- Allowable stresses (PT mode only)
- Post-tensioning parameters (PT mode only)
- Load combinations
- Other common entries for all support lines.

Design Strip Report includes:

- Definition of geometry (annotated graphics)
- Applied loads (annotated graphics)
- Tendon layout and values (graphics) with information on force, tendon height (PT mode only)
- Stress check results (PT mode only)
- Rebar report
- Punching shear
- Deflection graph
- Quantities

The following is the example of concise report:

A. Design Parameters and Load Combinations

A.1 Project Design Parameters

Parameter	Value	Parameter	Value
Concrete		Fy (Shear reinforcement)	66.72 ksi
F'c for BEAMS/SLABS	4061.00 psi	Minimum Cover at TOP	1.50 in
F'ci for BEAMS/SLABS	3045.80 psi	Minimum Cover at BOTTOM	1.50 in
For COLUMNS/WALLS	4061.00 psi	Post-tensioning	
Ec for BEAMS/SLABS	3607.00 ksi	SYSTEM	UNBONDED
For COLUMNS/WALLS	3607.00 ksi	Fpu	269.77 ksi
CREEP factor	2.00	Fse	174.04 ksi
CONCRETE WEIGHT	NORMAL	Strand area	0.153 in 2
UNIT WEIGHT	150.00 pcf	Min CGS from TOP	2.00 in
Tension stress limits / (f'c)1/2		Min CGS from BOT for interior spans	2.00 in
At Top	6.000	Min CGS from BOT for exterior spans	2.00 in
At Bottom	6.000	Min average precompression	149.39 psi
Compression stress limits / f'c		Max spacing / slab depth	8.00
At all locations	0.450	Analysis and design options	
Tension stress limits (initial) / (f'c)1/2		Structural system	BEAM
At Top	3.000	Moment of Inertia over support is	NOT INCREASED
At Bottom	3.000	Moments reduced to face of support	YES
Compression stress limits (initial) / f'c		Moment Redistribution	NO
At all locations	0.600	Effective flange width consideration	YES
Reinforcement		Effective flange width implementation method	ACI-318
Fy (Main bars)	66.72 ksi	DESIGN CODE SELECTED	American-ACI318 (2014)/IBC 2015

A.2 Load Combinations

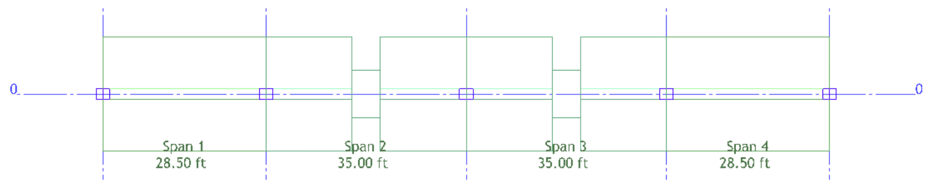
Strength load combinations
1. 1.2 SW + 1.6 LL + 1.2 SDL + 1.6 X + 1 HYP

Service load combinations
Sustained Load
1. 1 SW + 0.3 LL + 1 SDL + 0.3 X + 1 PT
Total Load
2. 1 SW + 1 LL + 1 SDL + 1 X + 1 PT

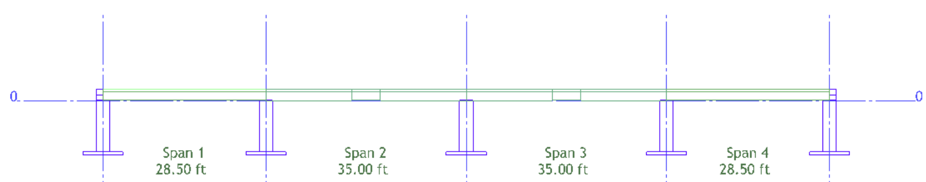
Initial load combinations
1 SW + 1.15 PT

B. Design Strip Report: Multi-span segmented beam frame system
B.1 Geometry

- Plan

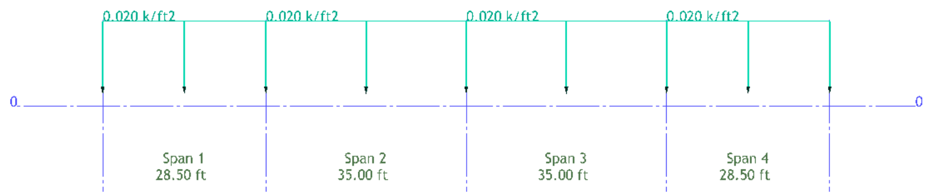


- Elevation

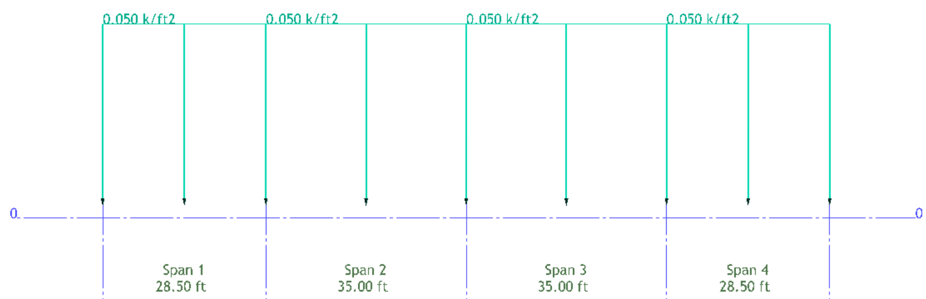


B.2 Applied loads

- Superimposed Dead Load

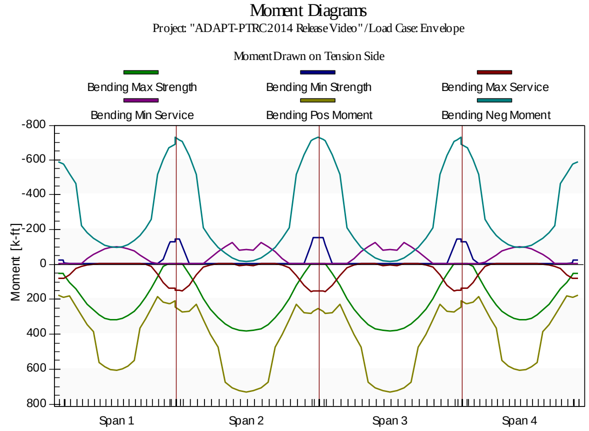


- Live Load



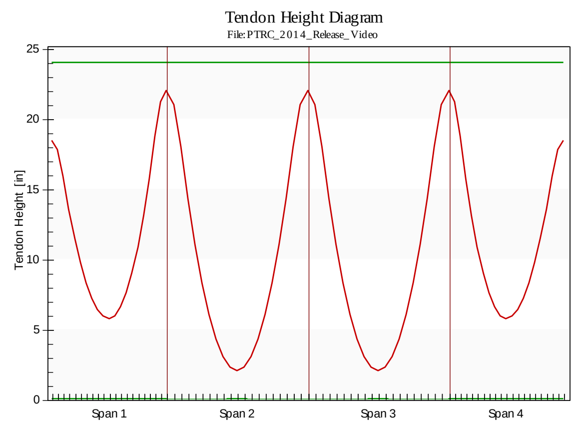
B.3 Design Moment

LOAD COMBINATION: Envelope



DESIGN MOMENT
(Moment is drawn on tension side)

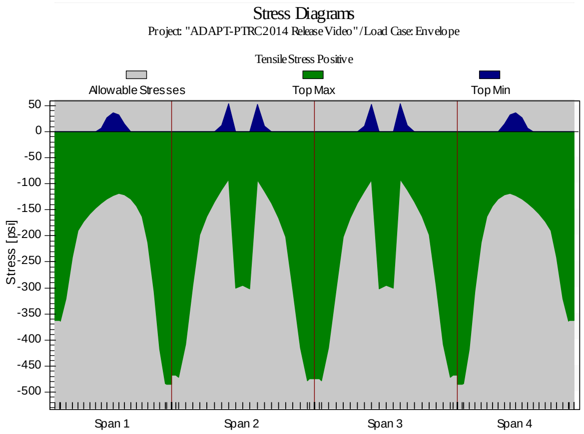
B.4 Tendon Profile

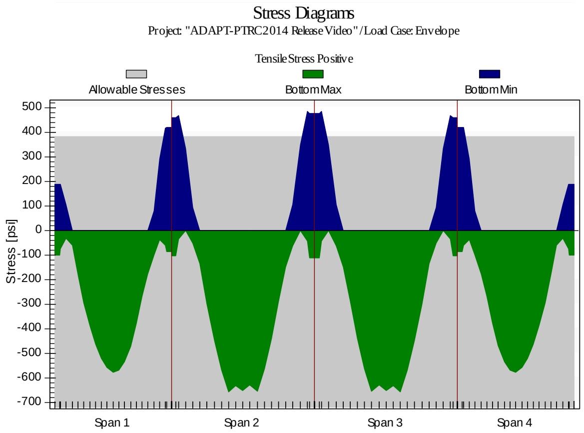


POST-TENSIONING PROFILE

B.5 Stress check results / Code check

LOAD COMBINATION: Envelope





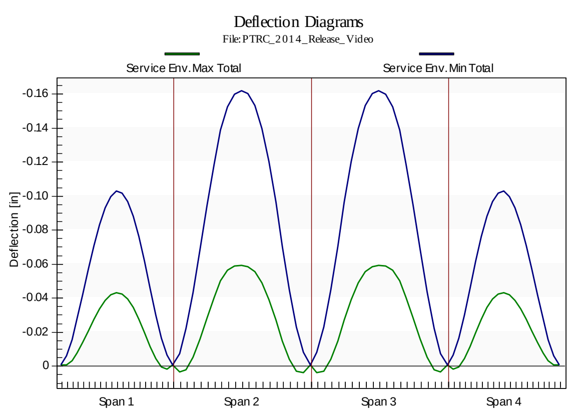
SERVICE COMBINATION STRESSES
(Tension stress positive)

B.6 Rebar Report

Total Strip Provided Rebar

Span	ID	Location	From	Quantity	Size	Length	Area
			ft			ft	in2
1	1	TOP	0.00	8	5	6.00	2.48
1	2	TOP	22.80	9	5	13.00	2.79
2	3	TOP	28.00	9	5	14.00	2.79
3	4	TOP	28.00	9	5	13.00	2.79
4	5	TOP	22.80	8	5	6.00	2.48
1	6	BOT	0.00	1	8	3.00	0.79
1	7	BOT	8.55	2	8	11.50	1.58
1	8	BOT	24.23	2	8	9.50	1.58
2	9	BOT	10.50	2	8	14.00	1.58
2	10	BOT	29.75	2	8	10.50	1.58
3	11	BOT	10.50	2	8	14.00	1.58
3	12	BOT	29.75	2	8	9.50	1.58
4	13	BOT	8.55	2	8	11.50	1.58
4	14	BOT	25.65	1	8	3.00	0.79
1	16	BOT	25.65	1	8	6.50	0.79
2	17	BOT	31.50	1	8	7.00	0.79
3	18	BOT	31.50	1	8	6.50	0.79

B.8 Deflection



DEFLECTION

B.9 Quantities

CONCRETE

Total volume of concrete = 1552.21ft3 (57.49 yd3)
Area covered = 2423.33 ft2

MILD STEEL

Total weight of rebar = 985.05 lbs
Average rebar usage = 0.41 psf, 0.63 pcf

PRESTRESSING MATERIAL
Total weight of tendon = 680.9 lb
Average tendon usage = 0.28 psf, 0.44 pcf

8.3.4 Compact Report

The compact report consists of the mirror image of user input, plus a tabular listing of critical information, such as post-tensioning and reinforcement, necessary for preparation of structural drawings. Also, it includes values of actions, such as moments, shears and stresses at left, center, and right of each span.

The following is the description of the available report sections:

8.3.4.1 Section 1- User Specified General Analysis and Design Parameters

This data block reflects the user's input in the selection of design parameters and design options. Some options related to post-tensioning may not be shown if the design is made in RC mode.

Parameter	Value	Parameter	Value
Concrete		Fy (Shear reinforcement)	66.72 ksi
F'c for BEAMS/SLABS	4061.00 psi	Minimum Cover at TOP	1.50 in
F'ci for BEAMS/SLABS	3045.80 psi	Minimum Cover at BOTTOM	1.50 in
For COLUMNS/WALLS	4061.00 psi	Post-tensioning	
Ec for BEAMS/SLABS	3607.00 ksi	SYSTEM	UNBONDED
For COLUMNS/WALLS	3607.00 ksi	Fpu	269.77 ksi
CREEP factor	2.00	Fse	174.04 ksi
CONCRETE WEIGHT	NORMAL	Strand area	0.153 in 2
UNIT WEIGHT	150.00 pcf	Min CGS from TOP	2.00 in
Tension stress limits / (f'c)1/2		Min CGS from BOT for interior spans	2.00 in
At Top	6.000	Min CGS from BOT for exterior spans	2.00 in
At Bottom	6.000	Min average precompression	149.39 psi
Compression stress limits / f'c		Max spacing / slab depth	8.00
At all locations	0.450	Analysis and design options	
Tension stress limits (initial) / (f'c)1/2		Structural system	BEAM
At Top	3.000	Moment of Inertia over support is	NOT INCREASED
At Bottom	3.000	Moments reduced to face of support	YES
Compression stress limits (initial) / f'c		Moment Redistribution	NO
At all locations	0.600	Effective flange width consideration	YES
Reinforcement		Effective flange width implementation method	ACI-318
Fy (Main bars)	66.72 ksi	DESIGN CODE SELECTED	American-ACI318 (2011)/IBC 2012

The following are parameters which enter the computations as recommended initial values but can be edited during the execution of the program. The final values are listed in the output section of the report. These parameters apply only to PT mode.

- Tendon CGS (Center of Gravity of Strand) at top of support and mid-spans: these are the user-suggested values. The actual cover used in the calculations, as modified by the user in the Recycle Window, are listed in Section 7 of the report.
- Minimum average precompression shows the value set by the user. The actual average post-tensioning is listed in Section 7 of the report.
- Max spacing between strands is also entered by the user. Refer to Section 7 where the force provided by each tendon reveals whether this postulation is adhered to.
- Tension stress limits are defined as multiples of the $(f'_c)^{1/2}$. Based on these values, the required post-tensioning along the member is determined. **Specifying a set of permissible values in this data block is no guarantee that the final stresses are, in fact, equal or less than the limits stated.** During execution in the Recycle Window the user may overwrite the previously set stress limits with the selected post-tensioning. The actual stresses are reported in data sections Section 7 of the report.

The following input data cannot be altered during the execution of the program:

- **Reinforcement:** This data block refers to the non-prestressed reinforcement in the beam/slab. The values for beam stirrups, where applicable, are given in Section 12 of the report.
- **Post-tensioning system:** Indicates the user's selection between grouted (bonded) or unbonded post-tensioning.

  The average effective stress in a strand (f_{se}) is the user's estimate of the stress in a strand after all losses have taken place. This value is used in the determination of the ultimate strength of a section if the "force selection" option of the program is used. It affects the amount of supplemental rebar which may be required to meet the strength stipulations of a section. If the "variable force" option (tendon selection) is used, the program does not use this value. It calculates that actual stress in the stand at each design section.

- Analysis option used: If the answer to moments reduced to face-of-support is YES, it indicates that the calculated centerline moments at each support are adjusted to face-of-support. In addition to the centerline moments, ADAPT prints out the moments reduced to face-of-support. Refer to moment data blocks for the description of printed values.
- Moment of Inertia over support: The beam or slab region over the width of a support (columns or walls) exhibits a greater stiffness than the unsupported regions. ADAPT has an option to allow for this greater stiffness by increasing the moment of inertia of region over the support. The increase is determined by a relationship proposed in ACI-318.

8.3.4.2 Section 2: Input Geometry

This data block reports model geometry as input by the user. It includes basic span geometry, effective width used in calculations, drop can, drop panel, transfer beam dimensions as well as support width and column dimensions. The geometry is described as follows:

- **2.1 Principal Span Data of Uniform Spans.** This section is available only if the user selects conventional geometry input.

Span	Form	Length	Width	Depth	TF Width	TF Thick.	BF/MF Width	BF/MF Thick.	Rh	Right Mult.	Left Mult.
		ft	in	In	in	in	in	in	in		
1	1	19.17	120.00	12.00					10.00	0.50	0.50
2	1	27.17	120.00	12.00					10.00	0.50	0.50
3	1	22.75	120.00	12.00					10.00	0.50	0.50
C	1	3.50	120.00	12.00					10.00	0.50	0.50

- **Detailed Data for Nonuniform Spans**. This section is available only if the user selects segmental geometry input.

