

PTData.Net

Post-Tensioning Design and Analysis Program

Application Manual

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

© 2011

Chapter 1 - Installing and Starting PTData.Net

- 1.1 General Information and Terminology
- 1.2 The Configuration File
- 1.3 The Datum Line
- 1.4 Dimensions Perpendicular to the Frame
- 1.5 Design Points
- 1.6 Sign Conventions
- 1.7 Units

Chapter 2 - Entering and Editing Input Data - The MAIN MENU

- 2.1 **PTData.Net** Menus
- 2.2 The **MAIN MENU**

Chapter 3 - The Input Data Screens

- 3.1 Entering and Editing Data in the Grids
- 3.2 The Start Over Command Button
- 3.3 General Input Data Screen
- 3.4 Column Input Screen
- 3.5 Transverse Beam Input Screen
- 3.6 Beam and Slab Input Data
 - a) **PTData.Net** Beam Input Screen
 - b) **PTData.Net** Two-Way Slab Input Screen
 - c) **PTData.Net** One-Way Slab Input Screen
- 3.7 Superimposed Load Input Screen
- 3.8 Tendon Profile Input Screen

Chapter 4 - The REVIEW MENU

- 4.1 Forces and Tendon Profiles - Review or Change
- 4.2 Tendon Data and Cross-Section Properties Screen
- 4.3 Flexural Stress Summary Screen
- 4.4 Unfactored Beam Moments Review Screen
- 4.5 Unfactored Beam Shear Review Screen
- 4.6 Flexural Concrete Stresses Review Screen
 - a) Service Load Stresses
 - b) Transfer Stresses
- 4.7 Deflection and Cracking Moment Review Screen
- 4.8 Tendon Balanced Load & Concrete Dead Loads Review Screen
- 4.9 Factored Load Review Screen
- 4.10 DL+LL/4 Rebar Review Screen
- 4.11 Beam Shear Design Review Screen
- 4.12 Punching Shear Stress Review Screen
- 4.13 Variable Prestress Force Option
- 4.14 Controlling Rebar Option

4.15 Return to the **MAIN MENU** Option

Chapter 5 - Variable Prestress Force Option

5.1 – The Tendon Stressing Patterns Input Screen

5.2 – The General variable Prestress Force Input Data Screen

5.3 – The Added Tendon Location Screen

5.4 – The Variable Prestress Force Review Menu

- 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)

Definitions

| | |
|---------------|--|
| 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 |

Definitions

| | |
|----------|---|
| 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 <u>only</u> 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) |

Definitions

- 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 ϕ factor)
- ϕM_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

Definitions

- 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 $A_s f_y$, 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)

Definitions

| | |
|----------------|--|
| 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 ϕ factor) |
| ϕV_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 |

Definitions

- 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**
- α_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
- β_1 Factor which varies with concrete strength f'_c : β_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.
- β_c Ratio of long side to short side of a rectangular column ($\beta_c = 1$ for round columns)
- γ_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
- γ_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
- ϕ Capacity reduction factor (0.9 for flexure, 0.75 for shear)
- ρ Reinforcing steel ratio = $A_s / B'd_s$
- ρ_b Balanced reinforcing steel ratio = $0.85\beta_1 f'_c / f_y [87000 / (87000 + f_y)]$
- ω Reinforcing index $(T_p + T_s - C_s) / (B' d_p f'_c)$

INSTALLING AND STARTING PTData.Net

1.1 - General Information and Terminology

PTData.Net 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 **PTData.Net**. A separate document, also furnished with **PTData.Net**, is called the **THEORY** manual, and it describes in technical detail what **PTData.Net** actually does. The **THEORY** manual is referenced often in this document.

PTData.Net 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 **PTData.Net** 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 **PTData.Net**, 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 . When referring to command buttons, the terms "press" and "click on" are used interchangeably.

1.2 - The Change Configuration Menu

PTDATA.NET 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, **PTData.Net** 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 **MAIN MENU** and the

“Change Configuration” accessed from the “Tools” pull-down menu. 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 = 10).
2. **ColModel** - The column modeling code (See **THEORY** Section 7.1(a)):
0 = Top and bottom column always present.
1 = Top column present for live loads only.
(Default = 0)
3. **CrackMom** - An indicator (1 or 0) which tells **PTData.Net** 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 in²).
5. **EffStressLR** - The assumed effective stress in low-relaxation tendons. (Default = 173 ksi).
6. **EffStressSR** - The assumed effective stress in stress-relieved tendons. (Default = 162 ksi).
7. **MinCapSizeEnabled** – If this option is enabled (Value=1) **PTData.Net** will not permit a shear cap size ($W1 \times W2$) 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).
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 318-08+ Check Box** – Checking this box tells the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases of the latest ACI 318 Code. This will be the default each time the program is opened. Unchecking 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.

WARNING: **PTData.Net** reads the data file each time it is **started**. When this file is changed **PTData.Net** will continue to use the new values until PTDATA.NET.INI is changed again. If you make a change in PTDATA.NET.INI for an atypical run, be sure to change the file back to its original values or **PTData.Net** 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 318-08+ box will be checked. The user can modify the load factors and uncheck the ACI318-08+ 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 **PTData.Net** 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 **PTData.Net** 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, **PTData.Net** 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, **PTData.Net** 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 **PTData.Net** 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 **PTData.Net** frame parameters are calculated at a set of **design points** in each span, some of which are determined by the user and some by **PTData.Net**.

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 **PTData.Net** 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, **PTData.Net** 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 **PTData.Net** depending on the Member Type.

1.6 - Sign Conventions

The following sign conventions are followed in **PTData.Net**:

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

PTData.Net 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 **PTData.Net** 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 **PTData.Net**. 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.

