SENECA STRUCTURAL ENGINEERING INC.

# PT Designer Ver 1.2.3

Post-Tensioning Design and Analysis Program

# Application Manual

**Seneca Software Solutions**
**23276 South Pointe Drive, Suite 209**
**Laguna Hills, CA 92653**
**949/595-8182**

# Table of Contents

## REVISION HISTORY

### Version 1.2.3

1. Updated licensing error.

### Version 1.2.1 & 1.2.2 & 1.2.3

1. Loading information was not getting saved properly when the user is trying to add different number of loads in each span.
2. Section names were not getting updated properly in Section Table for Two-Way slabs.
3. Section Manager was not updating properly upon editing the sections.
4. The Tree Menu was not updating the section names properly.

### Version 1.2.0

1. When the license is locked in other computers, the program shows the list of computer names, that are using the license. This list was not being displayed properly.
2. The license activation process has changed. Before if the license for a user was activated on a LAN Connection, and later if the program was used on a WiFi Network, the program would not run. This has been fixed.
3. Section Manager was not updating the section names properly when edited through the dialog box. Changing the names in the Section Manager Table directly was not saving the same in the respective Section Properties. Additionally, the Tree Menu was not reflecting such changes in the section names.
4. Earlier the use of special character in section names such as "24"x24" Column" led to errors on opening the files. To avoid that, the program displayed a warning to avoid using special characters in section name. This issue has been fixed and the user can use any special character.
5. The new trackbar introduced in Version 1.1.0 to directly adjust the number of output points in the result diagrams was getting reset every time the user clicked on Force and Tendon Profile.
6. Multiple display scaling issues have been fixed. The program would not scale properly if opened on different screen sizes.
7. Other display issues have been fixed. This version will be much smoother with much less flickering.

### Version 1.1.0

1. The users can now have multiple segments for 2-Way Slabs and One-Way Slabs as well.
2. Diagrams for bending moments, deflections, and stresses have been added.
3. Bug fixes regarding opening and saving a file.
4. Interface changes.

## DEFINITIONS

$A$ Cross-sectional concrete area

$A_c$ Cross-sectional concrete area of the critical punching shear section $A'_s$ Cross-sectional area of unstressed longitudinal compression steel $A_{ps}$ Cross-sectional area of prestressed steel

$A_s$ Cross-sectional area of unstressed longitudinal tension steel

$A_v$ Cross-sectional area of shear reinforcement (stirrups)

$a$ Tendon sag ( the maximum offset from the chord, the line connecting the two highpoints in each span)

$a_c$ Depth of rectangular compression stress block at nominal strength

$B, B_w$ Minimum web width of a T-beam

$B'$ Width of rectangular concrete compression stress block at nominal strength

$b_o$ Perimeter of the critical punching shear section

$C$ Total compression force acting on free body cross-section at nominal strength ($= C_C + C_S = T_p + T_S = T$)

$C_c$ Compression force acting on free body cross-section ***resisted by concrete*** at nominal strength

*CGC* Centroid of concrete cross-section

*CGR* Center of gravity of unstressed steel

*CGS* Center of gravity of prestressing steel

$C_s$ Compression force acting on free body cross-section ***resisted by unstressed compression reinforcement*** at nominal strength ( $=A'_s f_y$ )

$C_t$ Constant used in the stiffness calculation for the torsional member in the equivalent frame method

$c$ Distance from extreme compression fiber to neutral axis

$c_1$ Column dimension parallel to beam span ($c_{1L}$ at the left end of a span, $c_{1R}$ at the right end of a span)

$c_2$ Column dimension perpendicular to beam span ($c_{2L}$ at the left end of a span, $c_{2R}$ at the right end of a span)

$c_L$ Distance from support centerline to high point tendon profile "bend" at the left end of the beam

$c_R$ Distance from support centerline to high point tendon profile "bend" at the right end of the beam

$D$ Dead load moment or shear at a design point

$d$ Distance from extreme compression fiber to the centroid of the resultant total tension force ($T_p + T_s$). In shear calculations ***only*** (**THEORY** Chapter 14) $d$ need not be less than $0.8h$

$d'_s$ Distance from extreme compression fiber to centroid of unstressed compression steel $A'_s$

$d_p$ Distance from extreme compression fiber to centroid of prestressing steel $A_{ps}$

$d_s$ Distance from extreme compression fiber to centroid of unstressed tension steel $A_s$

$E_b$ Modulus of elasticity of beam concrete

$E_c$ Modulus of elasticity of column concrete

$E_{ps}$ Modulus of elasticity of prestressing steel

$E_s$ Modulus of elasticity of unstressed tension or compression steel

$e$ Eccentricity, distance between the *CGS* and the *CGC*

$e_x$ Horizontal distance from column centerline to centroid of the full critical punching shear section

$e_y$ Vertical distance from the datum line to centroid of the variable-stress sides of the critical punching shear section

$F$ Effective prestress force

*FLANGE* Width of slab assumed effective in beam section properties

$f$ Flexural concrete stress

$f'_c$ Concrete compression strength at 28 days

$f'_{ci}$ Concrete compression strength at time of stressing

$f_{dl}$ Extreme fiber flexural ***tensile*** stress caused by unfactored dead load

$f_{pc}$ Average concrete compression F/A

$f_{pe}$ Extreme fiber flexural compressive stress caused by equivalent tendon loads at the fiber ***where tension is caused by applied gravity loads***

$f_{ps}$ Stress in prestressing steel at nominal member strength (ultimate stress)

$f_{pu}$ Specified maximum tensile stress in prestressing steel

$f_r$ Modulus of rupture in concrete, the flexural tensile strength or the stress assumed to produce first cracking (normally $7.5\sqrt{f'_c}$)

$f_s$ Stress in unstressed tensile steel at nominal strength (normally = $f_y$)

$f_v$ Combined shear stress acting on the punching shear critical section due to direct shear and a portion of the unbalanced moment

$f_y$ Yield stress of unstressed steel

$H_w$ For a transverse equivalent frame beam, the dimension from the lowest slab soffit on either side of the joint to the soffit of the transverse beam

$h$ Total member depth

$I$ Moment of inertia

$I_s$ Moment of inertia of the slab portion only of a flanged beam section including the full slab tributary and excluding any portion of the beam web extending below the lowest slab soffit (used in the equivalent frame method).

$I_{sb}$ Moment of inertia of an entire flanged beam section including the full slab tributary and the entire beam web (used in the equivalent frame method).

$J_c$ "Polar" moment of inertia of the critical punching shear section about a horizontal centroidal axis perpendicular to the plane of the equivalent frame

$L$ Beam span between support centerlines

$L^+$ Most positive live load moment or shear at a design point

$L^-$ Most negative live load moment or shear at a design point

$L_{2L}$ The dimension from the centerline of the equivalent frame beam to the centerline of the adjacent equivalent frame beam to its **left**, looking towards the left (towards Joint 1)

$L_{2R}$ The dimension from the centerline of the equivalent frame beam to the centerline of the adjacent equivalent frame beam to its **right**, looking towards the left (towards Joint 1)

$L_c$ Column length from centerline of beam depth to point of fixity or pin at far end

$L_{clr}$ Beam clearspan between support faces

$M_2$ Secondary moment

$M_{bal}$ Balanced or equivalent load moment

$M_{cmax}$ Maximum moment permissible on any cross-section without compression reinforcement

$M_{cr}$ Moment ***in excess*** of the unfactored dead load moment which produces an extreme fiber tensile stress of $6\sqrt{f'_c}$ (used in beam shear calculations for $V_{ci}$)

$M_{design}$ $M_u + M_2$ (the *demand* moment)

$M_{dl}$ Unfactored dead load moment

$M_{equiv}$ Moment which equilibrates the tendon balanced, or equivalent, loads ***only*** (***not*** including the ***reactions*** to those loads, which are called the ***secondary*** reactions)

$M_f$ Portion of the total unbalanced moment $M_u$ at a joint which is transferred by direct flexure between slab and column

$M_{fr}$ Moment which produces a flexural tensile stress equal to the modulus of rupture

$f_r$ (the ***cracking moment*** referenced in ACI 318 Section 18.8.3)

$M_{ll}$ Unfactored live load moment

$M_{ll\,max}$ $M_u$ - $M_{dl}$

$M_n$ Nominal moment capacity (without ø factor)

$øM_n$ Useable moment capacity

$M_{net}$ $M_{tl} + M_{bal}$ $M_{tl}$ $M_{dl} + M_{ll}$

$M_u$ Applied moment caused by factored dead and live loads

$M_v$ Portion of the total unbalanced moment $M_u$ at a joint which must be transferred by eccentric shear stresses on the critical punching shear section

$M_{wind}$ Unfactored wind moment

$N$ Total number of segments into which each span is divided, each representing a potentially different cross-section

$P$ Number of equal spaces into which each clearspan is divided, with all design parameters (moments, shears, stresses, deflections, reinforcing, etc.) calculated at each end of each space

$R_2$ Secondary reaction

$\%R$ Maximum permissible percentage of inelastic negative moment redistribution

$S$ Number of spans in the frame, not counting cantilevers

$S_m$ Section modulus

$S_{mb}$ Section modulus at the bottom beam fiber

$S_{mt}$ Section modulus at the top beam fiber

$s$ Stirrup spacing measured along length of beam

$T$ Total tension force acting on free body cross-section at nominal strength ($= T_p + T_s = C$)

$T_p$ $A_{ps}f_{ps}$, tensile force in prestressing steel at nominal member strength (the ultimate prestress force)

*TRIB* Tributary, the perpendicular distance supported by a frame beam (*TRIBL* + *TRIBR* )

*TRIBL* Dimension from the centerline of a beam to a point midway to the adjacent beam (or support) to its **left**, looking towards the left (towards Joint 1)

*TRIBR* Dimension from the centerline of a beam to a point midway to the adjacent beam (or support) to its **right**, looking towards the left (towards Joint 1)

$T_s$ $A_sf_y$, tensile force in unstressed tension steel (normally rebar) at nominal member strength (the yield rebar tensile force)

$t$ Slab thickness

$U$ Required flexural or shear strength at a design point

$V_c$ Controlling nominal concrete shear strength (determined from $V_{cn}$, $V_{ci}$, $V_{cw}$ )

$V_{ci}$ Nominal shear strength for "inclined cracking" type of shear failure. (ACI 318 Eqn. 11-11)

$V_{cn}$ Nominal concrete shear strength (ACI 318 Eqn. 11-10). Can be used for $V_c$ in lieu of $V_{ci}$ or $V_{cw}$

$V_{cw}$ Nominal shear strength for "web cracking" type of shear failure (ACI 318 Eqn. 11-13)

$V_{dl}$ Unfactored dead load shear

$V_{ll}$ $V_u - V_{dl}$

$V_{ll}$ Unfactored live load shear

$V_n$ Nominal shear capacity $V_c + V_s$ (without ø factor)

$øV_n$ Useable shear capacity

$V_p$ Vertical component of prestress force (the shear "carried" by the tendons)

$V_s$ Nominal shear strength of shear reinforcement (stirrups)

$V_{tl}$ $V_{dl} + V_{ll}$

$V_u$ Applied factored total load shear (the demand shear)

$v_c$ Allowable combined shear stress acting on the critical punching shear section

$V_{wind}$ Unfactored wind shear

$W$ Wind moment or shear

$w_{bal}$, $w_p$ Tendon balanced, or equivalent, load

$w_{dl}$ Unfactored dead load $w_{ll}$ Unfactored live load $w_{net}$ $w_{tl}+w_{bal}$

$w_{tl}$ $w_{dl}+w_{ll}$

$w_u$ Factored dead plus live load

$x_{L,R}$ Distance from the ***centroid of the critical punching shear section*** to its left and right faces

$X_{L,R}$ Distance from the ***centroid of the lower column*** to the left and right faces of the critical punching shear section

$Y_L$ Distance from datum line to *CGS* at left end beam highpoint

$Y_m$ Distance from datum line to *CGS* at a lowpoint

$Y_R$ Distance from datum line to *CGS* at right end beam highpoint

$Y_{ref}$ Distance from datum line to top of beam

$Y_{tb}$ For a transverse equivalent frame beam, the vertical distance from the datum line to the soffit of the transverse beam

$y_t$ Distance from concrete centroid to the extreme fiber ***where tension is caused by applied gravity loads***

$\alpha_s$ A term used in determining $v_c$. $\alpha_s$ = 40 for interior columns, 30 for edge or edge parallel columns, and 20 for corner columns

$\beta_1$ Factor which varies with concrete strength $f'_c$: $\beta_1$ is 0.85 for strengths up to and including 4000 psi, then reduces continuously at a rate of 0.05 for each 1000 psi of strength in excess of 4000 psi down to a minimum of 0.65.

$\beta_c$ Ratio of long side to short side of a rectangular column ($\beta_c$ = 1 for round columns)

$\gamma_p$ A factor used in the calculation of $f_{ps}$ for bonded tendons, 0.40 for stress-relieved steel, 0.28 for low-relaxation steel

$\gamma_v$ The decimal fraction of the total unbalanced moment at any joint of a two-way system which must be transferred from slab to column by eccentric shear stresses on the critical punching shear section

ø Capacity reduction factor (0.9 for flexure, 0.75 for shear)

$\rho$ Reinforcing steel ratio = $A_s / B'd_s$

$\rho_b$ Balanced reinforcing steel ratio = $0.85 \beta_1 f'_c / f_y [87000/(87000+f_y)]$

# CHAPTER 1 INSTALLING AND STARTING PT DESIGNER

## 1-1 General Information and Terminology.

**PT Designer** is a Windows-based computer program for the design and analysis of linear post-tensioned concrete frames. This document, the APPLICATION Manual, describes how to install and use **PT Designer**. A separate document, also furnished with **PT Designer**, is called the **THEORY** manual, and it describes in technical detail what **PT Designer** actually does. The **THEORY** manual is referenced often in this document.

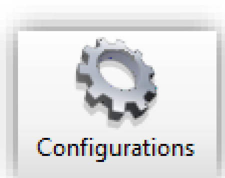
**PT Designer** is used to design and analyze prismatic frames where the cross-section of the beams or slabs can vary in any span; however the cross-section is constant between supports in any given span. The maximum number of spans in the **PT Designer** frame is 25 plus a cantilever at either or both ends. The term "span" herein is defined as a length of beam or slab supported at both ends, as opposed to a cantilever which is supported at one end only. Spans are numbered consecutively from left to right starting with 1 and ending with *S*, where *S* is the total number of spans excluding cantilevers. A left cantilever is identified as Span 0 and a right cantilever is identified as Span *S*+1. A maximum of 20 superimposed dead or live loads may be applied in any span or cantilever. The applied loads can be uniform "line" loads over all or part of a span or cantilever, point loads, concentrated moments, or applied wind moments acting at each end of each beam. In **PT Designer**, the cross-sectional geometry for beams and slabs is prismatic (constant) between supports except for two- way slabs where a "drop panel" is permitted at each column or a slab-band (a shallow wide beam) may be modeled. Each span or cantilever can contain one of a library of available cross-sectional "types". Each span or cantilever can contain one of 7 different tendon profiles (5 for cantilevers), including simple parabolas, compound parabolas, single point harps, and double point harps.

Throughout this document the term "beam" is used to address the horizontal frame members, regardless of whether they are actually beams, girders, or slabs in common engineering vernacular.

When describing keystrokes from the keyboard, the actual key is shown in carets. For example, the Escape key is indicated by <Esc>; the letter "B" by <B>. When referring to command buttons, the terms "press" and "click on" are used interchangeably.

## 1-2 The Change Configuration Menu.

**PT DESIGNER** creates a configuration file containing certain input data which, once established, rarely changes. To enter this data with each run would be time consuming and repetitious, however if it was "hard wired" permanently into the program it could never be changed by the user. To solve this, **PT Designer** puts this data in a data file which is read into the program each time it is run. When necessary, *some of* the data in this file can be changed by modifying the values using the **RIBBON MENU** > **CONFIGURATIONS**



The data items contained in the file *that can be permanently modified by the user* are as follows:

1. **Spaces** - The number of spaces *P* into which each clearspan is to be divided, design values calculated at each end of each space (see **THEORY** Section 8.1).
(Default = 12).

2. **ColModel** - The column modeling code (See **THEORY** Section 7.1(a)):
0 = Top and bottom column always present.
1 = Top column present for superimposed loads only.
(Default = 0)

3. **CrackMom** - An indicator (1 or 0) which tells **PT Designer** to consider (1) or ignore (0) the "cracking moment" calculation of Section 18.8.3, ACI 318-95 (See **THEORY** Section 9.1).
(Default = 0).

4. **$A_{ps}$** - The cross-sectional area of ***one*** post-tensioned strand. (Default = 0.153 $in^2$).

5. **EffStressLR** - The assumed effective stress in low-relaxation tendons. (Default = 174 ksi).

6. **EffStressSR** - The assumed effective stress in stress-relieved tendons. (Default = 162 ksi).

7. **MinCapSizeEnabled** – If this option is enabled (Value=1) **PT Designer** will not permit a shear cap size *(W1xW2)* to be entered which is smaller than the first (inside) critical section (Section #1). If this option is disabled any shear cap size can be entered. (Default=1). (*This function is still under development, however the users can check for Minimum Cap Size from the workspace; defined later*)

8. **ColumnStiffnessFactor** – The ratio between the effective (cracked) to gross (uncracked) moment of inertia *($I_e/I_{gross}$)* for ***all*** columns. (Default=1.0).

The data items contained in the configuration file *that **cannot** be **permanently** modified by the user* are as follows:

9. **KDL** - The load factor for dead loads except when combined with wind load ***only*** (see Item 10 below). (Default = 1.2).

10. **KLL** - The load factor for live loads (Default = 1.6).

11. **KW1** - The load factor for wind loads (*KW1*) when combined with dead ***and*** live loads.
(Default = 1.6).

12. **KW2** - The load factor for wind loads (*KW2*) when combined with dead load ***only***.
(Default = 1.6).

13. **KDW** - The load factor for dead loads (*KDW*) when combined with wind load only.
(Default = 0.9).

14. **KCOMB** - The multiplier (*KCOMB*) for combined factored dead, live, and wind loads. (Default = 0.75).

15. **KDL1** - The load factor for dead loads (*KDL1*) when combined with live and wind loads. (Default = 1.2).

16. **KLL1** - The load factor for live loads (*KLL1*) when combined with dead and wind loads. (Default = 1.6).

17. **The ACI 318-99 Check Box** – Checking this box tells the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases of the ACI 318-99 Code. Unchecking this box will tell the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases of the code after ACI 318-99. This will be the default each time the program is opened. The user may then over-ride any of the load factors for his/her individual requirements. The capacity reduction factors and redistribution cases cannot be modified and will remain consistent with either ACI 318-99 or later versions of ACI 318, depending upon the state of the check box.

**WARNING: PT Designer** reads the data file each time it is ***started***. When this file is changed **PT Designer** will continue to use the new values until **PTDATA.INI** is changed again. If you make a change in **PTDATA.INI** for an atypical run, be sure to change the file back to its original values or **PT Designer** will continue to use the atypical values. However, each time the program is started the current ACI load factors will be loaded and the ACI 318-99 box will be unchecked. The user can modify the load factors and check the ACI318-99 box *for individual runs*, and this will be saved permanently *for those runs*, but the load factors and check box in the initialization file for future runs will not be modified.

### 1-3 The Datum Line.

The vertical position of many **PT Designer** program parameters is determined by their distance from a constant horizontal line called the ***datum line***. The datum line can be anywhere and its location is determined in each run by the user. Dimensions ***below*** (down from) the datum line are positive; dimensions ***above*** (up from) the datum line are negative. The most convenient location for the datum line is at the top of the topmost beam segment in the entire frame. All dimensions from the datum line are then either zero or positive.

### 1-4 Dimensions Perpendicular to the Frame.

Many **PT Designer** dimensions are perpendicular to the plane of the frame. Examples of this type of dimension are the horizontal dimensions for the beam or slab cross- section; the equivalent frame $L_2$ dimensions at each joint, and the $c_2$ dimension of a rectangular column perpendicular to the span. In some cases, **PT Designer** supports ***different*** "perpendicular" dimensions on either side of the centerline of the frame and these dimensions are identified as dimensions on the "left" side and/or the "right" side of the frame centerline. For these perpendicular dimensions, **PT Designer** assumes that you are always looking ***towards the left*** end of the frame, i.e., towards the left end of the left cantilever if there is one, or towards Joint 1 if there is no left cantilever. For example, if you are looking at the **PT Designer** frame (as in Figure 1.1), the ***left*** $L_2$ dimension is on ***your*** side of the frame; the ***right*** $L_2$ dimension is on the ***far*** side of the frame. Similarly, for a beam cross-section which supports different *TRIBUTARIES* on either side of the frame centerline, the ***left*** *TRIB* is on ***your*** side of the frame (the near side); the ***right*** *TRIB* is on the ***far*** side of the beam section #2 as viewed in Figure 3.7.

## 1-5 Design Points.

Most **PT Designer** frame parameters are calculated at a set of ***design points*** in each span, some of which are determined by the user and some by **PT Designer**.

Parameters which are calculated at the design points include:

- Concrete section properties.
- Bending moments.
- Shears.
- Concrete flexural and average compression stresses.
- Tendon CGS, slope, and effective prestress force.
- Unstressed flexural reinforcement.
- Stirrup design.
- Minimum bonded reinforcement.

The user-determined design points are a function of the value *P* which is specified by the user and which appears as the ***SPACES*** item in the configuration file (see Section 1.2, Item #1). *P* is the number of equal spaces into which each clearspan is divided. Each end of each of these *P* spaces is a design point. There are, therefore, a total of *P*+1 design points in each span which are specified by the user. In addition to the user- specified design points **PT Designer** adds a point at a distance ***h/2*** from each support face, where *h* is the depth of the beam segment immediately adjacent to the appropriate support (Segment #1 at the left support, Segment #*N* at the right support). This adds two design points to each span and one to each cantilever. Finally, **PT Designer** adds two design points at each change in cross-section, one immediately to the left of the change, one immediately to the right. These add 2(*N*-1) design points to the set in each span or cantilever. The total number of design points is therefore *P*+3+2(*N*-1) for each span and *P*+2+2(*N*-1) for each cantilever. The design points are shown in Figure 1.2. An exception to the above occurs when a user-specified point occurs at exactly the same location as a section change. In that case the user- specified point will be omitted, as it would contain exactly the same data as one of the two section-change design points.

*N=1* for beams and one-way slabs, and *N=3* for two-way slabs where a drop cap is permitted at each column. *N* is automatically determined in **PT Designer** depending on the Member Type.

## 1-6 Sign Conventions.

The following sign conventions are followed in **PT Designer**:

- Internal Bending Moments in Beams - Positive causes tension on the bottom beam fiber, negative causes tension on the top fiber.
- Internal Bending Moments in Columns - Clockwise positive, counterclockwise negative, acting on the top of the bottom column or the bottom of the top column.
- Flexural Stresses - Tensile stresses are positive, compressive stresses are negative.
- Deflection - Down (sag) is positive, up (camber) is negative.
- Shear - An upward load to the left of a section causes positive shear at the section.

- Applied Loads - Loads acting down are positive, loads acting up are negative. Concentrated moments are clockwise positive, counterclockwise negative. Applied wind moments follow internal beam moment conventions (positive causes tension in the bottom beam fibers, negative causes tension in the top beam fibers).

## 1-7 Units.

**PT Designer** is unit-specific, which means that each parameter must be entered with specific units, and output data is presented in specific units. The units used in **PT Designer** are, in the author's opinion, those most commonly used by structural engineers for each program parameter and are clearly identified in input and output routines. Only English units are used **in PT Designer**. Loads are expressed in kips and feet, moments in kip-feet, stresses in kips per square inch and deflections in inches.

Spans, tributaries and column lengths are in feet and all cross-section dimensions are in inches except for T-beam flange widths, which are in feet.

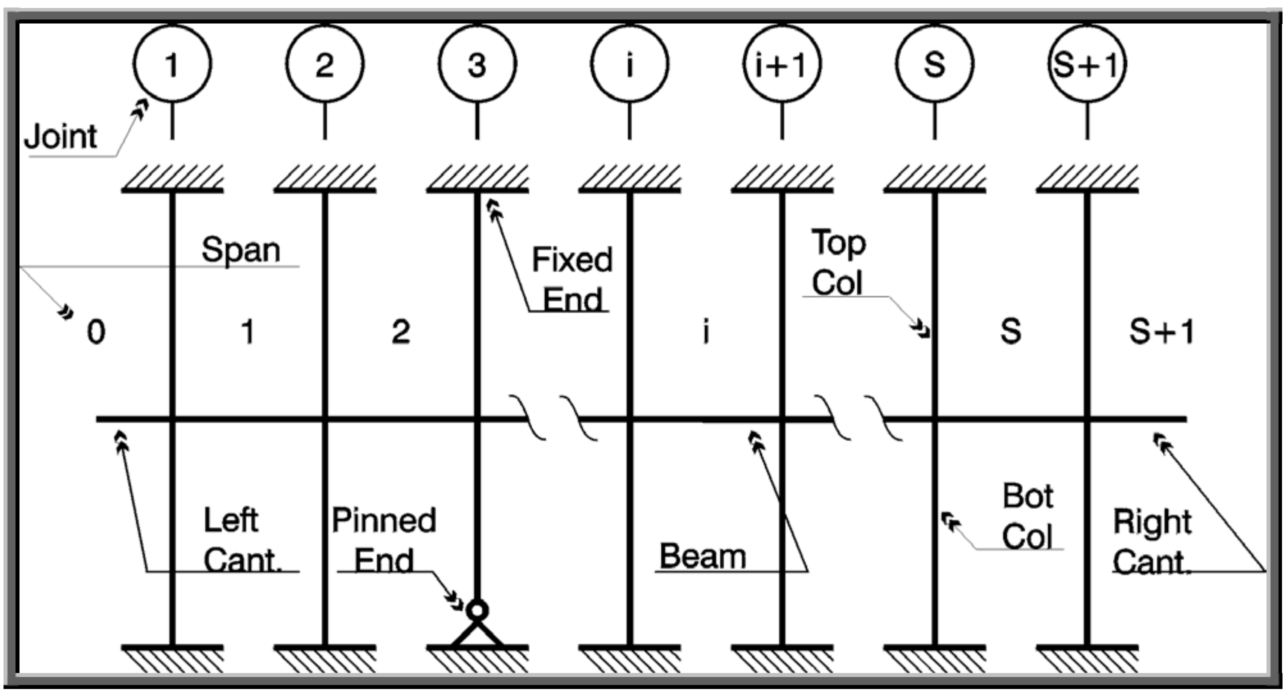



**Figure 1.1 Span & Joint Identification**

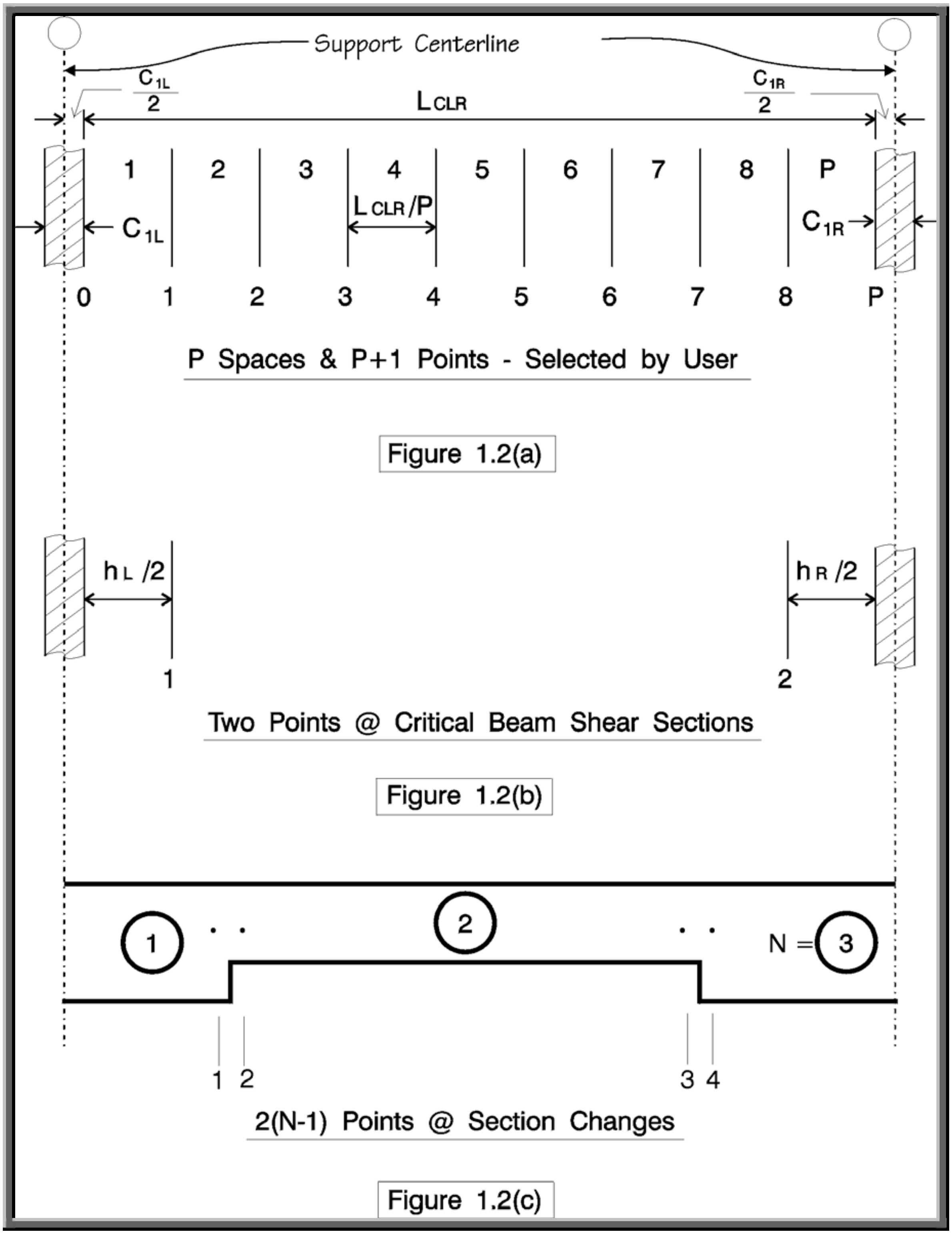



**Figure 1.2 Design Points**

# CHAPTER 2 GENERAL LAYOUT

## 2-1 PT Designer Menus

**PT Designer** operation is controlled mainly by the **RIBBON MENU / MAIN MENU** at the top of the window. This menu is further divided into 6 submenus consisting of **FILE**, **SECTION AND MATERIAL PROPERTIES**, **LOADING AND PRESTRESS**, **ANALYSIS AND RESULTS**, **TOOLS** and **HELP**.