Span	Seg.	Form	Left Dist.	Width	Depth	TF Width	TF Thick.	BF/MF Width	BF/MF Thick.	Rh	Right Mult.	Left Mult.
			ft	In	in	in	in	in	in	in		
1	1	2	0.00	24.00	30.00	216.00	8.00			10.00	0.56	0.44
1	2	2	0.33	24.00	30.00	216.00	8.00			10.00	0.56	0.44

1	3	2	5.00	24.00	30.00	216.00	8.00			10.00	0.56	0.44
1	4	1	18.42	24.00	30.00					10.00	0.50	0.50

The following is the description of the data:

- **Span**. This column shows the span number (ID). If the problem has a cantilever at left, its data precedes the first span by a line starting with “C.” Likewise, in the case of a cantilever at right, the last line will start with “C” describing the geometry of the right cantilever.
- **Seg**. This column shows the segment number (ID).
- **Form**. Identifies the cross-sectional geometry of the slab at mid-span. **Figure 8.3-2** illustrates the cross-sectional options. The same figure also gives the definition of parameters Depth, Width, TF Width (top flange width), TF Thick. (Top flange thickness) , BF/MF Width (bottom flange/middle flange width) and BF/MF Thick. (bottom flange/middle flange thickness) .

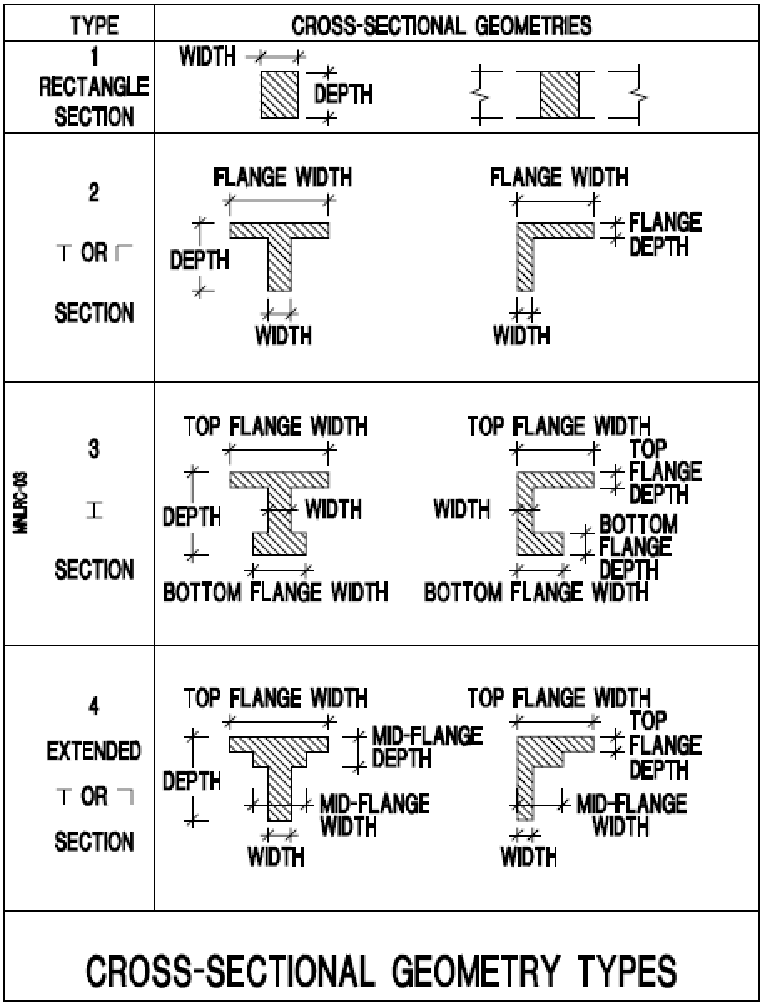


Figure 8.3-2

- **Rh (Reference height)**. The distance from a user-specified reference line to the top of the slab. For example, if a slab is 7” thick and Rh is set to 7”, the datum is located at the slab soffit. This data identifies the location from which tendon height control points are measured and is also used to input steps in a member.
- **Right or Left (Width) Multiplier.** A parameter which describes the factor by which the slab unit strip must be multiplied to cover the total tributary of a given span.

  It should be noted that the results printed in the output, such as the moments and reactions, refer to the total tributary (not effective tributary) unless indicated otherwise.

- **2.3 Effective Width Data of Uniform Spans.** This section applies to Conventional geometry input.

Span	Effective Width
	in
1	57.51
2	81.51
3	68.25

- **Effective Width Data for Non-Uniform Spans.** This section applies to Segmental geometry input.

Span	Seg.	Effective Width
		in
1	1	57.51
1	2	57.51
1	3	57.51

- **Span**. This column shows the span number (ID). If the problem has a cantilever at left, its data precedes the first span by a line starting with "C". Likewise, in the case of a cantilever at right, the last line will start with "C" describing the geometry of the right cantilever.
- **Seg**. This column shows the segment number (ID).
- **Effective width**. Mirrors the data in the Geometry-Effective Flange width input form.
- **2.5 Drop Cap and Drop Panel Data.** This data block gives the dimensions of drop caps/panels for each support if a two-way slab is the system type. **Figure 8.3-3** illustrates the definition of data columns 2 through 10.

Joint	Cap T	Cap B	Cap DL	Cap DR	Drop TL	Drop TR	Drop B	Drop L	Drop R
	in	in	in	in	in	in	in	in	in
1	36.00	48.00	0.00	24.00	0.00	30.00	100.00	0.00	50.00
2	36.00	48.00	24.00	24.00	24.00	30.00	100.00	50.00	50.00
3	36.00	48.00	24.00	24.00	24.00	30.00	100.00	50.00	50.00
4	36.00	48.00	24.00	24.00	24.00	30.00	100.00	50.00	50.00

For example, Cap T, which is the heading of column 2, is shown at the bottom right hand side of **Fig. 8.3-3** to indicate the total depth of "CAP." STEP 1 in the figure indicates the first thickening of the slab past the support, and is referred to as DROP CAP regardless of its size. The second change in thickness is called DROP PANEL, or STEP 2.

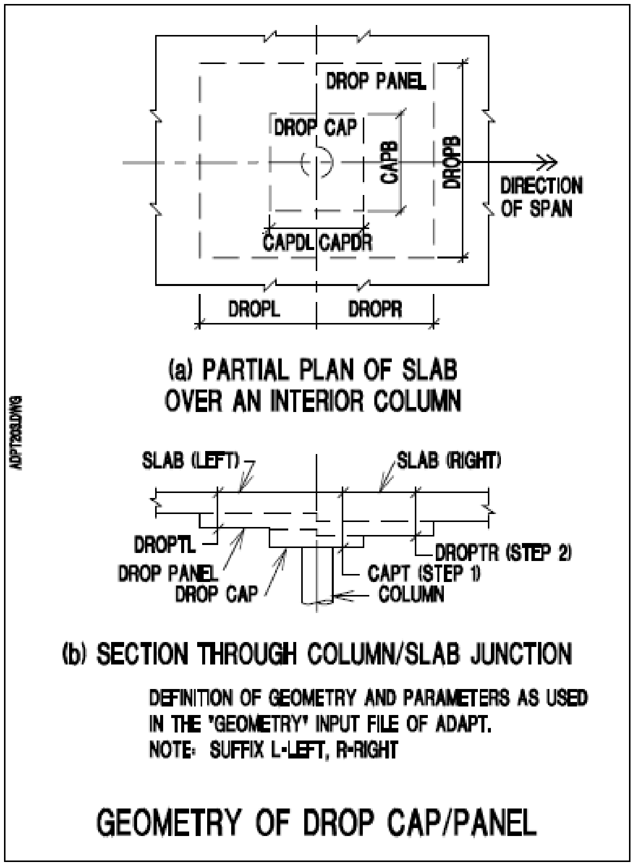


Figure 8.3-3

If no drop caps or panels are present, zeros are shown in this data block. A zero indicates that the user has not entered any value for a parameter. However, as far as the computations are concerned, ADAPT selects a minimum default value if necessary. For example, CAPT equal zero will result in a default value of CAPT equals span thickness for calculations.**2.6 Transverse Beam Data.** Transverse beam data are reported in the table of Section 2.5.

- **2.7 Support Width and Column Data.** This data block is only printed if column data is input, or if support widths are specified for reduction of moments to face-of-support. Otherwise the following sentence is printed: “NO COLUMN STIFFNESS IS INCLUDED IN THE ANALYSIS.”

Joint	Support Width	Length LC	B(DIA.) LC	D LC	% LC	CBC LC	Length UC	B(DIA.) UC	D UC	% UC	CBC UC
	in	ft	in	in			ft	in	in		
1	8.0	10.0	216.0	8.0	100	(2)					
2	18.0	10.0	12.0	18.0	100	(1)					

3	18.0	10.0	12.0	18.0	100	(1)					
4	8.0	10.0	216.0	8.0	100	(2)					

- **Support Width.** Is the support width at each joint. These values are used in the reduction of moments to the face-of-support. This value may be different from the column dimensions.
- **Length LC.** Is the height of the lower column measured from the center of the slab to the top of the bottom slab.
- **Length UC.** Is the height of the upper column measured from the center of the slab to the bottom of the top slab.
- **B(DIA).** Is the dimension of the column cross-section normal to the direction of the frame. A circular column is entered using B(DIA) only.
- **D.** Is the column dimension parallel to the frame.
- **CBC** is the Column Boundary Condition parameter. B and D can also describe the horizontal dimensions of the structural element supporting the slab, such as the thickness and length of a continuous wall. Whether or not a given wall/column support dimensioned through B and D is taking moments is determined by the way the wall/column is connected to the slab. The nature of the connection of the support to the slab at the slab/support junction is indicated by the CBC parameter as defined by the user and reflected in columns 6 and 10 of this data block. Note that the CBC parameter also describes the condition of fixity of the column at its far end away from the beam/slab, namely at its connection to the slab above and the slab/footing below.
- **%** is the percentage of the column stiffness included in the analysis.

8.3.4.3 Section 3: Input Applied Loading

This data block reports model geometry as input by the user. Loads entered by the user are sorted according to the span on which they act and are listed in the loading data block.

- **3.1 Loading A_s Appears in User's Input Screen.** This section mirrors the data as shown in the Loads input screen.

Span	Class	Type	W	P1	P2	A	B	C	F	M
			k/ft2	k/ft	k/ft	ft	ft	ft	k	k-ft
1	LL	U	0.050							
1	SDL	U	0.020							
2	LL	U	0.050							
2	SDL	U	0.020							
3	LL	U	0.050							
3	SDL	U	0.020							
4	LL	U	0.050							
4	SDL	U	0.020							

NOTE: SELFWEIGHT INCLUSION REQUIRED (SW= SELF WEIGHT Computed from geometry input and treated as dead loading. Unit self-weight W = 150.0 pcf
NOTE: LIVE LOADING is SKIPPED with a skip factor of 1.00

- **3.2 Compiled loads.** This section shows frame loads calculated by the program. If you specified uniformly distributed or partial loadings, the program would calculate frame loading based on the tributary width.

Span	Class	Type	P1	P2	F	M	A	B	C	Reduction Factor
			k/ft	k/ft	k	k-ft	ft	ft	ft	%
1	LL	U	1.000							0.000
1	SDL	U	0.400							
1	SW	U	1.891							
2	LL	P	1.000				0.000	15.000		0.000
2	LL	P	0.417				15.000	20.000		0.000
2	LL	P	1.000				20.000	35.000		0.000
2	SDL	P	0.400				0.000	15.000		
2	SDL	P	0.167				15.000	20.000		
2	SDL	P	0.400				20.000	35.000		
2	SW	P	1.912				0.000	15.000		
2	SW	P	1.029				15.000	20.000		
2	SW	P	1.912				20.000	35.000		
3	LL	P	1.000				0.000	15.000		0.000

- **Class**. Specifies load class for each span. Class LL is live load, class SDL is superimposed dead load, class SW is self-weight and class X is other loading.

- **Type**. There are 8 different load types:
  - **U** is for a uniformly distributed load acting on the entire tributary.
  - **C** is for a concentrated load. It acts at a point entered by the user and measured from the left support of the respective span.
  - **P** is for a partial uniform load that acts on the entire width (tributary) of a span over the length entered by the user.
  - **M** is for an applied moment that acts on the entire tributary at a distance from the left support entered by the user.
  - **L** is for a Line load that acts along the frame line of the slab. Line loads are entered in the same manner as partial loads.
  - **R** is for triangle load that acts along the frame line of the slab.
  - **V** is for variable load that acts along the frame line of the slab.
  - **T** is for trapezoidal load that acts along the frame line of the slab.

The user can also select the self-weight option. Using the geometry and unit weight entered by the user, ADAPT calculates the self-weight of the entire beam/slab and automatically amends the loading file. The value of the self-weight loading will appear in the output data.

Live load is not skipped unless stipulated by the user, in which case the following sentence appears at the end of the loading table:

LIVE LOADING is SKIPPED with a skip factor of x.xx.

When the skip load option is activated, ADAPT obtains two sets of solutions. (i) In the first set, live loading is assumed to act without the skip factor on all spans. (ii) In the second set, live load multiplied by the specified skip factor is selectively placed on different spans. Solutions of the second set are combined to yield the maximum possible negative and positive moments at each location. Results of moment combinations from (i) and (ii) are then merged to arrive at the governing moments for design.

8.3.4.4 Section 4: Calculated Section Properties

The data block on section properties gives the cross-sectional area, moment of inertia, and the location of the neutral axis of the entire tributary perpendicular to the direction of the span.

- **4.1 Section Properties of Uniform Spans and Cantilevers**. The table below shows the data for rectangular cross-sections.

Span	Area	Yb	Yt	b_eff	I	Yb	Yt
	in2	in	in	in	in4	in	in
1	1815.40	18.40	5.58	85.50	0.4423E+05	15.75	8.23
2	---	---	---	---	---	---	---
3	---	---	---	---	---	---	---
4	1815.40	18.40	5.58	85.50	0.4423E+05	15.75	8.23

In the case of flanged T-beams, there are two cross-sectional properties computed as shown above. One is for the section reduced by “effective” width, and the other is for the entire tributary. The reduced values are used for the general frame analysis of the model and to calculate stresses.

- **4.2 Section Properties for Non-Uniform Spans**. The table below shows the data for rectangular cross-sections.

Span	Seg.	Area	Yb	Yt	b_eff	i	Yb	Yt
		in2	in	in	in	in4	in	in
2	1	1836.00	18.41	5.59	105.00	0.4760E+05	16.37	7.63
2	2	988.00	16.21	7.77	100.00	0.4666E+05	16.21	7.77
2	3	1836.00	18.41	5.59	105.00	0.4760E+05	16.37	7.63
3	1	1836.00	18.41	5.59	105.00	0.4760E+05	16.37	7.63
3	2	988.00	16.21	7.77	100.00	0.4666E+05	16.21	7.77
3	3	1836.00	18.41	5.59	105.00	0.4760E+05	16.37	7.63

The calculated section properties are given in terms of span segments, for both customary and segmental input geometries. A non-segmental span with no drop caps or drop panels has one segment. A span with drop caps at either end has three segments. A span with drop caps and drop panels has five segments. Finally, a segmental span can have up to seven segments.

Yb and Yt refer to the distance from the section centroid to the top and bottom fibers.

When there is a change in cross-section of a span at the line of support, as shown in the idealized **Fig. 8.3-4(a)**, two options regarding the face-of-support arise. Over the support line, ADAPT considers the cross-section at the face-of-support of the shallower member to be the same as that of the deeper member. But, recognizing that the deeper span does not penetrate the shallower one, ADAPT assumes a zero length for the geometry of the deeper section into the shallower span. The same assumption is used for change of geometry over the supports of finite width as shown in **Fig. 8.3-4(b)**.

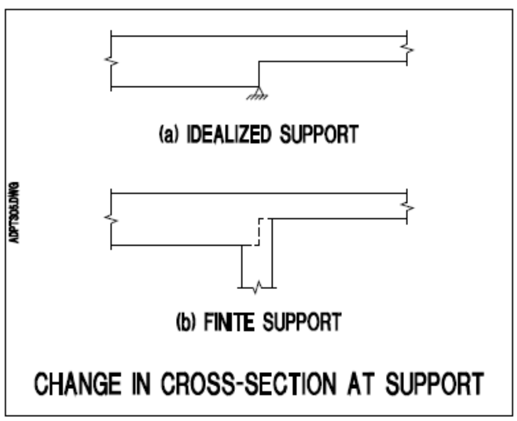


Figure 8.3-4

If the analysis is done with finite support widths, and the user has included the option to increase moment of inertia over the support, then ADAPT adds an additional segment over each support.