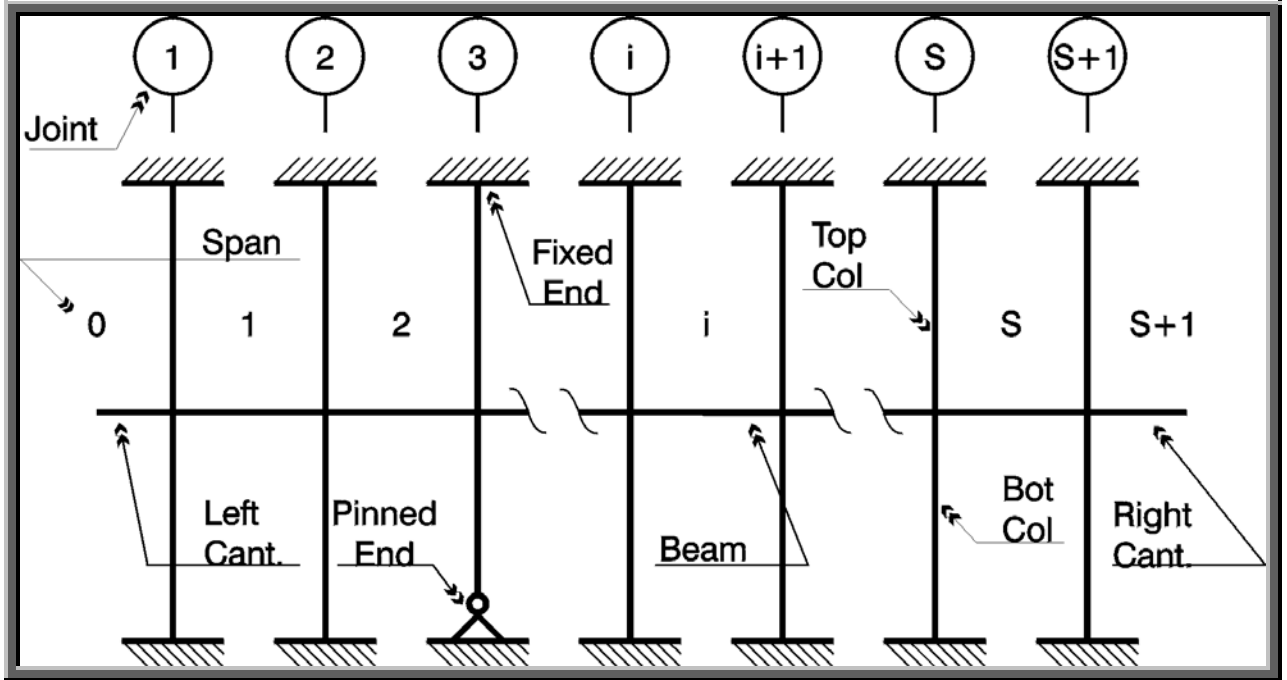


Figure 1.1 – Span & Joint Identification

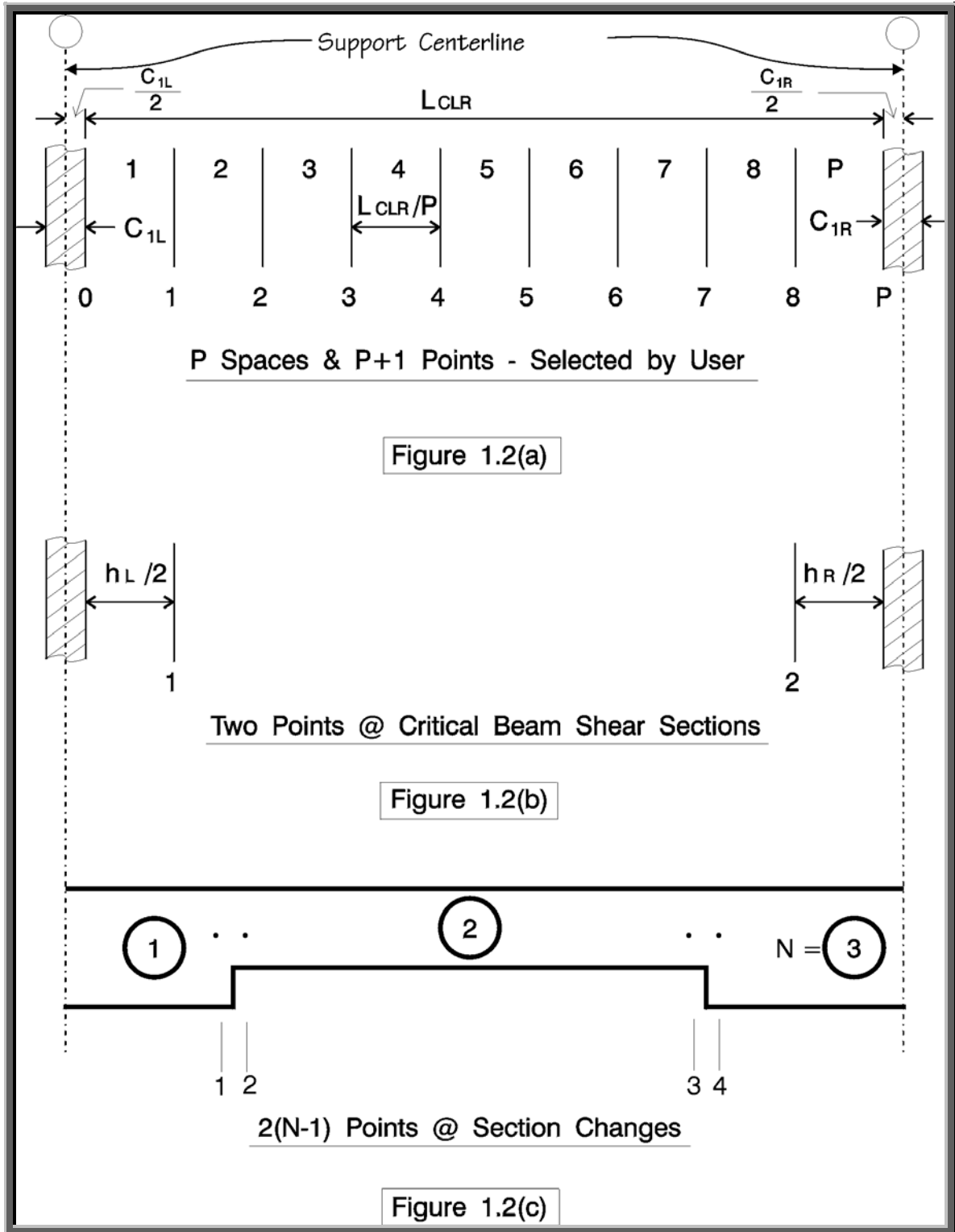


Figure 1.2 – Design Points

ENTERING AND EDITING INPUT DATA - THE MAIN MENU

2.1 - PTData.Net Menus

PTData.Net operation is controlled by two menus, the **MAIN MENU**, which appears each time the program is started and the **REVIEW MENU**, which appears after the user has inputted all of the design parameters and PTData.Net has performed an initial design for a particular run. The **MAIN MENU** is used to enter or modify input data for a particular run, and to save the data. The **REVIEW MENU** is used to review calculated values for the run in progress. The **REVIEW MENU** is discussed in Chapter 4. This chapter discusses the entering and editing of input data with the **MAIN MENU**.

Actions can be initiated and data can be entered and edited in **PTData.Net** 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.

2.2 - The MAIN MENU

The **MAIN MENU** is shown in Figure 2.1. The **MAIN MENU** consists of familiar Windows icons, command buttons and pull-down menus. The following actions can be initiated from the **MAIN MENU**:

- The input data procedure can be started for a new beam, one-way slab or two-way slab run from the command buttons on the toolbar, or from the “File” drop-down menu.
- Data can be recalled from a previously stored run by either the “File/Open” drop-down menu or the “Open File” icon on the toolbar.
- Input data can be edited, either for the current new run or a recalled run from the “Edit Input” command button on the toolbar or from the “Tools” drop-down menu.
- The program configuration can be changed from the “Tools” drop-down menu.
- Information about the licensed user of the program can be viewed from the “Help” drop-down menu.
- The **REVIEW MENU** can be accessed from either the command button on the toolbar or from the “Tools” drop-down menu.
- The program can be terminated.