Actions can be initiated and data can be entered and edited in **PT Designer** using either the mouse or the keyboard. With the mouse, initiate an action by clicking or double- clicking on the appropriate control. With the keyboard use the Tab key to navigate through the controls until the one you want has the focus (see Windows documentation), and then use the Enter key or the spacebar, as described in this manual, to initiate the action.

Command and option buttons can be either enabled or disabled. If enabled, the text in the button is dark and distinct. If disabled, the text is dimmed or "grayed". If a button is disabled it means the action associated with that button is inapplicable at that time.

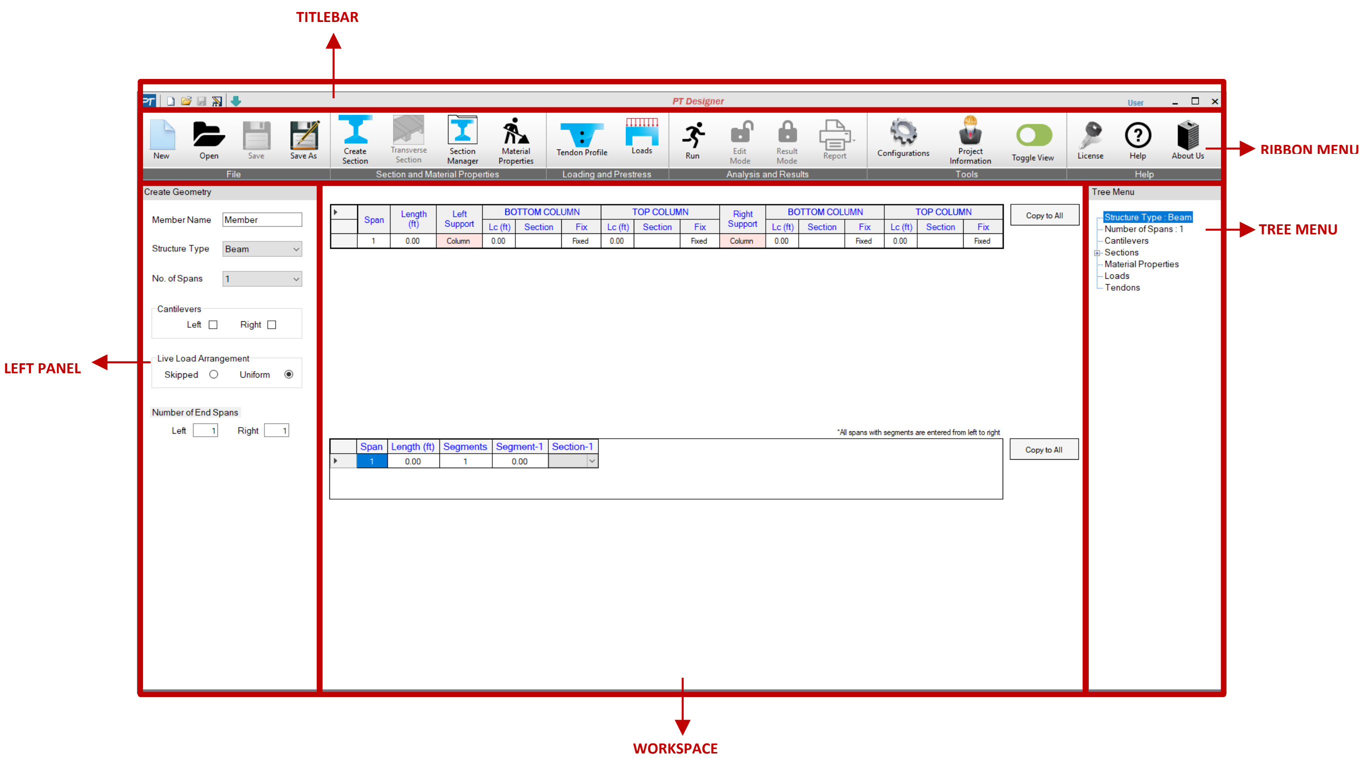



**Figure 2.1 PT Designer General Layout**

## 2-2 TITLEBAR.

The **TITLEBAR** is the topmost menu bar as shown in Figure 2.1. It consists of familiar Windows icons, command buttons and pull-down menus. The following actions can be initiated from the **TITLEBAR:**

- The NEW button will start a new project at any instant.
- The Open button will allow the user to open an existing project.

- Save button will enable the user to save the existing project. This button will be inactive if the project has not been saved even once. In such a case the user needs to go to Save As button to save the project with a new name.

- The Save As button will allow the user to save the current project with a new name. This option will be deactivated once the user saves it. Then the user might go to Save option to save the project

- The next button provides license and user information.

The buttons on the right are the standard windows minimize, maximize and exit buttons.

Apart from this the TITLEBAR will also show an indication whenever the project is saved as "***SAVED…***".

To the right side of the TITLEBAR, besides the minimize/maximize button, the name of the user is displayed. By default, it shows as "**User**". This can be changed by updating the user details by clicking on the "***Project Information***" from ***Ribbon Menu***.

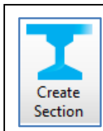
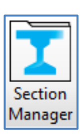
## 2-3 RIBBON MENU.

The RIBBON MENU lies beneath the TITLEBAR at the top of the window. This comprises of all the inputs and options provided by the software. It has been divided 6 submenus: File, Section and Material Properties, Loading and Prestress, Analysis and Results, Tools and Help.

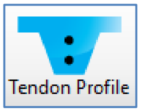
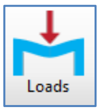
### 2-3(a) File.

This submenu has the same options as that found in the title bar. Check Section 2-2 for more information.

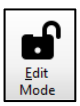
### 2-3(b) Section and material properties.

| | |
|---|---|
|   | **Create Section:** This command enables the user to add section properties for Column, Beam, Two Way Slab, and Two Way Mat Foundation. |
|   | **Transverse Section:** This command will be only active for two-way systems. It is used to input the geometry of beams at any joint perpendicular to the span of the equivalent frame beams (in the L2 direction). See **THEORY** Section 7.2(b) for a discussion of the equivalent frame transverse beams. |
|   | **Section Manager:** This command will list out all the sections added by the user. It also provides option to Add, Delete or Modify an existing section. |
|   | **Material Properties:** This command enables the user to add Concrete, Rebar and Tendon Material Properties for Beam, Slab or Two-Way systems. |

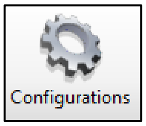
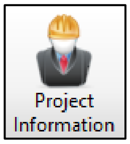
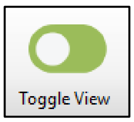
### 2-3(c) Loading and prestress.

| | |
|---|---|
|  | **Tendon Profile:** The user can enter the Tendon Profile in the beam, slab or two-way systems by clicking on this command. |
|  | **Loads:** The user can enter Uniform, Point or Line loads on the system by go to this command. |

### 2-3(d) Analysis and results.

| | |
|---|---|
|  | **Run:** With this command the user can analyze the structure. Once, analyzed the program will go into "Result Mode". The results will then be displayed on the "WorkSpace" and the "Left Panel" will provide options to navigate between different results. |
|  | **Edit Mode:** This command will be disabled by default when the program starts or when the user is working on a new file. Once the user analyzes by clicking on the "Run" button, this command will become active. The user can then go back to the Pre-Processing mode or the Input mode to change or view any of the inputs provided by the user. Once any input is changed, the program will delete all the results, thereby deactivating the result mode and the edit mode commands and the user will have to analyze the structure again, in order to show the updated results. |
|  | **Result Mode:** Once the structure has been analyzed, this option will become active for the users. When in this mode, the user cannot change any of the inputs. In order to change the inputs, the user need to go to the Pre-Processing mode or the Input mode by clicking on the command "Edit Mode". |
|  | **Report:** This command will generate an entire pdf report comprising of all the results that the program shows. Provides options to generate a Simple or Detailed Output. |

## 2-3(e) Tools.

| | |
|---|---|
|   | **Configurations:** This command will enable user to change some configurations which will affect the design and load combinations and save them as default, so that the same configurations are loaded every time the program starts. |
|   | **Project Information:** This command will allow the user to enter some general information about the user, like the Project Name, Company Name etc. The information presented in this dialog box will be shown at the top of the report generated. |
|   | **Toggle View:** This is a switch button, which will show the graphical representation of the inputs provided by the user on the workspace itself. By switch, it means that the user can switch between the tabular input and the graphical representation by clicking on this button itself. |

## 2-3(f) Help.

| | |
|---|---|
|   | **License:** This command provide the user with his/her license information for the current edition of PT Designer. It will also provide information of the company or individual who owns the license for the program. |
|   | **Help:** Through this command the user can access the User Manual and the Analysis Manual that comes along with the software. |
|   | **About Us:** This provides the information about the developers and creators of the program. |

## 2-4 TREE MENU

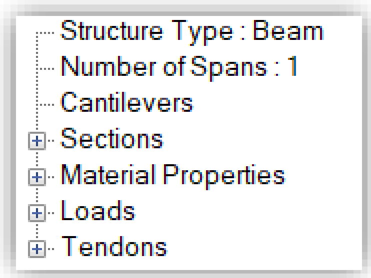



**Figure 2.2 Tree Menu layout**

This menu presents an intuitive layout of all the inputs entered by the user. It branches out steps for Structure Type, Number of spans, Cantilevers, Sections, Material Properties, Loads, Tendons and Transverse Section Properties if they have been entered. Once the user completes the input, information related to that input will be added in the tree menu. Some of the labels in the Tree Menu have a '+' sign besides them. As shown in Figure 2.3, the user can click on the '+' sign to maximize that particular level which lists out all the details of the inputs provided by the user.

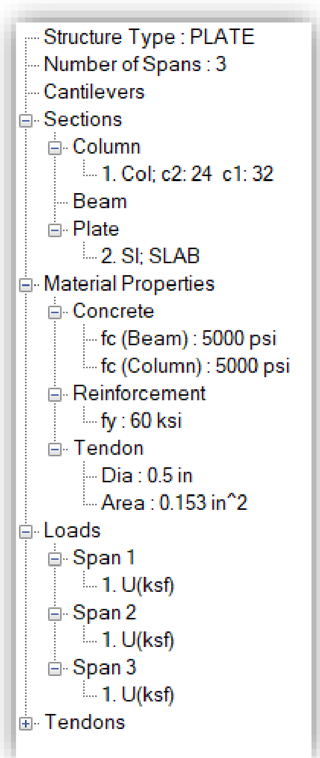



**Figure 2.3 Tree Menu Maximized**

*Structure Type*: This will show the type of the structure entered by the user which can be BEAM, SLAB (1-Way Slab), PLATE (2-Way Slab) or MAT (2-Way Mat).

*Number of Spans*: This will show the number of spans entered by the user.

*Cantilevers*: The if the user has entered any of the cantilevers like left and right cantilever

*Sections*: This shows a list of sections that user has added in the project. The list is bifurcated into 3 types of sections: Column, Beam and Plate. Plate will carry the information for 2- Way Slab and Mat Sections.

*Material Properties*: This list provides a short summary of the material properties being used in the project. The list is bifurcated into 3 sections: Concrete, Reinforcement, and Tendon, providing the information for the respective material property.

*Loads*: This provides the list of loads applied on each span defined by the user.

*Tendons*: This provides the Type of Tendons provided for each span in the project.

*Transverse Section*: This section will be only visible on the Tree Menu when the user adds a Transverse Section in the project.

## 2-5 Workspace

The workspace is the main interface of the program. When in the pre-processing or modeling mode, the workspace will present the users with Tables, where the dimensions and other structural data can be provided. Once the analysis is run, the workspace will show the results.

## 2-6 Left Panel

The left panel, in the pre-processing or modeling presents a “Create Geometry” form, where the user can enter geometry information specific to spans in the structure. Once the analysis is run, this panel lists out all the result commands, and the user can switch between the results by click on them.

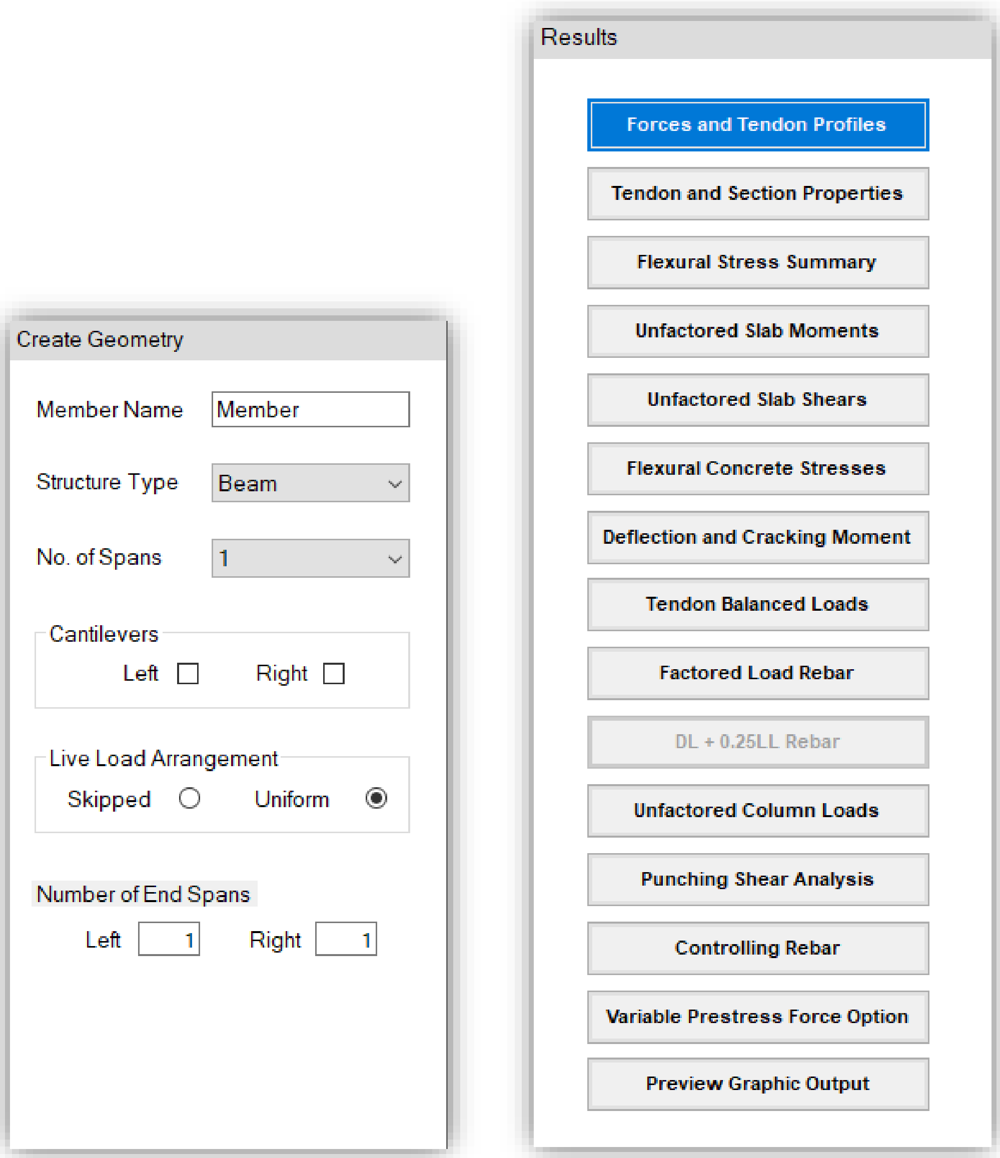



**Figure 2.4 Left Panel (Edit Mode and Result Mode)**

# CHAPTER 3 INPUT DATA FOR THE STRUCTURE

## 3-1 Create Geometry

When you start the program, the program interface is in Pre-processing or Edit Mode. In this mode the *Left Panel* shows the Create Geometry form and the *Workspace* will show tables to input data for your project.

In the Create Geometry form:

1. **Member Name** Enter the name of the specific member being designed into the text box. It will be printed on each page of printed output.

2. **Structure Type** Select the type of Structure: Beam, 1-Way Slab, 2-Way Slab and 2-Way Mat.

3. **Number of Spans** Using the pull-down combo box enter the number of spans *S* having a support at each end, i.e., exclusive of cantilevers. **PT Designer** accepts a maximum of 25 spans. **PT Designer** numbers spans consecutively from left to right, Span 1 on the left, Span *S* on the right.

4. **Cantilevers** Click on the "Left" check box if there is a cantilever at the left end of the frame, adjacent and immediately to the left of Span 1. Click on the "Right" check box if there is a cantilever at the right end of the frame, adjacent and immediately to the right of Span S.

5. **Live Load Arrangement** Click on the "Uniform" option button if the live load is applied uniformly in all spans under all loading conditions. Click on the "Skipped" option button if the live load is to be "skipped", arranged in a pattern which will produce the maximum possible positive and negative moments at each design point. See **THEORY** Section 8.3 for how **PT Designer** determines maximum positive and negative live load moments in each span.

6. **Number of End Spans** An endspan is a span which can contain an added tendon. An added tendon is a tendon which is not present in all spans of the frame, i.e., it "dead-ends" at some interior point, as opposed to "continuous" or "through" tendons which are present in *all* spans. A left endspan is an endspan whose added tendon is stressed at the left end of the frame. The use of added tendons permits the progressive decrease, or dropping off, of prestress force from span to span as it is no longer needed, starting at each end of the frame. This input item tells **PT Designer** how many spans at the ***left*** end of the frame may have added tendons which are stressed at the ***left*** end of the frame. This restriction on endspans applies to the *automatic design procedure only*. *Any* prestress force may be applied in *any* span of the frame with the manual force Chapter 3 **PT Designer** – Application Manual 3-3 selection procedure contained in the **RESULT MENU**, Chapter 4 (Also see **THEORY** Section 9.2). Enter the number of endspans at each end of the frame in the appropriate text box (Left or Right).

## 3-2 Workspace Tables

In the Pre-processing mode, the workspace will present 2 tables (one in case of 1-Way slab). The first table will be used to enter the span and the column information. The second table will assign the sections to different segments in the span. The layout and presentation of table changes as the *Structure Type* is changed.

*Efficient User Tip*: The values in both these tables can be entered swiftly by clicking the **Enter Key**. On pressing **Enter Key**, the next cell will become active to accept values from user. Also, in case of multiple spans having similar values, the user can use the **Copy to All** button to copy the values from one span to all the spans.

### 3-2(a) Geometry Table

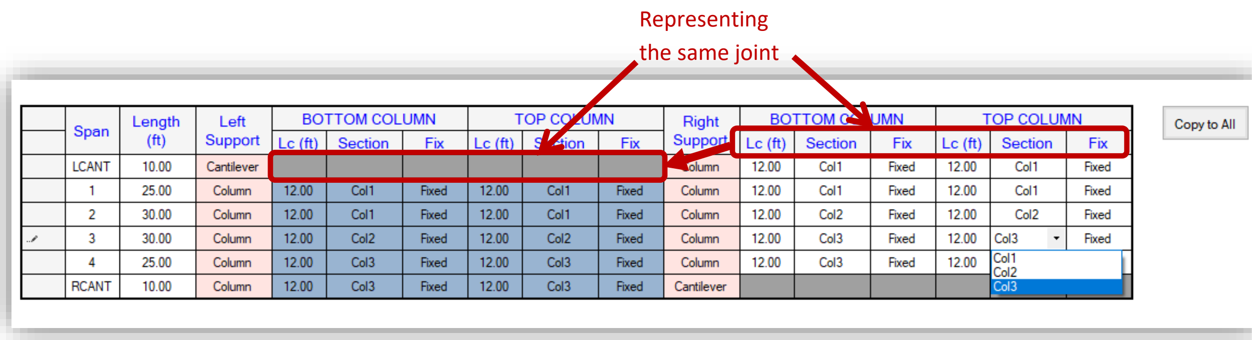
**BEAM, 2-WAYSLAB AND MAT**

1. **Span** This column is automatically entered based on the number of spans and cantilevers entered by the user in the Create Geometry panel. This column cannot be overwritten by the user.
2. **Length(ft)** Enter the span lengths in this column. The span lengths entered in this column will automatically be entered in the *Length(ft)* column for Table 2 as well; this will work the other way round too.
3. **Left Support** This column shows the support condition at "Left Support" of a span. If there is a cantilever at the left end, then it will show cantilever, else it will show Column, which signifies that the left support carries a column. This column cannot be edited by the user.

The next 6 columns are dedicated to enter the "Column Dimensions" at the left support. If the "Left Support" is a "Cantilever", then the following 6 cells will be greyed out, which means there is no column at that support. For both Bottom and Top Column:\

4. **Lc(ft)** Height of Column in ft.
5. **Section** This column provides a dropbox listing out all the column sections added by the user. The user can choose the desired section for the column from the dropdown list.
6. **Fix** Enter the support condition as: Fixed or Pinned.
7. **Right Support** Shows the type of support at the right end of the span.

The next 6 columns are dedicated to column dimensions at the right support. Again, if we have a cantilever at the right end, then then cells will be greyed out. Also, the support dimensions that you enter in these 6 cells will be automatically copied to 6 cells for the *Left Support* of the next span (the ones shaded in blue) and the user cannot edit the values in these cells. This is because the right support of the current span and the left support of the next span are representing the same joint.




| | Span | Length (ft) | Left Support | BOTTOM COLUMN | | | TOP COLUMN | | | Right Support | BOTTOM COLUMN | | | TOP COLUMN | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Lc (ft) | Section | Fix | Lc (ft) | Section | Fix | | Lc (ft) | Section | Fix | Lc (ft) | Section | Fix |
| | LCANT | 10.00 | Cantilever | | | | | | | Column | 12.00 | Col1 | Fixed | 12.00 | Col1 | Fixed |
| | 1 | 25.00 | Column | 12.00 | Col1 | Fixed | 12.00 | Col1 | Fixed | Column | 12.00 | Col1 | Fixed | 12.00 | Col1 | Fixed |
| | 2 | 30.00 | Column | 12.00 | Col1 | Fixed | 12.00 | Col1 | Fixed | Column | 12.00 | Col2 | Fixed | 12.00 | Col2 | Fixed |
| | 3 | 30.00 | Column | 12.00 | Col2 | Fixed | 12.00 | Col2 | Fixed | Column | 12.00 | Col3 | Fixed | 12.00 | Col3 | Fixed |
| | 4 | 25.00 | Column | 12.00 | Col3 | Fixed | 12.00 | Col3 | Fixed | Column | 12.00 | Col3 | Fixed | 12.00 | Col1 Col2 Col3 | |
| | RCANT | 10.00 | Column | 12.00 | Col3 | Fixed | 12.00 | Col3 | Fixed | Cantilever | | | | | | |

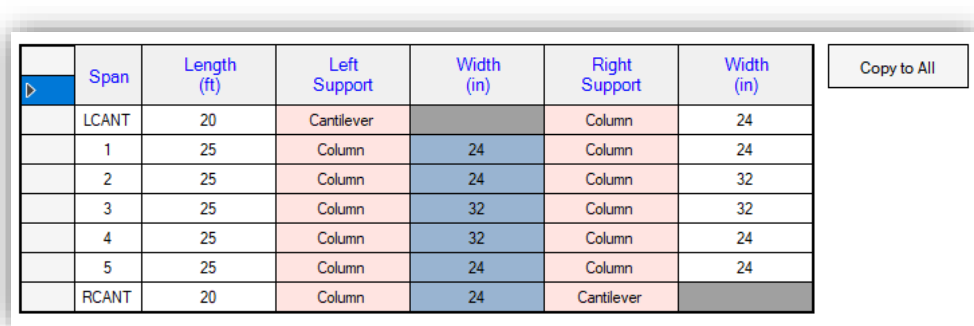
**Figure 3.1 Geometry Table layout**

8. **Copy to All Button** This button can be used to copy values in selected cells from the current span to all the spans.

## 1-WAY SLAB

Span, Length(ft) and Left Support are the same as that in case of BEAM. Please refer to the section above.

1. **Left Support Width / Right Support Width (in)** Enter the width of beam in inches at the left support or the right support over which the slab is supported. In case the span is a cantilever then the subsequent 'Width' section will be greyed out.
2. **Yref** This will adjust the datum for the section to the top of the beam. By default it is set to 0.
3. **Copy to All** Works in the same way as mentioned in BEAM section.




| | Span | Length (ft) | Left Support | Width (in) | Right Support | Width (in) |
|---|---|---|---|---|---|---|
| | LCANT | 20 | Cantilever | | Column | 24 |
| | 1 | 25 | Column | 24 | Column | 24 |
| | 2 | 25 | Column | 24 | Column | 32 |
| | 3 | 25 | Column | 32 | Column | 32 |
| | 4 | 25 | Column | 32 | Column | 24 |
| | 5 | 25 | Column | 24 | Column | 24 |
| | RCANT | 20 | Column | 24 | Cantilever | |

**Figure 3.2 1-Way Slab layout**

4. **Suggest Slab t** This button lies besides the Section Input for One-Way Slabs. Click on this will open up a form, which can be used to predict a suitable thickness of slab based on the span length and loadings on the span.

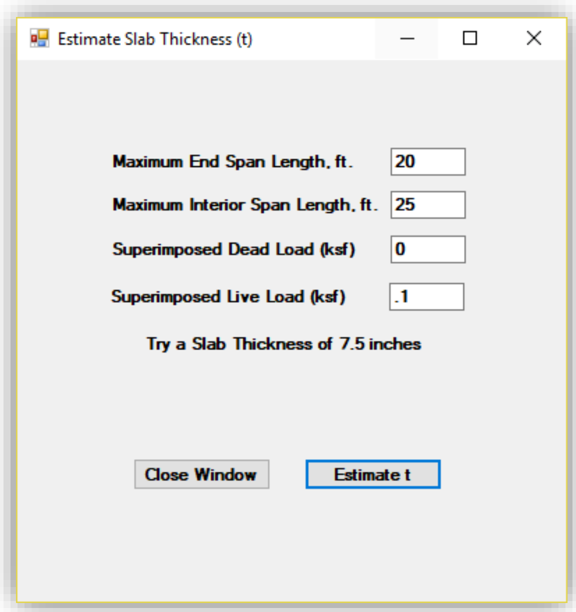



**Figure 3.3 Suggest Slab t**

## 3-2(b) Section assignment table

### BEAM

1. **Span** - This column is automatically entered based on the number of spans and cantilevers entered by the user in the Create Geometry panel. This column cannot be overwritten by the user.