8.3.4.5 Section 5: Moments, Shears and Reactions

Values given as moments, shears and reactions all refer to the *total tributary* and not the unit strip. Moments in this data block are moments at the center of supports (system line moments).

- 5.1 Span Moments and Shears (Excluding Live Load).

Span	Load Case	Moment Left	Moment Midspan	Moment Right	Shear Left	Shear Right
		k-ft	k-ft	k-ft	k	k
1	SW	-89.64	71.05	-152.26	-24.75	29.14
2	SW	-170.13	82.29	-179.23	-31.00	31.52
3	SW	-179.24	82.29	-170.11	-31.52	31.00
4	SW	-152.25	71.05	-89.65	-29.14	24.75
1	SDL	-19.08	15.12	-31.89	-5.25	6.15
2	SDL	-34.81	16.22	-36.29	-6.37	6.46
3	SDL	-36.29	16.22	-34.80	-6.46	6.37
4	SDL	-31.89	15.12	-19.08	-6.15	5.25
1	XL	0.00	0.00	0.00	0.00	0.00
2	XL	0.00	0.00	0.00	0.00	0.00
3	XL	0.00	0.00	0.00	0.00	0.00
4	XL	0.00	0.00	0.00	0.00	0.00

Moment Left and Moment Right relate to centerline moments in the slab at the left and right of each span, respectively.

Moment Mid-span refers to the moment at mid-span. The moment at mid-span is not necessarily the largest value. For the location and value of the maximum moment refer to Section 23 that gives detailed report at 1/20th points.

Shear Left and Shear Right are the centerline shear forces at the left and right of each span.

- **5.2 Reactions and Column Moments (Excluding Live Load)**

Joint	Load Case	Reaction	Moment Lower Column	Moment Upper Column
		k	k-ft	k-ft
1	SW	24.75	-89.64	0.00
2	SW	60.15	-17.87	0.00
3	SW	63.04	-0.02	0.00
4	SW	60.14	17.86	0.00
5	SW	24.75	89.65	0.00
1	SDL	5.25	-19.08	0.00
2	SDL	12.52	-2.91	0.00
3	SDL	12.92	0.00	0.00
4	SDL	12.52	2.91	0.00
5	SDL	5.25	19.08	0.00
1	XL	0.00	0.00	0.00
2	XL	0.00	0.00	0.00
3	XL	0.00	0.00	0.00
4	XL	0.00	0.00	0.00
5	XL	0.00	0.00	0.00

Reaction is the centerline dead load reaction at each support line.

Moment Lower Column and Moment Upper Column are upper and lower column moments and are given for each support at the connection of column to slab/beam. If a support does not have a column, or if moment transfer between the support and slab is inhibited by the user through the specification of an appropriate column boundary condition, a zero (0) is printed.

- **5.3 Span Moments and Shears (Live Load).** This section is a summary of maximum and minimum live load moments, and corresponding shear forces at the left and right centerlines, as well as at center span. Live load moments, shears and reactions are values reported at the center of supports and refer to the entire tributary.

Span	Moment Left Max	Moment Left Min	Moment Midspan Max	Moment Midspan Min	Moment Right Max	Moment Right Min	Shear Left	Shear Right
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft	k	k
1	-55.97	8.26	44.35	-6.54	-81.90	-19.00	-14.16	15.48
2	-95.17	-6.62	50.42	-9.86	-98.44	-11.59	-16.72	16.81
3	-98.45	-11.59	50.42	-9.86	-95.15	-6.62	-16.81	16.72
4	-81.89	-18.99	44.35	-6.53	-55.97	8.26	-15.48	14.16

Maximum and minimum values at a section may occur due to the skipping of live loading.

- **5.4 Reactions and Column Moments (Live Load).** This section is a summary of the maximum and minimum live load reactions and column moments given for each support at the connection of column to slab/beam.

Joint	Reaction Max	Reaction Min	Moment Lower Column Max	Moment Lower Column Min	Moment Upper Column Max	Moment Upper Column Min
	k	k	k-ft	k-ft	k-ft	k-ft
1	14.16	-1.04	8.26	-55.97	0.00	0.00
2	32.20	14.15	51.79	-59.07	0.00	0.00
3	33.62	15.49	64.50	-64.51	0.00	0.00
4	32.20	14.15	59.06	-51.78	0.00	0.00
5	14.16	-1.04	55.97	-8.26	0.00	0.00

If a support does not have a column, or if moment transfer between the support and slab is inhibited by the user through the specification of an appropriate column boundary condition, a zero (0) is printed.

These are minimum and maximum centerline values based on skipped loading case.

Values given as moments, shears and reactions all refer to the total tributary and not the unit strip. Moments in this data block are centerline moments.

8.3.4.6 Section 6: Moments Reduced to Face of Support

If the option of reducing moments to the face-of-support is invoked by the user, ADAPT adjusts the centerline moments to the face-of-support. The adjustments are based primarily on support widths. The adjusted values are printed in Sections 6.1 and 6.2.

- **6.1 Reduced Moments at Face of Support (Excluding Live Load)**

Span	Load Case	Moment Left	Moment Midspan	Moment Right
		k-ft	k-ft	k-ft
1	SW	-62.10	71.05	-119.58
2	SW	-135.33	82.29	-143.83
3	SW	-143.83	82.29	-135.33
4	SW	-119.58	71.05	-62.11
1	SDL	-13.24	15.12	-25.00
2	SDL	-27.65	16.23	-29.03
3	SDL	-29.04	16.23	-27.65
4	SDL	-25.00	15.12	-13.24
1	XL	0.00	0.00	0.00
2	XL	0.00	0.00	0.00
3	XL	0.00	0.00	0.00
4	XL	0.00	0.00	0.00

- **6.2 Reduced Moments at Face of Support (Live Load)**

Span	Moment Left Max	Moment Left Min	Moment Midspan Max	Moment Midspan Min	Moment Right Max	Moment Right Min
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
1	-40.15	7.05	44.34	-6.53	-64.55	-17.92
2	-76.38	-6.83	50.42	-9.87	-79.54	-11.45
3	-79.55	-11.45	50.42	-9.86	-76.36	-6.83
4	-64.54	-17.92	44.34	-6.53	-40.15	7.05

8.3.4.7 Section 7: Selected Post-tensioning Forces and Tendon Profiles (PT mode only)

- **7.1 Tendon Profile** Tendon profile types available in the library of the ADAPT version used are listed in this data block.

Tendon A

Span	Type	X1/L	X2/L	X3/L	A/L
1	1	0.100	0.500	0.100	---
2	1	0.100	0.500	0.100	---
3	1	0.100	0.500	0.100	---
4	1	0.100	0.500	0.100	---

- **Type**. Reflects the profile type chosen for each span. Parameters X1/L, X2/L, X3/L and A/L are tendon profile inflection points in terms of a fraction of the span length. These are illustrated in **Fig. 8.3-5**. Note that some parameters are unnecessary for describing certain profile types.

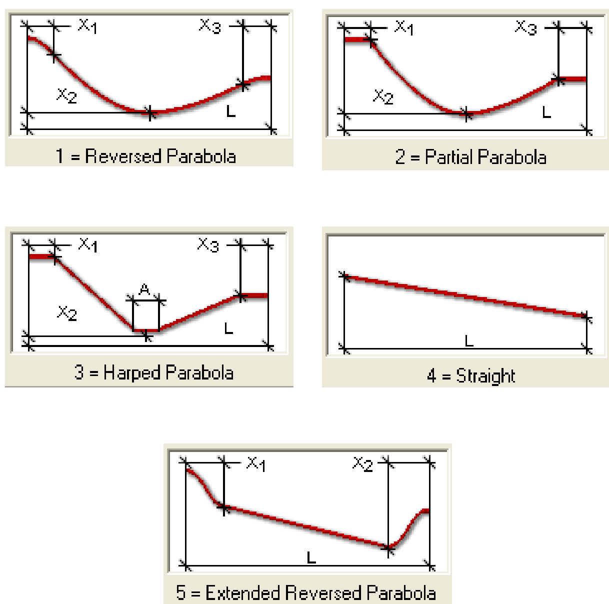


Figure 8.3-5

- **7.2 Selected Post-Tensioning Forces and Tendon Drape (PT mode only)** Columns 2 through 6 of this data block give the total post-tensioning forces and tendon heights selected by the user for tendon type A. Columns 7 and 8 are values calculated by ADAPT based on the post-tensioning selected in the preceding data block.

Tendon A

Span	Force	CGS Left	CGS C1	CGS C2	CGS Right	P/A	Wbal	WBal (%DL)
	k	in	in	in	in	psi	k/-	
1	274.554	18.42	---	5.75	22.00	151.24	3.259	142
2	274.554	22.00	---	2.02	22.00	277.89	2.985	139
3	274.554	22.00	---	2.02	22.00	277.89	2.985	139
4	274.554	22.00	---	5.75	18.42	151.24	3.259	142

The *All Tendons* table lists the sums of the PT force, P/A and Wbal of Tendon A, Tendon B, and Tendon C selected.

All Tendons

Span	Force	Total P/A	Total WBal (%DL)
	k	psi	
1	274.554	151.24	142
2	274.554	277.89	139
3	274.554	277.89	139
4	274.554	151.24	142

During the execution of the program, ADAPT calculates the required post-tensioning forces and displays them on the screen. The execution may pause, requiring the user to confirm the prompted values or modify them. The modification of the values calculated by ADAPT and prompted on the screen is referred to as Selection of post-tensioning by the user. After the user’s selection/ modification of forces and drapes, the program recalculates all the parameters and stresses based on the user’s input and prompts a new set of suggested values. The important issue for a reviewer to note is that, at this stage, the user has the option to override ADAPT’s calculated/displayed values. The Recycle Window is completely interactive and allows user modification.

It is recommended that the reviewer pay particular attention in comparing the user-selected post-tensioning (as displayed in data block 7.2) with the calculated required minimum values given in data block 7.4.

Columns 3 through 5 of data block 7.2 refer to the height of the tendon’s centroid (CGS) from a user-defined reference line. The left and right locations refer to the maximum height of tendon at left and right of the span relative to this reference line. In the case of a reversed parabola, these are the heights over the centerline of supports. For simple parabolas with straight portions over the supports, the heights refer to maximum rises at left and right of the span at location of transition to a straight line. The straight portion is called the Tendon Support Width and is defined through the coefficients in data block 7.1.

P/A in column 7 is the calculated average compression at mid-span, based on the user selected post-tensioning.

Wbal in column 8 is the balanced loading % in each span due to post-tensioning force and profile selected. It is the average uplift force over the entire span divided by the dead load tributary to the span. For purposes of calculating Wbal%, the program averages resulting concentrated forces from equivalent PT loading. For the frame analysis, the program uses the actual equivalent balanced loads due to PT at the proper, applied locations.

A positive value of Wbal means load acting upward against gravity. A negative value indicates a load in the direction of gravity.

- **7.4 Required Minimum Post-Tensioning Forces (PT mode only).** The forces in this data block refer to the required forces at left, center, and right for the entire tributary.

	Based on Stress Conditions			Based on Minimum P/A		
Type	Left	Center	Right	Left	Center	Right
	k	k	k	k	k	k
1	0.00	62.98	35.52	271.20	271.20	271.20
2	33.92	75.78	43.41	274.28	274.28	274.28
3	43.42	75.79	33.91	274.28	274.28	274.28
4	35.52	62.98	0.00	271.20	271.20	271.20

The required forces determined are the net effective forces after the immediate and long-term stress losses have been deducted. From the effective forces determined by ADAPT, the post-tensioning supplier calculates the initial forces required at time of stressing of tendons.

This data block shows the post-tensioning required to meet design criteria. Columns 2 through 4 are based on maintaining the tensile stresses in concrete at the location of maximum span moment, to the limit specified by the user in data block 1. Each row includes three regions for a given span. Data column 2 relates to the span's left support region; data column 3 is for the mid-span region; data column 4 shows the required force at the right support region of the same span. The "Left" region is 20% of the span length from the left support. The "Center" region is the 20-80% the span distance in the interior portion of span. The "Right" region is 20% of the span length from the right support.

For example, if the specified permissible stress for the exterior span is input as x(f'c)1/2 in data block 1, the number printed on column 3 in row of span 1 is the post-tensioning force necessary to meet that requirement.

At a given support, the post-tensioning required at the left of a support may be different from the force at its right.

The following considerations are observed in calculating the required post-tensioning in the support region:

- Stresses are calculated at 1/20th points in the span. In the region closest to the left support, the highest stress value is selected, and the required post-tensioning force provided to meet this condition is printed. If user wishes to know the exact location of the highest stress, detailed reports of stresses at 1/20th points given in report Section 27 should be reviewed.
- If moments are not reduced to the face-of-support, the centerline moments are used in lieu of moments reduced to face.
- The cross-section associated with the centerline of a support is that of the slab at the support line without any contribution from the supporting structure.

If at any location the existing moments are such that no post-tensioning is required, a zero (0) will be printed at that location.

MIN P/A in column 5 through 7 are the post-tensioning forces required to provide the user-specified minimum average compression (see data block 1) in the left, right, and center regions of the span.

- **7.5 Service Stresses (PT mode only).** Based on the post-tensioning forces and profiles confirmed or selected by the user, the top and bottom fiber stresses are calculated for each span and printed out in this data block. The stresses refer to concrete.

Envelope of Service 1

Span	Left Top Max-T	Left Top Max-C	Left Bot Max-T	Left Bot Max-C	Center Top Max-T	Center Top Max-C	Center Bot Max-T	Center Bot Max-C	Right Top Max-T	Right Top Max-C	Right Bot Max-T	Right Bot Max-C
	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi
1	-----	-272.94	81.66	-----	-----	-53.08	-----	-404.28	-----	-330.87	192.52	-----
2	-----	-319.20	214.34	-----	-----	-224.12	-----	-465.39	-----	-320.63	217.42	-----
3	-----	-320.66	217.48	-----	-----	-224.12	-----	-465.39	-----	-319.14	214.21	-----
4	-----	-330.84	192.46	-----	-----	-53.08	-----	-404.29	-----	-272.94	81.65	-----

Envelope of Service 2

Span	Left Top Max-T	Left Top Max-C	Left Bot Max-T	Left Bot Max-C	Center Top Max-T	Center Top Max-C	Center Bot Max-T	Center Bot Max-C	Right Top Max-T	Right Top Max-C	Right Bot Max-T	Right Bot Max-C
	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi
1	-----	-283.96	102.74	-98.93	-----	-122.39	-----	-423.83	-----	-330.87	192.52	-83.27
2	-----	-319.20	214.34	-100.84	-----	-294.67	-----	-494.16	-----	-320.63	217.42	-110.82
3	-----	-320.66	217.48	-110.79	-----	-294.67	-----	-494.16	-----	-319.14	214.21	-100.89
4	-----	-330.84	192.46	-83.30	-----	-122.39	-----	-423.83	-----	-283.95	102.73	-98.93

Tensile stresses are shown as positive and compressive stresses as negative. Maximum tension and compression, at the top and bottom at the left and right face-of-support and mid-span, are printed in their respective columns. Note that the maximum stress in a span may occur elsewhere. For the location and magnitude of the maximum stress, refer to either the graphical display of stress distribution, or the detailed report of stresses at 1/20th point of each span.

- **7.6 Post-Tensioning Balance Moments, Shears and Reactions (PT mode only).** The post-tensioning balanced moments are moments generated in the slab because of post-tensioning. These are also referred to as equivalent loads. These are displayed in a manner like the moments, shears, reactions, and column moments in Section 5. They refer to the total tributary.

Span Moments and Shears

Span	Moment Left	Moment Center	Moment Right	Shear Left	Shear Right
	k-ft	k-ft	k-ft	k	k
1	127.75	-143.42	225.00	-0.70	-0.70
2	251.17	-140.50	261.75	-0.33	-0.33
3	261.83	-140.58	251.17	0.33	0.33
4	225.00	-143.42	127.75	0.69	0.69

Reactions and Column Moments

Joint	Reaction	Moment Lower Column	Moment Upper Column
	k	k-ft	k-ft
1	0.695	136.583	0.000
2	-0.370	22.383	0.000
3	-0.652	0.026	0.000
4	-0.368	-22.367	0.000
5	0.695	-136.583	0.000

If the reduction of moments to the face-of-support option is used in the data input (refer to data block 1), slab moments printed are those reduced to face-of-support. Otherwise, they are centerline moments. Shears, reactions, and column moments are centerline values.

It is reiterated that values printed herein are due only to post-tensioning. Since post-tensioning forces are in self-equilibrium, the sum of external reactions generated by them must add up to zero. The reactions reported in this data block are the hyperstatic (secondary) actions due to post-tensioning.