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.

Commands:

1. **New Beam, New Two-Way Slab, New One-Way Slab** – These commands start the data input procedure for a new run. **PTData.Net** will proceed through all applicable Input Data Screens in sequence. The user is given the opportunity to modify the input from the Edit Input Data Screen (Figure 2.2). Once the input is complete and the user determines that it is correct, the calculations will be initiated and the “Forces and Profiles” screen will be presented. Once the forces and profiles are finalized the **REVIEW MENU** will be presented.
2. **Open** - This command allows recalling and editing input *and output* data from a run that was previously made and whose data file is saved (with the extension *.ptd). Note that when data is recalled from a previous run, *all data* pertaining to that run is recalled, including data from the Configuration Screen which will replace the current Configuration Screen data.
3. **Edit Input Data** - Pressing this button opens the appropriate Edit Input Data Screen. This option is enabled only when input data is available for editing, i.e., after a *.PTD file has been recalled or after input data for a new run has been entered. This screen permits editing of each of the eight Input Data Screens. These screens are described in detail in Chapter 3. Note that the “Transverse Beam” button is enabled only when the Member Type is a Two-Way Slab. When all input data is correct, click on the “OK - Data is Correct” button and this will initiate the calculations and present the Forces and Profiles Screen.
4. **Change Configuration** - 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.NET.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:

1. **Top and Bottom Columns Always Present**
2. **Top Column Present for Live Load Only**

If the first choice is selected, **PTData.Net** 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, **PTData.Net** 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 **PTData.Net** 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 **General Input Data Screen**, Section 3.3). 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 **PTData.Net** will not permit a shear cap size ($W1 \times W2$) 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.

h) Non-Controlling Cantilever Iteration - **PTData.Net** 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 318-08+ Check Box** – Checking this box tells the program to use the Load Factors, Capacity Reduction Factors and Redistribution Cases of the latest ACI 318 Code. This will be the default each time the program is opened. Unchecking 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.NET.INI file** by clicking on the “OK - Change PTDATA.NET.INI File” button. Use this last action with care because the modified data will continue to be used in all future runs until the **PTDATA.NET.INI** file is modified again. When in doubt as to the contents on the **PTDATA.NET.INI** file you can always bring up the Configuration Screen and review the current configuration data.

- 7. **The Review Menu** When valid output data is available for review (a *.PTD run has been recalled without editing or calculations have been performed on the current data) selecting this button will take you directly to the **REVIEW MENU**. The “Review Menu” option will be disabled whenever valid output data does not exist.