2. **Length (ft)** - Enter the span lengths in this column. The span lengths entered in this column will automatically be entered in the *Length(ft)* column for Geometry Table as well; this will work the other way round too.

3. **Segments** – Enter the number of segments for each span, if the number of spans is greater than 1 then program will automatically add new cells in the span in order to enter the length of segment and its corresponding section.

4. **Segment-1** – This column is automatically set to the length of span specified in the *Length(ft)* column. The column cannot be edited by the user. However, when we have more than 1 segment, the cell value for Segment-1 will change according to the segment lengths provided for the other segments, such that the sum of all the segment length is equal to the length of the span. So, the user just has to enter the lengths of

Segment-2 and the following segments. The length of Segment-1 will be automatically calculated under Segment-1 column.

5. **Section-1** – Users can choose the section from the list of added sections for the corresponding segment.

| Span | Length (ft) | Segments | Segment-1 | Section-1 | Segment-2 | Section-2 | Segment-3 | Section-3 |
|---|---|---|---|---|---|---|---|---|
| LCANT | 20 | 1 | 20.00 | Section1 | | | | |
| 1 | 25 | 2 | 15.00 | Section1 | 10.00 | Section2 | | |
| 2 | 30 | 3 | 10.00 | Section2 | 10.00 | Section3 | 10.00 | Section2 |
| 3 | 30 | 3 | 10.00 | Section2 | 5.00 | Section3 | 15.00 | Section2 |
| 4 | 25 | 2 | 15.00 | Section2 | 10.00 | Section1 | | |
| RCANT | 20 | 1 | 20.00 | Section1 | | | | |

**Figure 3.4 Section Assignment Table**

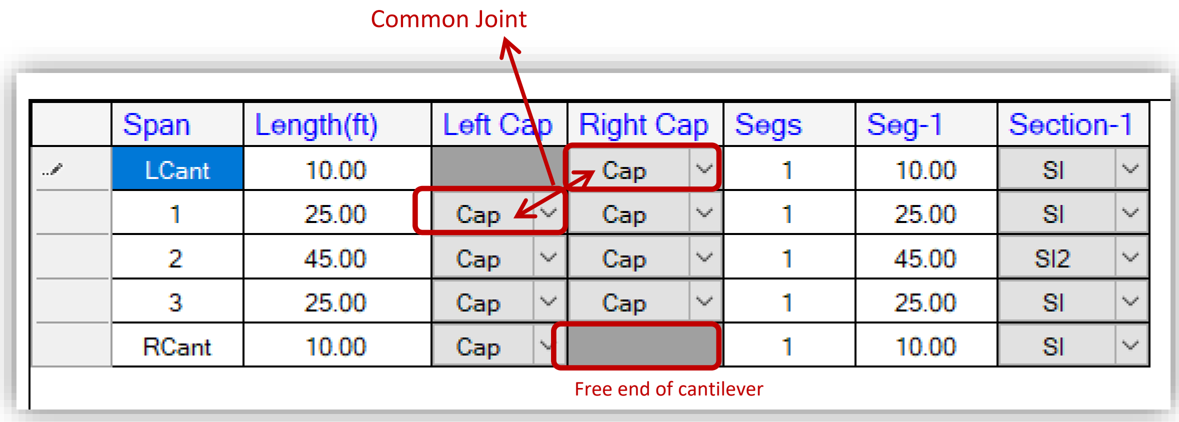
## 1-WAY SLAB

For 1-Way Slab, the section table is the same as that of a beam. Similar to beams, the user can enter multiple segments with different cross-sections in one span. The multiple segments can be used to model openings in a slab for lifts and other things.

## 2-WAY SLAB

Every span will have two end sections dedicated to caps. We can enter the cap sections for both the ends. The sections for common joints will be automatically entered once one of them is entered. Examples of such common joints are right end for one span and the left end for the following span. In case the span is a cantilever, the Left Cap section cell will be disabled for the free end. Also PT Designer has the capability to enter more than one segment in a span to accommodate slabs with different sections along the span. Common examples include slabs with openings in between. The reduced cross-section area accommodating the opening in the slab can be provided by the user.

For the sections either a slab section or a beam section can be provided. Please note for the spans containing beam sections, the caps cannot be provided. The tributary width for the caps and the slab thickness on the caps will be automatically taken from the adjacent slab section.




| | Span | Length(ft) | Left Cap | Right Cap | Segs | Seg-1 | Section-1 |
|---|---|---|---|---|---|---|---|
| | LCant | 10.00 | | Cap | 1 | 10.00 | Sl |
| | 1 | 25.00 | Cap | Cap | 1 | 25.00 | Sl |
| | 2 | 45.00 | Cap | Cap | 1 | 45.00 | Sl2 |
| | 3 | 25.00 | Cap | Cap | 1 | 25.00 | Sl |
| | RCant | 10.00 | Cap | | 1 | 10.00 | Sl |

**Figure 3.5 Section Assignment for 2-Way Slab/Mat**

**Copy to All** to copy all the slab features from one span to all the spans

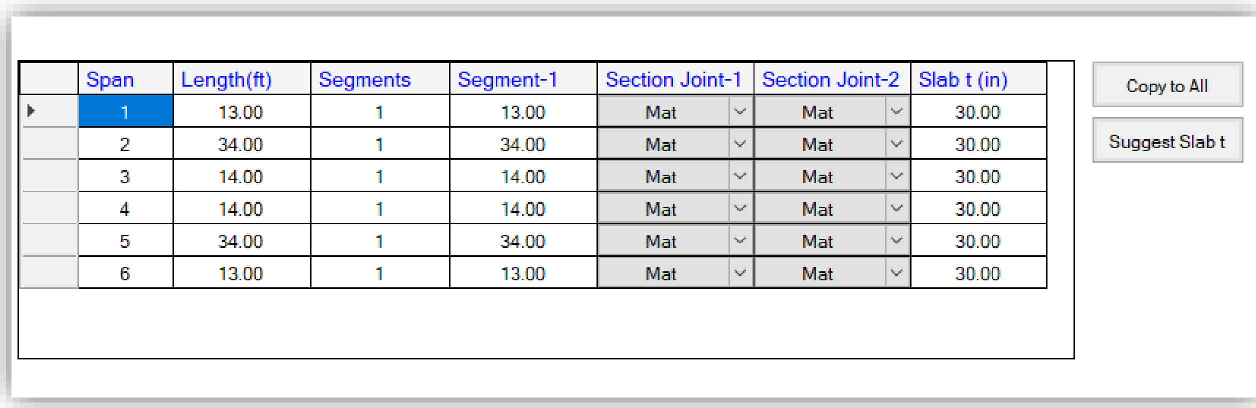
**Suggest Slab t** will provide an estimate of the slab thickness needed to sustain a given loading. May also suggest if caps are required or not.

**Check Min Cap** will check the minimum Cap Size for all the spans and change the cap sizes automatically to satisfy the minimum criteria.

*Efficient user tip*: The values in both these tables can be entered swiftly by clicking the **Enter Key**. On pressing **Enter Key**, the next cell will become active to accept values from user. Also, in case of multiple spans having similar values, the user can use the **Copy to All** button to copy the values from one span to all the spans.

### MAT

This is a table similar to that of Beams and 2-Way Slab, except for now PT Designer only supports 1 segment per span. User can provide caps at the end of the sections by defining sections in Section Joint-1 and Section Joint-2. The slab thickness can be directly entered in the last column. Again the Cap section at the right joint of one span will be copied automatically to the left joint of the subsequent span, because they represent the same joint.



| | Span | Length(ft) | Segments | Segment-1 | Section Joint-1 | Section Joint-2 | Slab t (in) |
|---|---|---|---|---|---|---|---|
| ▸ | 1 | 13.00 | 1 | 13.00 | Mat | Mat | 30.00 |
| | 2 | 34.00 | 1 | 34.00 | Mat | Mat | 30.00 |
| | 3 | 14.00 | 1 | 14.00 | Mat | Mat | 30.00 |
| | 4 | 14.00 | 1 | 14.00 | Mat | Mat | 30.00 |
| | 5 | 34.00 | 1 | 34.00 | Mat | Mat | 30.00 |
| | 6 | 13.00 | 1 | 13.00 | Mat | Mat | 30.00 |



**Figure 3.6 Section Assignment for Mat Foundation**

## 3-3 Creating Sections

To add sections, the user can go to the "Create Section" button in the Ribbon Menu. This opens up a windows form to add Section Properties. This form is divided into 5 tabs: Beam, Column, Slab, Caps, and Mat.

### 3-3(a) Beam Section

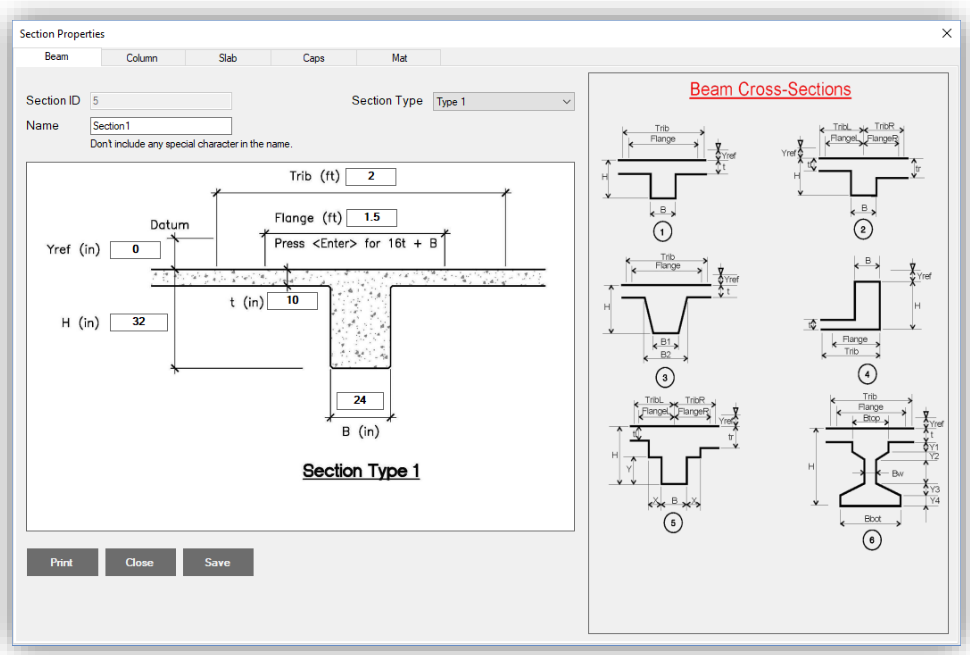



**Figure 3.7 Beam Section Property**

1. **Section ID** – The section id will be automatically added to the section number that we are creating now.

2. **Name** – Enter the name for the section. Same name cannot be used twice. The program will pop a warning box.

1. **Section Type** – Select the section type from the 8 beam cross-sections. Depending upon the choice the input window will change and allow the user to enter the dimensions. A description about all the section types is given on the right hand side of the window.

2. **Print** – This will print the current section into a pdf.

3. **Close** – This will close the window.

4. **Save** – This will add the section, but the window does not close giving the user the ability to add further sections.

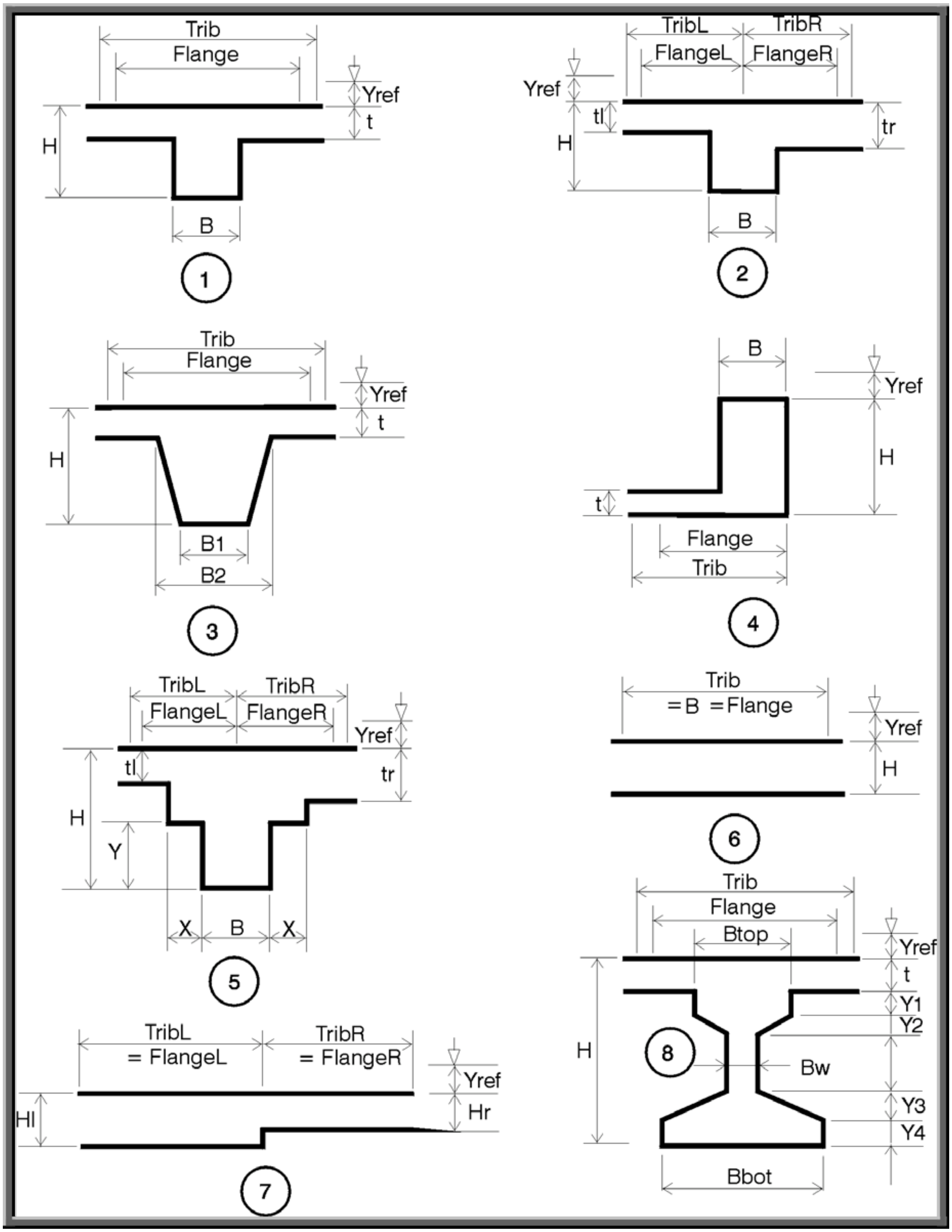



**Figure 3.8 Beam and Slab Section Types**

## 3-3(b) Column Section

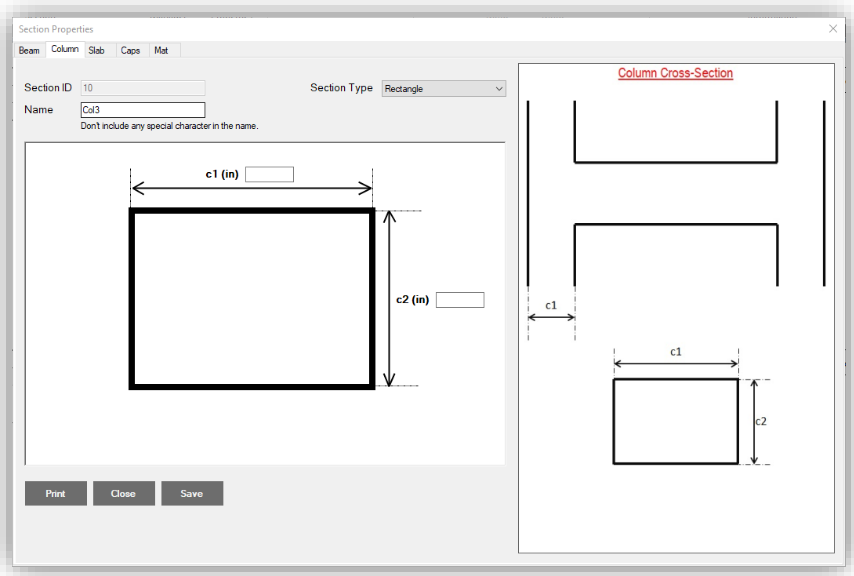



**Figure 3.9 Column Section Property**

Enter the c1 and c2 values for the column section. The description of c1 and c2 is provided in the box on the right side of the window. For "*Round Columns*" choose the section type as Circular and then provide the diameter.

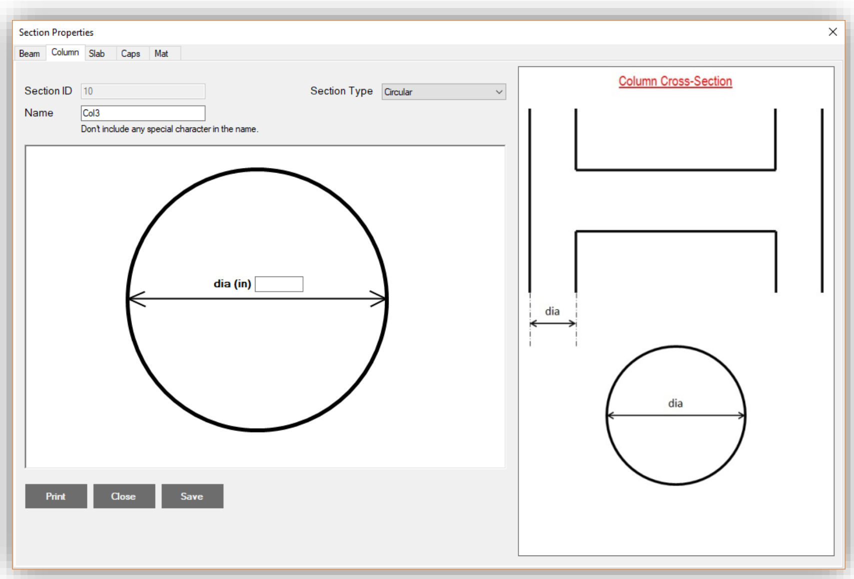



**Figure 3.10 Round Columns**

### 3-3(c) Slab

Most of the entries are same as that for Beam except for two section types (Type 6 and Type 7) which are specifically slab sections are added to the list of sections.

### 3-3(d) Caps

Provide cap length along the span W1 dimension, cap width perpendicular to the span W2 dimension and cap depth for the 2-Way Slab. Please note for Mat foundation, the cap information should be entered from the Mat Section Input.

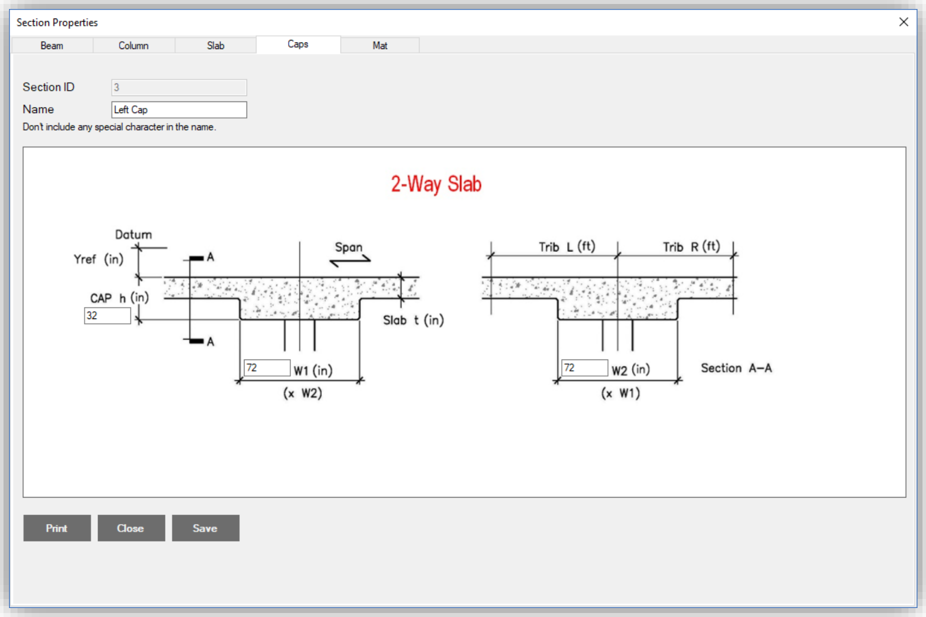



**Figure 3.11 2-Way Slab Cap Section Property**

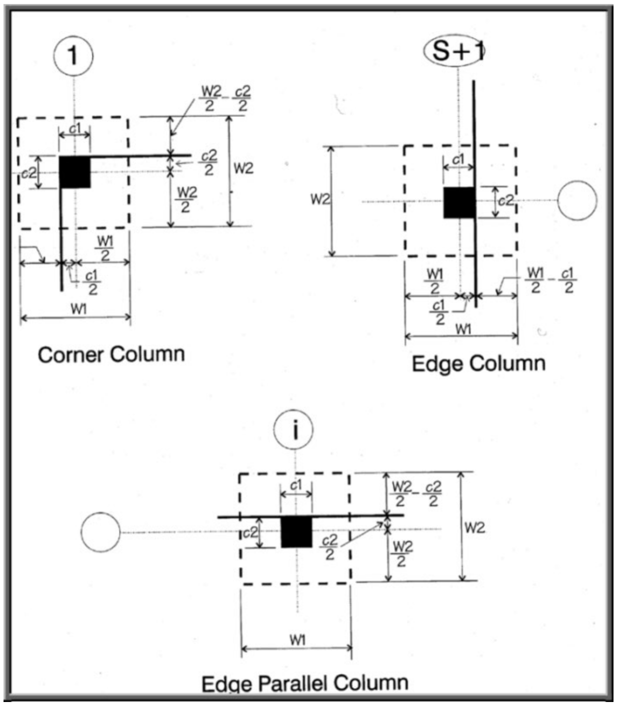



**Figure 3.12 Edge, Corner, & Edge Parallel Drop Caps**

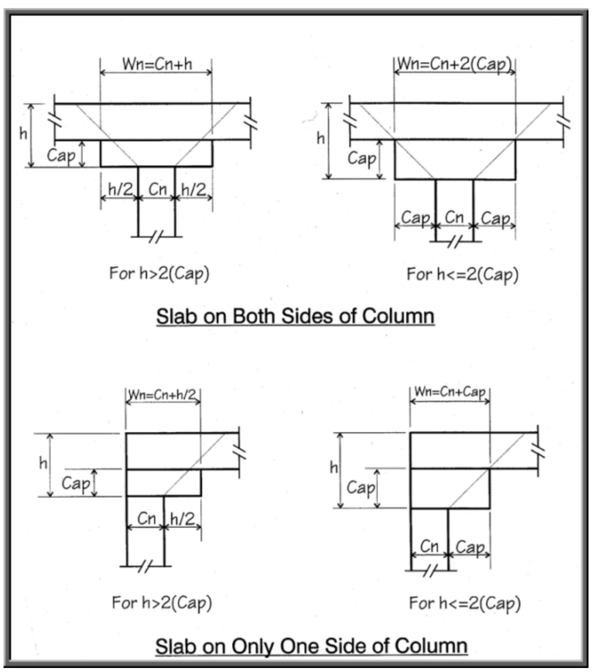



**Figure 3.13 Minimum Cap Plan Dimensions**

### 3-3(e) Mat

In the mat section tab, the cap sections for a mat foundation can be provided. The Tributary widths should also be provided in this case. The slab thickness is provided directly in the section assignment table for Mat foundations (Refer Section 3-2b).

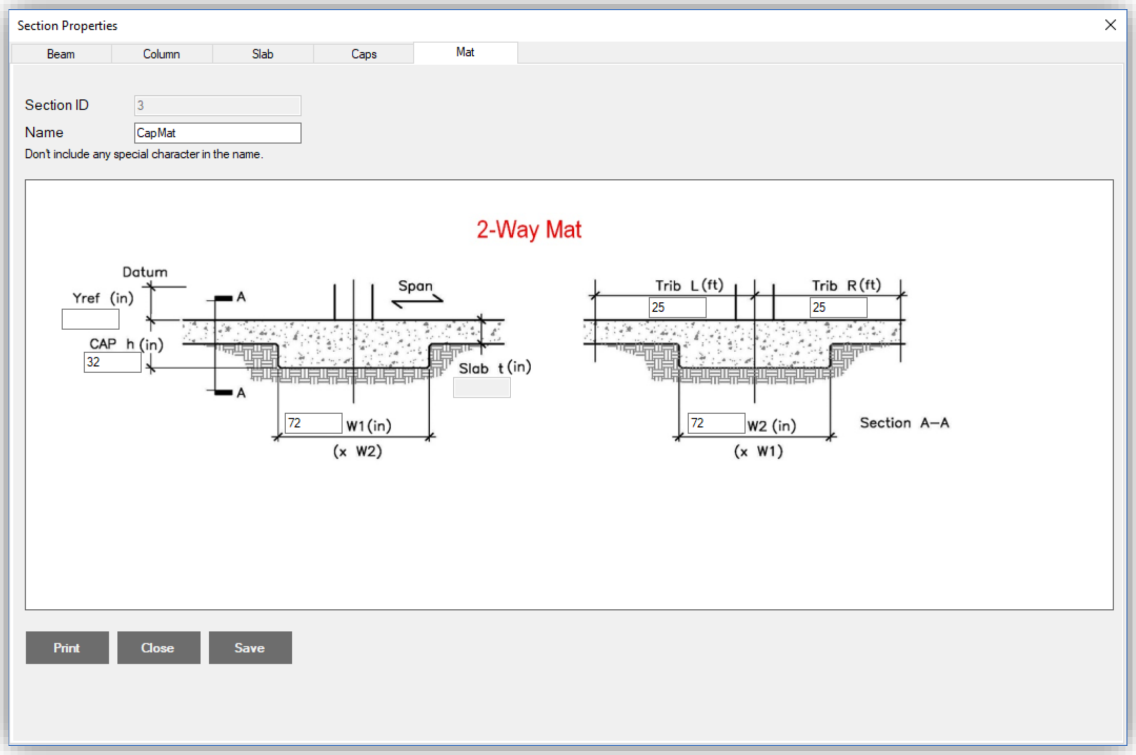



**Figure 3.14 Mat Cap Section Property**

## 3-4 Transverse Section Property

The user can enter transverse section property by clicking on transverse section on Ribbon Menu. This option will appear ***only*** for two-way systems. It is used to input the geometry of beams at any joint ***perpendicular*** to the span of the equivalent frame beams (in the $L_2$ direction). See **THEORY** Section 7.2(b) for a discussion of the equivalent frame transverse beams.

A rectangular transverse beam can occur to the left or to the right of each joint. Data is thus required for *2S+2* transverse beams. The beam geometry is defined by entering two values for each beam:

- $B_w$ the width of the transverse beam web in inches.
- $Y_{tb}$ the vertical distance in inches from the datum line to the soffit of the transverse beam.

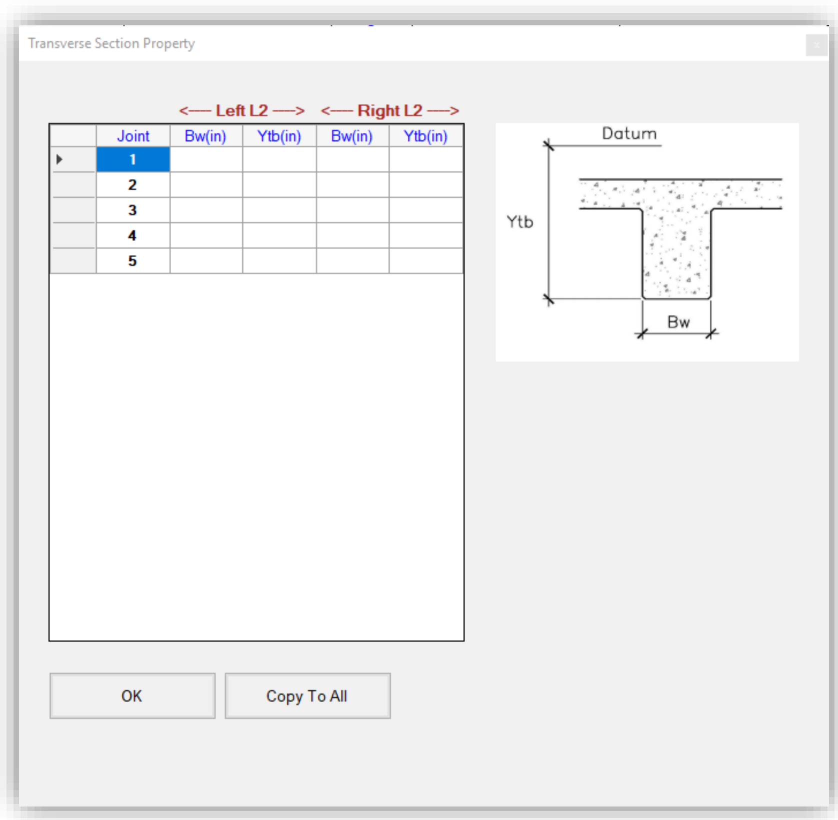



**Figure 3.15 Transverse Beam Input Screen**

The **Transverse Beam Input Screen** is shown in Figure 3.9. One can use the “Copy to All” button to copy a value to all the joints. If existing transverse beam data is being edited, all of the previously entered data will appear in the data grid.