8.3.4.8 Section 8: Factored Moments and Reactions Envelope

This data block lists the duly combined actions for the evaluation of the member's ultimate strength.

- **8.1 Factored Design Moments (Not Redistributed).** The factored and combined actions, or design actions, are the sum of dead loading, live loading, and secondary effects, each multiplied by a coefficient. If reduction to face-of-support is invoked by the user, the factored moments given relate to face-of-support; else, they represent centerline moments.

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
1	-17.23	58.29	320.86	239.45	-121.20	-46.58
2	-138.61	-27.35	383.40	286.93	-144.96	-36.01
3	-144.90	-35.94	383.40	286.95	-138.59	-27.35
4	-121.18	-46.58	320.86	239.46	-17.24	58.28

- **8.2 Reactions and Column Moments.** The factored support reactions and moments apply to the analysis solution considering the effective geometry with application of load from the full tributary entered in data block 2.1.

Joint	Reaction Max	Reaction Min	Moment Lower Column Max	Moment Lower Column Min	Moment Upper Column Max	Moment Upper Column Min
	k	k	k-ft	k-ft	k-ft	k-ft
1	59.35	35.04	19.30	-83.46	0.00	0.00
2	138.35	109.47	80.30	-97.06	0.00	0.00
3	144.29	115.28	103.20	-103.21	0.00	0.00
4	138.34	109.46	97.05	-80.30	0.00	0.00
5	59.35	35.03	83.46	-19.30	0.00	0.00

For the design of columns, total factored reactions and factored column moments are normally used. If design handbooks are used for column design, the relating eccentricity of the axial loading commonly required for use in such handbooks is readily obtained by dividing the printed factored column moment by the corresponding total factored reaction. Shears, reactions, and column moments are centerline values.

- **8.3 Secondary Moments (PT mode only).** Secondary (hyperstatic) moments are caused by post-tensioning forces. These are induced in the member by the constraints of the supports to the member's free movement. They are calculated from the secondary actions at the supports.

Span	Left	Midspan	Right
	k-ft	k-ft	k-ft
1	137.42	146.50	155.58
2	179.17	184.50	189.75
3	189.83	184.50	179.17
4	155.58	146.50	137.42

Secondary moments are adjusted to the face-of-support, provided this option is selected during input into ADAPT.

- **8.4 Factored Design Moments (Redistributed).** These section shows moments are listed in Section 8.1 after redistribution. This section is available only if you selected to redistribute moments.

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min	Redist. Coef. Left	Redist. Coef Right
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft		
1	59.09	-17.48	326.72	240.93	-124.16	-41.98	0.00	11.24
2	-124.21	-30.44	387.72	305.89	-129.95	-32.48	11.24	11.21
3	-129.95	-32.48	387.72	305.91	-124.24	-30.52	11.24	-11.24
4	-124.19	-41.98	326.72	240.91	59.07	-17.49	-1.53	0.00

8.3.4.9 Section 9: Factored Lateral Moments Envelope

This section shows the results for the combination of lateral and gravity moments.

- **9.1 Input Lateral Moments.** This section mirrors the input data specified in the Lateral Input Data screen.

Span	Left	Right
	k-ft	k-ft
1	30.00	-30.00
2	45.00	-45.00
3	45.00	-45.00
4	30.00	-30.00

- **9.2 Factored Lateral Moments (Not Redistributed).** This section shows factored and combined actions of dead loading, live loading, secondary effects, and lateral moments each multiplied by a coefficient.

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
1	+97.16	+34.41	+294.25	+224.05	-110.01	-2.09
2	+74.52	+18.80	+353.14	+273.17	-139.18	-7.78
3	+76.22	+12.91	+353.14	+273.17	-134.81	-9.55
4	-54.91	-8.28	+294.25	+224.05	+42.05	-20.70

- **9.3 Factored Lateral Moments (Redistributed).** This section shows moments listed in Section 9.2 for redistribution. This section is available only if you selected to redistribute moments.

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min	Redist. Coef. Left	Redist. Coef Right
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft		
1	98.14	35.26	300.79	226.41	-99.14	-1.88	0.00	11.24
2	75.30	19.25	360.71	276.34	-125.49	-7.01	0.00	11.24
3	77.02	13.29	360.38	276.44	-121.57	-8.60	0.00	11.24
4	-61.43	-9.27	296.73	226.29	42.47	-21.16	-11.24	0.00

If reduction to face-of-support is invoked by the user, the factored moments given relate to face-of-support; else, they represent centerline moments.

8.3.4.10 Section10: Mild Steel - No Redistribution

The mild reinforcement is calculated based on one of the following three sets of criteria:

- One-way systems.
- Two-way systems; and
- Beam system.

The applicable set of criteria is invoked by the user through the choice of the structural system (one-way, two-way, beam) and the design type (post-tensioned or mild-reinforced). The user's selection is shown clearly in data block 1.

- **10.1 Required Rebar.** This section shows required rebar based on ultimate, minimum, initial (PT mode only) and the UBC special load case requirement (PT mode only). Column 5 reports the maximum enveloped value of reinforcement for the span and position in span.

10.1.1 Total Strip Required Rebar

Span	Location	From	To	As Required	Ultimate	Minimum	Initial	UBC
		ft	ft	in2	in2	in2	in2	in2
1	TOP	0.00	4.27	2.23	0.00	2.23	0.00	0.00
1	TOP	24.23	28.50	2.23	0.00	2.23	0.00	0.00
2	TOP	0.00	5.25	2.66	0.00	2.66	0.00	0.00
2	TOP	29.75	35.01	2.66	0.00	2.66	0.00	0.00
3	TOP	0.00	5.25	2.66	0.00	2.66	0.00	0.00
3	TOP	29.75	35.01	2.66	0.00	2.66	0.00	0.00
4	TOP	0.00	4.27	2.23	0.00	2.23	0.00	0.00
4	TOP	24.23	28.50	2.23	0.00	2.23	0.00	0.00
1	BOT	0.00	1.42	0.57	0.00	0.00	0.57	0.00
1	BOT	9.97	18.52	1.38	0.00	1.38	0.00	0.00
1	BOT	25.65	28.50	1.72	0.00	0.00	1.72	0.00
2	BOT	0.00	3.50	2.07	0.36	0.00	2.07	0.00
2	BOT	12.25	22.75	1.44	0.00	1.44	0.00	0.00
2	BOT	31.50	35.01	2.15	0.00	0.00	2.15	0.00
3	BOT	0.00	3.50	2.15	0.38	0.00	2.15	0.00
3	BOT	12.25	22.75	1.44	0.00	1.44	0.00	0.00
3	BOT	31.50	35.01	2.07	0.00	0.00	2.07	0.00
4	BOT	0.00	2.85	1.72	0.19	0.00	1.72	0.00
4	BOT	9.97	18.52	1.38	0.00	1.38	0.00	0.00
4	BOT	27.07	28.50	0.57	0.00	0.00	0.57	0.00

- **Provided Rebar.** This section lists provided rebar details as calculated by the program.

10.2.1 Total Strip Provided Rebar

Span	ID	Location	From	Quantity	Size	Length	Area
			ft			ft	in2
1	1	TOP	0.00	8	5	6.00	2.48
1	2	TOP	22.80	9	5	13.00	2.79
2	3	TOP	28.00	9	5	14.00	2.79
3	4	TOP	28.00	9	5	13.00	2.79
4	5	TOP	22.80	8	5	6.00	2.48
1	6	BOT	0.00	1	8	3.00	0.79
1	7	BOT	8.55	2	8	11.50	1.58
1	8	BOT	24.23	2	8	10.50	1.58
2	9	BOT	10.50	2	8	14.00	1.58
2	10	BOT	29.75	2	8	11.50	1.58
3	11	BOT	10.50	2	8	14.00	1.58
3	12	BOT	29.75	2	8	9.50	1.58
4	13	BOT	8.55	2	8	11.50	1.58
4	14	BOT	25.65	1	8	3.00	0.79
1	16	BOT	25.65	1	8	7.50	0.79
2	17	BOT	31.50	1	8	8.00	0.79
3	18	BOT	31.50	1	8	6.50	0.79

10.2.2 Total Strip Steel Disposition

Span	ID	Location	From	Quantity	Size	Length
			ft			ft
1	1	TOP	0.00	8	5	6.00
1	2	TOP	22.80	9	5	5.70
2	2	TOP	0.00	9	5	7.30
2	3	TOP	28.00	9	5	7.00
3	3	TOP	0.00	9	5	7.00
3	4	TOP	28.00	9	5	7.00
4	4	TOP	0.00	9	5	6.00
4	5	TOP	22.80	8	5	6.00
1	6	BOT	0.00	1	8	3.00
1	7	BOT	8.55	2	8	11.50
1	8	BOT	24.23	2	8	4.27
1	16	BOT	25.65	1	8	2.85
2	8	BOT	0.00	2	8	6.23
2	9	BOT	10.50	2	8	14.00
2	10	BOT	29.75	2	8	5.25
2	16	BOT	0.00	1	8	4.65
2	17	BOT	31.50	1	8	3.50
3	10	BOT	0.00	2	8	6.25
3	11	BOT	10.50	2	8	14.00
3	12	BOT	29.75	2	8	5.25
3	17	BOT	0.00	1	8	4.50
3	18	BOT	31.50	1	8	3.50
4	12	BOT	0.00	2	8	4.25
4	13	BOT	8.55	2	8	11.50
4	14	BOT	25.65	1	8	3.00
4	18	BOT	0.00	1	8	3.00

- **10.3 Base Reinforcement.** This section describes base reinforcement as entered by the user.

Base Reinforcement

Isolated bars

Span	Location	From	Quantity	Size	Cover	Length	Area
--	--	ft	--	--	in	ft	in2
1	TOP	.00	5	5	1.50	72.59	1.55

Mesh Reinforcement

#	Span	Location	From	Spacing	Size	Cover	Length	Area
--	--	--	ft	in	--	in	ft	in2
1	1	BOT	.00	12.00	4	1.50	19.17	3.60
	2	BOT	.00	12.00	4	1.50	27.17	3.60

8.3.4.11 Section11: Mild Steel –Redistributed

This section is the same as Section 10. The difference is that the values for reinforcement are based on the redistributed moments.

8.3.4.12 Section 12: Shear Reinforcement

Depending on the structural system selected, the results are based on the type of shear check made, punching shear check (two-way systems) or a one-way shear check (one-way systems).

A one-way shear check is conducted for beams and one-way slabs.

- **12.1 Shear Calculation Envelope**. Each span is subdivided into 20 equal parts. Shear is checked at each subdivision.

12.1 Shear Calculation Envelope
SPAN 1

X/L	X	d	Vu,max	Mu,max	Ratio	Req.	Spacing	Base
	ft	in	k	kft		in2/ft	in	
0.00	0.00	19.20	-111.91	-0.01	1.11	0.000	0.00	OK
0.05	1.75	19.20	-100.72	187.89	1.26	0.023	18.00	NS
0.10	3.50	19.20	-89.54	356.00	1.62	0.213	18.00	NS
0.15	5.25	19.20	-78.33	504.32	1.65	0.166	18.00	NS
0.20	7.00	19.20	-67.15	632.92	1.62	0.095	18.00	NS
0.25	8.75	19.20	-55.96	741.97	1.39	0.000	0.00	OK
0.30	10.50	19.20	-44.76	830.56	1.11	0.000	0.00	OK
0.35	12.25	20.20	-33.57	899.96	0.79	0.000	0.00	OK

0.40	14.00	21.20	-22.39	949.42	0.50	0.000	0.00	OK
0.45	15.75	21.80	-11.20	978.95	0.24	0.000	0.00	---
0.50	17.50	22.00	0.00	277.81	0.00	0.000	0.00	---
0.55	19.25	21.80	11.20	978.95	0.24	0.000	0.00	---
0.60	21.00	21.20	22.39	949.42	0.50	0.000	0.00	OK
0.65	22.75	20.20	33.57	899.96	0.79	0.000	0.00	OK
0.70	24.50	19.20	44.76	830.56	1.11	0.000	0.00	OK
0.75	26.25	19.20	55.96	741.97	1.39	0.000	0.00	OK
0.80	28.00	19.20	67.15	632.92	1.62	0.095	18.00	NS
0.85	29.75	19.20	78.33	504.32	1.65	0.166	18.00	NS
0.90	31.50	19.20	89.54	356.00	1.62	0.213	18.00	NS
0.95	33.23	19.20	100.72	187.89	1.26	0.023	18.00	NS
1.00	35.01	19.20	111.91	-0.01	1.11	0.000	0.00	OK

Note: "Ratio" is calculated using paired shear (V) and moment (M) design values resulting in the lowest concrete capacity. For ACI and CSA codes, the lowest value of V*d/M is used.
Note: Sections with **** have exceeded the maximum allowable shear stress.
Note: Base stirrups flagged as OK satisfy the requirements - additional reinforcement is not needed. Base stirrups flagged as NS are not sufficient - additional stirrups are reported in the table.

The first and last points refer to the system line at support (X/L=0 and X/L=1). It is not required by some codes to check the shears at the system line. The first point for which shear is to be checked is recommended to be taken a distance equal to the depth of member from the face-of-support. Hence, the values given for X/L=0 and X/L=1 are to be considered as a guideline.

The depth 'd' used for stirrup calculations is based on the total depth of the section, reinforcement cover, post-tensioning CGS (where applicable) and bar size.

For post-tensioned, one-way systems, the concrete capacity for ACI and CSA designs are dependent on the ratio of Vu*d/Mu, where 'Vu' and 'Mu' are ultimate demand shears and moments at the individual design sections. When Live Load skipping is active, the shear capacity at each design check is governed by the worst-case ratio. The program conservatively uses the Vu and Mu values from the load skipping data set which result in the lowest ratio. The values do not necessarily pair from the same skip pattern result.

When base shear reinforcement is applied to beams or one-way slab designs, the program reports the design status in the "Base" column of the report. Where base stirrups are applied, the program will report a status check as "OK" or "NS."

If the base stirrups meet the required area and spacing by calculation, "OK" will be reported and no additional reinforcement will be reported. If “NS” is reported, the base stirrups are not sufficient to meet the demand and the program will report the demand balance for required reinforcement along with the necessary spacing. This reported reinforcement in Report 12 is in addition to the base reinforcement entered by the user.

8.3.4.13 Section 13: Punching Shear Reinforcement

A punching shear check is carried out if the structural system is two-way.

- 13.1 Critical Section Geometry

Column	Layer	Cond.	a	d	b1	b2
			in	in	in	in
1	2	1	4.69	9.38	52.37	45.37
2	1	1	8.19	16.38	28.84	28.84
3	1	1	7.19	14.38	28.38	30.37
4	1	1	4.44	8.88	38.87	56.87

Layer : The layer of the reinforcement for each column
Cond. : 1 = Interior, 2 = End, 3 = Corner, 4 = Edge
a : The distance between the layer and face of column or drop cap(*)
d : Effective depth
b1 : length of section parallel to span line
b2 : length of section normal to span line

- **13.2 Critical Section Stresses.** The outcome of the punching shear analysis is summarized in data column entitled Stress ratio. This is the total stress from two-way action divided by the allowable stress. If the stress ratio for any support exceeds limits imposed by the code the cross section must be enlarged and/or the concrete capacity increased and/or punching shear reinforcement provided.

Label	Layer	Cond.	Factored shear	Factored moment	Stress due to shear	Stress due to moment	Total stress	Allowable stress	Stress ratio
			k	k-ft	ksi	ksi	ksi	ksi	
1	2	1	-211.90	+163.79	0.12	0.026	0.142	0.186	0.764
2	1	1	-341.03	-56.61	0.18	0.014	0.194	0.190	1.025
3	1	1	-395.71	+17.12	0.23	0.005	0.239	0.190	1.259
4	1	1	-196.22	-263.75	0.12	0.046	0.162	0.183	0.884

Four location classifications exist. These are Corner, Interior, End and Edge. These conditions are differentiated in the punching shear calculations by the number of edges that participate in development of shear resistance. These conditions are identified at the top of the data block. From the geometry of the problem input by the user, ADAPT determines which of the conditions is applicable at each support. The condition as identified by ADAPT is listed in data column 3. Condition 1 is Interior, Condition 2 is End, Condition 3 is Edge and Condition 4 is corner.

If ADAPT determines that a punching shear check is not applicable for a support, such as in the case of a wall support, no values will be printed for that joint.

Data columns 4 and 5 are the applicable factored shear and moment at the joints. Calculated stresses due to the factored shears and moments are shown in data columns 6 and 7.