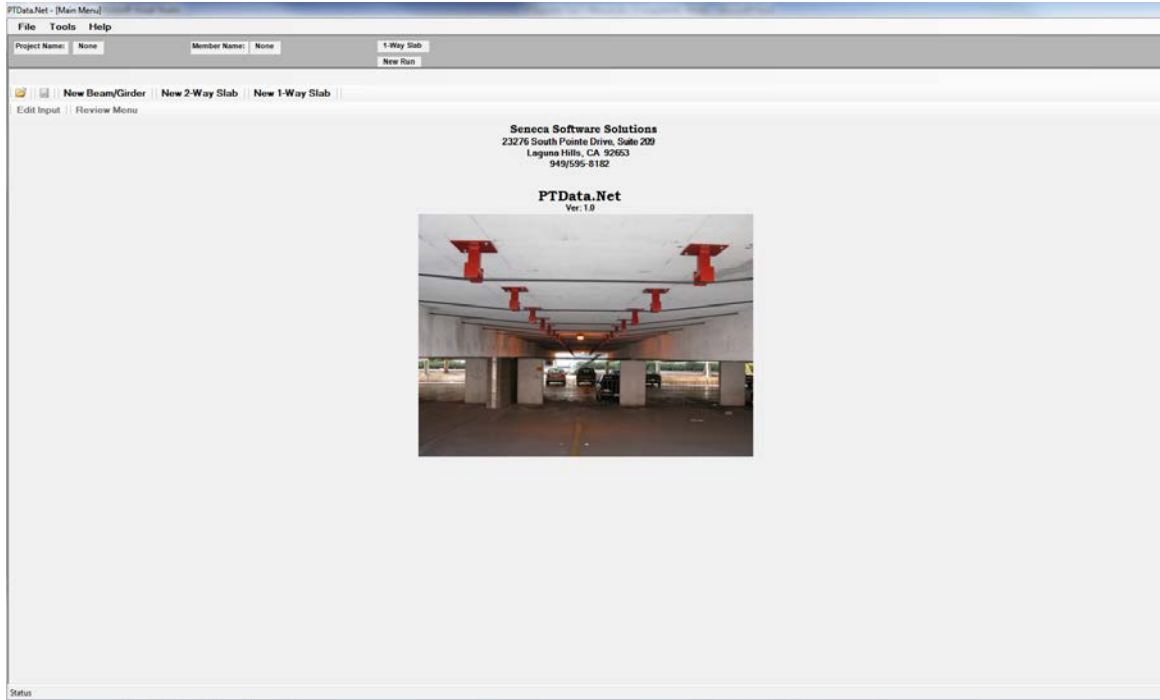


Figure 2.1 – The Main Menu

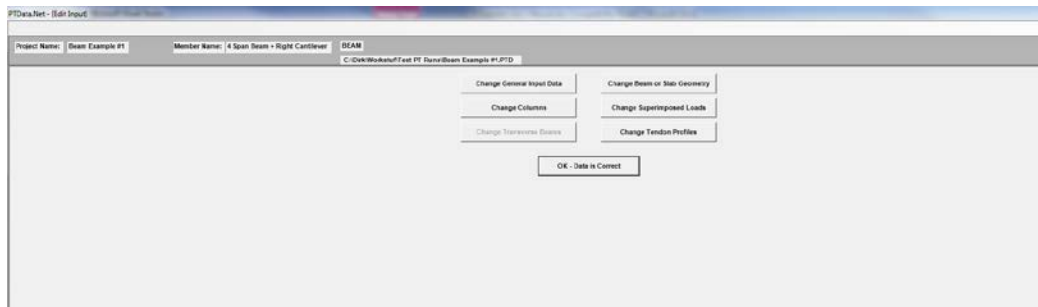


Figure 2.2 – The Edit Input Data Screen

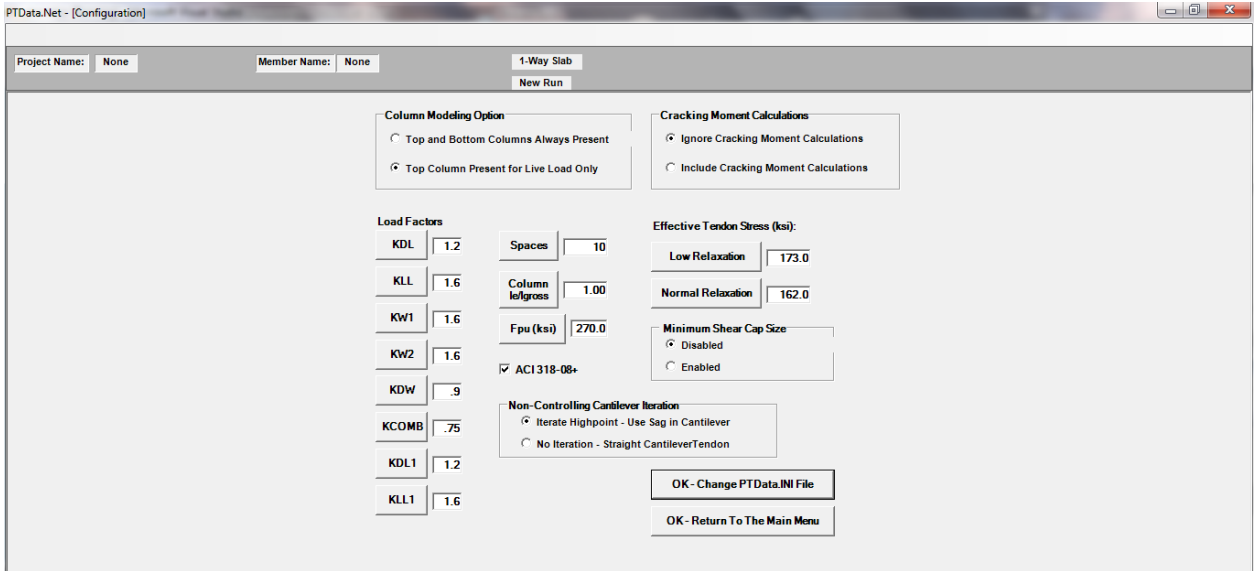


Figure 2.3 – The Configuration Screen

THE INPUT DATA SCREENS

This chapter contains detailed descriptions of the six screens which are used to manage **PTData.Net** input data.

When a **New Run** command button in the **MAIN MENU** is pressed **PTData.Net** presents each required input screen consecutively and automatically (i.e., starting with the **General Input Data Screen** and proceeding in order through the **Tendon Profile Screen** - see Figure 2.2 for the precise input screen sequence). After all of the six input data screens have been completed, **PTData.Net** returns to the **Edit Input Data** screen (Figure 2.2), where the user has the opportunity to edit any of the input data by accessing the input screens individually. This is done by pressing the appropriate **Edit Input Data** screen command button. After each input data screen is edited, **PTData.Net** returns to the **Edit Input Data Screen**. When all input data is correctly entered, press the “OK - Data is Correct” command button in the **Edit Input Data** screen and the program will initiate the calculations and present the **Tendon Profile Screen**.

When data is recalled from a previously saved run the recalled data may also be edited by pressing the “Edit Input Data” command button from the **MAIN MENU** and accessing the input screens individually from the **Edit Input Data** screen.

3.1 - Entering and Editing Data In The Grids

All Input Data Screens except one (the **General Input Data Screen**) use a grid system for entering and displaying data. The grids are a series of rows and columns with a space for data at the intersection of each row and column (called a “cell”.) There are two ways to enter or edit data in each cell:

- Using the mouse, double-click on the cell you want to change. A dialog box will appear and prompt you for the input data.
- Using the keyboard arrow keys (up, down, left, right) move the focus to the cell you want to change and then press either <Enter> or the <Space Bar>. A dialog box will appear and prompt you for the input data. All grid data can be entered or edited with this method using the keyboard alone, without the mouse.

3.2 - The “Start Over” Command Button

Each Input Data Screen contains the “Start Over” command button. Pressing the “Start Over” button during data entry for a New Run clears all previously entered data and returns the program to the **MAIN MENU**. This is particularly useful if the user notices, during data entry, that the wrong member type has been selected.

Pressing the “Print” button in any of the review menus causes **PTData.Net** to print the current screen to the active Windows printer. For screens which show only one span of data, pressing the “Print This Span” button prints the current screen only.

Following are descriptions of each of the six input data screens:

3.3 - General Input Data Screen

The **General Input Data Screen** is shown in Figures 3.1 through 3.3. Figure 3.1 shows default values for the beam or girder member type (BEAM), 3.2 shows defaults for 2-way slabs (PLATE), and 3.3 shows defaults for 1-way slabs (SLAB). Following is a description of each of the 28 General Input Items.

The following items describe the general frame configuration and design criteria:

- 1. Project Name** Enter the name of the project into the text box. It will be printed on each page of printed output.
- 2. 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.
- 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. **PTData.Net** accepts a maximum of 25 spans. **PTData.Net** 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 **PTData.Net** 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 **PTData.Net** 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

selection procedure contained in the **REVIEW 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).

- 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. **PTData.Net** 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.

- 9. 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 **PTData.Net** 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.

- 10. 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.

The following **General Input Data Screen** items describe the post-tensioned tendons:

- 11. Tendon Type** **PTData.Net** 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.

- 12. 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.
- 13. 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.
- 14. 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.
- 15. 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.
- 16. Cross-Sectional Area of One Strand** Enter this area in square inches in the text box.

The following **General Input Data Screen** items describe properties and criteria relating to the frame concrete:

- 17. Beam Strength** The 28-day beam concrete compressive strength in pounds per square inch (psi).
- 18. Beam Density** The beam concrete weight (density) in pounds per cubic foot (pcf).
- 19. Column Strength** The 28-day column concrete compressive strength in pounds per square inch (psi).
- 20. Column Density** The column concrete weight (density) in pounds per cubic foot (pcf).
- 21. Tensile Stress Coefficient Top** The allowable flexural tensile stress in the top concrete fiber expressed as a multiplier of $\sqrt{f'_c}$.
- 22. Tensile Stress Coefficient Bottom** The allowable flexural tensile stress in the bottom concrete fiber expressed as a multiplier of $\sqrt{f'_c}$.

- 23. 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 **PTData.Net**.

The following **General Input Data Screen** items describe the unstressed reinforcing steel:

- 24. Yield Strength** Enter the value in kips per square inch (ksi) for both longitudinal steel and stirrups in the text box provided
- 25. 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, **PTData.Net** 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.
- 26. Stirrup Size** Entered as a number (#3-#9) using the pull-down combo box provided and used to establish the required stirrup spacing.
- 27. 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.
- 28. 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.
- 29. 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.

3.4 - Column Input Screen

This screen is used to input all data related to the frame columns. There are two columns at each joint, one above the joint (the TOP column) and one below the joint (the BOTTOM column). The total number of joints is $S+1$ therefore data for $2S+2$ columns must be entered for each frame.