To leave the screen and accept the transverse beam data as shown press the “OK” command button.

## 3-5 Section Manager

The section manager provides a platform to the user to delete, modify or add a section.

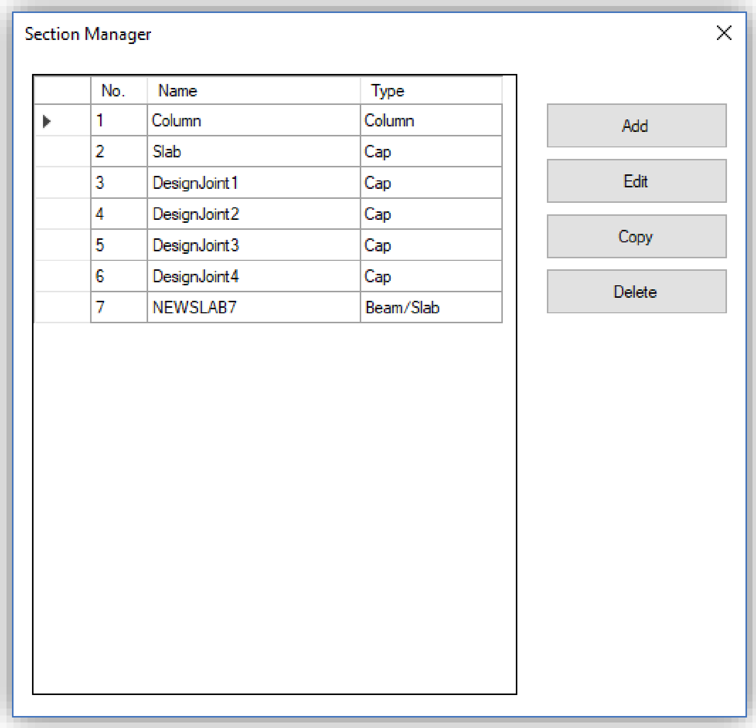



**Figure 3.16 Section Manager**

6. **Add** – To add a new section.

7. **Edit** – Select a section (select anywhere in the row) and click on edit to modify the section.

8. **Copy** – Creates a copy of the section selected.

9. **Delete** – Select a section and click on Delete to delete that section data.

***Note**: For 2-Way Slab / Mat Foundation, when designing the caps for punching shear, the program will redirect user to workspace. The user can then click on Section Manager in order to check and modify the designed sections.*

## 3-6 Material Properties

This window enables user to add Concrete, Reinforcement and Tendon material properties. The window is organized into 3 different tabs, showing related properties for each.

### 3-6(a) Concrete

1. **Beam Strength** The 28-day beam concrete compressive strength in pounds per square inch (psi).

2. **Beam Density** The beam concrete weight (density) in pounds per cubic foot (pcf).

3. **Column Strength** The 28-day column concrete compressive strength in pounds per square inch (psi).

4. **Column Density** The column concrete weight (density) in pounds per cubic foot (pcf).

5. **Tensile Stress Coefficient Top** The allowable flexural tensile stress in the top concrete fiber expressed as a multiplier of $\sqrt{f'_c}$.

6. **Tensile Stress Coefficient Bottom** The allowable flexural tensile stress in the bottom concrete fiber expressed as a multiplier of $\sqrt{f'_c}$.

7. **Minimum F/A** The minimum average compression stress (in psi) acting on the gross concrete cross-section at any point which is permissible in the "automatic" design generated by **PT Designer**.

### 3-6(b) Reinforcement

1. **Yield Strength** Enter the value in kips per square inch (ksi) for both longitudinal steel and stirrups in the text box provided

2. **Long. Bar Size** Entered as a number (#3-#11, #14, #18) using the pull-down combo box provided. The nominal bar diameter is used, along with the cover specified in items 12 and 13, to determine the center of gravity of the longitudinal unstressed steel. For example, with a #11 bar and a top 2" concrete cover, **PT Designer** would calculate the dimension from the top concrete fiber to the *CGS* of the #11 bar as 2 + 1.41/2 = 2.71". This value is used for the top *CGS* of the unstressed longitudinal reinforcement in all flexural strength calculations. If a smaller bar is actually used, the calculations are conservative. If a larger bar is actually used, the calculations are non- conservative.

3. **Stirrup Size** Entered as a number (#3-#9) using the pull-down combo box provided and used to establish the required stirrup spacing.

4. **Concrete Cover Top** Enter the *clear* dimension in inches from the top concrete fiber to the *top* of the longitudinal bar in the text box.

5. **Concrete Cover Bottom** Enter the *clear* dimension in inches from the bottom concrete fiber to the *bottom* of the longitudinal bar in the text box.

6. **Bot. Mat Spacing** Enter the typical bottom mat rebar spacing. The size of rebar will match the Long. Bar Size input value. If no bottom mat is used, a 0 value is inputted. This option is only available for two way slab designs.

7. **DL+LL/4 Rebar** Previous editions of the Uniform Building Code have required, in one-way post-tensioned members with unbonded tendons, sufficient bonded unstressed reinforcing steel to develop, using $\phi = 1.0$, the moments due to unfactored dead load plus 25% of the unfactored ***and unreduced*** live load. This input item tells **PT Designer**

whether this requirement is applicable. Click the "No" option button if this requirement does not apply. Click the "Yes" option button if it does apply.

8. **Full/Reduced LL Ratio** The number entered in this text box represents the ratio of unreduced to reduced live load for the ***entire*** frame. For example, if the Uniform Building Code section referenced in (9) above applies, and the frame beams require an unreduced live load of 50 psf which is reduced to a minimum of 30 psf, enter a value of 50/30 = 1.67 for this item. ***All*** input live loads will be multiplied by a factor of 1.67 in calculations ***pertaining to this item only***. This results in some minor conservatism in spans where the reduced live load is ***greater*** than 30 psf. This item will be disabled if the "No" option button is pressed in Item 9.

### 3-6(c) Tendon

1. **Tendon Type PT Designer** supports four types of tendons:

   - Unbonded low relaxation
   - Unbonded normal relaxation
   - Bonded low relaxation
   - Bonded normal relaxation

   The type of tendon is selected with the pull-down combo box provided.

2. **Bundle Diameter** The diameter or height of the tendon bundle in inches. This value is used to determine the dimension between the top and bottom concrete fibers and the tendon *CGS*, which is equal to the appropriate cover plus half the bundle diameter. Enter the diameter in the text box.

3. **Concrete Cover Top** The *clear* distance in inches from the top concrete fiber to the *top* of the tendon bundle (***not*** to the *CGS*.) Enter the distance in the text box.

4. **Concrete Cover Bottom** The *clear* distance in inches from the bottom concrete fiber to the *bottom* of the tendon bundle, for all spans *except* spans 1 and *S*. Enter the distance in the text box.

5. **Concrete Cover Bottom End Spans** The *clear* distance in inches from the bottom concrete fiber to the *bottom* of the tendon bundle for spans 1 and *S* only. Many building codes, notably the Uniform Building Code, require more fire cover in endspans than in interior spans for the same fire rating. This input item allows the user to address this code requirement. Enter the distance in the text box.

6. **Cross-Sectional Area of One Strand** Enter this area in square inches in the text box.

7. **%Sup. DL @ Transfer** In the text box for this item enter the percentage of superimposed dead load which is present and acting on the frame ***at the time the tendons are stressed***. This will be used in transfer stress calculations. For example, if half of the superimposed dead load is present at transfer, enter "50" for this item.

8. **Perpendicular Compression** Code requirements for punching shear capacity at critical sections of two-way slabs is a function of the average compressive stress acting on the faces of the critical section. **PT Designer** knows the compression stress acting on the two faces of the critical section which are normal to the plane of the equivalent frame (based on the prestress force at each joint), however it does not know the compressive stress on the faces of the critical section which are parallel to the equivalent frame (produced by prestress forces normal to the plane of the equivalent frame). This input value is the compressive stress acting on the faces of the critical section which are parallel to the equivalent frame. The default value for this variable is 125 psi.

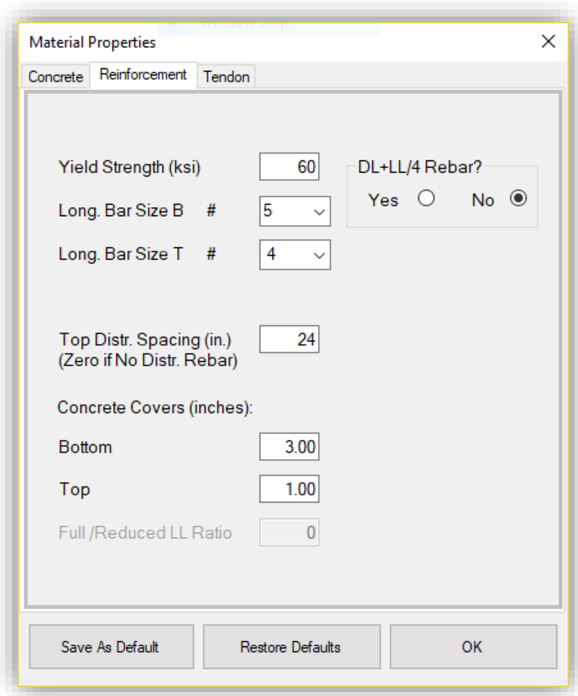



**Figure 3.17 Material Properties**

9. **Save As Default** – Saves the current data in the window as default. The next time the program will open with these default values.

10. **Restore Defaults** – Restores the default values stored in the program.

11. **OK** – Stores the data and closes the window.

### 3-7 Tendon Profile

The last of the input data screens is the **Tendon Profile Input Screen**, which is used to define the shape of the tendon profile in each span and cantilever. **PT Designer** supports a library of tendon profiles which is shown in Figure 3.16 as well as on the screen. Note that tendon TYPES 1 through 7 are for spans and TYPES 8 through 12 are for cantilevers. Only one tendon TYPE may be used in each span, however the TYPE can be different in each span. The input data required to define the tendon TYPE includes the TYPE number and any literal values shown in Figure 3.17 for that tendon TYPE. For example, for a TYPE 1 centerline parabola the only input data required is the TYPE number (#1). For a face-to-face double harped profile (TYPE 7) you must input six numerical values, the TYPE number (#7), the distances from the left support centerline to the two loads (*A* and *B*), the distances from the left and right support centerlines to the tendon high point ($c_L$ and $c_R$ respectively), and the ratio between the two balanced loads $P_2/P_1$ (discussed below). In this screen the high point locator dimensions $c_L$ and $c_R$ are entered in inches and the dimensions *A* and *B* are entered in feet. $P_2/P_1$ is unitless. Note that the tendon highpoint and lowpoint dimensions ($Y_L$, $Y_m$, and $Y_R$) are not entered as input items in the **Tendon Profile Input Screen**. These dimensions are determined initially for all spans by **PT Designer** in the automatic design procedure (see **THEORY**, Section 9.2) and can then be modified, if necessary, by the user from the **RESULT MENU**.

To leave the screen and accept the tendon profile data as shown press the "OK" command button.

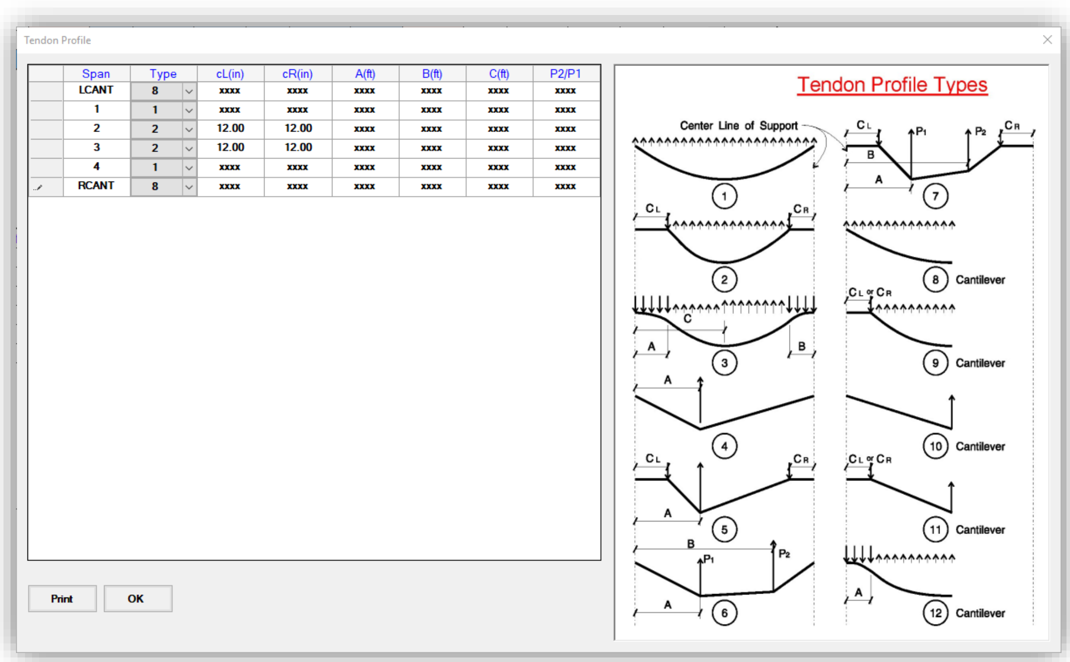



**Figure 3.18 Tendon Profile Input Screen**

## 3-8 Loads

This screen is used to input all of the superimposed dead, live, and wind loads which act on the frame beams. These loads ***do not include*** the weight of the beam concrete, which is calculated ***automatically*** by **PT Designer** for each segment, based upon the input geometry and concrete density. Loads entered in this screen are those acting ***in addition*** to the concrete weight. A maximum of 20 superimposed loads can be entered for each span in the frame. Each superimposed load can have a dead load portion and a live load portion acting at the same location. **PT Designer** supports five types of loads as shown in Figure 3.13. They are:

**UNIFORM LOAD (U)** A load in ***kips per square foot*** which acts over the full tributary of the beam starting at a distance *A* measured in feet from the left support centerline and ending at a distance *B*, also measured in feet from the left support centerline. For left cantilevers the distances *A* and *B* are measured from the right support centerline. The length of the load is *B - A* in feet. The uniform load option will always calculate a line load based upon the single tributary width specified. Therefore, if the beam or plate has various tributary widths along the span the Line Load (L) load type or additional Uniform Load (U) type can be used to model varying tributary widths.

**LINE LOAD (L)** A load in ***kips per lineal foot*** starting at a distance *A* measured in feet from the left support centerline and ending at a distance *B*, also measured in feet from the left support centerline. The length of the load is *B - A* in feet. A **LINE LOAD** is independent of the beam TRIBUTARY and can be used to model various tributary widths.

**POINT LOAD (P)** A concentrated load in ***kips*** located at a distance *A* measured in feet from the left support centerline (right support centerline in a left cantilever).

**CONCENTRATED MOMENT (M)** A concentrated moment in ***kip-feet*** located at a distance *A* measured in feet from the left support centerline (right support centerline in a left cantilever).

**WIND MOMENT (W)** A set of ***two*** applied beam moments in kip-feet, one acting at each end of the beam, caused by lateral wind loads. Wind moments are assumed to vary linearly between beam ends. Normally these moments will be obtained from a separate frame analysis for wind loads only. The signs of the input wind moments must be consistent with ***one*** direction of applied wind loads throughout the frame (either direction). **PT Designer** knows these moments are reversible and will consider both directions of applied wind loads in the analysis.

Loads in Figure 3.13 are shown acting in the ***positive*** direction, except for wind moments where the left-end moment is shown positive and the right-end moment is shown negative.

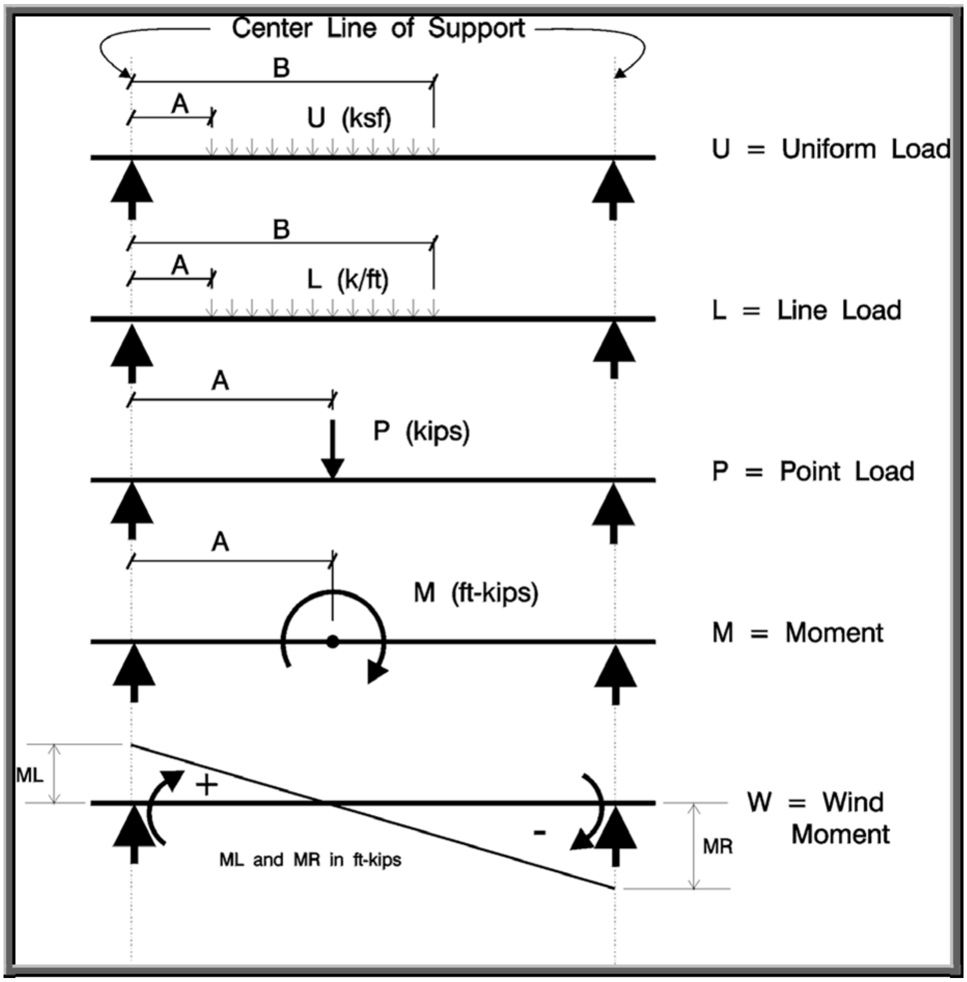



**Figure 3.19 Applied Loads**

The **Superimposed Load Input Screen** is shown in Figure 3.15. One screen appears for each span and the current span is shown at the top of the data grid just below the Project Heading Bar. The first screen to ***appear*** will be for the left cantilever, if there is one, or Span 1 if there is no left cantilever, however loads can be entered in any span in any order. Loads for any one span can be input ***in any order*** independently of the TYPE of load or the location in the span. **PT Designer** identifies each load in each span with a number which appears in the LOAD # column at the left of the data grid. Loads can be added using the "Add a Load", "Typical Load", and "Uniform Load" command buttons, described more fully below. For each load you will be prompted, as appropriate, for the TYPE of load, the magnitude of the dead and live load portions (DL, LL) in the correct units, and the *A* and *B* dimensions in feet. When prompted for the TYPE of load, enter the appropriate identifying letter (<U>,<L>, <P>, <M>, or <W>) in either upper or lower case. **PT Designer** will insert this letter in the TYPE column for that load along with the correct units for that TYPE of load as a reminder. For wind moments, input the left-end moment in the DL column, the right-end moment in the LL column. If more than one set of wind moments is entered in any span, **PT Designer** considers them as additive.

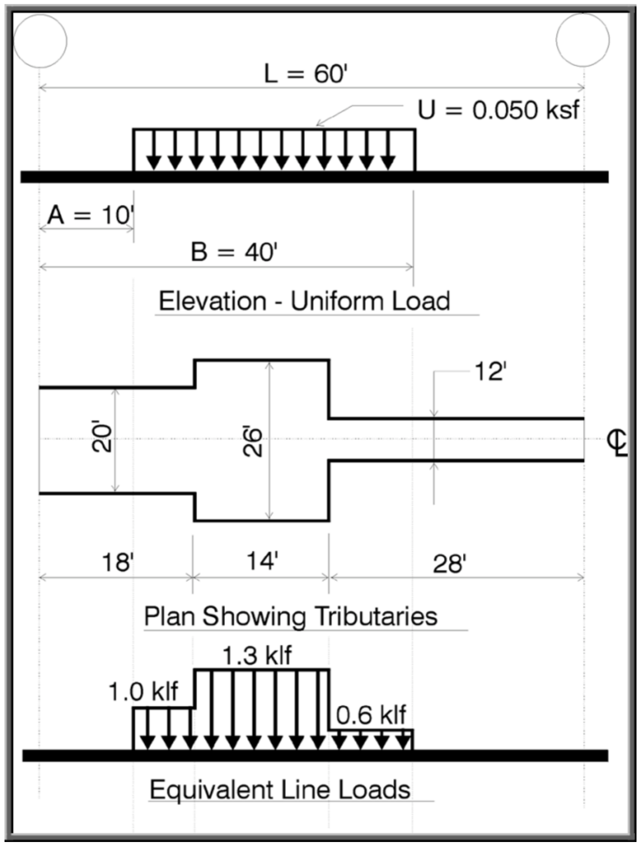



**Figure 3.20 Uniform Load (U) Example**

Following is a description of the function of each of the command buttons which appears in the **Superimposed Load Input Screen**:

- **Choose Span** - Changes the screen display to any other specified span in the frame.
- **Add a Load** - Starts the input dialog for a load which is added to the current span only. Under Type, the user can choose the type loading from the dropdown box.
- **Remove a Load** - Removes the load in this span identified by "Load #". All subsequent loads will move up one load number.
- **Remove All Loads** – Removes all loads in all spans.
- **Next Span** - Changes the screen display to the next consecutive span in the frame.
- **Previous Span** - Changes the screen display to the previous span in the frame.
- **Typical Load** – Shows an input dialog for an ***identical*** load which is added to the current span and to every other span in the frame. In each span the load will have ***exactly*** the same TYPE, DL, LL, *A*, and *B* values as input in the current span. Underneath Loading Pattern, there are 2 options, on full length and user defined. If we select "On full Length", then the loading will automatically be applied to all the

spans. If we want the loading pattern on a specific length then select "User Defined", which will then activate the "A" and "B" input boxes.

- **Uniform Load** – Shows an input dialog similar to that of a typical load, for a load with Load Type = Uniform (U) which is added to the current span. The added loads will have the same DL and LL values in all spans and will extend over the full length of each span from *A=0* to *A=L*. The uniform load is applied over the tributary width specified for each span.

- **OK** – Press this command button only when all of the load data has been entered correctly for ***all spans***. It will close the **Superimposed Load Input Screen**.

For **UNIFORM LOADS (U)**, which often occur over an entire span (*A* = 0*, B* = *L*), **PT Designer** will automatically insert into the data grid a value of *L*, the span length in feet, for the *B* dimension and a value of zero for *A*. These values can be accepted or changed. For uniform or line loads **PT Designer** will not accept a value for *B* which is less than or equal to the value already input for *A*. For point loads and concentrated moments **PT Designer** skips the prompt for *B*. and inserts "xxxx" in the *"B"* column. Similarly for wind moments both the *A* ***and*** *B* prompt are skipped and "xxxx" is inserted in those columns. **PT Designer** will not accept an *A* ***or*** *B* value greater than *L*. Figure 3.15 shows a **Superimposed Load Input Screen** for Span 1 which contains an example of each of the five types of loads.

When all loads in all spans are correct press the "OK" command button.

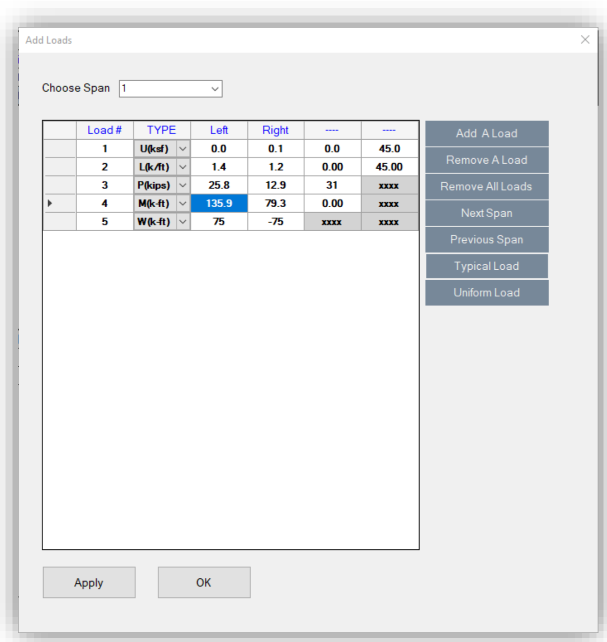



**Figure 3.21 Loading Input Screen**

.

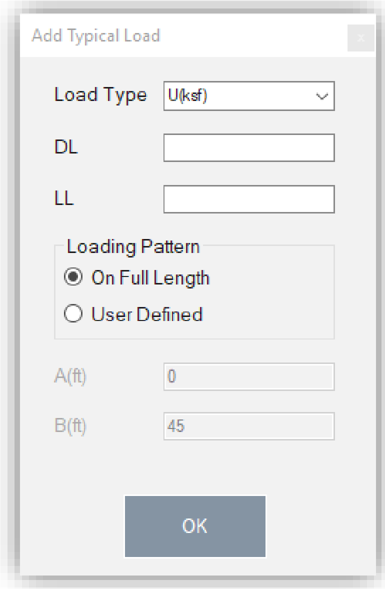



**Figure 3.22 Typical Load Dialog**

### 3-9 Configurations

Select this button to open the Configuration Screen, shown in Figure 2.3. This screen allows editing of the configuration options contained in the **PTDATA.INI** file (see Section 1.2 for a description of which items can be permanently modified, and which can only be modified for individual runs). Data in this screen is selected by means of *option buttons* and *text boxes*. Option buttons are contained in frames and only one button (option) may be selected (by clicking on the button) in each frame. The option button selected contains a black dot. Text boxes contain numerical values and can be modified by directly editing the value in the box or by clicking on the command button adjacent to the box. Following are the available configuration options:

a. **Column Modeling** Use this option to change the frame modeling method for columns (see **THEORY** Section 7.1(a)). Two choices are available:

   i. **Top and Bottom Columns Always Present**

   ii. **Top Column Present for Live Load Only**

   If the first choice is selected, **PT Designer** assumes that the top and bottom columns at each joint are both present and contribute stiffness to the joint under ***all*** loading conditions. If the second choice is selected, **PT Designer** assumes that ***only the bottom column*** is present to resist dead and balanced loads and ***both top and bottom columns*** are present to resist live loads. Select the option you want by clicking on the option button in the Column Modeling Option frame.

b. **Cracking Moment Calculations** - Use this Option to include or ignore the cracking moment requirements of ACI 318-95 Section 18.8.3 (see **THEORY**

Section 9.1). The requirement to consider cracking moment calculations has been eliminated in ACI 318-08, so this option is only applicable to analysis of older designs or new designs where ACI 318-08 has yet to be adopted.

c. **Spaces** - Enter in the text box the number of equal spaces (unlimited) into which each clearspan is to be divided, design values calculated at each end of each space (see **THEORY** Section 8.1).

d. **Column $I_e/I_{gross}$** – Enter in the text box the ratio between the effective (cracked) and gross (uncracked) column moment of inertia, $I_e/I_{gross}$. The default value of 1.0 will result in gross column section properties for column stiffnesses. This value will apply to ***all*** columns in the frame, top and bottom.

e. **Effective Tendon Stress** - These options change the effective tendon stresses used by **PT Designer** for flexural strength and shear calculations in the Constant Prestress Force mode. One effective tendon stress is input for low relaxation tendons, one for stress relieved tendons (the tendon type is determined by the user in the **Material Properties** 3-6). These values will also be used to determine the number of tendons required in the member being designed.

f. **Load Factors** - These options change the eight load factors for dead, live, and wind loads in strength demand equations as described in **THEORY** Section 9.1(a). These values can be modified for individual runs, but will not be permanently changed in the Configuration file file.

g. **Minimum Shear Cap Size** - If this option is enabled **PT Designer** will not permit a shear cap size *(W1xW2)* to be entered which is smaller than the first (inside) critical section (Section #1). If this option is disabled any shear cap size can be entered. *Currently the user has the flexibility to check for Minimum Cap Size from the workspace itself. This option is under development.*

h. **Non-Controlling Cantilever Iteration** - **PT Designer** can determine, by iteration, the tendon CGS at a cantilever support to optimize the balanced load in the cantilever and the adjacent span. If you want the iteration to occur, select the "Iterate Highpoint" option button. If you want the cantilever tendon to be straight (no sag) between the tip of the cantilever and the cantilever support (no iteration), select the "No Iteration" option button. This latter option can be useful in the case of very short cantilevers.

i. **The ACI 318-99+ Check Box** – Unchecking this box tells the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases from the codes after ACI 318-99. This will be the default each time the program is opened. Checking this box will tell the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases of the 1997 Uniform

Building Code. The user may then over-ride any of the load factors for his/her individual requirements. The capacity reduction factors and redistribution cases cannot be modified and will remain consistent with either the latest ACI 318 Code or the 1997 Uniform Building Code, depending upon the state of the check box.