Total stress (column 8) is the sum of stresses due to shear and bending (sum of columns 6 and 7). Column 9 reports the allowable stress, as determined by the applicable design code. Column 10 reports the stress ratio. This is the total stress divided by the allowable stress.

The contribution of the vertical component of force from post-tensioning is conservatively omitted in determining the allowable stress.

- **13.3 Punching Shear Reinforcement.** This section lists required punching shear reinforcement. The term "Dist" refers to the distance from the face of column or drop that the layer of studs is located. If the maximum allowable shear stress is exceeded, the program will report "*". If a value is followed by "*" (e.g. 8*) this refers to the distance from face of drop to the stud layer. This would apply only if drop panels or drop caps are modeled.

Reinforcement option: Shear Studs
Stud diameter: 0.38
Number of rails per side: 1

Col.	Dist	Dist	Dist	Dist	Dist	Dist	Dist	Dist	Dist	Dist
	in	in	in	in	in	in	in	in	in	in
1										
2	1.6	3.3	4.9	6.5	8.2					
3	1.0	2.1	3.1	4.1	5.1	6.2	7.2			
4										

Dist. = Distance measured from the face of support

Note: Columns with --- have not been checked for punching shear.

Note: Columns with *** have exceeded the maximum allowable shear stress.

8.3.4.14 Section 14: Deflections

The deflection data block in the report is a summary of the calculations showing only the maximum values obtained for each span. Deflections for each stage shown in the data block are calculated with due consideration of cracking and loss of stiffness. The program utilizes hard-coded values for modulus of rupture dictated by the selected design code.

Deflections are reported for sustained and total long-term conditions with and without consideration of incremental live load applied as sustained load an exposed to creep. For example, the reports blocks shown below are for “Service Combination 1” (SC1) and “Service Combination 3” (SC3). These are the combinations:

SC1: 1.0*SW + 1.0*SDL + 1.0*PT + 0.3*LL + 0.3*XL, referred to as “Sustained Load” in the load combination input.

SC3: 1.0*SW + 1.0*SDL + 1.0*PT + 1.0*LL + 1.0*XL, referred to as “Total Load” in the load combination input.

Note that the program allows for 2 “Sustained Load” combinations and 2 “Total Load” combinations.

The “Sustained” and “Total” load combination values are determined as follows:

Example 1 – Span 2 – Long-Term Deflections with consideration of incremental Live Load:

SC1 – Sustained = -0.14” + (-0.06”*0.3*3) = -0.20”

SC1 – Total = -0.20” + (-0.06”*0.7) = -0.24”

The sustained result is a long-term sustained deflection that includes the factor of 0.3 multiplied by the live load deflection. In effect, this is the portion of live load that is sustained through the life of the structure and subject to creep. The factor of 0.3 is taken directly from the load combination defined by the user.

The total result is the long-term total deflection which includes the additional increment of live load taken by the factor 0.7* the live load deflection. The factor of 0.7 is 1 – the user defined live load factor of 0.3.

The results for SC1 and SC2 (if used) are referred to as long-term deflections.

Example 2 – Span 2 – Long-Term deflection without consideration of incremental Live Load:

SC3 – Sustained = -0.14" (Note that no live load is included)

SC3 – Total = --0.14" + -0.06" = -0.21" (Note that this displacement is the instantaneous deflection + creep + live load deflection as a short-term load not subject to long-term effects).

NOTE: The second data block for combinations SC3 and SC4 are calculated identical to previous versions of the program.

14.1 Maximum Span Deflections - Service Combination 1

Span	SW	SW+PT	SW+PT+ SDL	SW+PT+SDL +Creep	LL	X	Sustained	Total
	in	in	in	in	in	in	in	in
1	0.04	-0.04	-0.03	-0.08(4079)	-0.03(10660)	0.00(*****)	-0.11(3114)	-0.13(2605)
2	0.07	-0.06	-0.05	-0.14(2902)	-0.06(6864)	0.00(*****)	-0.20(2102)	-0.24(1731)
3	0.07	-0.06	-0.05	-0.14(2902)	-0.06(6865)	0.00(*****)	-0.20(2103)	-0.24(1731)
4	0.04	-0.04	-0.03	-0.08(4079)	-0.03(10662)	0.00(*****)	-0.11(3114)	-0.13(2606)

14.3 Maximum Span Deflections - Service Combination 3

Span	SW	SW+PT	SW+PT+ SDL	SW+PT+SDL +Creep	LL	X	Sustained	Total
	in	in	in	in	in	in	in	in
1	0.04	-0.04	-0.03	-0.08(4079)	-0.03(10660)	0.00(*****)	-0.08(4079)	-0.11(3029)
2	0.07	-0.06	-0.05	-0.14(2902)	-0.06(6864)	0.00(*****)	-0.14(2902)	-0.21(2040)
3	0.07	-0.06	-0.05	-0.14(2902)	-0.06(6865)	0.00(*****)	-0.14(2902)	-0.21(2040)
4	0.04	-0.04	-0.03	-0.08(4079)	-0.03(10662)	0.00(*****)	-0.08(4079)	-0.11(3029)

Note: Deflections are calculated using effective moment of inertia of cracked sections.

The concrete's modulus of elasticity (E_C) used for the deflection is calculated by ADAPT using the concrete strength input by the user and the selected code's formula for normal-weight and light-weight concrete. The user has the option to overwrite the code-based modulus of elasticity and enter his/her choice.

The Creep Factor (K) is input by the user and is used in determining long-term, time-dependent deflections as described above.

A negative deflection value indicates deflection upward.

Values in the parentheses are the deflection ratios computed as the length of each span divided by its maximum deflection. Deflection ratios are entered as positive regardless of direction of deflection. If a deflection ratio exceeds 100000, a value equal to 99999 is entered within the parentheses.

The program checks for cracked deflection for multiple stages. Data column 2 (SW) is deflection due only to self-weight. It should be noted that this column of data serves only as background information for the user, for at no time during its function is the slab expected to be subjected to self-weight alone. Under normal conditions either post-tensioning or shoring will be present.

SW+PT in column 3 is the immediate elastic deflection of the slab due to the self-weight and the post-tensioning.

SW+PT+SDL in column 4 is the immediate elastic deflection of the slab due to the self-weight, user-defined dead loading, and the post-tensioning.

SW+PT+SDL+CREEP in column 5 is the sum of the immediate deflection (column 4) and the deflection due to creep. The deflection due to creep is applied to the live load as outlined in Example 1 above. Column 5 is (1+K) times the deflection due to SW+PT+SDL (column 4).

Deflection due to live loading (LL) is listed in column 6 and is calculated by taking (SW+PT+SDL+LL) – Column 4. This represents the total live load deflection. From this value, the program applies the necessary sustained load factor (0.3 by default) and creep factor to determine that portion of long-term deflection influenced by sustained live load.

Where X load is applied, the program determines this by taking (SW+PT+SDL+LL+XL) – (SW+PT+SDL+LL). The deflection for X load is added directly to the sustained or total load result with no creep factor applied.

8.3.4.15 Section15: Friction, Elongation and Long-Term Stresses (PT mode only)

- **15.1 Input Parameters**. This section shows the values entered during data entry.

Parameter	Value	Parameter	Value
Type of Strand	Low Relaxation	Coefficient of Angular Friction (meu)	0.07000 1/rad
Age of Concrete at Stressing	5 days	Coefficient of Wobble Friction (K)	0.00050 rad/m
Ec at Stressing	10500.00 MPa	Ratio of Jacking Stress	0.90
Average Relative Humidity	80.00 percent	Anchor Set	6.00 mm
Volume to Surface Ratio of Members	130.00 mm	Tendon_A Stressing Method	Both side
Es of Strand	190000.00 MPa		

- **15.2 Long-term Losses**. This section can report different information depending on the option that the user selected for the long-term stress loss calculation. There are three options for long-term stress loss calculations:
  - Lump sum entry. A lump sum value may be calculated by the user and entered during data input. The effective stresses in the tendon are calculated by subtracting this value from the initial stresses. Since the friction and seating losses cause the initial stresses to vary along the tendon, the effective stresses will also vary. The lump sum is reported in the Section 15.1 Input parameters. In this case the section 15.2 is not reported.
  - Long-Term Loss calculations for unbonded tendons. For unbonded tendons, the strain in the tendon at any given point is not directly related to the local strain in the concrete. The program can calculate and report an average long-term loss value for the entire tendon based on the average precompression in the member and expected losses due to shrinkage, creep, elastic shortening and relaxation of the prestressing steel. The effective stresses in the tendon are calculated by subtracting the average long-term loss value from the initial stresses. To obtain these results the user must execute ADAPT in Force Selection mode. The section table will show as follows:

Tendon	Elastic shortening	Shrinkage	Creep	Relaxation	Total
	MPa	MPa	MPa	MPa	MPa
TENDON_A	28.52	17.28	34.96	42.49	123.20

- Long-Term Loss computation for grouted tendons. Long-term stress losses in grouted tendons are a function of the local strain in the concrete. To calculate long-term stress loss for a grouted system, a detailed strain computation must be performed along the path of a tendon. A detailed listing of stresses and long-term losses is available if ADAPT is executed in Tendon Selection mode. In this case the values at left, center and right of each span are listed as shown in the following table:

Tendon	Span	Left	Center	Right
		ksi	ksi	ksi
TENDON_A	CL	7.29	7.57	8.16
TENDON_A	1	8.26	8.87	9.46
TENDON_A	2	9.58	9.05	9.03

ADAPT calculates the losses at 1/20th points along each span and lists them in a file for interested users (friction_detail.dat). This is a text file and can be viewed with any text editor or word processor.

If a friction and long-term losses calculation need to be done for a grouted system designed with Force Selection, a lump sum must be entered for long-term loss.

- **15.3 Calculated Stresses After Friction and Long-Term Losses (PT mode only).** Shows calculated stresses after friction and long-term loss at left, center and right of span.

Tendon	Span	Stress Left FL Only	Stress Center FL Only	Stress Right FL Only	Stress Left FL+LTL	Stress Center FL+LTL	Stress Right FL+LTL
		ksi	ksi	ksi	ksi	ksi	ksi
TENDON_A	1	215.59	218.49	224.01	203.97	206.88	212.54
TENDON_A	2	224.01	225.32	220.53	212.54	213.70	208.91
TENDON_A	3	220.53	213.85	210.36	208.91	202.23	198.75

- **15.4 Summary (PT mode only).** Gives the average initial stress, total long-term losses, final average stress, and final average force. The section shows the left, right and total elongation after anchor set. It also gives the left and right anchor set influence distances.

Tendon	Avg. Initial Stress	LTL	Avg. Final Stress	Avg. Final Force	Elongation Left	Elongation Right	Elongation Total	Left Anchor Set	Right Anchor Set

	ksi	ksi	ksi	k	in	in	in	ft	ft
TENDON_A	219.45	11.60	207.85	31.89	5.80	-0.06	5.74	28.00	32.00

- **15.5 Critical Stress Ratios (PT mode only).** The critical stress ratios show the ratios of the calculated tendon stress to the strand's specified ultimate strength.

Tendon	Stressing Left	Stressing Right	Anchorage Left	Anchorage Right	Max
TENDON_A	0.90	0.90	0.80	0.78	0.84

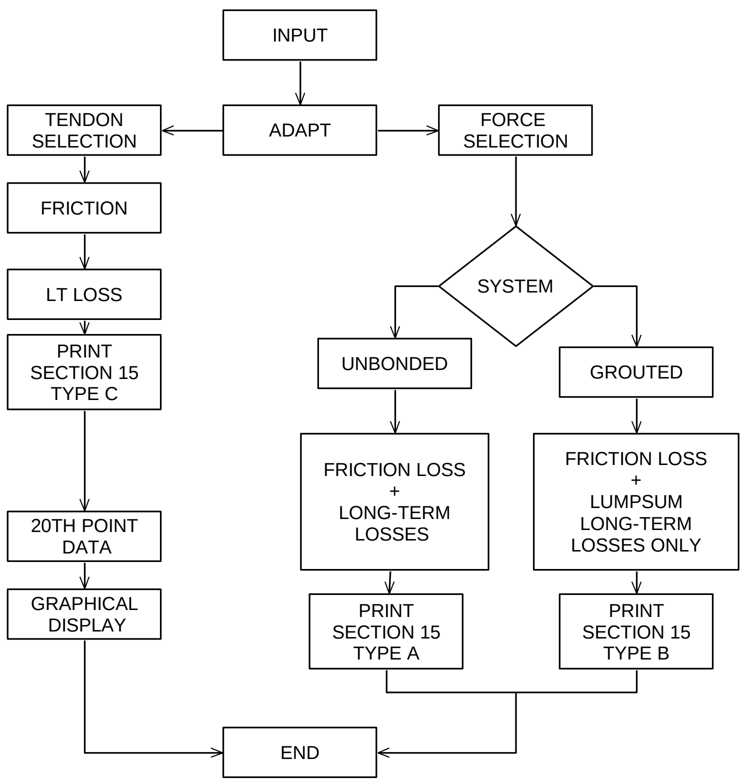
- **15.6 Summary (PT mode only).** Gives the average force per span, tendon extents, elongation left and right of span, anchor set left and right of span, stress at anchorage and maximum stress along the tendon. This data block only applies if the calculation method is Tendon Selection Method.

Tendon	Force	Ext. Left	Start Span	End Span	Ext. Right	Elong. Left	Elong Right	Anchor Left	Anchor Right	Max Stress ratio
	k					in	in			
TENDON_A	33.58	0.00	1	3	0.00	5.80	-0.06	0.80	0.78	0.84
TENDON_B	33.63	0.00	1	1	0.20	2.68	0.00	0.80	0.84	0.84
TENDON_C	32.83	0.20	3	3	0.00	0.00	2.16	0.83	0.78	0.83

- **Tendon** column lists the types of tendons used. These may be tendons A, B and C.
- **Force** is the average force in each strand along the length of each tendon. The average force is not used in the computations. It is listed as an indicator since this force is often viewed as a measure of effectiveness of a strand when comparing alternatives.
- **Ext. Left and Ext. Right** illustrate the extent of each tendon type into left and right spans.
- **Elong. Left and Elong. Right** are the elongations at left and right ends of each tendon. If a tendon type is stressed at one end only, the elongation at the dead end will be shown by a zero (0) in the report.
- **Anchor Left and Anchor Right** is a ratio determined by taking the stress after seating loss at anchorage divided by the strand's ultimate strength.
- **Max. Stress Ratio** is the ratio of the maximum stress along each tendon to the strand's ultimate strength immediately after jacking and seating of tendon.

Example Output for: Type A, Type B, and Type C

- A solution for a grouted system in effective-force mode with lump sum stress specification (TYPE B).
- A solution for an unbonded tendon system in effective-force mode, together with computed long-term stress losses (TYPE A); and,
- A solution for a grouted system in tendon-selection mode with computed long-term stress loss (TYPE C). An unbonded tendon system with tendon-selection mode and computed long-term loss calculations also has a report type (C) as indicated in the flow chart.