The **Column Input Screen** is shown in Figure 3.4. For a new run **PTData.Net** first requests input dimensions for the "most typical column", the one that repeats the most times. If there are no repeating columns, any one of the columns will do for this "typical" column. Note that column dimensions are accepted for a TOP column (above the joint) and a BOTTOM column (below the joint). If there are no columns in the frame, enter "N" or "n" for the first typical column entry (the Bottom Column L_c

dimension) and the entire column data entry routine will be skipped. If existing column data is being edited, all of the previously entered data will appear in the data grid. For each column, top and bottom, **PTData.Net** needs to know:

- L_c The column length in feet (from the midpoint of the beam depth to the point of fixity or the pin at the far end - see **THEORY** Figure 7.1)
- c_2 The cross-sectional column dimension in inches perpendicular to the beam span
- c_1 The cross-sectional column dimension in inches parallel to the beam span
- **FIX** The far-end column fixity condition
Enter "P" or "p" for a pinned end
Enter "F" or "f" for a fixed end

Input the actual physical dimensions of the column. Adjustments for equivalent frame column stiffnesses in two-way systems are made automatically by the program. Round columns can be input by entering the **diameter** for c_2 and **zero** for c_1 .

For a new run, **PTData.Net** makes all the columns the same as the "typical column". Changes to the typical column data, or the previously entered data, can then be made as necessary using the methods described in Section 3.1.

To leave the screen and accept the column data as shown press the "OK - Data is Correct" command button.

3.5 - Transverse Beam Input Screen

This screen 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.

The **Transverse Beam Input Screen** is shown in Figure 3.5. For a new run **PTData.Net** first requests input dimensions for the "most typical transverse beam", the one that repeats the most times. If there are no repeating transverse beams, any one of the transverse beams will do for this "typical" transverse beam. If there are no

transverse beams enter “N” or “n” for the first typical transverse beam dimension and the program will skip the entire transverse beam input routine. If existing transverse beam data is being edited, all of the previously entered data will appear in the data grid.

For a new run, **PTData.Net** makes all the transverse beams the same as the “typical transverse beam”. Changes to the typical transverse beam data, or the previously entered data, can then be made as necessary using the methods described in Section 3.1.

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

3.6 – Beam and Slab Input Data

The beam and slab input data is different for the three member types, Beam, Two-Way Slab, and One-Way Slab.

3.6(a) – Beam Input Screen

Figure 3.6 is the **Beam Geometry Input Screen**. Span lengths are entered from support centerline to support centerline for spans and from support centerline to the extreme end of the cantilever for cantilevers. A different cross section can be used in each span. Enter an alphanumeric name (two characters maximum) for the cross-sectional shape (Section ID) in each span. For each cross-section shape ID define the cross-section geometry by selecting the Section ID and pressing the “Define or Review Geometry” command button. Cross-section data need be entered only once for each different cross-section ID name. **PTData.Net** will make all of the identically named sections the same. **PTData.Net** supports a library of cross-sectional TYPES which are shown in Figure 3.7 and on the input screen. Units for *all* cross-section dimensions are inches **except** for FLANGES and TRIBUTARIES which are in feet.

After the “Define or Review Geometry” command button is pressed, a dialog box will appear and prompt for the Cross-Section Type for the section ID selected. In Figure 3.6 Cross Section Type 1 is specified for Segment A. Selecting OK in the dialog box will cause the cross-section geometry input data screen to appear for the cross-section type selected. Figure 3.8 shows the input screen for the Type 1 Cross-Section. After the cross-section geometry data is entered for each Cross-Section Type, program returns to the Beam Geometry Input Data screen as shown in Figure 3.6. Note that the column titled “Section ID/Type” shows “A/1” for each span, indicating Section ID “A” and Cross-Section Type “1” for each span. To modify a Section ID use the procedures detailed in Section 3.1. To modify the Cross-Section geometry highlight the appropriate cell in the “Section ID/Type” column and press the “Define or Review Geometry” command button. When all data is correct, press the “OK – Data is Correct” command button.

3.6(b) – PTData.Net Two-Way Slab Input Screen

Figure 3.9 shows the input data screen for two-way slabs. In the grid titled “Span Geometry” enter, for each span, the span length L between support centerlines, the slab thickness, the tributary lengths on the left and right sides of the equivalent frame

centerline, and the distance Y_{ref} from the datum line to the top of the slab. Use the “Typical” command buttons as appropriate, or enter the data into each cell as detailed in Section 3.1.

ACI 318 defines an edge column as column with a face closer than 4 times the slab thickness to the free edge of the slab. **PTData.Net** understands that the column is an edge column through the modeling of the tributary area. It is important to model edge columns properly since the allowable punching shear stresses are different for edge and corner columns than they are for interior columns. Also, when **PTData.Net** determines that the column is an edge column it will not model a torsional beam to the edge side in the equivalent frame analysis. If a cantilevered tributary area exists in a span that is larger than 4 times the slab thickness from the face of column, a tributary area less than 4 times the slab thickness should be modeled with the remainder added to the tributary area of the interior side.

Enter the dimensions of a rectangular drop cap, if one exists, at each joint in the grid titled “Drop Cap Geometry”. At each joint enter the dimension of the cap parallel to the span of the frame ($W1$), the dimension of the cap perpendicular to the span ($W2$), and the total depth of the cap, including the slab, (h). **PTData.Net** assumes that the rectangular cap is centered on the column. For edge, corner, or edge parallel columns enter the cap size as if it was a “full-sized” cap centered on the column and extending beyond the slab edge. Enter “full size” $W1$ and $W2$ dimensions for edge, corner, and edge parallel drops as shown in Figure 3.12. **PTData.Net** will automatically truncate the sides of the drops to the slab edge as necessary to conform to the column/slab edge relationships shown in Figure 3.12. Figure 3.12 shows rectangular columns, these relationships are similar for round columns.

If the “Minimum Cap Size” option is enabled in the Configuration screen, **PTData.Net** will determine the plan dimensions of the cap ($W1 \times W2$) which are required, based upon the column dimensions $c1$ and $c2$, and the cap depth h , to ensure that the inner Critical Section #1 falls within the cap, and will not permit $W1$ or $W2$ dimensions which are smaller than those minimum dimensions. These minimum dimensions, and their relationship to $c1$, $c2$, and h , are shown in Figure 3.13. If the “Minimum Cap Size” option is disabled, any cap dimensions can be entered; however in that case **PTData.Net** will base the d dimension for the inner Critical Section on the total cap depth, rather than the slab thickness. The designer must enter $W1$ and $W2$ dimensions which are compatible with this assumption.

To the right of the column of buttons that aid in entering the slab and column capital geometry, a button titled “Suggest Slab t” exists (Figure 3.9). This function will provide a suggested slab thickness based upon the maximum end and interior span lengths in conjunction with the superimposed dead and live load. The trial slab thickness does not account for the use of column capitals. For higher superimposed loads and/or span lengths, **PTData.Net** will provide an estimate of the slab thickness which may contain a recommendation that capitals be considered.

When all data is correct, press the “OK – Data is Correct” command button.