You can modify data in the Configuration Screen and use the modified data in the ***current run only*** by clicking on the "OK - Return To The Main Menu" button. You can use the modified data in the current run ***and insert the modified data into the PTDATA.INI file*** by clicking on the "Set As Default" button. Use this last action with care because the modified data will continue to be used in all future runs until the **PTDATA.INI** file is modified again. When in doubt as to the contents on the **PTDATA.INI** file you can always bring up the Configuration Screen and review the current configuration data.

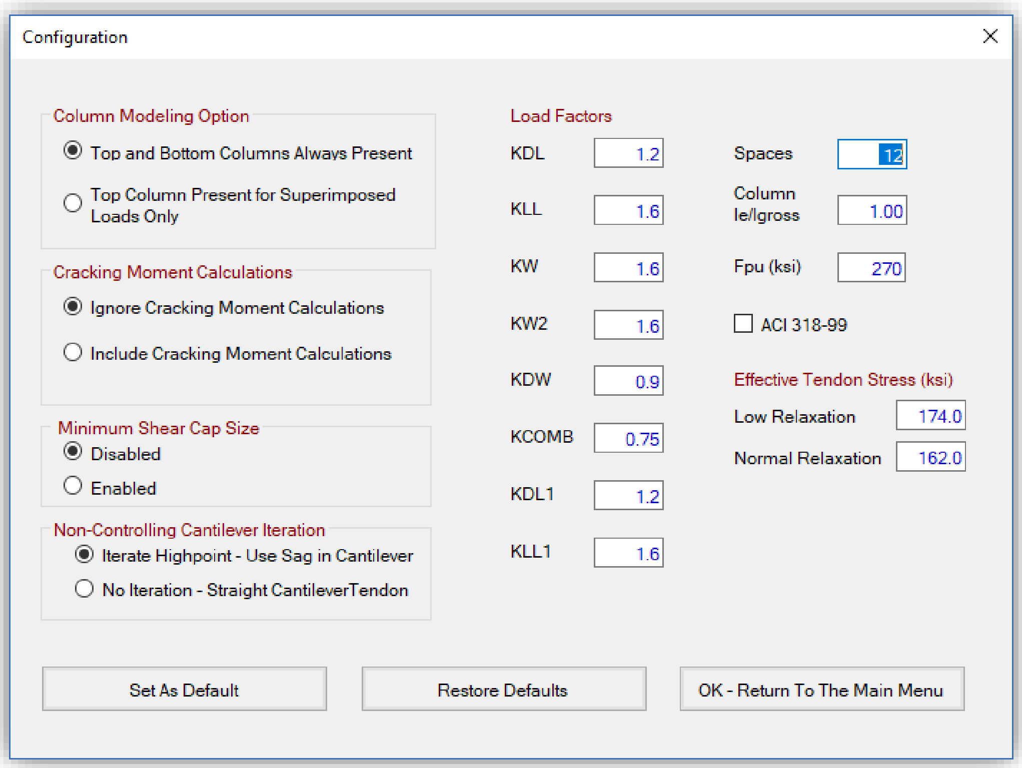



**Figure 3.23 Change Configurations Input Screen**

# CHAPTER 4 RESULTS

Click on the “Run” button to run the analysis. Once the structure is analysed successfully then we get the results panel on the left panel. The first result that is “Forces and Tendon Profiles” is selected by default, the workspace showing the corresponding results.

The **Result Panel** contains various Options which can be selected by clicking on the appropriate command button. **Result Panel** Options permit the screen review of forces, profiles, flexural stresses, percent dead load balanced (only when parabolic tendon profiles are specified), moments, shears, punching shear stresses, and deflections; the initiation of the Variable Prestress Force Mode.

All **Result Panel** command buttons are always available ***except*** the ***Forces and Tendon Profiles*** button (Section 4.1) which is enabled ***only*** in the Constant Prestress Force Mode, the ***DL+0.25LL Rebar*** button which is enabled only if it applies to the current run, the ***Punching Shear Analysis*** button which is enabled only if the member type is “2-Way Slab”, and the ***Variable Prestress Force Option*** button.

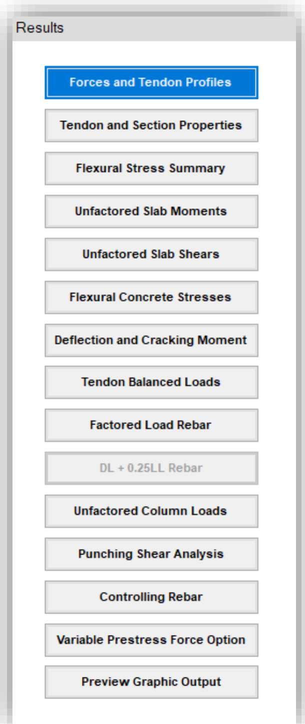



**Figure 4.1 Result Menu**

Following is a detailed description of each of the various Result Panel Options:

## 4-1 Forces and Tendon Profiles- Result or Change

Select this Option to review the current tendon forces and profiles for the frame in the Constant Prestress Force mode. The first time this Option is selected for a run, the forces and profiles will be those calculated by **PT Designer** in its automatic design. The screen used to display the forces and profiles in **PT Designer** is shown in Figure 4.2. This is the only **Result Option** which permits direct editing of calculated values. Any prestress force or tendon profile may be changed in this screen, and the corresponding joint and span maximum stresses will immediately be calculated in the adjacent grid. If a change is made, **PT Designer** recalculates all other frame values which are a function of the tendon force and profile immediately for the other results. All the other results specified in the Results menu are always consistent with the currently selected forces and profiles.

The Forces and Tendon Profile screen shows the force per foot across the entire tributary for each span, and the percent concrete self-weight balanced for spans in which parabolic tendon profiles are specified. Caution flags will appear on the lower right hand side of the window for balance loads below 65% and above 125%. For designs that use harped stands, a balance load percentage is not calculated in this window. With no value in the cell, a low balance percentage flag will appear but this should be disregarded. The balance load for harped stands can be viewed by using the "Tendon Balance Load" command button (See Section 4-8).

Forces are shown in kips and are the ***entire*** effective ***constant*** prestress force for each frame beam. To determine the required number of strands, **PT Designer** uses the cross-sectional area of one tendon ($A_{ps}$) which is input in the **Material Properties** (Section 3-6) and the value $f_{se}$ for effective tendon stress which is input in the **Configuration Screen** (Section 3-9). The required number of strands is $F_e/(A_{ps}f_{se})$.The user may change either the force or the number of strands, and **PT Designer** will modify the other value accordingly. The force or number of strands can be changed in only one span by changing the value in the appropriate cell, or it can be changed in all spans by using the "Typical Force Fe" or "Typical No. of Strands" command buttons.

In the automatic design procedure **PT Designer** calculates the precise force required to satisfy the design requirements without regard as to whether or not this force represents an integer number of strands. Thus the first time this Option is viewed in the Results, the number of strands will, in general, be a non-integer value (i.e., 10.6, 12.4, etc.). The user can then modify the number of strands to an integer number, if desired, and **PT Designer** will adjust the force accordingly.

All profile dimensions (both high points and low points) are measured ***from the datum line***, rather than from the top or bottom of the beam concrete. For Tendon Types 1 through 5 and 8 through 12 (Figure 3.16) one low point is sufficient, along with the tendon high points, to define the tendon profile. In this case the single lowpoint is shown in Figures 4.2 as Lo1. It is the dimension from the datum line to the ***lowest vertical point*** in the tendon profile. For the remaining Tendon Types (the double harps 6 and 7) two low points are required to define the profile. They are shown in Figures 4.2 as Lo1 and Lo2.

Lo1 is the vertical dimension from the datum line to the tendon at a distance *A* from the left support centerline. Lo2 is the vertical dimension from the datum line to the tendon at a distance *B* from the left support centerline. Dimensions for Lo2 will appear in the **Forces and Tendon Profiles Result Screen** only for Tendon Types 6 or 7. See Figure 3.15 for a graphical description of the tendon profile dimensions.

**PT Designer** reviews each span and cantilever and determines the minimum and maximum value of the average concrete compression *F/A* for the span or cantilever. These extreme values of concrete compression (in ksi) are shown in the two columns headed *"F/A"* in Figures 4.2.

To edit the data in the **Forces and Tendon Profiles Result Screen** see the procedure described in Section 3.1. If calculations for this run have been made previously, **PT Designer** retains the previous values for forces and profiles, and the "Previous Force and Profile" command button will be enabled. To revert to the previous forces and profiles press the "Previous Force and Profile" command button. This option is extremely useful when you are "tuning" a run and you want to begin with an existing force and profile.

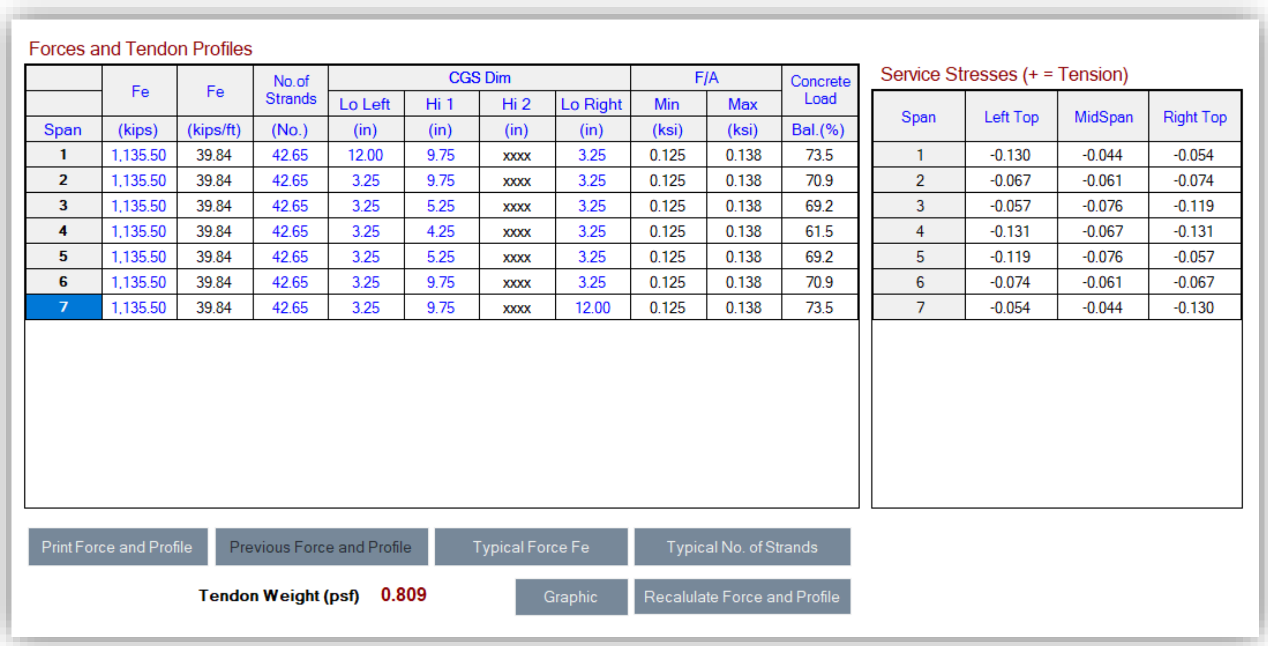
If changes have occurred inside the RESULT MENU windows and/or you would like to have **PT Designer** to provide revised starting points for the design of the tendons and their profiles, use the "Re-Calculate Force and Profile" command button. This function will enable the same algorithm used after the initial input was completed.

The Tendon Weight in psf, based upon the total weight of all tendons in the frame and the total tributary of the entire frame, is calculated and presented in this screen. Allowance is made for added tendon "tails" and excess strand protruding from the edges at exterior stressing ends.

If any value has been changed in the Forces and Tendon Profiles Result Screen, (this includes pressing the "Previous Force and Profile" command button) **PT Designer** will, immediately upon leaving the screen, recalculate all values which are a function of tendon force and profile, including unstressed reinforcing steel data. All calculated values accessible from the RESULT MENU are thus consistent with the currently selected forces and profiles.

To the right of the tendon force and profile chart, the flexural stresses are tabulated at the joints and midpoint of each span. These stresses will change concurrently with any modification to the tendon force or profile with the screen remaining open. If the flexural stress limits exceed the values input into the Material Properties (Section 3-6) a flag will appear with a corresponding asterisk (*) in the flexural stress chart at the location of exceedance. The flags will not be a part of the printed output but identify areas for possible further review. Flexural compressive stresses are **not** included in this table.

Those values are listed in the Flexural Stress Summary Screen per section 4.3 and are recommended to be reviewed to verify code compliance and design intent.



Forces and Tendon Profiles

| | Fe | Fe | No. of Strands | CGS Dim | | | | F/A | | Concrete Load |
|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Lo Left | Hi 1 | Hi 2 | Lo Right | Min | Max | |
| Span | (kips) | (kips/ft) | (No.) | (in) | (in) | (in) | (in) | (ksi) | (ksi) | Bal.(%) |
| 1 | 1,135.50 | 39.84 | 42.65 | 12.00 | 9.75 | xxxx | 3.25 | 0.125 | 0.138 | 73.5 |
| 2 | 1,135.50 | 39.84 | 42.65 | 3.25 | 9.75 | xxxx | 3.25 | 0.125 | 0.138 | 70.9 |
| 3 | 1,135.50 | 39.84 | 42.65 | 3.25 | 5.25 | xxxx | 3.25 | 0.125 | 0.138 | 69.2 |
| 4 | 1,135.50 | 39.84 | 42.65 | 3.25 | 4.25 | xxxx | 3.25 | 0.125 | 0.138 | 61.5 |
| 5 | 1,135.50 | 39.84 | 42.65 | 3.25 | 5.25 | xxxx | 3.25 | 0.125 | 0.138 | 69.2 |
| 6 | 1,135.50 | 39.84 | 42.65 | 3.25 | 9.75 | xxxx | 3.25 | 0.125 | 0.138 | 70.9 |
| 7 | 1,135.50 | 39.84 | 42.65 | 3.25 | 9.75 | xxxx | 12.00 | 0.125 | 0.138 | 73.5 |

Service Stresses (+ = Tension)

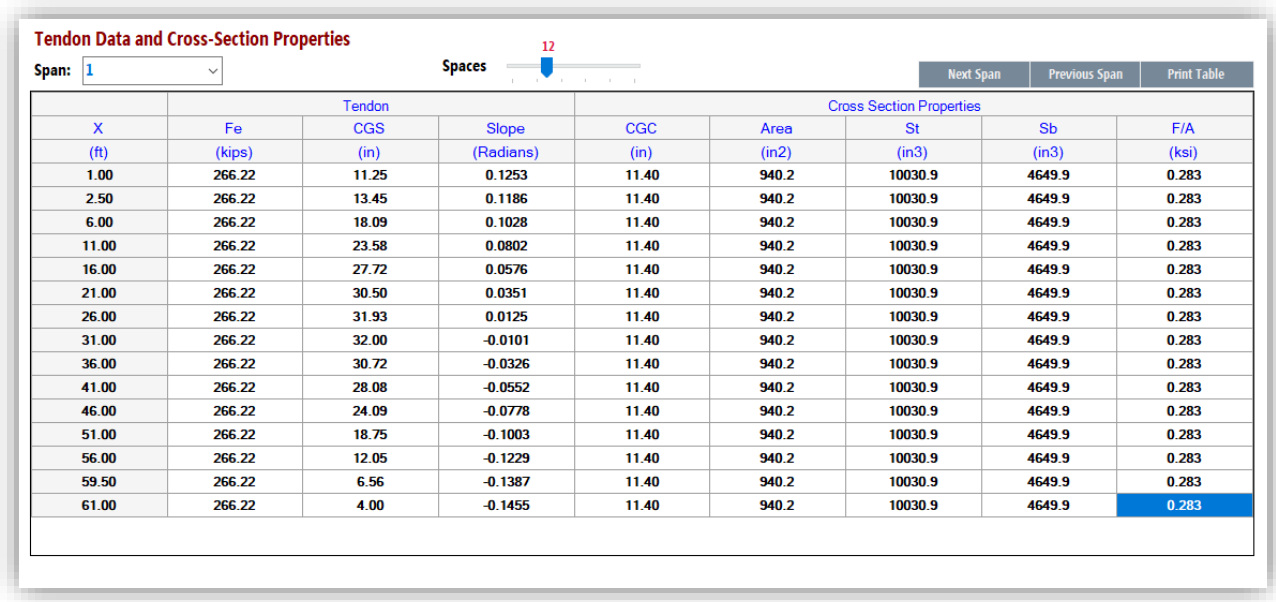
| Span | Left Top | MidSpan | Right Top |
|---|---|---|---|
| 1 | -0.130 | -0.044 | -0.054 |
| 2 | -0.067 | -0.061 | -0.074 |
| 3 | -0.057 | -0.076 | -0.119 |
| 4 | -0.131 | -0.067 | -0.131 |
| 5 | -0.119 | -0.076 | -0.057 |
| 6 | -0.074 | -0.061 | -0.067 |
| 7 | -0.054 | -0.044 | -0.130 |



**Figure 4.2 Forces & Tendon Profiles Results**

## 4-2 Tendon Data and Cross-Section Properties Screen

This screen is shown in Figure 4.3. It shows, for each span and at each design point, the effective tendon force Fe; the tendon CGS dimension (measured in inches from the datum line); the tendon Slope in radians (clockwise tangent rotation positive); the centroid of the gross concrete cross-section CGC (measured from the datum line); the cross-sectional concrete Area in in$^2$; the top and bottom section moduli St and Sb in in$^3$; and the average compressive stress F/A in ksi.


Tendon Data and Cross-Section Properties
Span: 1
Spaces 12
Next Span | Previous Span | Print Table

| | Tendon | | | Cross Section Properties | | | | |
|---|---|---|---|---|---|---|---|---|
| X | Fe | CGS | Slope | CGC | Area | St | Sb | F/A |
| (ft) | (kips) | (in) | (Radians) | (in) | (in2) | (in3) | (in3) | (ksi) |
| 1.00 | 266.22 | 11.25 | 0.1253 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 2.50 | 266.22 | 13.45 | 0.1186 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 6.00 | 266.22 | 18.09 | 0.1028 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 11.00 | 266.22 | 23.58 | 0.0802 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 16.00 | 266.22 | 27.72 | 0.0576 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 21.00 | 266.22 | 30.50 | 0.0351 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 26.00 | 266.22 | 31.93 | 0.0125 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 31.00 | 266.22 | 32.00 | -0.0101 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 36.00 | 266.22 | 30.72 | -0.0326 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 41.00 | 266.22 | 28.08 | -0.0552 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 46.00 | 266.22 | 24.09 | -0.0778 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 51.00 | 266.22 | 18.75 | -0.1003 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 56.00 | 266.22 | 12.05 | -0.1229 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 59.50 | 266.22 | 6.56 | -0.1387 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |
| 61.00 | 266.22 | 4.00 | -0.1455 | 11.40 | 940.2 | 10030.9 | 4649.9 | 0.283 |

**Figure 4.3 Tendon And Section Properties**

The number of design points can be changed by dragging the trackbar from 5 to 30. If you need to provide more design points then you can provide it in the Configurations window.

## 4-3 Flexural Stress Summary Screen

If you select this Option **PT Designer** will sort the currently calculated values for concrete flexural tensile and compressive stresses and select the critical ones for your review. The **Flexural Stress Summary Screen** is shown in Figure 4.4. This screen shows you the critical ***tensile*** stresses in end spans ***and*** through (interior) spans separately. Critical ***compressive*** stresses are displayed for ***all*** spans, both end and interior.

Critical stresses are shown for the ***top*** and ***bottom*** beam fibers, along with the location in the span where the critical stresses occur (measured from the left support centerline in all spans except a left cantilever where the location is measured from the right support centerline).

In the example shown in Figure 4.2, the frame has five spans and two cantilevers. One left end span and one right end span have been specified in the **Create Geometry Section** (see Section 3-1). The heading for Left End Spans identifies those spans as the left cantilever through Span 2 (**CL to 2**) and the heading for Right End Spans identifies them as Span 6 through the Right Cantilever (**6 to CR**). The remaining spans are identified as interior span in the heading for Interior through Spans (**3 to 5**). The maximum tensile stress in the top beam fiber in the left end spans is 0.405 ksi and it occurs in Span 2 at a distance of 25.5 feet from the left support centerline (Joint 2).

The maximum top fiber tensile stress in the interior Spans 3 through 5 is 0.408 ksi and it occurs in Span 5 at a distance of 25.5 feet from the left support centerline of Span 5 (also at Joint 5). The maximum tensile stresses at the bottom beam fiber are also tabulated in a similar manner for both end and interior spans.

The sorting of maximum ***compressive*** stresses is not separated into end and interior spans as is the sorting for tensile stresses. Rather, the ***single*** maximum top and bottom fiber compressive stress ***in any span*** (end or interior) is found and displayed in this screen. In the example shown, the maximum compressive stress which exists at the top beam fiber ***anywhere in the frame*** is -0.527 ksi and it occurs in Span 2, 14.25 feet from the left support centerline (in Span 2 between Joints 2 and 3). At the bottom beam fiber the maximum compressive stress is –0.723 ksi and it occurs in Span 5 at a distance of 25.5 feet from the left support centerline (Joint 5).

Users must pay particular attention to the critical compressive stresses shown in this summary. The **PT Designer** automatic design procedure is based upon ***tensile stresses only*** (see **THEORY** Section 9.2). The user must verify that the compressive stresses for any design are within Code allowables. This screen facilitates that since it displays the maximum top and bottom compressive stresses which exist anywhere in the frame. Users are cautioned against the use of any post-tensioned concrete member whose design is controlled by flexural compressive stresses.

SUMMARY

| | Left Endspans<br>CL to 2 | Interior Spans<br>3 to 5 | Right Endspans<br>6 to CR |
|---|---|---|---|
| Max. Tension at Top (ksi)<br>Span Number<br>Distance From Left Support (ft) | 0.405<br>2<br>25.5 | 0.408<br>5<br>25.5 | 0.405<br>6<br>3.0 |
| Max. Tension at Bottom (ksi)<br>Span Number<br>Distance From Left Support (ft) | 0.212<br>2<br>14.25 | 0.176<br>4<br>14.25 | 0.212<br>6<br>14.25 |
| Max. Compression at Top (ksi)<br>Span Number<br>Distance From Left Support (ft) | | -0.527<br>2<br>14.25 | |
| Max. Compression at Bot (ksi)<br>Span Number<br>Distance From Left Support (ft) | | -0.723<br>5<br>25.5 | |

Print

**Figure 4.4 Flexural Stress Summary**

## 4-4 Unfactored Beam Moments

The ***Unfactored Beam Moments Result Screen*** is shown in Figure 4.5. It tabulates, in kip-feet, the ***unfactored*** dead load, balanced load, live load, secondary, and wind moments for each ***design point*** in each span of the frame. See Section 1.8 for a discussion of the design points. Positive moments cause tension in the bottom beam fiber, negative moments cause tension in the top beam fiber.

There are two columns of live load moments displayed in this screen, one with the live load arranged to produce the most negative live load possible (Max M-) at each point, one with the live load arranged to produce the most positive live load moment possible (Max M+). These two live load columns thus bound the envelope of live load moments possible at each point. For example, in Figure 4-5 at x = 31 feet (the midpoint of the span) the most positive live load moment possible (Max M+) is +129.08 foot-kips; the most negative live load moment possible (Max M-) is –5.47 foot-kips. For non-skipped live loads (see Section 3-1, Item #5) the moments shown in the two live load columns will be equal.

**Unfactored Beam Moments (k-ft)**

Span: 1 Spaces 19 Next Span Previous Span Print Table Show Diagram

| X(ft) | DL | Bal Load | LL(Max-) | LL(Max+) | M2 | Mwind |
|---|---|---|---|---|---|---|
| 1.00 | -410.63 | 162.88 | -188.11 | 8.42 | 162.88 | 0.00 |
| 2.50 | -340.81 | 135.58 | -156.86 | 7.73 | 161.52 | 0.00 |
| 4.16 | -267.94 | 107.09 | -124.21 | 6.96 | 160.02 | 0.00 |
| 7.32 | -141.65 | 57.69 | -67.48 | 5.50 | 157.16 | 0.00 |
| 10.47 | -31.77 | 14.68 | -17.94 | 4.04 | 154.29 | 0.00 |
| 13.63 | 61.70 | -21.93 | 0.00 | 27.00 | 151.43 | 0.00 |
| 16.79 | 138.76 | -52.14 | 0.00 | 60.72 | 148.56 | 0.00 |
| 19.95 | 199.41 | -75.96 | -0.35 | 87.61 | 145.70 | 0.00 |
| 23.11 | 243.66 | -93.38 | -1.81 | 108.43 | 142.84 | 0.00 |
| 26.26 | 271.49 | -104.41 | -3.27 | 122.08 | 139.97 | 0.00 |
| 29.42 | 282.92 | -109.04 | -4.73 | 128.54 | 137.11 | 0.00 |
| 32.58 | 277.94 | -107.28 | -6.20 | 127.82 | 134.25 | 0.00 |
| 35.74 | 256.56 | -99.12 | -7.66 | 119.92 | 131.38 | 0.00 |
| 38.89 | 218.76 | -84.57 | -9.12 | 104.85 | 128.52 | 0.00 |
| 42.05 | 164.56 | -63.62 | -10.58 | 82.59 | 125.66 | 0.00 |
| 45.21 | 93.95 | -36.27 | -12.04 | 53.15 | 122.79 | 0.00 |
| 48.37 | 6.93 | -2.53 | -13.51 | 16.54 | 119.93 | 0.00 |
| 51.53 | -96.50 | 37.61 | -42.23 | 0.00 | 117.06 | 0.00 |
| 54.68 | -216.34 | 84.14 | -94.67 | 0.00 | 114.20 | 0.00 |
| 57.84 | -352.58 | 137.07 | -154.29 | 0.00 | 111.34 | 0.00 |
| 59.50 | -430.68 | 167.42 | -188.46 | 0.00 | 109.83 | 0.00 |
| 61.00 | -505.24 | 196.39 | -221.09 | 0.00 | 108.47 | 0.00 |

**Figure 4.5 Unfactored Beam Moments**

Click on **SHOW DIAGRAM** to view the bending moment diagram for the current span.