15 - FRICTION, ELONGATION AND LONG-TERM LOSSES

15.1 Input Parameters

Parameter	Value	Parameter	Value
Type of Strand	Low Relaxation	Coefficient of Angular Friction (meu)	0.07000 1/rad
Age of Concrete at Stressing	5 days	Coefficient of Wobble Friction (K)	0.00140 rad/ft
Ec at Stressing	1523.00 ksi	Ratio of Jacking Stress	0.80
Average Relative Humidity	80.00 percent	Anchor Set	0.25 in
Volume to Surface Ratio of Members	0.00 in	Tendon_A Stressing Method	Both sides
Es of Strand	29000.00 ksi		

15.2 Long-term Losses

Tendon	Elastic Shortening	Shrinkage	Creep	Relaxation	Total
	ksi	ksi	ksi	ksi	ksi
TENDON_A	1.19	4.04	1.61	3.54	10.48

15.3 Calculated Stresses After Friction and Long-term Losses

Tendon	Span	Stress Left FL Only	Stress Center FL Only	Stress Right FL Only	Stress Left FL+LTL	Stress Center FL+LTL	Stress Right FL+LTL
		ksi	ksi	ksi	ksi	ksi	ksi
TENDON_A	1	177.67	182.56	189.66	167.19	172.08	179.18
TENDON_A	2	190.64	196.29	190.93	180.16	185.81	180.45
TENDON_A	3	189.97	183.44	176.94	179.49	172.96	166.46
TENDON_A	CR	176.47	173.32	170.50	165.99	162.84	160.02

15.4 Summary

Tendon	Avg. Initial Stress	LTL	Avg. Final Stress	Avg. Final Force	Elongation Left	Elongation Right	Elongation Total	Left Anchor Set	Right Anchor Set
	ksi	ksi	ksi	k	in	in	in	ft	ft
TENDON_A	183.20	10.48	172.72	26.94	5.59	0.00	5.59	32.03	32.5

15.5 Critical Stress Ratios

Tendon	Stressing Left	Stressing Right	Anchorage Left	Anchorage Right	Max
TENDON_A	0.80	0.80	0.66	0.64	0.73

Type A

15 - FRICTION, ELONGATION AND LONG-TERM LOSSES

15.1 Input Parameters

Parameter	Value	Parameter	Value
Long term Lump Loss	1.16 ksi	Ratio of Jacking Stress	0.80
Es of Strand	29000.00 ksi	Anchor Set	0.25 in
Coefficient of Angular Friction (meu)	0.07000 1/rad	Tendon_A Stressing Method	Both sides
Coefficient of Wobble Friction (K)	0.00140 rad/ft		

15.3 Calculated Stresses After Friction and Long-term Losses

Tendon	Span	Stress Left FL Only	Stress Center FL Only	Stress Right FL Only	Stress Left FL+LTL	Stress Center FL+LTL	Stress Right FL+LTL
		ksi	ksi	ksi	ksi	ksi	ksi
TENDON_A	1	178.42	183.36	190.47	177.26	182.20	189.31
TENDON_A	2	190.47	197.01	190.91	189.31	195.84	189.75
TENDON_A	3	190.91	184.38	175.52	189.75	183.21	174.36
TENDON_A	CR	175.52	173.78	172.18	174.36	172.62	171.02

15.4 Summary

Tendon	Avg. Initial Stress	LTL	Avg. Final Stress	Avg. Final Force	Elongation Left	Elongation Right	Elongation Total	Left Anchor Set	Right Anchor Set
	ksi	ksi	ksi	k	in	in	in	ft	ft
TENDON_A	184.00	1.16	182.84	27.98	5.65	-0.01	5.64	32.76	39.83

15.5 Critical Stress Ratios

Tendon	Stressing Left	Stressing Right	Anchorage Left	Anchorage Right	Max
TENDON_A	0.80	0.80	0.66	0.64	0.73

Type B

15 - FRICTION, ELONGATION AND LONG-TERM LOSSES

15.1 Input Parameters

Parameter	Value	Parameter	Value
Type of Strand	Low Relaxation	Coefficient of Wobble Friction (K)	0.00140 rad/ft
Age of Concrete at Stressing	5 days	Ratio of Jacking Stress	0.80
Ec at Stressing	1523.00 ksi	Anchor Set	0.25 in
Average Relative Humidity	80.00 percent	Tendon_A Stressing Method	Both sides
Volume to Surface Ratio of Members	0.00 in	Tendon_B Stressing Method	Left side
Es of Strand	29000.00 ksi	Tendon_C Stressing Method	Right side
Coefficient of Angular Friction (meu)	0.07000 1/rad		

15.2 Long-term Losses

Tendon	Span	Left	Center	Right
		ksi	ksi	ksi
TENDON_A	1	7.92	10.17	10.97
TENDON_A	2	10.90	11.20	10.89
TENDON_A	3	10.95	10.45	8.77
TENDON_A	CR	9.26	7.65	7.30
TENDON_B	1	7.73	10.17	10.85
TENDON_B	2	10.86	0.00	0.00
TENDON_B	3	0.00	0.00	0.00
TENDON_B	CR	0.00	0.00	0.00
TENDON_C	1	0.00	0.00	0.00
TENDON_C	2	0.00	0.00	10.84
TENDON_C	3	10.95	10.45	8.77
TENDON_C	CR	9.26	7.65	7.30

15.3 Calculated Stresses After Friction and Long-term Losses

Tendon	Span	Stress Left FL Only	Stress Center FL Only	Stress Right FL Only	Stress Left FL+LTL	Stress Center FL+LTL	Stress Right FL+LTL
		ksi	ksi	ksi	ksi	ksi	ksi
TENDON_A	1	178.42	183.36	190.47	170.44	173.20	179.44
TENDON_A	2	190.47	197.01	190.91	179.58	185.83	180.02
TENDON_A	3	190.91	184.38	175.52	180.02	173.92	166.95
TENDON_A	CR	175.52	173.78	172.18	166.23	166.08	164.78
TENDON_B	1	177.12	182.05	189.17	169.42	171.89	178.28
TENDON_B	2	189.17	0.00	0.00	178.42	0.00	0.00
TENDON_B	3	0.00	0.00	0.00	0.00	0.00	0.00
TENDON_B	CR	0.00	0.00	0.00	0.00	0.00	0.00
TENDON_C	1	0.00	0.00	0.00	0.00	0.00	0.00
TENDON_C	2	0.00	0.00	191.05	0.00	0.00	180.17
TENDON_C	3	191.05	184.38	175.52	180.17	173.92	166.95
TENDON_C	CR	175.52	173.78	172.18	166.23	166.08	164.78

15.6 Summary

Tendon	Force	Ext. Left	Start Span	End Span	Ext. Right	Elong. Left	Elong Right	Anchor Left	Anchor Right	Max Stress ratio
	k					in	in			
TENDON_A	28.15	0.00	1	CR	0.00	5.65	-0.01	0.66	0.64	0.73
TENDON_B	28.13	0.00	1	1	0.20	1.88	0.00	0.66	0.71	0.71
TENDON_C	27.58	0.20	3	CR	0.00	0.00	2.43	0.72	0.64	0.72

Type C

8.3.5 Detailed Report

The detailed report consists of listing of values at 1/20th points along each span. As an example, the following tables illustrate partial listings of those results available in the Detailed Report settings. Selected detailed results of the analysis and design at 1/20th points along each span may be included in the comprehensive output using the report setup dialog box. Detailed output for a portion of the results are available after completion of the run. The remainder of the detailed results are available after execution of the analysis and design post-processors. All the detailed output files are written to separate data files with the .DAT extension and located in the model files location. These files may be accessed by the user.

8.3.5.1 Section 21: Tendon Heights (PT mode only)

The detailed tendon height output reports the height of the centroid of each tendon type at 1/20th points along each span. The data is stored in the PTCGS.DAT file.

X/L	X	CGS A	CGS B	CGS C
	ft	in	in	in
	SPAN 1			
0.00	0.000	18.42	18.42	18.42
0.05	1.425	17.81	17.81	17.81
0.10	2.850	15.99	15.99	15.99
0.15	4.275	13.71	13.71	13.71
0.20	5.700	11.73	11.73	11.73
0.25	7.125	10.05	10.05	10.05
0.30	8.550	8.68	8.68	8.68
0.35	9.975	7.62	7.62	7.62
0.40	11.400	6.86	6.86	6.86
0.45	12.825	6.40	6.40	6.40
0.50	14.250	6.25	6.25	6.25
0.55	15.675	6.45	6.45	6.45
0.60	17.100	7.04	7.04	7.04
0.65	18.525	8.02	8.02	8.02
0.70	19.950	9.40	9.40	9.40
0.75	21.375	11.17	11.17	11.17
0.80	22.800	13.34	13.34	13.34
0.85	24.225	15.90	15.90	15.90
0.90	25.650	18.85	18.85	18.85
0.95	27.075	21.21	21.21	21.21
1.00	28.500	22.00	22.00	22.00

8.3.5.2 Section 22: Post-tensioning Balanced Loading (PT mode only)

This option lists the equivalent loading (balanced loads) generated to substitute the post-tensioning obtained in ADAPT's final design. These loads can be used as an applied loading in an independent general frame program to verify the accuracy of ADAPT post-tensioning solutions.

Span	Type	W	F	M	a	b
		k/ft	k	k-ft	ft	ft
1	3	13.928			0.00	2.85
1	3	18.021			25.65	28.50
1	3	-3.482			2.85	14.25
1	3	-4.505			14.25	25.65
2	3	15.158			0.00	3.50
2	3	15.158			31.50	35.00
2	3	-3.789			3.50	17.50
2	3	-3.789			17.50	31.50
3	3	15.158			0.00	3.50
3	3	15.158			31.50	35.00
3	3	-3.789			3.50	17.50
3	3	-3.789			17.50	31.50
4	3	18.021			0.00	2.85
4	3	13.928			25.65	28.50
4	3	-4.505			2.85	14.25
4	3	-3.482			14.25	25.65
1	4			-0.26	28.50	
4	4			0.26	0.00	
2	4			-50.77	15.00	
2	4			50.77	20.00	
3	4			-50.77	15.00	
3	4			50.77	20.00	

8.3.5.3 Section 23: Detailed Moments

This section reports moments for all load cases and enveloped (max/min) results when skip loading is active at 1/20th points along each span.

SPAN 1

X/L	X	SW	SDL	XL	LL Min	LL Max	PT	Secondary
	ft	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
0.00	0.00	-89.64	-19.08	0.00	-223.87	33.04	133.05	133.08
0.05	1.43	-56.29	-12.01	0.00	-147.19	27.12	119.88	134.06
0.10	2.85	-26.79	-5.74	0.00	-78.64	21.20	78.44	135.04
0.15	4.28	-1.12	-0.29	0.00	-18.21	15.28	26.39	136.01
0.20	5.70	20.71	4.35	0.00	8.34	44.42	-18.60	136.99
0.25	7.12	38.70	8.17	0.00	3.07	82.08	-56.51	137.97
0.30	8.55	52.85	11.19	0.00	-2.47	114.35	-87.35	138.95
0.35	9.98	63.16	13.39	0.00	-8.39	142.29	-111.12	139.92
0.40	11.40	69.63	14.78	0.00	-14.31	162.11	-127.82	140.90
0.45	12.83	72.26	15.36	0.00	-20.22	173.81	-137.45	141.88
0.50	14.25	71.05	15.12	0.00	-26.14	177.38	-140.01	142.85
0.55	15.68	66.00	14.08	0.00	-32.06	172.83	-134.45	143.83
0.60	17.10	57.10	12.22	0.00	-37.98	160.16	-119.75	144.81
0.65	18.53	44.37	9.55	0.00	-43.90	139.37	-95.91	145.78

0.70	19.95	27.80	6.06	0.00	-49.81	110.45	-62.91	146.76
0.75	21.38	7.39	1.77	0.00	-55.73	73.41	-20.77	147.74
0.80	22.80	-16.86	-3.34	0.00	-61.65	28.25	30.53	148.72
0.85	24.23	-44.95	-9.26	0.00	-99.45	-25.03	90.97	149.69
0.90	25.65	-76.88	-15.99	0.00	-167.37	-65.45	160.56	150.67
0.95	27.08	-112.65	-23.54	0.00	-243.42	-70.72	216.42	151.65
1.00	28.50	-152.26	-31.89	0.00	-327.59	-75.99	235.70	152.62

8.3.5.4 Section 24: Detailed Shears

This section reports shears for all load cases and enveloped (max/min) results when skip loading is active at 1/20th points along each span.

SPAN 1

X/L	X	SW	SDL	XL	LL Min	LL Max	PT	Secondary
	ft	k	k	k	k	k	k	k
0.00	0.00	-24.75	-5.25	0.00	4.15	-56.66	-0.68	-0.69
0.05	1.43	-22.06	-4.68	0.00	4.15	-50.96	19.17	-0.69
0.10	2.85	-19.36	-4.11	0.00	4.15	-45.26	39.02	-0.69
0.15	4.28	-16.67	-3.54	0.00	4.15	-39.56	34.06	-0.69
0.20	5.70	-13.97	-2.97	0.00	4.15	-33.86	29.09	-0.69
0.25	7.12	-11.28	-2.40	0.00	4.15	-28.16	24.13	-0.69
0.30	8.55	-8.58	-1.83	0.00	4.15	-22.46	19.17	-0.69
0.35	9.98	-5.89	-1.26	0.00	4.15	-16.76	14.21	-0.69
0.40	11.40	-3.19	-0.69	0.00	4.15	-11.06	9.25	-0.69
0.45	12.83	-0.50	-0.12	0.00	4.15	-5.36	4.29	-0.69
0.50	14.25	2.20	0.45	0.00	4.92	0.00	-0.68	-0.69
0.55	15.68	4.89	1.02	0.00	10.62	0.00	-7.10	-0.69
0.60	17.10	7.59	1.59	0.00	16.32	0.00	-13.52	-0.69
0.65	18.53	10.28	2.16	0.00	22.02	0.00	-19.94	-0.69
0.70	19.95	12.98	2.73	0.00	27.72	0.00	-26.36	-0.69
0.75	21.38	15.67	3.30	0.00	33.42	0.00	-32.78	-0.69
0.80	22.80	18.37	3.87	0.00	39.12	0.00	-39.20	-0.69
0.85	24.23	21.06	4.44	0.00	44.82	0.00	-45.62	-0.69
0.90	25.65	23.75	5.01	0.00	50.52	0.00	-52.03	-0.69
0.95	27.08	26.45	5.58	0.00	56.22	0.00	-26.36	-0.69
1.00	28.50	29.14	6.15	0.00	61.92	0.00	-0.68	-0.69

8.3.5.5 Section 25: Factored Moments and Reactions

This section reports factored design moments for Strength load combinations. The envelope of factored moments is shown in Section 8.

Load Combination: 1.20SW + 1.60LL + 1.20SDL + 1.60XL + 1.00SEC

Factored Design Moments (Not Redistributed)

Span	Left	Left	Middle	Middle	Right	Right

	Max	Min	Max	Min	Max	Min
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
1	-213.43	88.63	530.06	204.43	-434.73	-136.36
2	-503.88	-58.81	627.93	242.08	-523.01	-87.22
3	-523.02	-87.22	627.93	242.09	-503.83	-58.90
4	-434.73	-136.35	530.05	204.43	-213.44	88.62

Reactions and Column Moments

Joint	Reaction Max	Reaction Min	Moment Lower Column Max	Moment Lower Column Min	Moment Upper Column Max	Moment Upper Column Min
	k	k	k-ft	k-ft	k-ft	k-ft
1	127.34	30.04	55.45	-355.55	0.00	0.00
2	293.00	177.48	333.93	-375.53	0.00	0.00
3	305.55	189.47	412.80	-412.80	0.00	0.00
4	292.99	177.47	375.54	-333.93	0.00	0.00
5	127.34	30.04	355.55	-55.44	0.00	0.00

Factored Design Moments (Redistributed)

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min	Redist. Coef. Left	Redist. Coef Right
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft		
1	-216.34	90.13	543.04	196.91	-457.58	-123.20	0.00	-4.38
2	-458.36	-65.47	635.81	295.13	-477.58	-78.85	9.94	9.58
3	-477.58	-78.85	635.81	295.13	-458.25	-65.47	9.59	9.94
4	-457.47	-123.19	542.94	196.87	-216.35	90.12	-4.35	0.00

8.3.5.6 Section 26: Factored Lateral Moments

This section reports factored design moments for lateral load combinations. The envelope of factored moments is shown in Section 9.

Load Combination: 1.20 SW +1.00 LL +1.20 SDL +1.00 XL +1.00 LAT

Factored Lateral Moments (Not Redistributed)

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft
1	+99.26	-89.52	+423.70	+220.16	-307.43	-120.94
2	-278.60	-0.44	+506.92	+265.77	-374.08	-101.71
3	-290.08	-17.70	+506.92	+265.77	-362.54	-84.45
4	-252.34	-65.84	+423.60	+220.06	-144.63	+44.15

Factored Lateral Moments (Redistributed)

Span	Left Max	Left Min	Middle Max	Middle Min	Right Max	Right Min	Redist. Coef. Left	Redist. Coef Right
	k-ft	k-ft	k-ft	k-ft	k-ft	k-ft		
1	100.80	-89.02	434.67	239.48	-277.68	-109.24	0.00	11.02
2	-275.06	-0.11	517.65	291.89	-337.23	-91.74	1.82	11.07
3	-322.97	-19.47	515.72	272.48	-328.07	-76.23	-11.07	11.07
4	-281.80	-73.53	424.18	208.22	-147.23	44.29	-11.02	0.00

8.3.5.7 Section 27: Detailed Stresses (PT mode only)

This section reports top and bottom fiber stresses for individual load cases. When skip loading is active, stresses due to live load are enveloped. Initial and Service combination results are also reported with enveloped results provided when skip loading is active.