3.6(c) – PTData.Net One-Way Slab Input Screen

Figure 3.12 shows the data input screen for one-way slabs. Enter, for each span, the span L between support centerlines, the slab thickness h , and the dimension Y_{ref} between the datum line.

When all data is correct, press the “OK – Data is Correct” command button.

3.7 - Superimposed Load Input Screen

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 **PTData.Net** 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. **PTData.Net** supports five types of loads as shown in Figure 3.13. They are:

UNIFORM LOAD (U) A load in **kip_s 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 **kip_s 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 **kip_s** 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). **PTData.Net** 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.

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. **PTData.Net** 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. **PTData.Net** 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, **PTData.Net** considers them as additive.

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

- **Change 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.
- **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** - Starts the 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.
- **Uniform Load** - Starts the input dialog for a load with Load Type = Uniform (U) which is added to the current span and to every other span in the frame. 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 - Data is Correct FOR ALL SPANS** - 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** and the program will proceed to the next step in the data input or editing process.

For **UNIFORM LOADS (U)**, which often occur over an entire span ($A = 0$, $B = L$), **PTData.Net** 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 **PTData.Net** 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 **PTData.Net** 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. **PTData.Net** 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 - Data is Correct FOR ALL SPANS” command button.

3.8- Tendon Profile Input Screen

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. **PTData.Net** 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 **PTData.Net** in the automatic design procedure (see **THEORY**, Section 9.2) and can then be modified, if necessary, by the user from the **REVIEW MENU**.

The **Tendon Profile Input Screen** is shown in Figure 3.17. **PTData.Net** prompts you for input data required for the profile TYPE entered and skips data not required for that TYPE, inserting an uneditable “xxxx” in each grid cell not required. For double-harped profiles (TYPES 6 and 7) **PTData.Net** requires that you input the *ratio* between the two balanced loads, P_2/P_1 . **PTData.Net** will adjust the profile dimensions *in the automatic design* such that the two balanced loads will always have this same ratio (i.e., $P_2/P_1 = \text{Constant}$). A reasonable starting value for P_2/P_1 is the ratio between the respective dead load values for the (presumed) applied loads acting down at the two harping points. If you want the two balanced loads to be equal, enter a value of 1.0 for P_2/P_1 . This restriction on the relative values of P_1 and P_2 applies only to the automatic design generated by **PTData.Net** and can be overridden by the user manually from the **REVIEW MENU**.

To leave the screen and accept the tendon profile data as shown press the “OK - Data is Correct” command button.

The screenshot shows the PTData.Net - [General Input] window. The window title is "PTData.Net - [General Input]". The window contains several sections of input fields:

- Project Name:** Application Manual
- Member Name:** Sample
- Number of Spans:** 1
- Cantilevers:** Left Right
- Live Load Arrangement:** Skipped Uniform
- Number of End Spans:** Left Right
- Concrete Data:**
 - Beam Strength (psi): 5000
 - Beam Density (pcf): 150
 - Column Strength (psi): 5000
 - Column Density (pcf): 150
 - Tensile Stress Coefficients: Top Bottom
 - Minimum F/A (psi): 125
- Reinforcing Steel Data:**
 - Yield Strength (ksi): 60
 - Long. Bar Size T #
 - Long. Bar Size B #
 - Stirrup Size #
- Tendon Data:**
 - Unbonded Low Relaxation
 - Bundle Diameter (in):
 - Concrete Covers (inches): Top Bottom
 - Bottom Endspans:
- Concrete Covers (inches):** Top Bottom
- DL+LL/4 Rebar?** Yes No
- Full/Reduced LL Ratio:**

Buttons at the bottom: Start Over, OK - Data is Correct

Figure 3.1 – The General Input Data Screen (Beam Defaults)

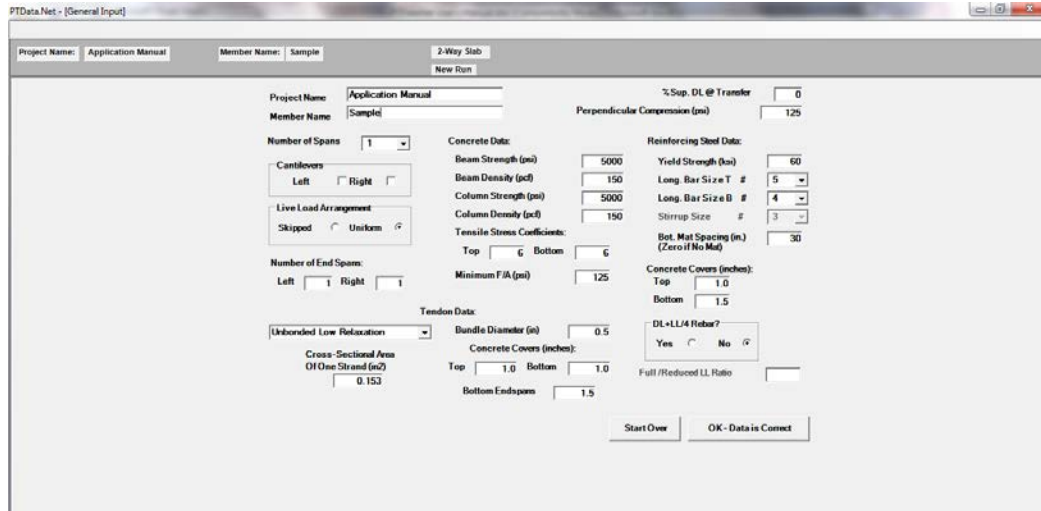


Figure 3.2 – The General Input Data Screen (Two Way Slab)

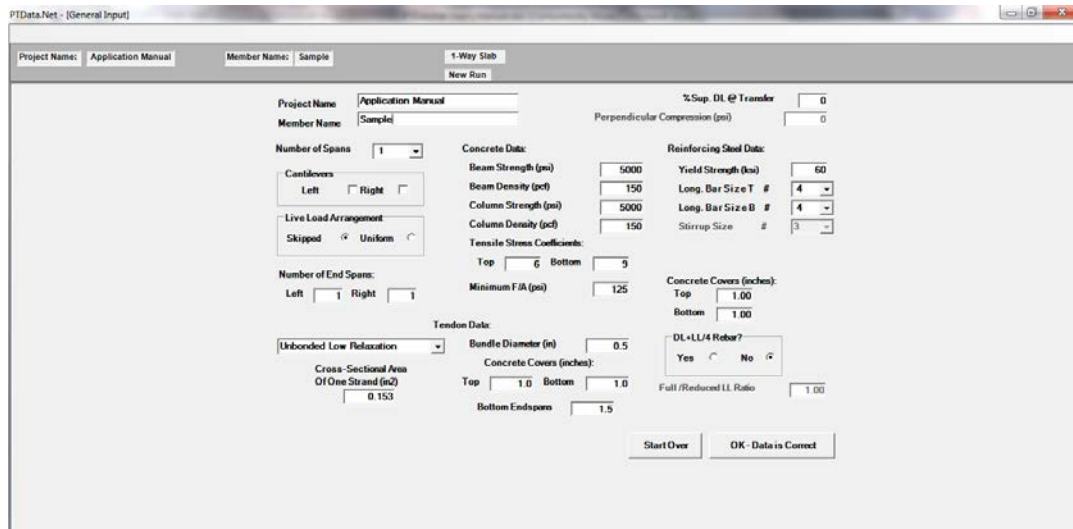


Figure 3.3 – The General Input Data Screen (One-Way Slab)