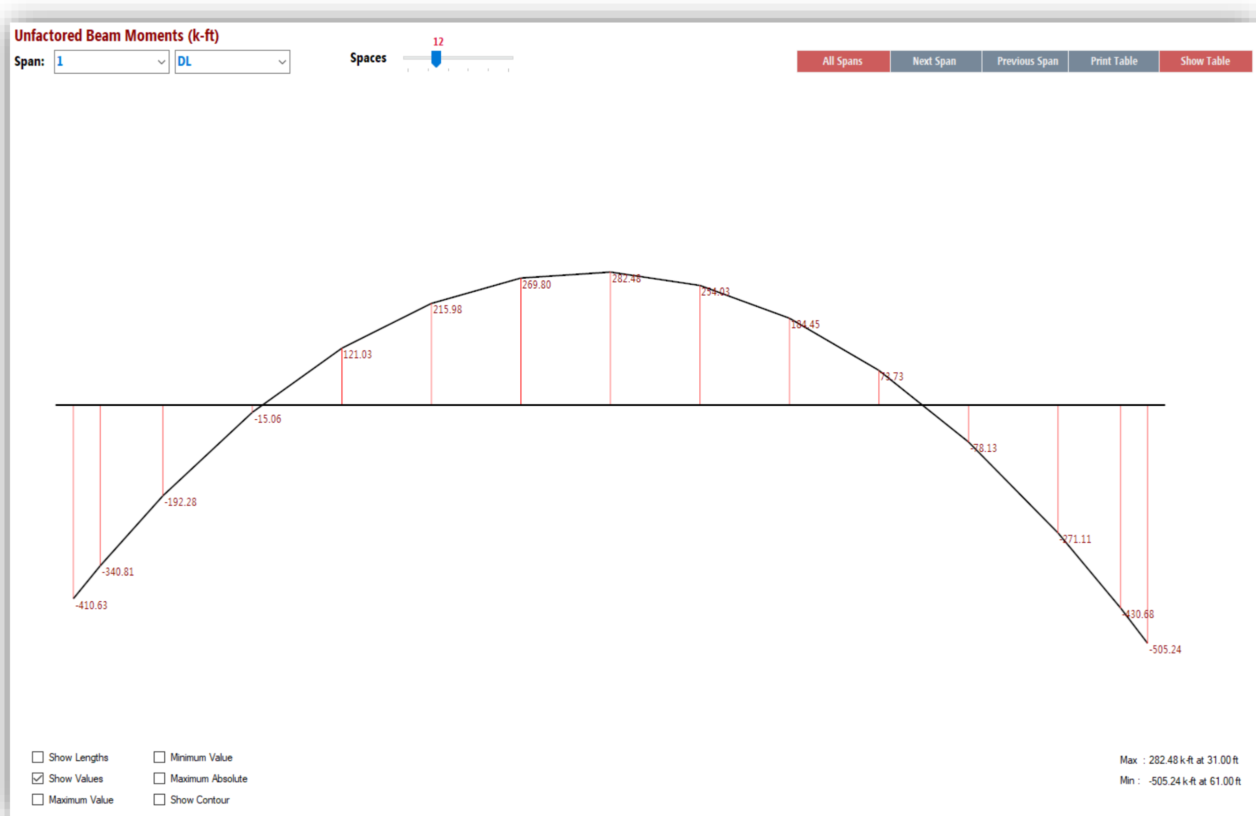



**Figure 4.6 Bending Moment for the current span**

To increase the number of value points, drag the tracker on the trackbar besides spaces. The trackbar can provide from 5 to 30 points. If you need more values, then you can change the value of Spaces in the Configuration window from the Ribbon Menu.

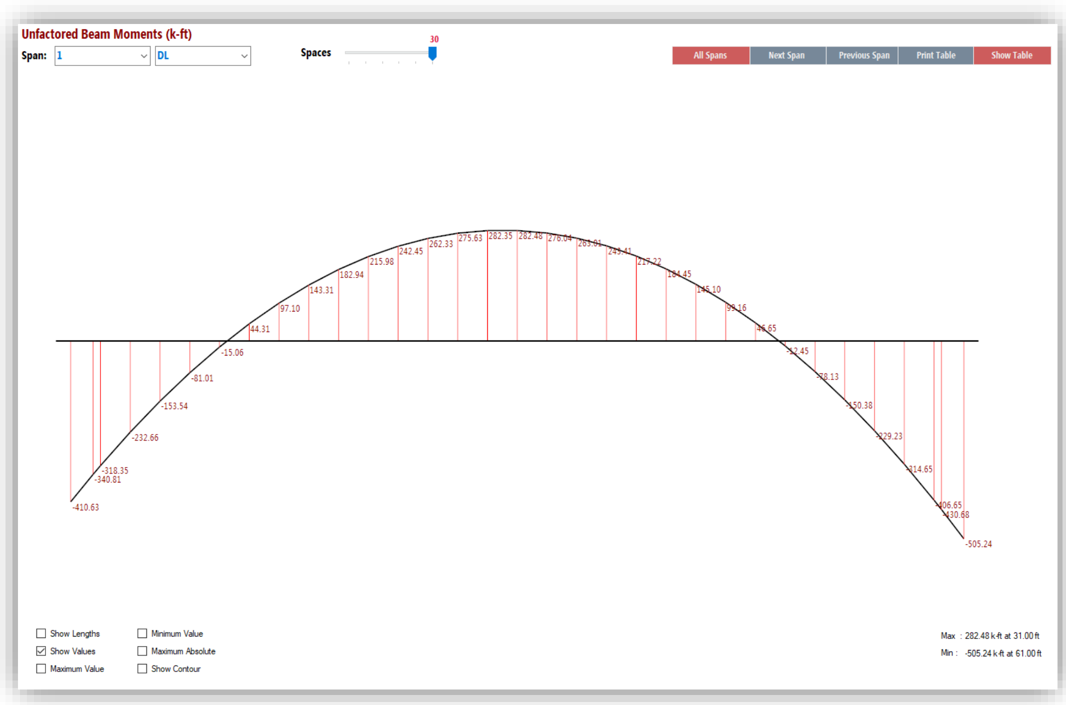


**Figure 4.7 Bending Moment with increased number of design points**

The window also shows the Max and Min value along with the position over the span where it occurs. In order to view only the maximum value, check off Show Values and check on Maximum Value.

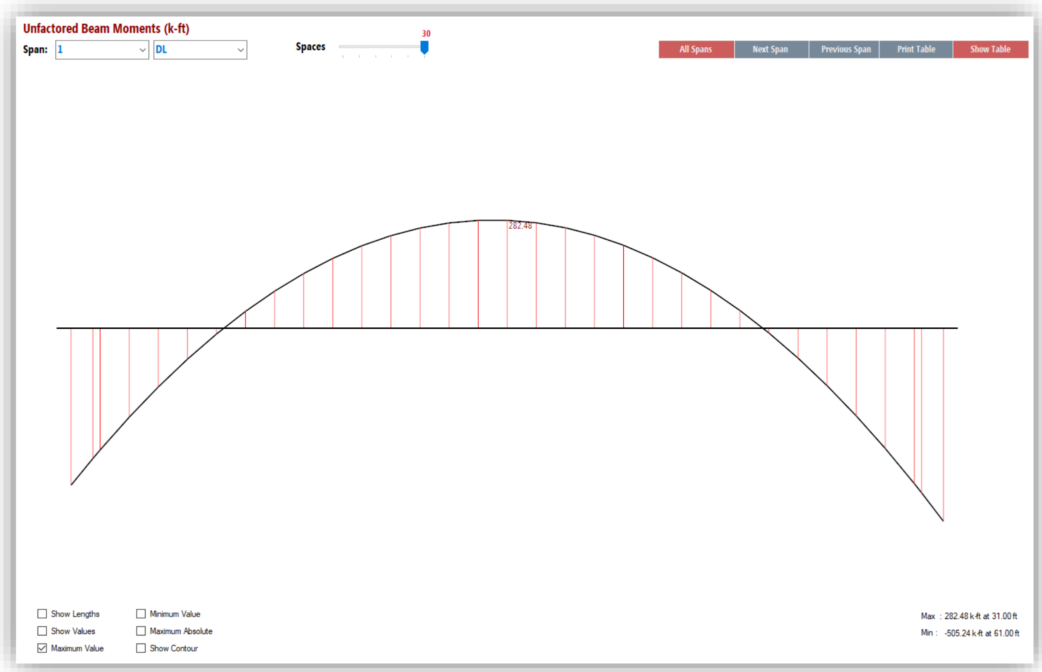



**Figure 4.8 Showing only the Maximum Value**

Similarly minimum value and maximum absolute value can also be observed. In order to see the contour or gradation of forces over the span check on Show Contour.

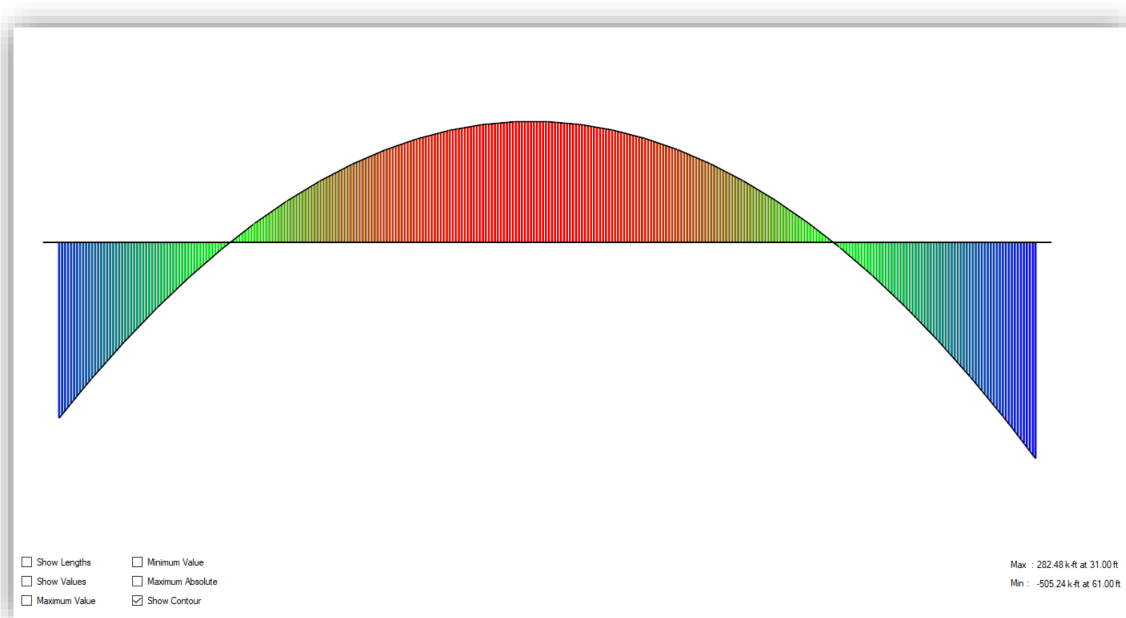



**Figure 4.9 Show Contour**

The gradation towards red shows the gradation to the maximum value and the gradation towards blue shows the gradation to the minimum.

In order to see the bending moment diagram for the entire structure, click on “All Spans” from the top right of the figure. The moment you click on it, the text changes to “Current Span”, to allow the user to switch between current span view and all span view.

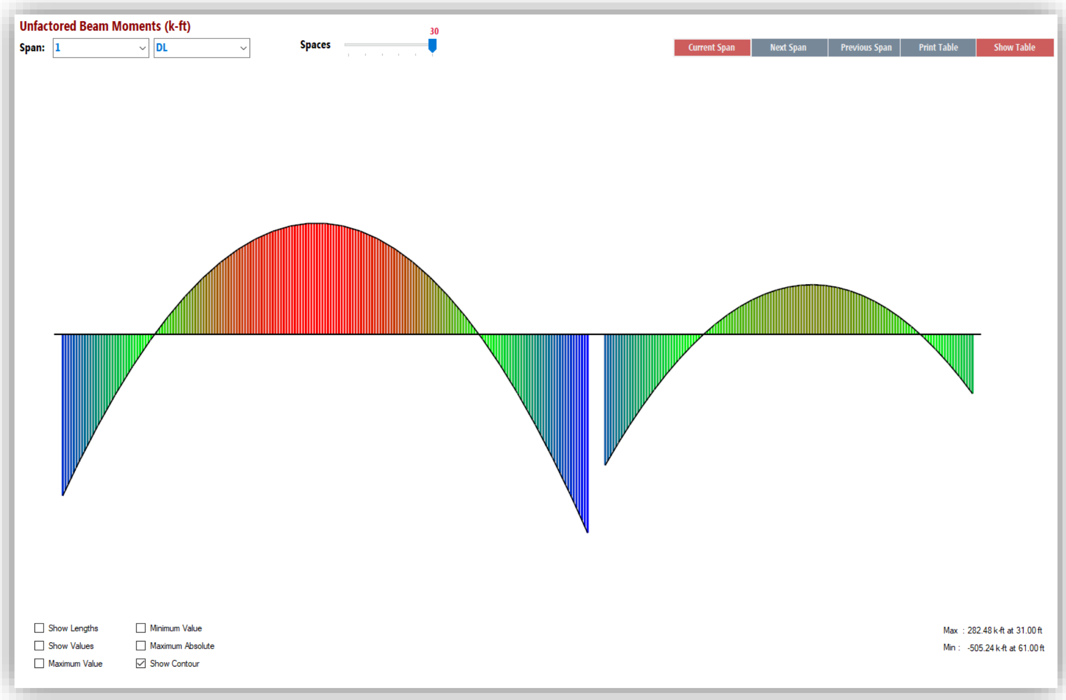



**Figure 4.10 Bending Moment for all the spans together**

The program shows the bending moments onto the faces of the column, that’s why there are gaps between the bending moments for the different spans.

To switch between different load cases, select the load case from the top of the screen.

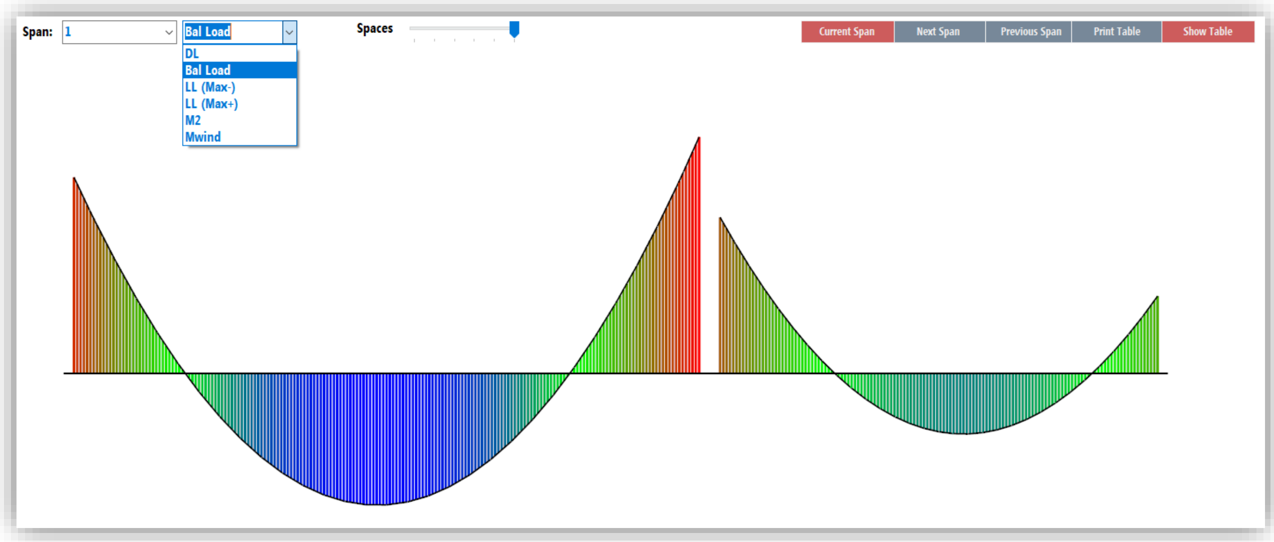



**Figure 4.11 Different Load Cases**

In order to return back to the tabular view, click on “Show Table” from top right of the window. Similarly, the diagrams for beam shear, stress and deflections can be seen from their respective result windows.

## 4-5 Unfactored Beam Shear

Unfactored dead load, live load, and wind shears (in kips) at all design points are tabulated in this screen which is shown in Figure 4-6. Positive shears at any design point are produced by upward loads to the ***left*** of the design point. The ***live load*** shears are shown for three loading conditions (see Figure 3.4 in the **THEORY Manual**):

- Live loads arranged to produce maximum negative moment at the left end of the span.
- Live loads arranged to produce maximum negative moment at the right end of the span.
- Live loads arranged to produce the maximum positive field moment in the span.

Unfactored Beam Shears (kips)

Span: 1 Spaces 12 Next Span Previous Span Print Table Show Diagram

| X(ft) | DL | LL(MaxML) | LL(MaxMR) | LL(MaxM+) | VWind |
|---|---|---|---|---|---|
| 1.00 | 47.78 | 21.37 | 20.91 | 21.37 | 0.00 |
| 2.50 | 45.32 | 20.29 | 19.83 | 20.29 | 0.00 |
| 6.00 | 39.56 | 17.77 | 17.31 | 17.77 | 0.00 |
| 11.00 | 31.33 | 14.17 | 13.71 | 14.17 | 0.00 |
| 16.00 | 23.10 | 10.57 | 10.11 | 10.57 | 0.00 |
| 21.00 | 14.88 | 6.97 | 6.51 | 6.97 | 0.00 |
| 26.00 | 6.65 | 3.37 | 2.91 | 3.37 | 0.00 |
| 31.00 | -1.58 | -0.23 | -0.69 | -0.23 | 0.00 |
| 36.00 | -9.80 | -3.83 | -4.29 | -3.83 | 0.00 |
| 41.00 | -18.03 | -7.43 | -7.89 | -7.43 | 0.00 |
| 46.00 | -26.26 | -11.03 | -11.49 | -11.03 | 0.00 |
| 51.00 | -34.48 | -14.63 | -15.09 | -14.63 | 0.00 |
| 56.00 | -42.71 | -18.23 | -18.69 | -18.23 | 0.00 |
| 59.50 | -48.47 | -20.75 | -21.21 | -20.75 | 0.00 |
| 61.00 | -50.94 | -21.83 | -22.29 | -21.83 | 0.00 |

**Figure 4.12 Unfactored Beam Shear**

## 4-6 Flexural Concrete Stresses

This screen, shown in Figure 4-7, shows the concrete flexural stresses at each design point produced by the ***unfactored*** service dead and live loads and by ***unfactored*** transfer loads.

The stresses are tabulated for the extreme top and bottom beam fibers, and for service loads, the most positive and most negative moments possible at each design point.

See Section 1-5 for a discussion of the design points, and **THEORY** Section 2.1 and the entire **THEORY** Chapter 10 for a discussion of how the flexural stresses are calculated. In **PT Designer** tensile stresses are positive in sign and compressive stresses are negative in sign.

### 4-6(a) Service Load Stresses

Service load flexural concrete stresses are produced by unfactored dead and live loads with the live loads arranged to produce maximum positive and negative moments at each design point.

In the example shown in Figure 4-7, at x = 31 feet the stress at the top beam fiber caused by the ***most negative moment*** (Max M-) which can occur at that design point is –0.369 ksi. The stress at the top beam fiber caused by the ***most positive moment*** (Max M+) which can occur at that design point is -0.530 ksi. Thus the ***range*** of flexural stresses which can occur at the top beam fiber at x = 31 feet is from -0.369 ksi in compression to -0.530 ksi in compression. Similarly at the ***bottom*** of the beam at x = 31 feet the flexural stresses can range between 0.259 ksi under the ***most negative moment*** possible and +0.607 ksi under the ***most positive moment*** possible. These ranges in flexural stresses are caused, of course, by the various arrangements of "skipped" live load. When the live load is not skipped the stresses for Max M- and Max M+ at each beam fiber will be equal. The ***tensile stresses*** shown in this screen are the controlling criteria for the **PT Designer** automatic design procedure for tendon force and profile.

### 4-6(b) Transfer Stresses

Transfer stresses are the concrete flexural stresses at each design point produced by the loads present immediately after stressing all of the post-tensioned tendons. This is commonly known as the ***transfer condition*** and the loads present at that time are known as the ***transfer loads***. **PT Designer** assumes that the transfer loads are the self weight of the concrete and the tendon balanced loads ***only*** (see **THEORY** Section 10.2). No live loads or superimposed dead loads are assumed present in the ***transfer condition***. Since no longterm prestress losses have occurred in this condition, tendon forces are at their initial maximum values. **PT Designer** assumes, in the calculation of transfer stresses, that initial tendon forces are 7/6 times effective forces.

The stresses are tabulated for the top and bottom beam fiber. Since there is no live load present in the transfer condition, there is only one moment possible at each design point and thus there is only one flexural stress possible at each beam fiber. See Section 1-5 for a discussion of the design points, and **THEORY** Section 2.1 and the entire **THEORY** Chapter 10 for a discussion of how the flexural stresses are calculated. In **PT Designer** tensile stresses are positive in sign and compressive stresses are negative in sign.

In the example shown in Figure 4-7, at x = 61 feet (the face of the right support) the flexural stress caused by transfer loads at the ***top*** beam fiber is -0.105 ksi. At the ***bottom*** of the beam at x = 61 feet the flexural stress is -0.852 ksi. The **PT Designer** automatic design procedure is based upon limiting ***service load***, not transfer stresses. The user must verify that the transfer stresses, as calculated by **PT Designer** and displayed in this screen, are acceptable.

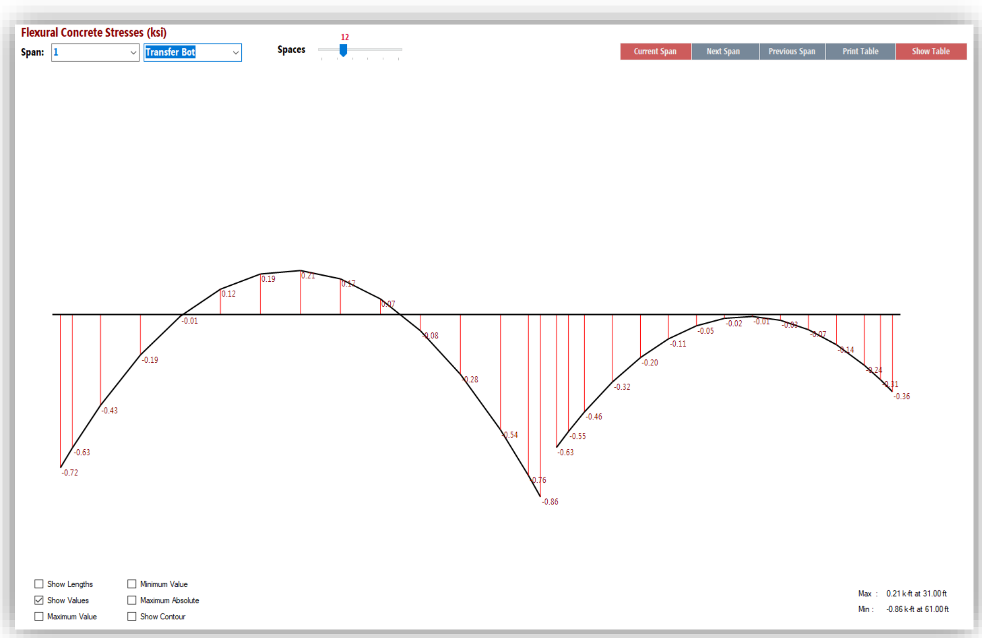



**Figure 4.13 Flexural Concrete Stresses**

## 4-7 Deflection and Cracking Moment Result Screen

This screen, shown in Figure 4-8, tabulates the deflections caused by unfactored dead, balanced, and live loads and the top and bottom fiber cracking moments at each ***design point*** in each span.

Deflections are shown in inches and are separated into dead load deflection (including balanced loads) and the maximum possible live load deflection at each design point. A positive deflection value is ***down*** (a sag), a negative value is ***up*** (a camber). There are no creep multipliers applied to the deflections shown in this screen.

Cracking moments (in kip-feet) for the top and bottom beam fibers are tabulated in the columns labeled Mcr Top and Mcr Bot. The cracking moment is the applied moment at the design point which, when combined with the effects of prestressing $(-\frac{F}{A} \pm \frac{M_{bal}}{S})$, produces a flexural tensile stress of $7.5\sqrt{f_c'}\ (= f_r)$ at the top or bottom fiber. Normally top fiber cracking moments are negative and bottom fiber cracking moments are positive, i.e., the cracking moment ***causes tension*** at the fiber in consideration. However, when the prestressing effects alone produce a flexural tensile stress greater than $7.5\sqrt{f_c'}$ the direction of the cracking moment will be reversed, i.e., the cracking moment ***causes compression*** at the point under consideration to ***reduce*** the tensile stress down to $7.5\sqrt{f_c'}$.

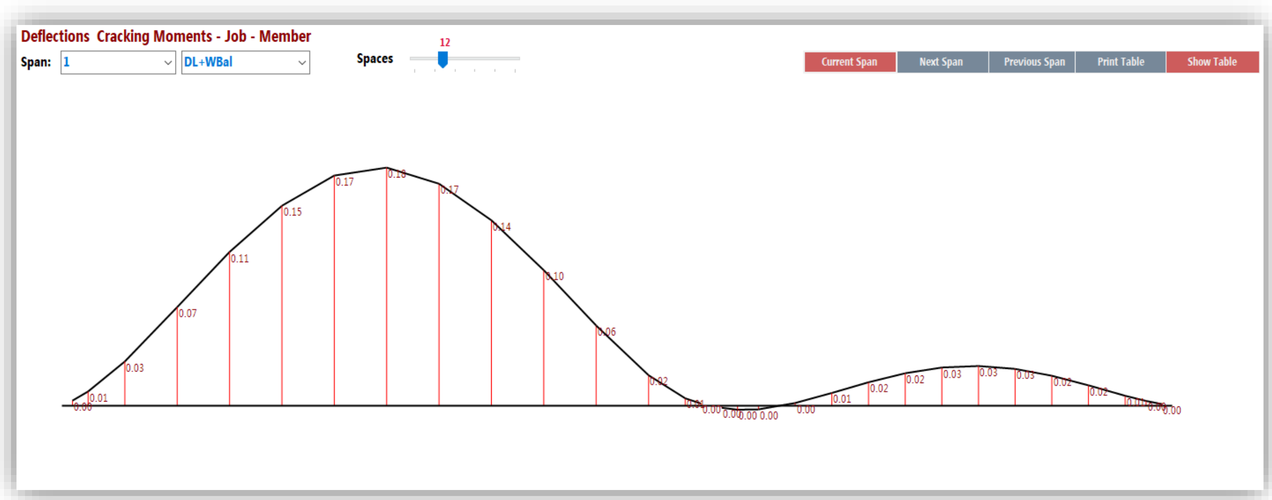



**Figure 4.14 Deflection and Cracking Moment**

## 4-8 Tendon Balanced Load & Concrete Dead Loads Result Screen

This screen, shown in Figure 4.9, tabulates all of the balanced, or equivalent, loads exerted by the tendon on the concrete in each span (see **THEORY** Chapter 6), and the concrete dead loads in each segment. The balanced loads are shown in Figure 4.10 for continuous tendons and Figures 4.11 and 4.12 for added tendons. Each load shown with a literal value in Figures 4.10 through 4.12 is tabulated in this screen if it is present in the Span.

There are three possible types of balanced loads, LINE loads in kips per foot, POINT loads in kips, and concentrated MOMENTS in kip-feet. The **Tendon Balanced Load Result Screen** shows the magnitude and location of each balanced load caused by either continuous or added tendons present in each span. See **THEORY** Chapter 6 for more information regarding balanced loads due to various continuous or added tendon configurations.

The bottom portion of the screen will identify the percentage of the concrete self weight that is balanced by the tendons in each span. Balance load percentages below 65%, between 65% and 125% and above 125% are listed as “Too Low”, “Nice” and “High (Be Careful)” respectively. These comments are not printed in the hard copy of the output and are presented solely to aid the user based upon the experience of the developers of **PT Designer**. Balanced loads are not a code issue but have historically been used by post-tensioning engineers as a tool to an efficient design. The appropriate percentage of balance load should be reviewed on a project specific basis and take into account shorter spans, cantilevers, high super imposed, etc.