SPAN 1

X/L	X	SW Top	SW Bot	SDL Top	SDL Bot	XL Top	XL Bot	LL Top Max-T	LL Top Max-C	LL Bot Max-T	LL Bot Max-C	PT Top	PT Bot
	ft	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi
0.00	0.00												
0.05	1.43	126.	-241.	27.	-51.	0.	0.	329.	-61.	116.	-629.	-421.	359.
0.10	2.85	60.	-114.	13.	-25.	0.	0.	176.	-47.	91.	-336.	-329.	182.
0.15	4.28	2.	-5.	1.	-1.	0.	0.	41.	-34.	65.	-78.	-212.	-41.
0.20	5.70	-46.	88.	-10.	19.	0.	0.	-19.	-99.	190.	36.	-112.	-233.
0.25	7.12	-86.	165.	-18.	35.	0.	0.	-7.	-183.	351.	13.	-27.	-395.
0.30	8.55	-118.	226.	-25.	48.	0.	0.	6.	-255.	489.	-11.	41.	-527.
0.35	9.98	-141.	270.	-30.	57.	0.	0.	19.	-318.	608.	-36.	95.	-628.
0.40	11.40	-155.	297.	-33.	63.	0.	0.	32.	-362.	693.	-61.	132.	-700.
0.45	12.83	-161.	309.	-34.	66.	0.	0.	45.	-388.	743.	-86.	153.	-741.
0.50	14.25	-159.	304.	-34.	65.	0.	0.	58.	-396.	758.	-112.	159.	-752.
0.55	15.68	-147.	282.	-31.	60.	0.	0.	72.	-386.	738.	-137.	147.	-728.
0.60	17.10	-127.	244.	-27.	52.	0.	0.	85.	-358.	684.	-162.	114.	-665.
0.65	18.53	-99.	190.	-21.	41.	0.	0.	98.	-311.	595.	-188.	61.	-563.
0.70	19.95	-62.	119.	-14.	26.	0.	0.	111.	-247.	472.	-213.	-13.	-422.
0.75	21.38	-17.	32.	-4.	8.	0.	0.	124.	-164.	314.	-238.	-107.	-242.
0.80	22.80	38.	-72.	7.	-14.	0.	0.	138.	-63.	121.	-263.	-222.	-23.
0.85	24.23	100.	-192.	21.	-40.	0.	0.	222.	56.	-107.	-425.	-357.	235.
0.90	25.65	172.	-328.	36.	-68.	0.	0.	374.	146.	-280.	-715.	-512.	532.
0.95	27.08	252.	-481.	53.	-101.	0.	0.	543.	158.	-302.	-1040.	-637.	771.
1.00	28.50												

X/L	X	Initial Top Max-T	Initial Top Max-C	Initial Bot Max-T	Initial Bot Max-C	Env-1 Top Max-T	Env-1 Top Max-C	Env-1 Bot Max-T	Env-1 Bot Max-C	Env-2 Top Max-T	Env-2 Top Max-C	Env-2 Bot Max-T	Env-2 Bot Max-C
	ft	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi	psi
0.00	0.00												
0.05	1.43	-----	-359.	172.	-----	-----	-287.	102.	-122.	60.	-329.	183.	-562.
0.10	2.85	-----	-318.	94.	-----	-----	-270.	70.	-58.	-----	-303.	133.	-293.
0.15	4.28	-----	-242.	-----	-52.	-----	-220.	-----	-70.	-----	-243.	18.	-125.
0.20	5.70	-----	-175.	-----	-179.	-----	-198.	-----	-115.	-----	-267.	64.	-90.
0.25	7.12	-----	-118.	-----	-289.	-----	-187.	-----	-191.	-----	-315.	156.	-182.
0.30	8.55	-----	-70.	-----	-380.	-----	-178.	-----	-256.	-----	-357.	235.	-264.
0.35	9.98	-----	-32.	-----	-453.	-----	-172.	-----	-312.	-----	-394.	307.	-337.
0.40	11.40	-----	-4.	-----	-507.	-----	-165.	-----	-357.	-----	-419.	354.	-400.
0.45	12.83	15.	-----	-----	-543.	-----	-159.	-----	-392.	3.	-430.	376.	-453.
0.50	14.25	24.	-----	-----	-561.	-----	-152.	-----	-417.	25.	-429.	374.	-495.
0.55	15.68	21.	-----	-----	-555.	-----	-148.	-----	-427.	39.	-418.	353.	-523.
0.60	17.10	3.	-----	-----	-521.	-----	-148.	-----	-418.	44.	-399.	315.	-531.
0.65	18.53	-----	-29.	-----	-458.	-----	-153.	-----	-389.	38.	-371.	263.	-521.

0.70	19.95	-----	-77.	-----	-367.	-----	-163.	-----	-342.	22.	-335.	194.	-491.
0.75	21.38	-----	-140.	-----	-247.	-----	-177.	-----	-275.	-----	-292.	110.	-441.
0.80	22.80	-----	-217.	-----	-99.	-----	-196.	-----	-188.	-----	-240.	11.	-373.
0.85	24.23	-----	-310.	78.	-----	-----	-219.	-----	-124.	-----	-180.	-----	-421.
0.90	25.65	-----	-417.	284.	-----	-----	-261.	52.	-79.	69.	-159.	-----	-579.
0.95	27.08	-----	-481.	405.	-----	-----	-285.	99.	-123.	211.	-175.	-----	-851.
1.00	28.50												

8.3.5.8 Section 28: Required Post-tensioning (PT mode only)

The post-tensioning force required from allowable tensile stress input at 1/20th point along each span is calculated and listed in this table.

X/L	X	PT	X	PT	X	PT	X	PT
	ft	k	ft	k	ft	k	ft	k
	SPAN 1		SPAN 2		SPAN 3		SPAN 4	
0.00	0.00	---	0.00	---	0.00	---	0.00	---
0.05	1.43	65.37	1.75	186.67	1.75	198.09	1.43	203.66
0.10	2.85	0.00	3.50	67.40	3.50	86.19	2.85	108.17
0.15	4.28	0.00	5.25	0.00	5.25	0.00	4.28	0.00
0.20	5.70	0.00	7.00	0.00	7.00	0.00	5.70	0.00
0.25	7.12	119.02	8.75	112.63	8.75	117.32	7.12	0.00
0.30	8.55	201.03	10.50	202.98	10.50	211.79	8.55	154.64
0.35	9.98	245.22	12.25	239.31	12.25	245.26	9.98	219.48
0.40	11.40	267.33	14.00	251.29	14.00	254.83	11.40	250.67
0.45	12.82	276.46	15.75	276.57	15.75	278.53	12.82	267.38
0.50	14.25	275.81	17.50	275.71	17.50	275.71	14.25	275.80
0.55	15.68	267.37	19.25	278.53	19.25	276.56	15.68	276.46
0.60	17.10	250.68	21.00	254.83	21.00	251.30	17.10	267.33
0.65	18.52	219.49	22.75	245.27	22.75	239.35	18.52	245.23
0.70	19.95	154.63	24.50	211.83	24.50	203.02	19.95	201.02
0.75	21.38	0.00	26.25	117.42	26.25	112.72	21.38	119.00
0.80	22.80	0.00	28.00	0.00	28.00	0.00	22.80	0.00
0.85	24.23	0.00	29.75	0.00	29.75	0.00	24.23	0.00
0.90	25.65	108.18	31.50	86.15	31.50	67.35	25.65	0.00
0.95	27.08	203.68	33.25	198.07	33.25	186.63	27.08	65.40
1.00	28.50	---	35.00	---	35.00	---	28.50	---

8.3.5.9 Section 29: Detailed Rebar

The rebar required at 1/20th point along each span is listed in this table. “Analysis” refers to rebar required for Strength (ultimate) conditions, including the UBC special load combination when applied to the model. “Minimum” refers to reinforcement required for Service and Initial conditions. The selected reinforcement is the enveloped amount from the Analysis and Minimum conditions.

SPAN 1

X/L	X	Analysis Top	Analysis Bot	Minimum Top	Minimum Bot	Selected Top	Selected Bot
	ft	in2	in2	in2	in2	in2	in2
0.04	1.16	0.00	0.00	2.23	0.48	2.23	0.48
0.05	1.42	0.00	0.00	2.23	0.49	2.23	0.49
0.10	2.85	0.00	0.00	2.23	0.00	2.23	0.00
0.15	4.27	0.00	0.00	2.23	0.00	2.23	0.00
0.20	5.70	0.00	0.00	0.00	0.00	0.00	0.00
0.25	7.13	0.00	0.00	0.00	0.00	0.00	0.00
0.30	8.55	0.00	0.25	0.00	0.00	0.00	0.25
0.35	9.97	0.00	0.55	0.00	1.38	0.00	1.38
0.40	11.40	0.00	0.75	0.00	1.38	0.00	1.38
0.45	12.82	0.00	0.89	0.00	1.38	0.00	1.38
0.50	14.25	0.00	0.88	0.00	1.38	0.00	1.38
0.55	15.68	0.00	0.80	0.00	1.38	0.00	1.38
0.60	17.10	0.00	0.61	0.00	1.38	0.00	1.38
0.65	18.52	0.00	0.43	0.00	1.38	0.00	1.38
0.70	19.95	0.00	0.16	0.00	0.00	0.00	0.16
0.75	21.37	0.00	0.00	0.00	0.00	0.00	0.00
0.80	22.80	0.00	0.00	0.00	0.00	0.00	0.00
0.85	24.23	0.00	0.00	2.23	0.00	2.23	0.00
0.90	25.65	0.00	0.00	2.23	1.01	2.23	1.01
0.95	27.07	0.00	0.00	2.23	1.63	2.23	1.63
0.96	27.34	0.03	0.00	2.23	1.64	2.23	1.64

8.3.5.10 Section 30: Punching Shear Reinforcement

Reinforcement option: Stud

Stud diameter: 0.38

Number of rails per side: 1

Column - 2

Layer	Cond.	a	d	b1	b2	Vu	Mu	Stress	Allow.	Ratio	As	NStuds	Dist.
		in	in	in	in	k	k-ft	ksi	ksi		in2		in
1	2	17.19	34.38	35.19	46.37	-250.14	-1.15	0.082	0.215	0.38	0.00	0	0.00
2	2	17.19	34.38	295.23	82.37	-250.14	-1.15	0.025	0.145	0.17	0.00	0	0.00

Dist. = Distance between shear studs between layers

Note: Columns with --- have not been checked for punching shear.
Note: Columns with *** have exceeded the maximum allowable shear stress.

Legend (30):

Layer	: The layer of the reinforcement for each column
Cond.	: 1 = Interior 2 = End 3 = Edge 4 = Corner
a	: The distance between the layer and face of column or drop cap (d/2)
d	: Effective depth
b1	: length of section parallel to span line
b2	: length of section normal to span line
Vu	: Factored shear
Mu	: Factored moment
Stress	: Maximum stress

Allow : Allowable stress
Ratio : Ratio of calculated to allowable stress
As : Required area of reinforcement
Nstud : Number of shear studs between layers on each rail

Dist : Distance between stud layers along each rail

8.3.5.11 Section 32: Detailed Friction and Long-Term Stress Losses (PT mode only)

This report is available when the Tendon Selection method is used as the calculation method. At 1/20th point along each span, the initial stress (stress at force transfer), long-term losses and final stresses for the tendons are reported.

X/L	X	Initial Stress	Longterm Loss	Final Stress	X	Initial Stress	Longterm Loss	Final Stress
	ft	ksi	ksi	ksi	ft	ksi	ksi	ksi
	SPAN 1				SPAN 2			
0.00	0.00	64.18	8.76	55.41	0.00	48.21	8.76	39.46
0.05	1.43	63.75	8.76	54.98	1.75	46.43	8.76	37.66
0.10	2.85	62.25	8.76	53.50	3.50	45.31	8.76	36.56
0.15	4.28	61.44	8.76	52.67	5.25	44.35	8.76	35.60
0.20	5.70	60.74	8.76	51.99	7.00	43.55	8.76	34.80
0.25	7.13	60.06	8.76	51.31	8.75	42.77	8.76	34.00
0.30	8.55	59.38	8.76	50.61	10.50	41.96	8.76	33.20
0.35	9.97	58.70	8.76	49.93	12.25	41.17	8.76	32.42
0.40	11.40	58.01	8.76	49.24	14.00	40.39	8.76	31.63
0.45	12.82	57.35	8.76	48.58	15.75	39.60	8.76	30.84
0.50	14.25	56.66	8.76	47.89	17.50	38.82	8.76	30.07
0.55	15.68	55.95	8.76	47.18	19.25	38.04	8.76	29.28
0.60	17.10	55.20	8.76	46.44	21.00	37.27	8.76	28.50
0.65	18.52	54.46	8.76	45.70	22.75	36.50	8.76	27.73
0.70	19.95	53.72	8.76	44.95	24.50	35.71	8.76	26.96
0.75	21.38	52.98	8.76	44.22	26.25	34.96	8.76	26.20
0.80	22.80	52.23	8.76	43.48	28.00	34.19	8.76	25.44
0.85	24.23	51.51	8.76	42.74	29.75	33.43	8.76	24.67
0.90	25.65	50.78	8.76	42.01	31.50	32.68	8.76	23.93
0.95	27.08	49.90	8.76	41.13	33.25	31.78	8.76	23.03
1.00	28.50	48.21	8.76	39.46	35.00	30.14	8.76	21.38

8.3.5.12 Section 32: Unbalanced Moment Reinforcement

- **32.1 Unbalanced Moment Reinforcement - No Redistribution.** This table shows unbalanced moments for Strength load combinations when redistribution is not applied.

Load Combination: STRENGTH_1_Max_LL

Joint	Gamma Left	Gamma Right	Width Left	Width Right	Moment Left	Moment Right	As Top	As Bot	n Bar Top	n Bar Bot
			ft	ft	k-ft	k-ft	in2	in2		
1	0.00	0.78	0.00	27.00	0.00	22.49	0.00	0.00	0	0
2	0.59	0.59	10.00	10.00	-5.94	0.00	0.00	0.00	0	0
3	0.59	0.59	10.00	10.00	-0.94	0.00	0.00	0.00	0	0

- **32.2 Unbalanced Moment Reinforcement – Redistributed.** This table shows unbalanced moments for Strength load combinations when redistribution is applied.

Load Combination: STRENGTH_2_Max_LL_Pos_Lat

Joint	Gamma Left	Gamma Right	Width Left	Width Right	Moment Left	Moment Right	As Top	As Bot	n Bar Top	n Bar Bot
			ft	ft	k-ft	k-ft	in2	in2		
1	0.00	0.78	0.00	27.00	0.00	105.65	0.00	0.00	0	0
2	0.59	0.59	10.00	10.00	-194.82	0.00	0.00	0.00	0	0
3	0.59	0.59	10.00	10.00	-186.15	0.00	0.00	0.00	0	0

8.3.5.13 Section 34: Demand Moment and Moment Capacity

This data block lists the demand moment and the moment capacity for the 1/20th points along each span based on the user selection in the Design settings input screen. Columns 7 and 8 of this table list the ratio of the demand over capacity for both positive and negative moments. The demand moment is the worst-case moment for all Strength combinations.

- **34.1 Based on User Entered Values.** The capacity listed in this section is calculated with respect to base (user-defined) reinforcement and PT.

SPAN 1

X/L	X	Demand Moment Pos	Demand Moment Neg	Moment Capacity Pos	Moment Capacity Neg	Demand/Capacity Pos	Demand/Capacity Neg
	ft	k-ft	k-ft	k-ft	k-ft		
0.00	0.00	100.15	-215.35	171.36	-585.08	0.58	0.37
0.04	1.16	100.70	-216.14	184.77	-572.90	0.55	0.38
0.05	1.42	107.89	-186.00	188.82	-570.62	0.57	0.33
0.10	2.85	143.78	-32.06	183.12	-519.05	0.79	0.06
0.15	4.28	174.15	0.00	245.74	-464.68	0.71	0.00
0.20	5.70	243.33	0.00	296.22	-224.36	0.82	0.00
0.25	7.12	332.31	0.00	344.59	-185.40	0.96	0.00
0.30	8.55	406.78	0.00	410.58	-155.31	0.99	0.00
0.35	9.97	468.90	0.00	559.64	-135.17	0.84	0.00
0.40	11.40	512.15	0.00	585.06	-117.75	0.88	0.00
0.45	12.83	536.70	0.00	596.54	-107.29	0.90	0.00
0.50	14.25	542.52	0.00	602.65	-104.00	0.90	0.00
0.55	15.68	529.39	0.00	595.42	-108.31	0.89	0.00
0.60	17.10	497.78	0.00	578.33	-121.87	0.86	0.00
0.65	18.52	447.32	0.00	547.56	-144.49	0.82	0.00
0.70	19.95	377.99	0.00	380.25	-171.14	0.99	0.00
0.75	21.38	290.02	0.00	312.04	-211.49	0.93	0.00
0.80	22.80	183.33	0.00	252.77	-262.37	0.73	0.00
0.85	24.22	81.19	-91.48	185.56	-516.83	0.44	0.18
0.90	25.65	44.10	-247.94	216.28	-602.64	0.20	0.41
0.95	27.07	2.62	-423.21	221.17	-666.89	0.01	0.63
0.96	27.34	0.00	-457.14	220.25	-670.85	0.00	0.68
1.00	28.50	0.00	-457.93	204.08	-686.51	0.00	0.67

- **34.2 Based on Designed Values.** The capacity listed in this section is calculated with respect to base (user-defined) reinforcement, PT, and program-calculated reinforcement.