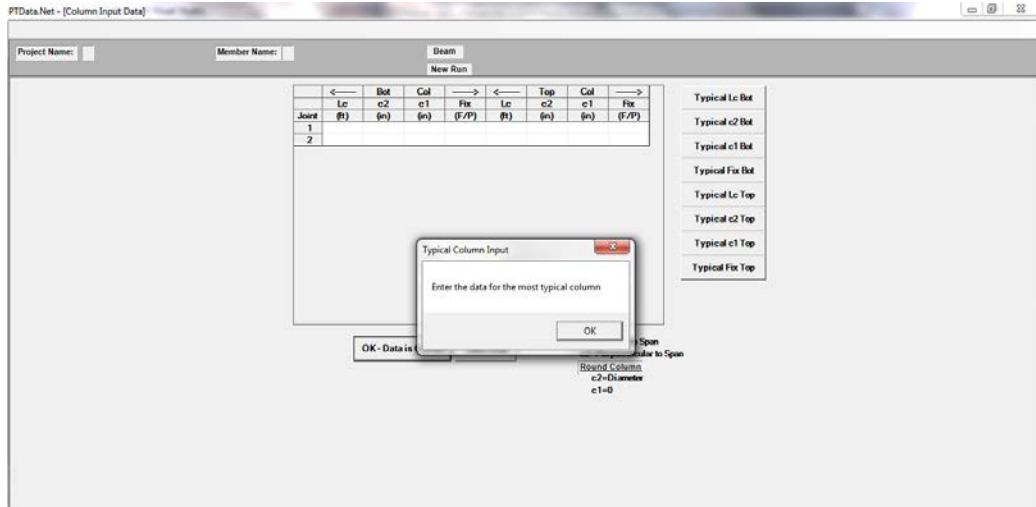


Figure 3.4 – The Column Input Screen

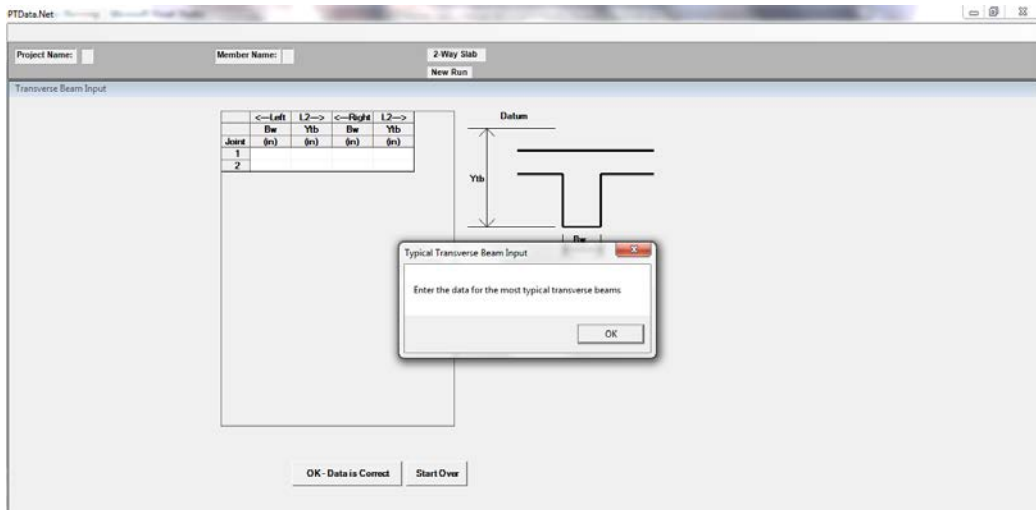


Figure 3.5 – The Transverse Beam Input Screen

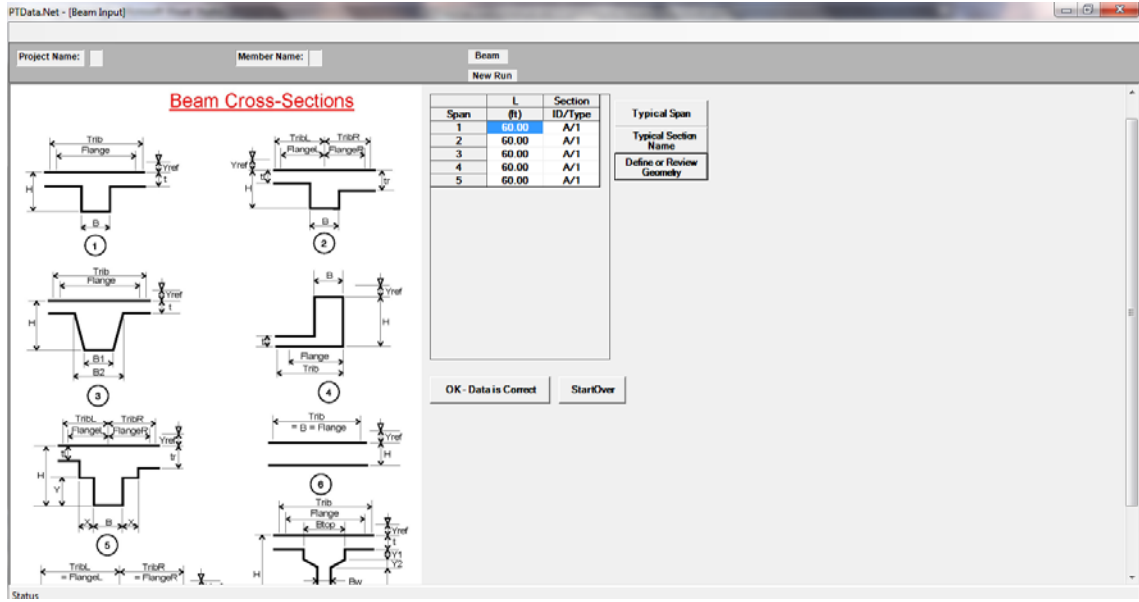


Figure 3.6 – The Beam Geometry Input Screen (with Scroll Bar)