A percentage is not calculated harped profiles. Since harped tendons are typically used to support point load(s), **PT Designer** does not know what percent of the applied load is the concrete self weight. The balance load percentage created by the harped tendon must be determined by the designer.

The concrete dead loads are tabulated in this screen, in each segment of each span, for easy comparison with the tendon balanced loads.

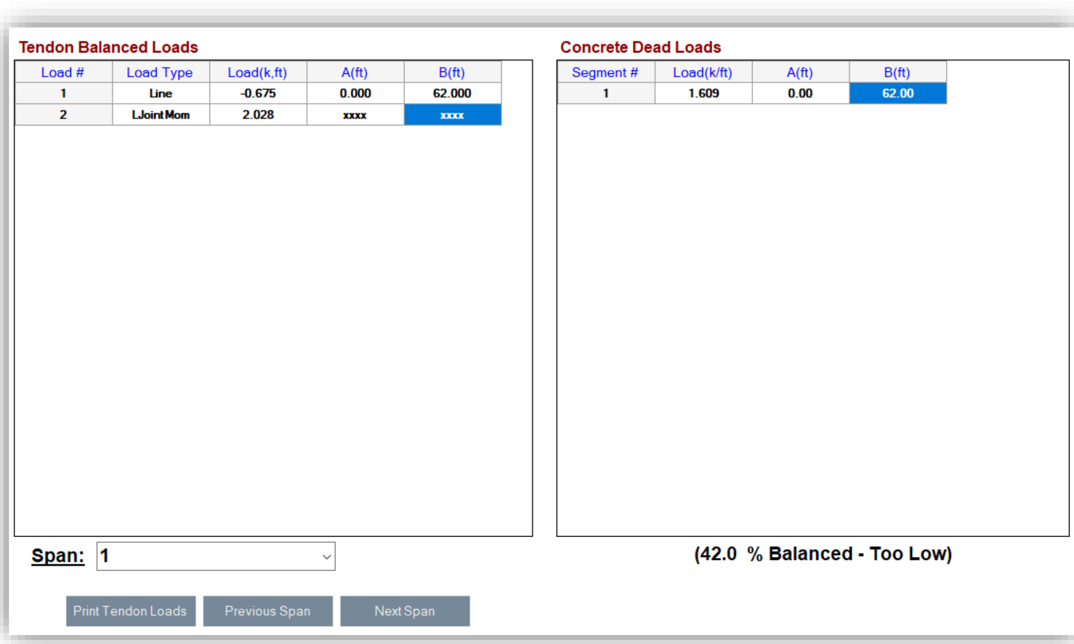



**Figure 4.15 Tendon Balanced Load & Concrete Dead Load**

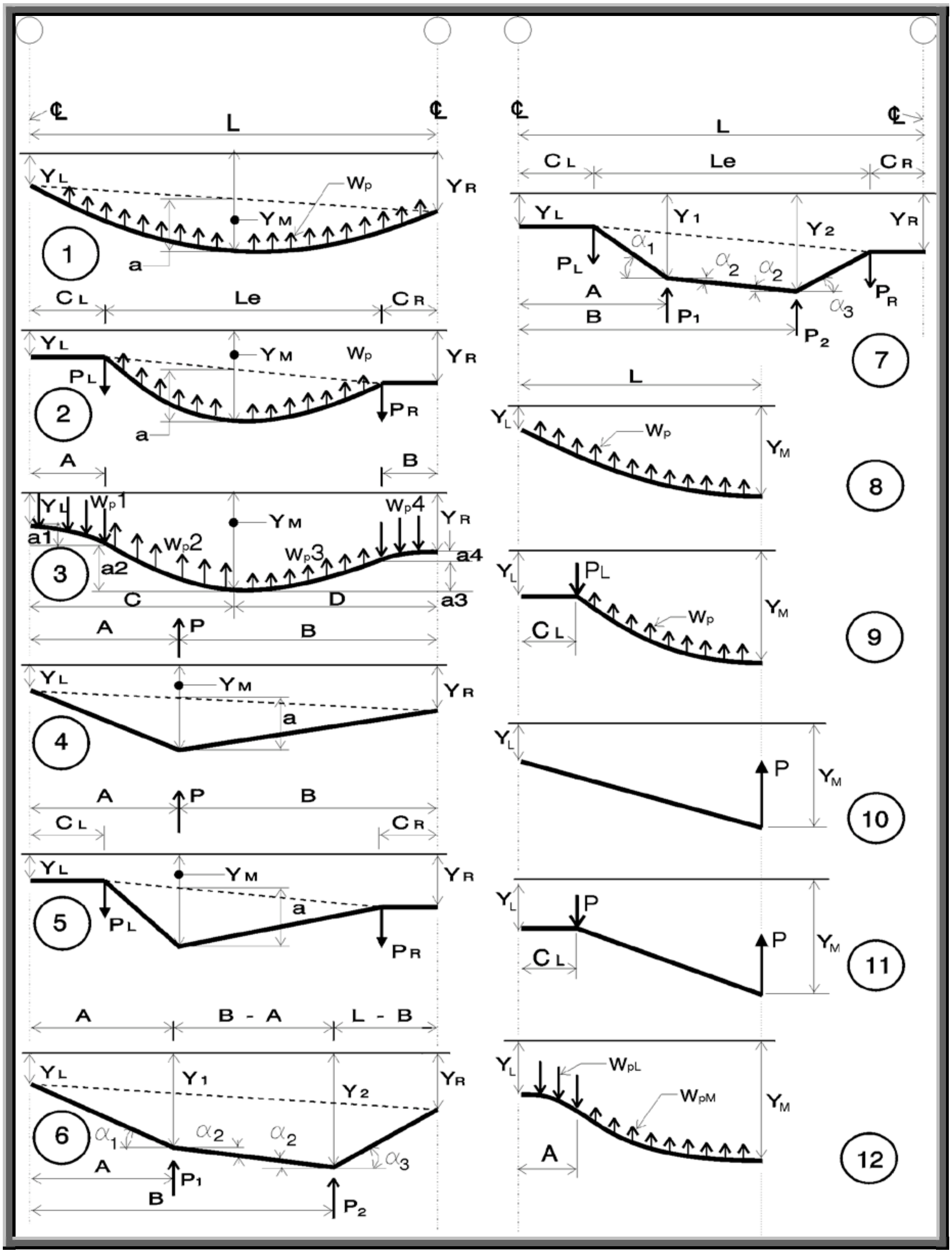


**Figure 4.16 Tendon Curvature Loads**

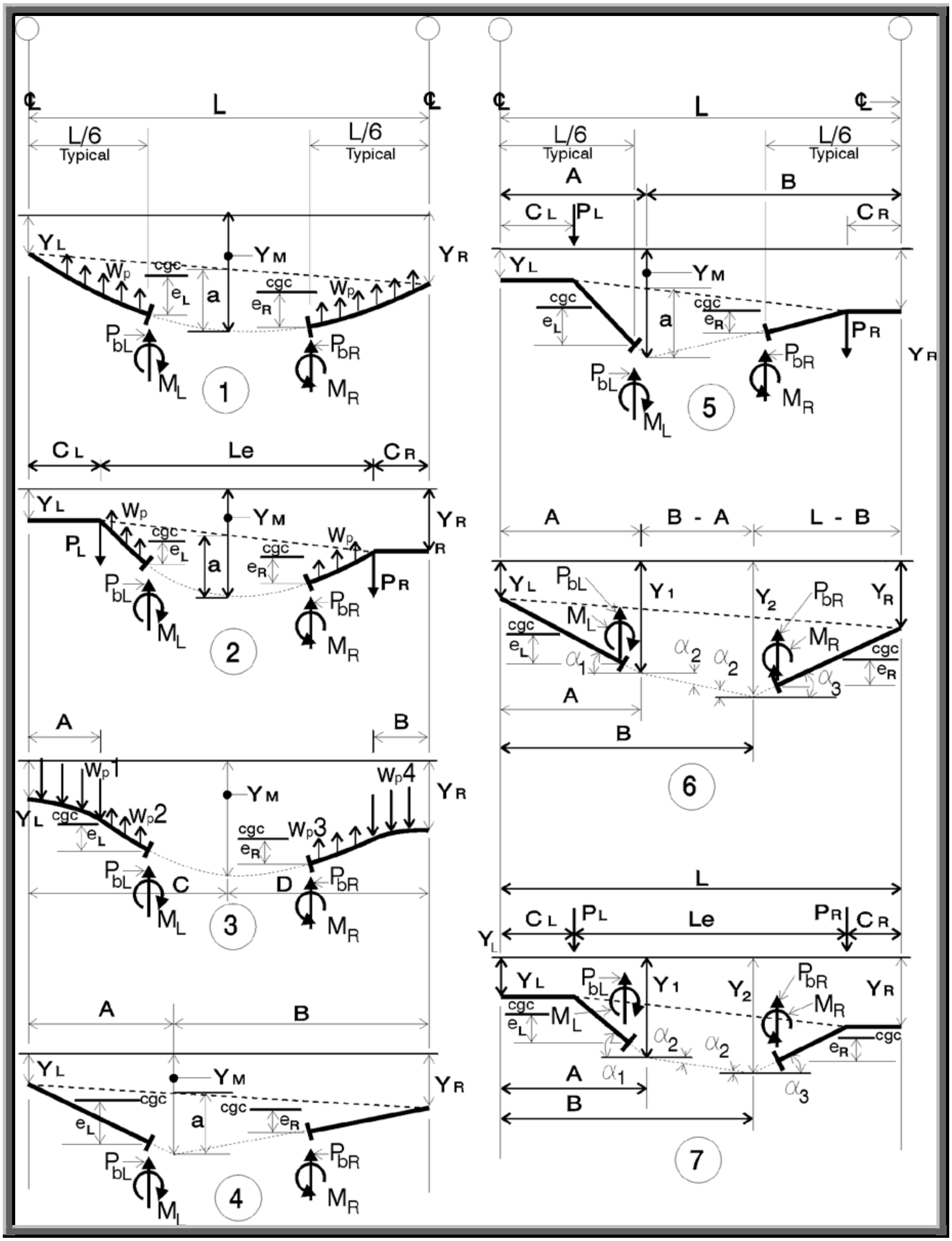



Figure 4.17 Added Tendon Curvature Loads - Spans

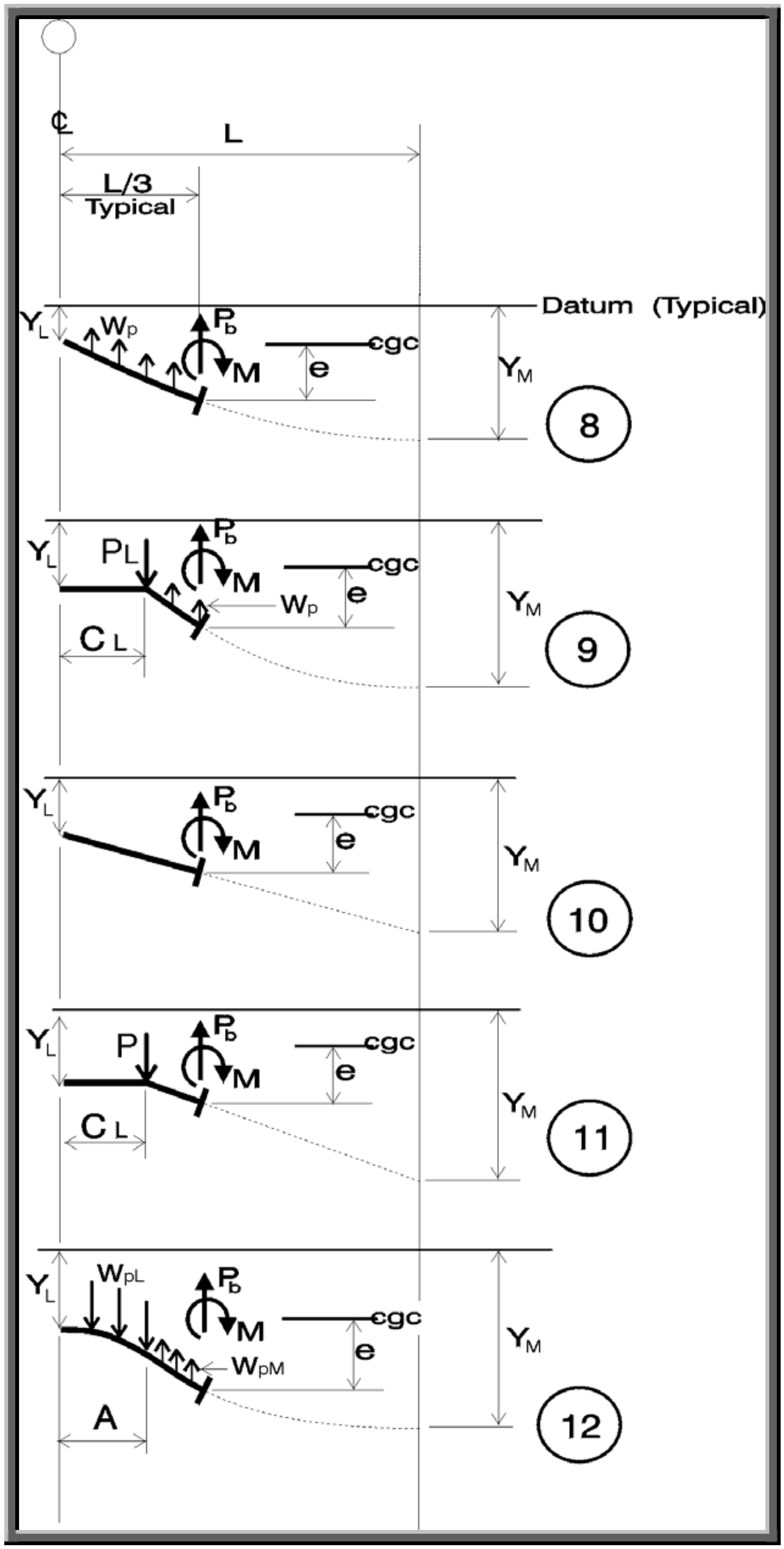



**Figure 4.18 Added Tendon Balanced Loads - Cantilevers**

## 4-9 Factored Load Rebar Result Screen

The area (in square inches) of unstressed reinforcing steel (rebar) required for ultimate strength at all design points for all three redistribution patterns can be reviewed in this screen which is shown in Figures 4.13 and 4.14. **PT Designer** calculates the required factored load rebar for three patterns of inelastic negative moment redistribution, %R=0%, 6.67% and 15% (see **THEORY** Chapter 12). Figure 4.13 shows the required rebar for a case with %R=0, Figure 4.14 shows it for %R=7.50%.

At each design point this screen shows the most positive (Max M+) and most negative (Max M-) design moments Mdesign possible and the required areas of tensile reinforcement (As) and compression reinforcement (A's) at the top and bottom of the beam, and the ultimate tendon stress fps used in the strength calculation at that design point. Note that Mdesign includes the factored dead and live load moments plus the secondary moment. The calculated areas of rebar are based upon the concrete covers and bar size entered in the **Material Properties**.

An asterisk (*) following any moment value in this screen indicates that the flexural design at this point either is or would be controlled by the cracking moment requirements of ACI 318-08 Section 18.8.2 (i.e., $1.2M_{fr} > M_{design}$ - see **THEORY** Section 9.1). If the cracking moment requirement applies (see Section 2.2.4.b), the moment with an asterisk is $1.2M_{fr}$. If the cracking moment requirement is waived, the moment shown with an asterisk is $M_{design}$ and the asterisk merely indicates that cracking moment requirements would have controlled at this point had they been applied. These conditions are indicated by a message at the bottom of this screen. In the example shown, the cracking moment requirement is waived and the asterisks indicate where it would have controlled had it been required.

To be code-conformant, the rebar selected for the final design must, at all design points, be equal to or greater than that shown for ***any one of the three redistribution patterns***. You must select the most favorable of the ***three redistribution patterns***, and provide at least that amount of rebar at each design point. You cannot select the most favorable rebar quantity of the three redistribution patterns ***at each point***. The final rebar selected must be consistent with one of the three patterns ***in its entirety***.

Top and bottom bar cutoff points can be determined from this screen by extending bars a development length past the last design point where they are no longer required.

Factored Load Rebar - Two-Span Example - Beam

**Span** 1 **% Redistribution =** 0

| X (ft) | Max M+ (ft-kips) | Max M- (ft-kips) | As Top (in2) | As Bot (in2) | A's Top (in2) | A's Bot (in2) | fps (ksi) |
|---|---|---|---|---|---|---|---|
| 1.00 | -242.81* | -557.27* | 1.889 | 0.000 | 0.000 | 0.000 | 203.1 |
| 2.50 | -164.79* | -428.14* | 1.136 | 0.000 | 0.000 | 0.000 | 201.4 |
| 6.00 | 0.35* | -153.81* | 0.000 | 0.000 | 0.000 | 0.000 | 197.8 |
| 11.00 | 195.23* | 172.56 | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 16.00 | 424.21 | 339.47* | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 21.00 | 594.05 | 440.16* | 0.000 | 0.708 | 0.000 | 0.000 | 234.0 |
| 26.00 | 689.18 | 490.20 | 0.000 | 1.209 | 0.000 | 0.000 | 234.0 |
| 31.00 | 707.22 | 491.96 | 0.000 | 1.326 | 0.000 | 0.000 | 234.0 |
| 36.00 | 648.19 | 445.43* | 0.000 | 1.058 | 0.000 | 0.000 | 234.0 |
| 41.00 | 512.07 | 350.63* | 0.000 | 0.409 | 0.000 | 0.000 | 234.0 |
| 46.00 | 298.87* | 207.54* | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 51.00 | 39.73* | -14.97* | 0.000 | 0.000 | 0.000 | 0.000 | 197.5 |
| 56.00 | -196.21* | -386.03* | 0.653 | 0.000 | 0.000 | 0.000 | 202.6 |
| 59.50 | -390.10* | -691.64* | 2.388 | 0.000 | 0.000 | 0.000 | 206.8 |
| 61.00 | -480.44* | -834.18* | 3.227 | 0.000 | 0.000 | 0.000 | 208.8 |

Next Span
Previous Span
Change %R
Next %R
Previous %R
Print This Span

**Figure 4.19 Factored Load Rebar for 0% Redistribution**

Factored Load Rebar - Two-Span Example - Beam

**Span** 1 **% Redistribution =** 7.5

| X (ft) | Max M+ (ft-kips) | Max M- (ft-kips) | As Top (in2) | As Bot (in2) | A's Top (in2) | A's Bot (in2) | fps (ksi) |
|---|---|---|---|---|---|---|---|
| 1.00 | -242.81* | -515.47* | 1.567 | 0.000 | 0.000 | 0.000 | 203.1 |
| 2.50 | -164.79* | -386.60* | 0.825 | 0.000 | 0.000 | 0.000 | 201.4 |
| 6.00 | 0.35* | -112.87* | 0.000 | 0.000 | 0.000 | 0.000 | 197.8 |
| 11.00 | 212.65* | 189.17 | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 16.00 | 461.09 | 339.47* | 0.000 | 0.092 | 0.000 | 0.000 | 234.0 |
| 21.00 | 632.45 | 440.16* | 0.000 | 0.968 | 0.000 | 0.000 | 234.0 |
| 26.00 | 726.73 | 490.20 | 0.000 | 1.463 | 0.000 | 0.000 | 234.0 |
| 31.00 | 743.92 | 491.96 | 0.000 | 1.575 | 0.000 | 0.000 | 234.0 |
| 36.00 | 684.04 | 445.43* | 0.000 | 1.301 | 0.000 | 0.000 | 234.0 |
| 41.00 | 547.07 | 350.63* | 0.000 | 0.645 | 0.000 | 0.000 | 234.0 |
| 46.00 | 333.02* | 207.54* | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 51.00 | 41.89* | 16.18* | 0.000 | 0.000 | 0.000 | 0.000 | 234.0 |
| 56.00 | -196.21* | -326.32* | 0.212 | 0.000 | 0.000 | 0.000 | 202.6 |
| 59.50 | -390.10* | -629.93* | 1.905 | 0.000 | 0.000 | 0.000 | 206.8 |
| 61.00 | -480.44* | -771.61* | 2.724 | 0.000 | 0.000 | 0.000 | 208.8 |

Next Span
Previous Span
Change %R
Next %R
Previous %R
Print This Span

**Figure 4.20 Factored Load Rebar for 7.5% Redistribution**

## 4-10 DL+LL/4 Rebar Result Screen

This screen, shown in Figure 4.15 for a case with %R=15%, is similar to **the Factored Load Rebar Result Screen** described in Section 4-9 with the following exceptions:

- Moments are those produced by unfactored dead loads and 25% of the ***unfactored*** and

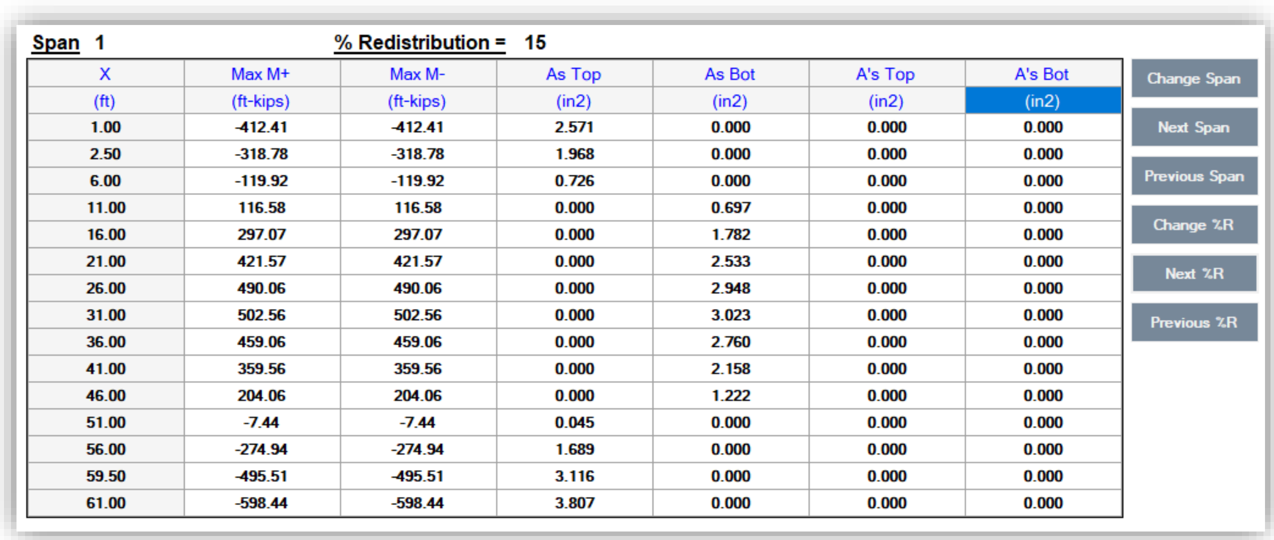
***unreduced*** live loads.

- Rebar areas satisfy the requirements of this UBC Code Section, i.e., prestressed reinforcement is ignored and $\phi$ = 1.0 (see **THEORY** Section 9.1).

- Redistribution patterns are for %R=0%, 10%, and 15%.

- Cracking moment requirements of ACI 318, Section 18.8.2 do not apply.

This screen can only be accessed if this Code requirement has been applied in the **Material Properties**.



Span 1 % Redistribution = 15

| X | Max M+ | Max M- | As Top | As Bot | A's Top | A's Bot |
|---|---|---|---|---|---|---|
| (ft) | (ft-kips) | (ft-kips) | (in2) | (in2) | (in2) | (in2) |
| 1.00 | -412.41 | -412.41 | 2.571 | 0.000 | 0.000 | 0.000 |
| 2.50 | -318.78 | -318.78 | 1.968 | 0.000 | 0.000 | 0.000 |
| 6.00 | -119.92 | -119.92 | 0.726 | 0.000 | 0.000 | 0.000 |
| 11.00 | 116.58 | 116.58 | 0.000 | 0.697 | 0.000 | 0.000 |
| 16.00 | 297.07 | 297.07 | 0.000 | 1.782 | 0.000 | 0.000 |
| 21.00 | 421.57 | 421.57 | 0.000 | 2.533 | 0.000 | 0.000 |
| 26.00 | 490.06 | 490.06 | 0.000 | 2.948 | 0.000 | 0.000 |
| 31.00 | 502.56 | 502.56 | 0.000 | 3.023 | 0.000 | 0.000 |
| 36.00 | 459.06 | 459.06 | 0.000 | 2.760 | 0.000 | 0.000 |
| 41.00 | 359.56 | 359.56 | 0.000 | 2.158 | 0.000 | 0.000 |
| 46.00 | 204.06 | 204.06 | 0.000 | 1.222 | 0.000 | 0.000 |
| 51.00 | -7.44 | -7.44 | 0.045 | 0.000 | 0.000 | 0.000 |
| 56.00 | -274.94 | -274.94 | 1.689 | 0.000 | 0.000 | 0.000 |
| 59.50 | -495.51 | -495.51 | 3.116 | 0.000 | 0.000 | 0.000 |
| 61.00 | -598.44 | -598.44 | 3.807 | 0.000 | 0.000 | 0.000 |



**Figure 4.21 DL+0.25LL Rebar for 15% Redistribution**

## 4-11 Beam Shear Design Result Screen

This screen, shown in Figure 4.16, reviews the major parameters in the beam shear design. The **Beam Shear Design Result Screen** is normally applicable only for beams and girders, however it can also be accessed for both one and two-way slabs where it typically does not control.

Parameters shown at each design point are the three concrete shear capacities $V_{cn}$, $V_{cw}$, and $V_{ci}$, (from which the controlling concrete shear capacity $V_c$ is selected), the controlling area of shear reinforcement $A_v$ (expressed in square inches of vertical web reinforcement per running foot of beam), the required *two-legged* stirrup spacing based upon the stirrup size entered in the **Material Properties**, and finally a CODE which identifies the ACI Code Section which controlled the shear design. The CODES are:

- **N/R** Shear reinforcement is not required in conformance with ACI 318 Section 11.4.5.1. In this case **PT Designer** will suggest a stirrup spacing of 24".

- **STR** Shear reinforcement is controlled by strength requirements (See **THEORY**, Section 14.1(b), Equation 14-8).
- **MIN** Shear reinforcement is controlled by one of the two minimum shear reinforcement requirements in ACI 318 Equations 11-13 and 11-14.
- **24MAX** Shear reinforcement is controlled by the 24" maximum stirrup spacing requirement of ACI 318 Section 11.4.5.
- **12MAX** Shear reinforcement is controlled by the 12" maximum stirrup spacing requirement of ACI 318 Section 11.4.5.
- **3/4H** Shear reinforcement is controlled by the 3/4h maximum stirrup spacing requirement of ACI 318 Section 11.4.5.
- **3/8H** Shear reinforcement is controlled by the 3/8h maximum stirrup spacing requirement of ACI 318 Section 11.4.5.

Beam Shear Design

Span 1

| X | Vcn | Vcw | Vci | Av | #3 @ | Code |
|---|---|---|---|---|---|---|
| (ft) | (kips) | (kips) | (kips) | (in2/ft) | (in) | ---- |
| 1.00 | 91.96 | 136.27 | 111.00 | 0.151 | 17.5 | STR |
| 2.50 | 96.22 | 136.27 | 119.19 | 0.046 | 24.0 | 24MAX |
| 6.00 | 121.32 | 136.27 | 157.88 | 0.040 | 24.0 | 24MAX |
| 11.00 | 152.74 | 136.27 | 531.27 | 0.040 | 24.0 | 24MAX |
| 16.00 | 127.26 | 136.27 | 72.26 | 0.000 | 24.0 | N/R |
| 21.00 | 68.64 | 144.95 | 55.23 | 0.039 | 24.0 | 24MAX |
| 26.00 | 67.78 | 151.18 | 57.61 | 0.000 | 24.0 | N/R |
| 31.00 | 67.88 | 151.41 | 57.70 | 0.000 | 24.0 | N/R |
| 36.00 | 65.30 | 145.65 | 55.50 | 0.000 | 24.0 | N/R |
| 41.00 | 88.30 | 136.27 | 53.65 | 0.040 | 24.0 | 24MAX |
| 46.00 | 152.74 | 136.27 | 122.96 | 0.000 | 24.0 | N/R |
| 51.00 | 152.74 | 136.27 | 278.27 | 0.040 | 24.0 | 24MAX |
| 56.00 | 108.87 | 136.27 | 128.05 | 0.040 | 24.0 | 24MAX |
| 59.50 | 97.50 | 136.27 | 106.63 | 0.222 | 11.9 | STR |
| 61.00 | 100.39 | 143.54 | 102.56 | 0.284 | 9.3 | STR |

Next Span

Previous Span

Print This Span

**Figure 4.22 Beam Shear Design**

## 4-12 Punching Shear Stress Result Screen

This screen can be accessed only for two-way systems. It shows the currently- calculated stresses acting on the critical punching shear section at each joint, caused by vertical shear and moment transfer from slab to column. This screen is shown in Figures 4.17 and 4.18. Stresses are in ksi and a ***positive*** stress acts ***down on the critical section***.