8.3.5.14 Section 35: Detailed Deflections

This data block lists sustained and total deflections at 1/20th points for Service combinations. Examples of how Sustained and Total deflection values in Columns 9 and 10 are calculated are described in detail in **Section 8.3.4.14** of this manual.

35.1 - Detailed Deflections - Service Combination 1

SPAN 1

X/L	X	SW	SW+PT	SW+PT+SDL	SW+PT+SDL+Creep	LL	XL	Sustained	Total
	ft	in	in	in	in	in	in	in	in
0.00	0.0	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
0.05	1.4	0.005	-0.002	-0.001	-0.004	0.012	0.000	0.006	0.014
0.10	2.8	0.011	-0.006	-0.004	-0.012	0.027	0.000	-0.016	0.031
0.15	4.2	0.017	-0.011	-0.007	-0.022	0.043	0.000	-0.028	0.047
0.20	5.7	0.024	-0.016	-0.011	-0.034	0.060	0.000	-0.043	0.062
0.25	7.1	0.030	-0.022	-0.016	-0.047	0.077	0.000	-0.059	0.076
0.30	8.5	0.035	-0.027	-0.019	-0.058	0.091	0.000	-0.073	0.088
0.35	9.9	0.039	-0.031	-0.023	-0.069	-0.020	0.000	-0.087	-0.101
0.40	11.4	0.042	-0.034	-0.026	-0.077	-0.023	0.000	-0.098	-0.114
0.45	12.8	0.043	-0.036	-0.027	-0.082	-0.026	0.000	-0.106	-0.124
0.50	14.2	0.043	-0.037	-0.028	-0.084	-0.029	0.000	-0.110	-0.130
0.55	15.6	0.041	-0.036	-0.027	-0.082	-0.031	0.000	-0.110	-0.131
0.60	17.1	0.037	-0.034	-0.026	-0.077	-0.032	0.000	-0.106	-0.128
0.65	18.5	0.033	-0.030	-0.023	-0.069	-0.032	0.000	-0.098	-0.120
0.70	19.9	0.027	-0.025	-0.019	-0.058	-0.031	0.000	-0.086	-0.108
0.75	21.3	0.021	-0.019	-0.015	-0.045	-0.030	0.000	-0.071	-0.092
0.80	22.8	0.015	-0.014	-0.010	-0.031	-0.027	0.000	-0.055	-0.074
0.85	24.2	0.009	-0.008	-0.006	-0.018	-0.022	0.000	-0.038	-0.054
0.90	25.6	0.004	-0.003	-0.003	-0.008	-0.017	0.000	-0.023	-0.034
0.95	27.0	0.001	-0.001	0.000	-0.001	0.011	0.000	-0.010	0.016
1.00	28.5	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000

8.3.5.15 Section 36: Detailed Deflections Live Load

This data block deflections at 1/20th points for live load skip patterns when applied to a model. The Max and Min values are reported along with Creep Max and Creep Min. deflections. The creep deflections are those used to determined sustained and total long-term deflections. The

Creep Max and Creep Min deflections are calculated as follows:

Example – Span 1 @ X/L 0.50 for SC1:

Creep Max = LL Max*0.3*(K+1)

= 0.120”*0.3*3 = 0.108”

Creep Min = LL Min*0.3*(K+1)

= -0.029”*0.3*3 = -0.026”

The Creep factor (K) is user-defined in the material assignment input and the factor of 0.3 is that defined in the Sustained Load combination.

Note that for Service Total Load combinations 3 and 4, the Creep Max and Creep Min values are reported and calculated as zero (0). For these combinations, long-term effects due to live load are not considered.

36.1 - Detailed Deflections Live Load - Service Combination 1

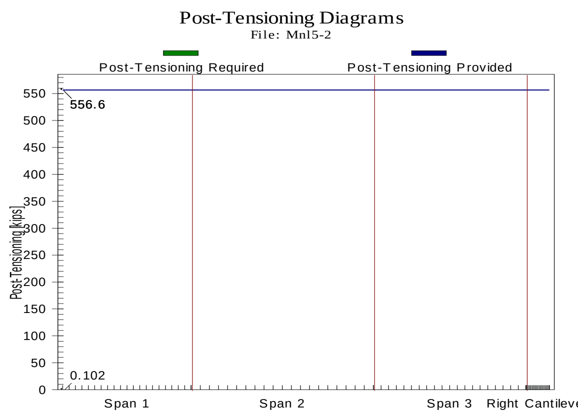
SPAN 1

X/L	X	LL1	LL2	LL3	LL4	LL5	LL6	LL Max	LL Min	Creep Max	Creep Min
	ft	in	in	in	in	in	in	in	in	in	in
0.00	0.0	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
0.05	1.4	0.010	-0.002	0.012	0.012	0.010	-0.002	0.012	-0.002	0.010	-0.002
0.10	2.8	0.022	-0.004	0.027	0.026	0.022	-0.004	0.027	-0.004	0.024	-0.004
0.15	4.2	0.036	-0.007	0.043	0.043	0.036	-0.006	0.043	-0.007	0.039	-0.006
0.20	5.7	0.050	-0.010	0.060	0.060	0.049	-0.009	0.060	-0.010	0.054	-0.009
0.25	7.1	0.063	-0.013	0.077	0.077	0.062	-0.012	0.077	-0.013	0.069	-0.012
0.30	8.5	0.074	-0.017	0.091	0.091	0.073	-0.015	0.091	-0.017	0.082	-0.015
0.35	9.9	0.083	-0.020	0.103	0.103	0.081	-0.018	0.103	-0.020	0.093	-0.018
0.40	11.4	0.089	-0.023	0.113	0.112	0.087	-0.021	0.113	-0.023	0.101	-0.021
0.45	12.8	0.092	-0.026	0.118	0.118	0.089	-0.023	0.118	-0.026	0.106	-0.024
0.50	14.2	0.091	-0.029	0.120	0.120	0.088	-0.026	0.120	-0.029	0.108	-0.026
0.55	15.6	0.087	-0.031	0.118	0.118	0.084	-0.027	0.118	-0.031	0.106	-0.028
0.60	17.1	0.080	-0.032	0.112	0.112	0.077	-0.028	0.112	-0.032	0.101	-0.029
0.65	18.5	0.070	-0.032	0.102	0.102	0.067	-0.029	0.102	-0.032	0.092	-0.029
0.70	19.9	0.059	-0.031	0.090	0.090	0.055	-0.028	0.090	-0.031	0.081	-0.028
0.75	21.3	0.046	-0.030	0.075	0.075	0.043	-0.026	0.075	-0.030	0.068	-0.027
0.80	22.8	0.032	-0.027	0.059	0.059	0.029	-0.024	0.059	-0.027	0.053	-0.024
0.85	24.2	0.019	-0.022	0.042	0.042	0.017	-0.020	0.042	-0.022	0.038	-0.020
0.90	25.6	0.009	-0.017	0.025	0.025	0.007	-0.015	0.025	-0.017	0.023	-0.015
0.95	27.0	0.002	-0.009	0.011	0.011	0.001	-0.008	0.011	-0.009	0.010	-0.008
1.00	28.5	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000

8.3.6 Graphical Report

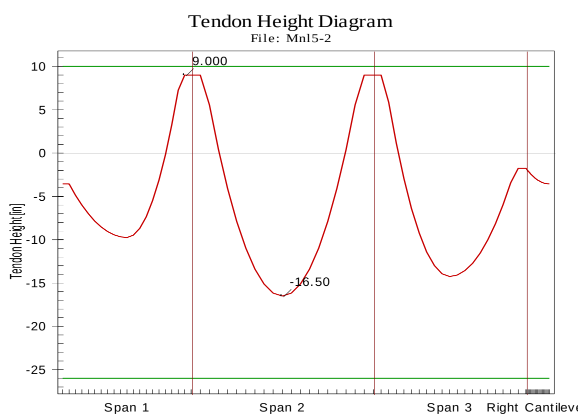
The graphical display includes print-plots of critical graphical information, such as moments, deflections, stresses, post-tensioning, and reinforcement.

8.3.6.1 PT Force



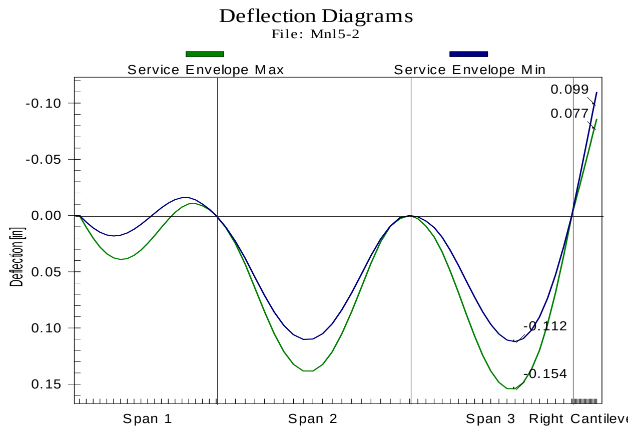
POST-TENSIONING REQUIRED AND PROVIDED

8.3.6.2 PT-Profile



POST-TENSIONING PROFILE

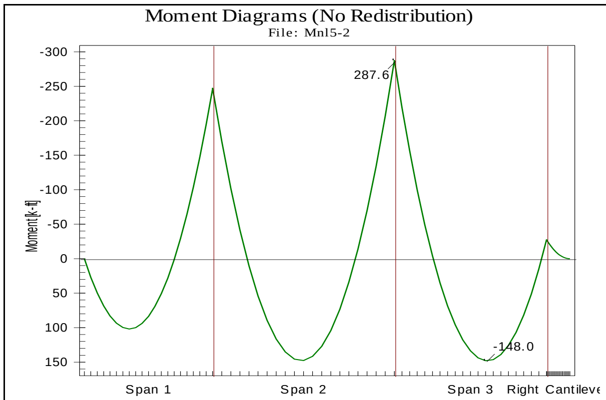
8.3.6.3 Deflection



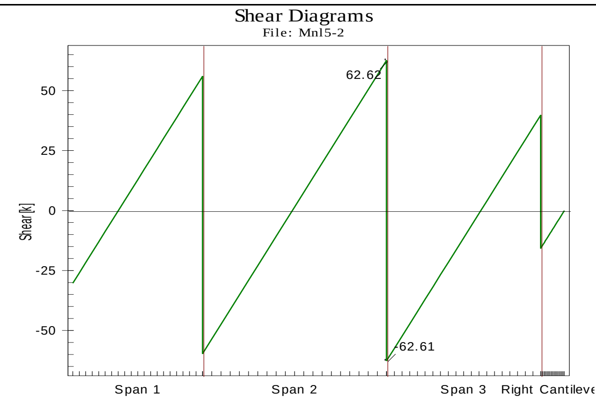
DEFLECTION

8.3.6.4 Load Cases

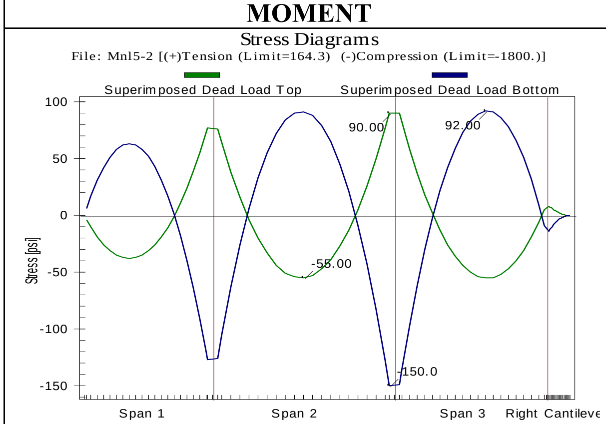
LOAD CASE: Super Imposed Dead Load



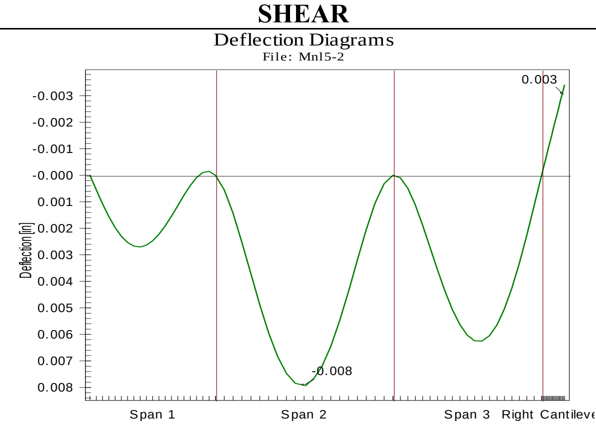
MOMENT



SHEAR



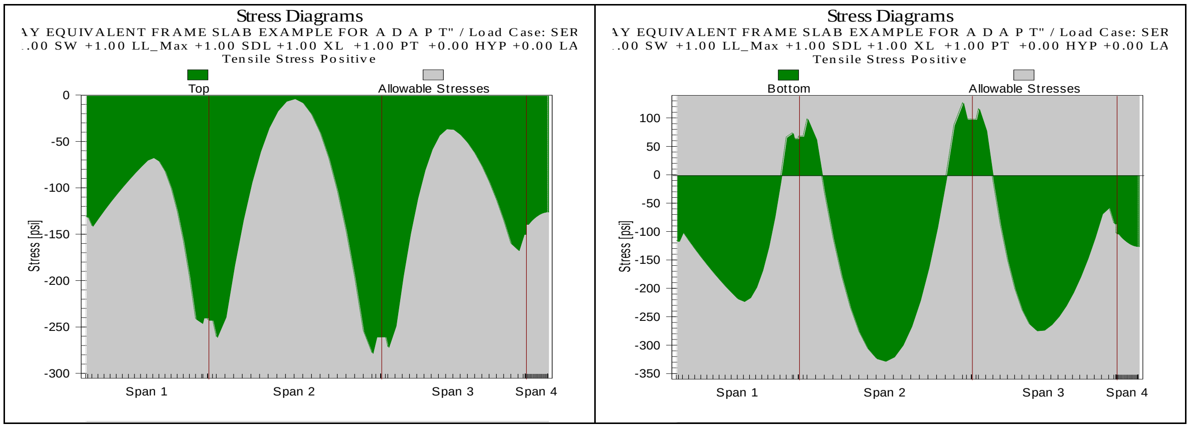
STRESS



DEFLECTION

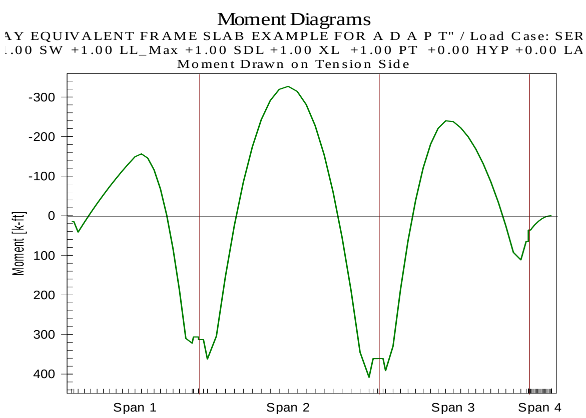
8.3.6.5 Load Combination

LOAD COMBINATION: SERVICE_1_Max_LL



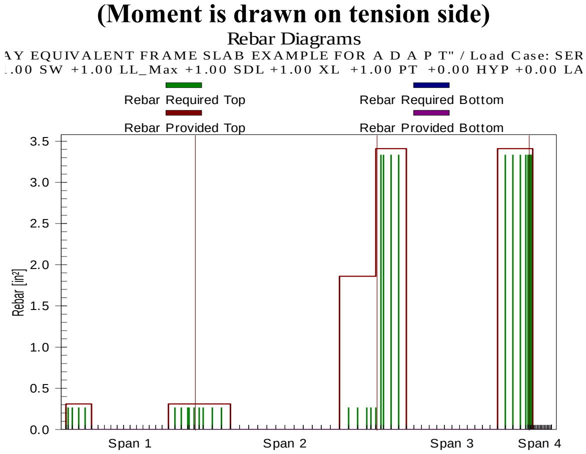
SERVICE COMBINATION STRESSES

(Tension stress positive)



DESIGN MOMENT

(Moment is drawn on tension side)



REINFORCEMENT

REQUIRED AND PROVIDED