Information included in this screen, at each joint, includes:

- *Ac* - Area of the critical section.
- *Jc* - Polar moment of inertia of the critical section.
- *Ex* - Dimension from the centerline of the column to the centroid of the critical section (right is positive, left is negative).
- *xL* - Dimension from the centroid of the critical section to the left face of the critical section.
- *xR* - Dimension from the centroid of the critical section to the right face of the critical section.
- *fL* – The maximum combined shear stress on the left face of the critical section.
- *fR* – The maximum combined shear stress on the right face of the critical section.
- *Allow* – the allowable shear stress on the critical section.
- *d/Bo* – The ratio of the average d dimension for the critical section to the perimeter of the critical section, used in calculating the allowable shear stress.

If capitals are used, their plan dimensions and total thickness at each joint will be indicated on a table to the left of the critical section information. This information will match the values input per section 3.6b or will be generated by **PT Designer** when the "Design Capitals" function was activated. In addition, the two critical shear planes will be shown on the right of the screen. The critical shear plane diagram is not drawn to match the specific design information, but provided to aid the designer. See **THEORY** Section 14.2(b) for a discussion of the allowable punching shear stress. Note that the allowable shear stress ***includes*** the appropriate $\phi$ factor depending upon which Code is specified (1997 UBC or ACI 318-08+). If an applied stress exceeds the allowable stress at any joint, **PT Designer** will alert the user to this condition with a warning banner at the bottom of the screen.

**PT Designer** supports ***two*** critical sections as shown in Figure 4-20. Critical Section #1, just outside the column perimeter, is present in ***every*** two-way run, and values for Critical Section #1 always appear in the **Punching Shear Stress Result Screen**. Values for Critical Section #2 are accessible only if that critical section is present in the run. If Critical Section #2 is not present in a particular run, that command buttons will be disabled. Figure 4.17 shows the review screen for Critical Section #1; Figure 4-18 shows it for Critical Section #2.

For the example shown in Figures 4-17 and 4-18, the maximum applied punching shear stress at Joints 2 and 5 of Critical Section #2 (Figure 4-18) is 0.205 ksi on the right face at Joint 3. The allowable stress at these joints is 0.224 ksi so the design is adequate for punching shear at this critical section.

In **PT Designer**, if any applied stress exceeds the allowable stress, not only will the warning banner appear but the "**Design Caps**" command button will be enabled.

Press this button and **PT Designer** will design the capitals first for Critical Section #1 and then for Critical Section #2 using a patterned process beginning with the two exterior joints and working inward. This is a rigorous process and may take a considerable amount of time. Since a change in the capital dimensions affects the stiffness of the frame members, along with the shear stresses, both the flexural design and the shear design must be addressed in the process. To accomplish this, **PT Designer** goes through an iterative process and recalculates the stiffness matrix, moment distribution, etc. after every incremental change in a capital. After the automatic design procedure is complete the user receives a message to verify that shear stresses are satisfied in both critical sections. It is possible that after first determining the capital depth in Critical Section #1 at each joint and then determining the capital plan dimensions in Critical Section #2 at each joint that the frame stiffness has been modified enough that one of the Critical Sections at one or more joints no longer satisfies the allowable stresses. One more iteration (click of the "**Design Caps**" button) would then be necessary.

The allowable punching shear stress (Figures 4.17 and 4.18) is a function of the precompression from the strands. After **PT Designer** determines the dimensions for the capitals, there may be a reduction in the number of strands to satisfy the allowable flexural stresses. This often occurs since the additional section modulus of the capital can reduce the flexural stresses over the columns which may allow a reduction in the tendon force. When any significant change to the precompression force occurs, it is recommended to verify the punching shear stress has not been exceeded.

If slab bands are used in the design, **PT Designer** assumes the slab band only occurs between the column faces in the span under design. For the slab band to extend into the adjacent spans to provide punching shear resistance, a column capital size will need to be inputted at each column. **PT Designer** will only use the capital in the adjacent spans where the slab band does not occur. Input the capital assuming it will occur on all sides of the column and **PT Designer** will automatically modify the capital dimensions to occur on the side of the column without the slab band. Without any capital geometry being entered, **PT Designer** calculates punching shear resistance on the adjacent spans (away from the slab band) based upon the slab thickness only.

One final important point regarding the automated capital design is that it needs to be run prior to adding any beams (or "slab-bands") in the model. This is because **PT Designer** automatically eliminates capitals where beams exist. Therefore, as the program attempts to add a capital it will be removed later in another routine. This could cause an “endless loop” or some very strange results in many cases. Beams, or slab- bands, should be added to the model *after* first designing the column capitals.

### 4-13 Variable Prestress Force Option

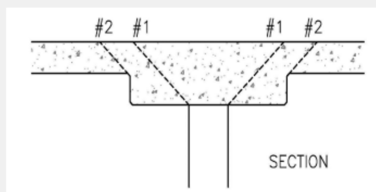
To initiate the Variable Prestress Force procedure for the current run, select this Option. See Chapter 5 for a detailed description of how to use the Variable Prestress Force Option.

### 4-14 Controlling Rebar Option

Press this button to review the controlling joint and span flexural ***tension*** rebar for each redistribution case. This is intended to provide a quick look at the mild tension reinforcing required. To view the complete controlling rebar requirements, including compression reinforcement and bar lengths, review the printed output.

Caps

| | W1 | W2 | h |
|---|---|---|---|
| Joint | (in.) | (in.) | (in.) |
| 1 | 72.00 | 72.00 | 24.00 |
| 2 | 72.00 | 72.00 | 24.00 |
| 3 | 72.00 | 72.00 | 24.00 |
| 4 | 72.00 | 72.00 | 24.00 |
| 5 | 72.00 | 72.00 | 24.00 |
| 6 | 72.00 | 72.00 | 24.00 |
| 7 | 72.00 | 72.00 | 24.00 |
| 8 | 72.00 | 72.00 | 24.00 |




Critical Section # 1

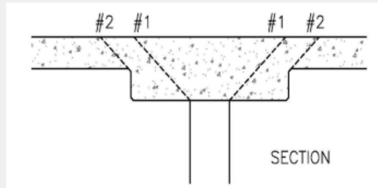
| | Ac | Jc | Ex | xL | xR | fL | fR | Allow | d/Bo | Vu(@Vmax) | Mu(@Vmax) | Vu(@Mmax) | Mu(@Mmax) |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Joint | (in.^2) | (in.^4) | (in.) | (in.) | (in.) | (ksi) | (ksi) | (ksi) | (in./in.) | (kips) | (k-ft) | (kips) | (k-ft) |
| 1 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.071 | 0.070 | 0.214 | 0.136 | 263.1 | -4.8 | 263.1 | -4.8 |
| 2 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.098 | 0.104 | 0.212 | 0.136 | 377.6 | 34.1 | 377.6 | 34.1 |
| 3 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.118 | 0.116 | 0.212 | 0.136 | 435.9 | -10.2 | 435.9 | -10.2 |
| 4 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.114 | 0.114 | 0.212 | 0.136 | 426.1 | 3.4 | 426.1 | 3.4 |
| 5 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.115 | 0.114 | 0.212 | 0.136 | 427.7 | -2.3 | 427.7 | -2.3 |
| 6 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.115 | 0.116 | 0.212 | 0.136 | 431.6 | 5.0 | 431.6 | 5.0 |
| 7 | 3,735 | 948,578 | 0.00 | 18.25 | 18.25 | 0.111 | 0.108 | 0.212 | 0.136 | 408.6 | -16.0 | 408.6 | -16.0 |
| 8 | 1,736 | 130,202 | -10.44 | 6.16 | 17.44 | 0.060 | 0.060 | 0.212 | 0.212 | 105.0 | -40.8 | 105.0 | -40.8 |

Section #1 Section #2 Print Design Caps

**Figure 4.23 Punching Shear Analysis – Section 1**

**Caps**

| | W1 | W2 | h |
|---|---|---|---|
| Joint | (in.) | (in.) | (in.) |
| 1 | 72.00 | 72.00 | 24.00 |
| 2 | 72.00 | 72.00 | 24.00 |
| 3 | 72.00 | 72.00 | 24.00 |
| 4 | 72.00 | 72.00 | 24.00 |
| 5 | 72.00 | 72.00 | 24.00 |
| 6 | 72.00 | 72.00 | 24.00 |
| 7 | 72.00 | 72.00 | 24.00 |
| 8 | 72.00 | 72.00 | 24.00 |




**Critical Section #** 2

| | Ac | Jc | Ex | xL | xR | fL | fR | Allow | d/Bo | Vu(@Vmax) | Mu(@Vmax) | Vu(@Mmax) | Mu(@Mmax) |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Joint | (in.^2) | (in.^4) | (in.) | (in.) | (in.) | (ksi) | (ksi) | (ksi) | (in./in.) | (kips) | (k-ft) | (kips) | (k-ft) |
| 1 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.084 | 0.084 | 0.170 | 0.029 | 263.1 | -4.8 | 263.1 | -4.8 |
| 2 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.119 | 0.122 | 0.172 | 0.029 | 377.6 | 34.1 | 377.6 | 34.1 |
| 3 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.140 | 0.139 | 0.172 | 0.029 | 435.9 | -10.2 | 435.9 | -10.2 |
| 4 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.136 | 0.136 | 0.172 | 0.029 | 426.1 | 3.4 | 426.1 | 3.4 |
| 5 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.137 | 0.136 | 0.172 | 0.029 | 427.7 | -2.3 | 427.7 | -2.3 |
| 6 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.137 | 0.138 | 0.172 | 0.029 | 431.6 | 5.0 | 431.6 | 5.0 |
| 7 | 3,133 | 3,489,399 | 0.00 | 40.80 | 40.80 | 0.131 | 0.130 | 0.172 | 0.029 | 408.6 | -16.0 | 408.6 | -16.0 |
| 8 | 1,701 | 423,201 | -27.91 | 12.89 | 34.91 | 0.062 | 0.062 | 0.192 | 0.054 | 105.0 | -40.8 | 105.0 | -40.8 |

Section #1 Section #2 Print Design Caps

**Figure 4.24 Punching Shear Analysis – Section 2**

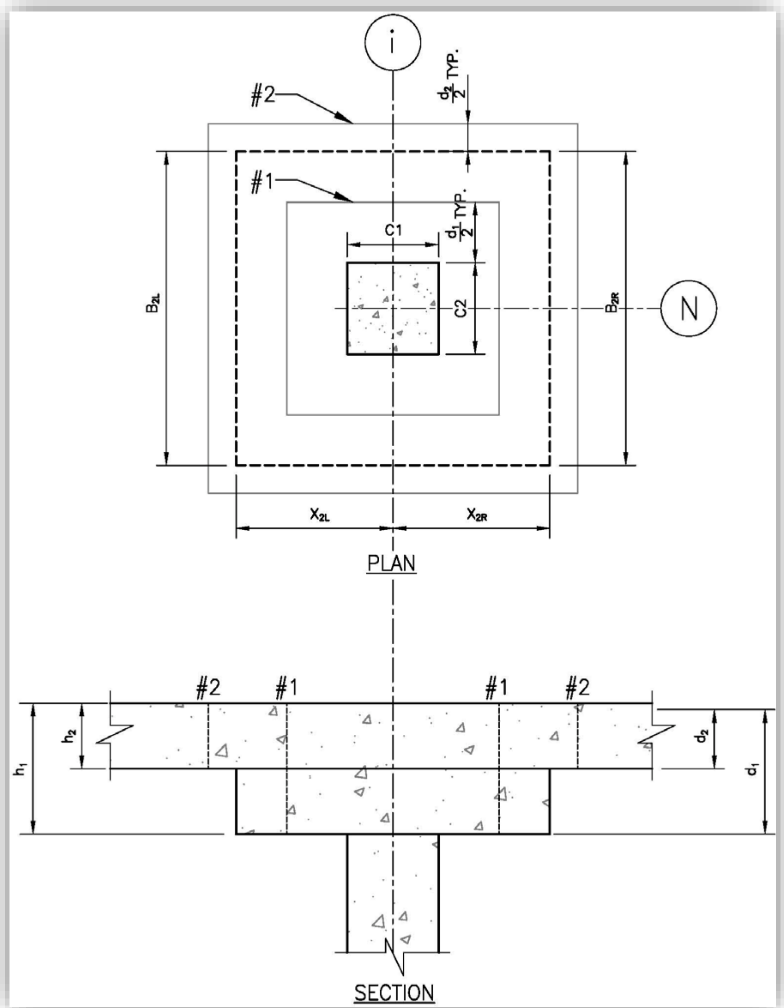



**Figure 4.25 Critical Punching Shear Sections**

Controlling Rebar - Job - Member

**% Redistribution = Ult: 7.5% ; DL + 0.25LL = 10%**

| Joint | Top Rebar | Span | Total (B) Rebar | Added (B) Rebar to ... |
|---|---|---|---|---|
| 1 | 9-#5 | 1 | 1-#4 (@ 99.9 in. o/c) | - |
| 2 | 9-#5 | 2 | 18-#4 (@ 15.3 in. o/c) | 7-#4 (@ 39.4 in. o/c) |
| 3 | 10-#5 | 3 | 11-#4 (@ 25.1 in. o/c) | - |
| 4 | 10-#5 | 4 | 13-#4 (@ 21.2 in. o/c) | 2-#4 (@ 99.9 in. o/c) |
| 5 | 10-#5 | 5 | 12-#4 (@ 23.0 in. o/c) | 1-#4 (@ 99.9 in. o/c) |
| 6 | 10-#5 | 6 | 15-#4 (@ 18.4 in. o/c) | 4-#4 (@ 69.0 in. o/c) |
| 7 | 9-#5 | 7 | 1-#4 (@ 99.9 in. o/c) | - |
| 8 | 9-#5 | | | |

Next %R

Previous %R

Print This Form

**Figure 4.26 Controlling Rebar for 2-Way Slabs**

# CHAPTER 5 THE VARIABLE PRESTRESS FORCE OPTION

The **Variable Prestress Force** Option is selected from the **RESULT MENU**. Following are definitions which are used in the Variable Prestress Force mode:

- ***Tendon*** – a complete assembly consisting of a number of strands, sheathing, and anchorages.

- ***Strand*** – the high-strength prestressing steel which extends between anchorages - the element of tendon which is elongated and anchored to provide the necessary prestressing force. Strands are most commonly supplied as a seven-wire cable with an ultimate strength of 270 ksi. Properties of the strand are input in the General Input Data Screen (Section 3.3, Items #11 and 16). Note in the Variable Prestress Force Option, a tendon is made up of a number of strands.

- ***Through Tendon*** - a tendon which is continuous from the left end of the frame to the right end of the frame. Each **PT Designer** frame has one through tendon containing any number of strands, including zero.

- ***Added Tendon*** - a tendon which is in only part of the frame. Any number of added tendons may be used in the frame.

- ***Stressing-End Anchor*** - an anchor point where a tendon is stressed.

- ***Dead-End Anchor*** - an anchor point where no stressing occurs.

Input for the Variable Prestress Force Option is managed through four screens which appear after the Option is selected from the **MAIN MENU**. These screens are described as follows in Sections 5.1 through 5.4:

## 5-1 The Tendon Stressing Patterns Input Screen

**PT Designer** supports twelve stressing patterns including every possible stressing arrangement for through tendons with up to two intermediate stressing construction joints. The stressing patterns are described graphically in the **Tendon Stressing Patterns Input Screen**, shown in Figure 5-1.

In Figure 5-1 the horizontal line in each of the 12 patterns represents the full length of the through tendon, i.e., the full length of the frame from left to right ends. A solid black arrow represents a stressing anchor point and the direction in which the arrow points indicates the direction in which the tendon is stressed (pulled) at that point. A short vertical bar at the left or right end of the tendon indicates a dead-end anchor point. The 12 patterns represent every possible combination of stressing for a through tendon with up to two intermediate stressing points. Patterns 1 through 3 have no intermediate stressing points (they are stressed only at the extreme ends of the frame). Patterns 4 through 8 have one intermediate stressing point. Patterns 9 through 12 have two intermediate stressing points.

The intermediate stressing points are located by the dimensions *J1* and *J2*, both measured from the left end of the frame. The left end of the frame is Joint 1 if there is no left cantilever or the left end of the left cantilever if there is one. *J2* must be larger than *J1* if they both exist.

Select one of the tendon stressing patterns by clicking on the desired option button. Press the “Continue” command button when the correct stressing pattern has been selected and you are ready to leave the screen.

## 5-2 The General Variable Prestress Force Input Data Screen

The **General Variable Prestress Force Input Data Screen** is shown in Figure 5-2. In this screen the user enters data required for determining short and long-term prestress losses, the number of strands in the through tendon, the number of added tendon locations, and the locations of the intermediate construction joints.

Figure 5-2 shows the default values for low-relaxation prestressing steel. **PT Designer** loss calculations follow the method presented in Estimating Prestress Losses, Zia, Preston, Scott, Workman, Concrete International, June, 1979, pp. 32-38. The 11 prestress loss input items on the left of this screen correspond to the variables in the referenced method:

1. **Friction Wobble Coefficient *k*** The value *k* in the friction loss equation:

$$T_o = T_x e^{kL_x + \mu\alpha}$$

   The wobble coefficient is a multiplier of the tendon length $L_x$ between the stressing end and the point *x* where the friction loss is being evaluated. The default value is 0.001.

2. **Friction Curvature Coefficient $\mu$** The value $\mu$ in the friction loss equation shown in Item #1 above. The curvature coefficient $\mu$ is a multiplier of $\alpha$, the total angular change (expressed in radians) through which the tangent to the tendon rotates in the length $L_x$ between the stressing end and the point *x*. The default value is 0.07.

3. **Maximum Jacking Stress** The maximum stress permitted in the prestressing tendon at the stressing end while the jack is still attached to the tendon (i.e., before anchorage seating losses). The ACI Code limits this value to $0.94f_{py}$ or $0.80f_{pu}$ whichever is less. The default value is 216 ksi.

4. **Modulus of Elasticity of P/S Steel $E_s$** The default value is 28,000 ksi.

5. **Anchorage Seating Loss** The distance in inches the wedges travel after the jack releases the tendon. The default value is 0.25 inches.

6. **P/S Steel Relaxation Coefficient $K_{re}$** A coefficient used in the calculation for steel relaxation found in Table 2 of *Estimating Prestress Losses*. Default values are 5,000 for low relaxation strand and 20,000 for stress-relieved strand.

7. **P/S Steel Relaxation Coefficient $J$** A coefficient used in the calculation for steel relaxation found in Table 2 of *Estimating Prestress Losses*. Default values are 0.04 for low relaxation strand and 0.15 for stress-relieved strand.

8. **Average $f_{pi}$ / $f_{pu}$ (For Relaxation Coefficient $C$)** The coefficient $C$ is used in the calculation for steel relaxation. It is found in Table 3 of *Estimating Prestress Losses*, and is based upon the ratio of initial (anchor) stress in the tendon ($f_{pi}$) to the tensile strength of the tendon ($f_{pu}$) which is entered here. **PT Designer** automatically determines $C$ based upon the input ratio of $f_{pi}$ / $f_{pu}$. The default value is 0.7.

9. **Age of Concrete At Stressing (Days)** The coefficient $K_{sh}$ is used in the calculation for concrete shrinkage. It is found in Table 1 of *Estimating Prestress Losses* and is a function of the concrete age (after the end of moist curing) at the time the tendons are stressed. **PT Designer** automatically determines $K_{sh}$ based upon the concrete age input here. The default value is 5 days.

10. **Average Ambient Relative Humidity (%)** This value ($RH$) is used in the calculations for concrete shrinkage. A map of the continental United States and Canada is presented on page 37 of *Estimating Prestress Losses* as an aid in determining the local $RH$ value. The default value is 60%.

11. **Initial Concrete Strength $f'_{ci}$ (psi)** The concrete compressive strength at the time the tendons are stressed. The default value is 3000 psi.

Enter in the appropriate text boxes on the right side of the screen the total number of strands in the through tendon, the total number of added tendon locations, and the intermediate stressing joint locations. The intermediate stressing joints are located by the dimensions *J1* and *J2*, both measured from the left end of the frame. The left end of the frame is Joint 1 if there is no left cantilever or the left end of the left cantilever if there is one. *J2* must be larger than *J1* if they both exist. If the selected stressing pattern has no intermediate stressing points (Types 1-3) the *J1* and *J2* dimension text boxes and captions will be disabled (as they are in Figure 7.2). If the selected stressing pattern does have intermediate stressing points (Types 4-12) the *J1* and *J2* dimension captions and text boxes will be enabled.

An added tendon location is a set of two dimensions, one which locates the added tendon stressing end, the other which locates its dead-end. Each unique set of added tendon stressing end and dead-end dimensions is one added tendon location. Added tendons can be stressed at a maximum of four locations, the left and right ends of the frame and at the intermediate locations *(J1 and J2)* present in the selected stressing pattern. Added tendon dead end locations can be anywhere in the frame, consistent with the direction of the stressing anchor point. Note that the actual dimensions to the stressing ends and dead ends are not entered in this screen, just the total number of added tendon locations.

When the input data in this screen is correct, press the "Continue" command button.

### 5-3 The Added Tendon Location Input Screen

Data for each added tendon location is entered in this screen which is shown in Figure 5-3. This screen will not appear if there are no added tendon locations. Each added tendon location requires three input items:

- The number of strands in the added tendon.

- The location of the added tendon stressing end. There are four possible locations, the left end of the frame (L), the right end of the frame (R), and at one of the two possible intermediate stressing points (*J1* or *J2*). Enter the literal value (*L, R, J1* or *J2*) for the location of the stressing end**. PT Designer** will not accept a *J1* or *J2* entry if the point does not exist in the selected stressing pattern. The added tendon must be stressed in the same direction as the through tendon at the same point. For example, for a Type 10 Stressing Pattern an added tendon with its stressing anchor at *J1* must be stressed to the left.

- The location of the added tendon dead-end, measured in feet from the left end of the frame. The dead end location must be consistent with the direction of the stressing anchor. If the added tendon stressing anchor points towards the left end of the frame, the location of the dead-end must be farther from the left end than the stressing anchor. If the added tendon stressing anchor points towards the right end of the frame, the dead end location must be closer to the left end than the stressing anchor. **PT Designer** checks this and will not accept a dead end location which is incompatible with the stressing anchor direction.

When the input data in this screen is correct, press the “Continue” command button.

## 5-4 The Variable Prestress Force Result Menu

Once all of the data required for the **Variable Prestress Force** mode is entered, **PT Designer** proceeds to the **Variable Prestress Force Result Menu**, shown in Figure 5-4.This screen permits, by pressing the appropriate command button, the user to edit the Variable Prestress Force input data, cancel the **Variable Prestress Force** mode and return to the **RESULT MENU**, and when all data is correctly input, start the Variable Prestress Force Calculations.

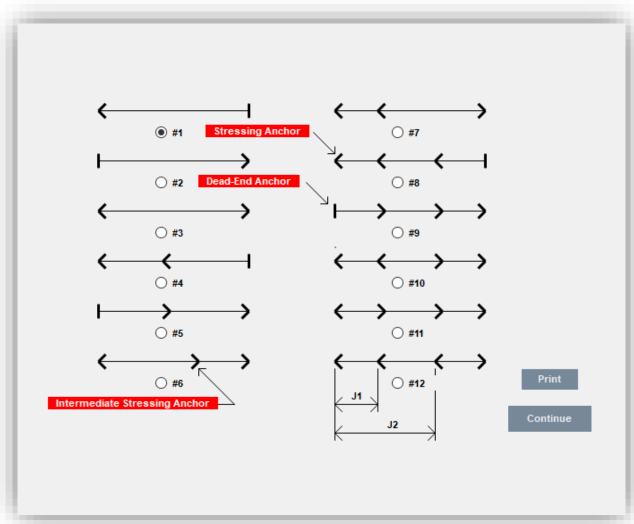



**Figure 5.1 Tendon Stressing Patterns Input Screen**

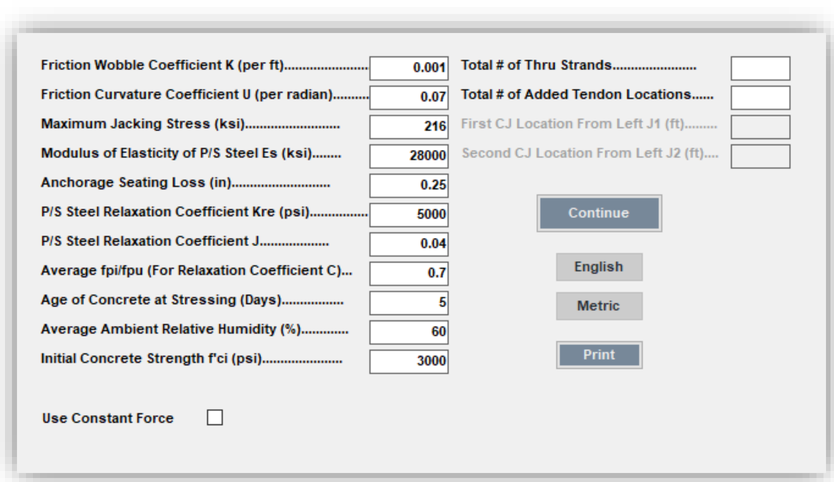



**Figure 5.2 General Variable Prestress Force Input Data Screen**

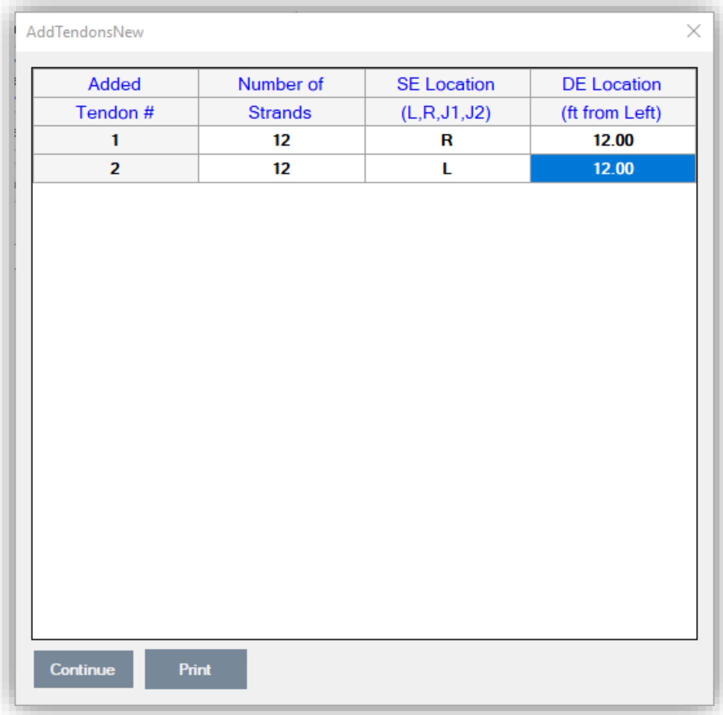



**Figure 5.3 The Added Tendon Location**

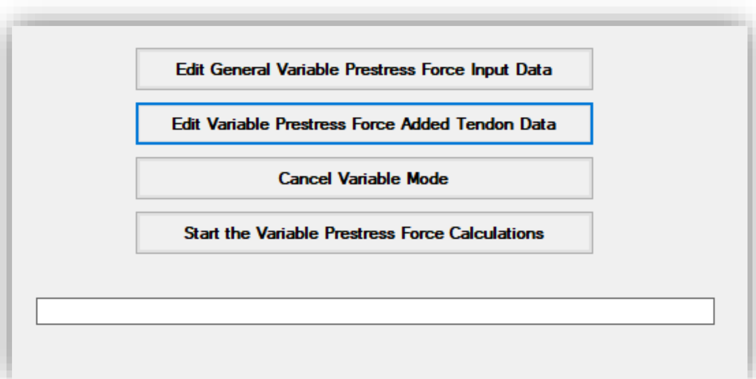



**Figure 5.4 Variable Prestress Force Result Menu**