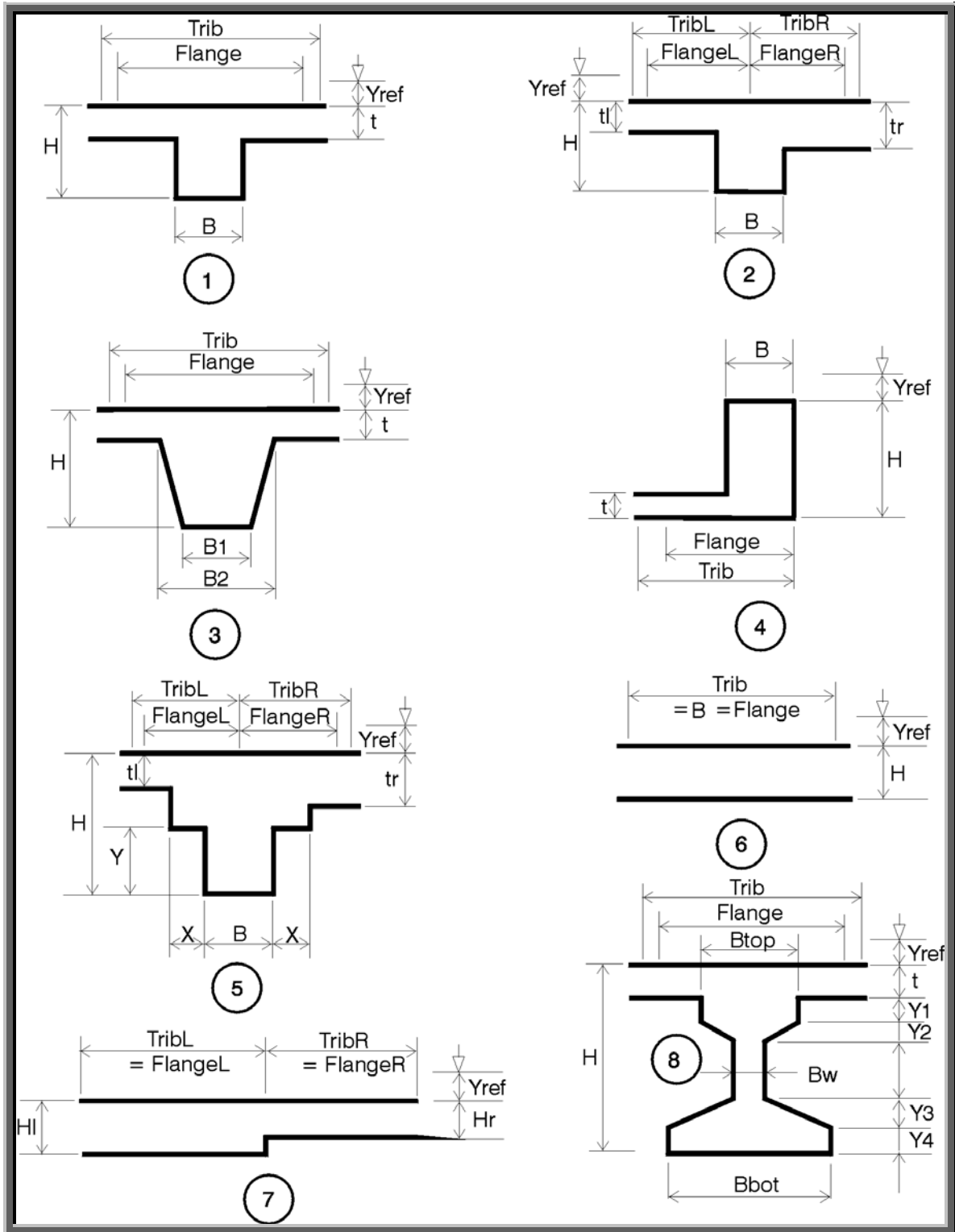


Figure 3.7 – Beam Types

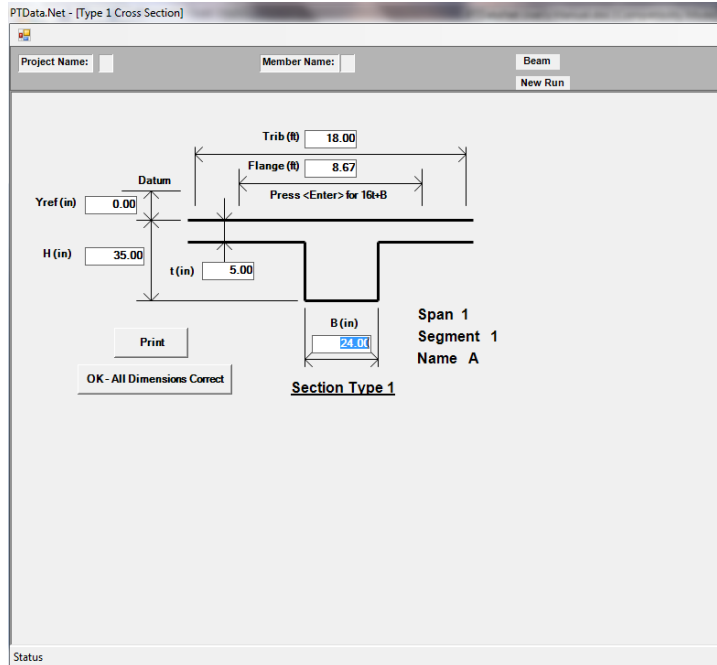


Figure 3.8 – The Type 1 Cross Section Input Screen

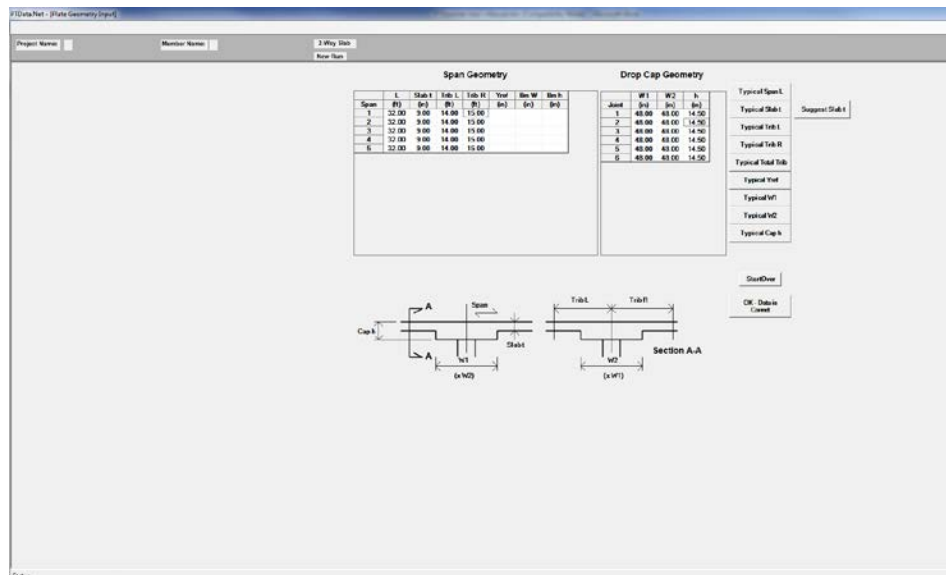


Figure 3.9 – The Two Way Slab Geometry Input Screen

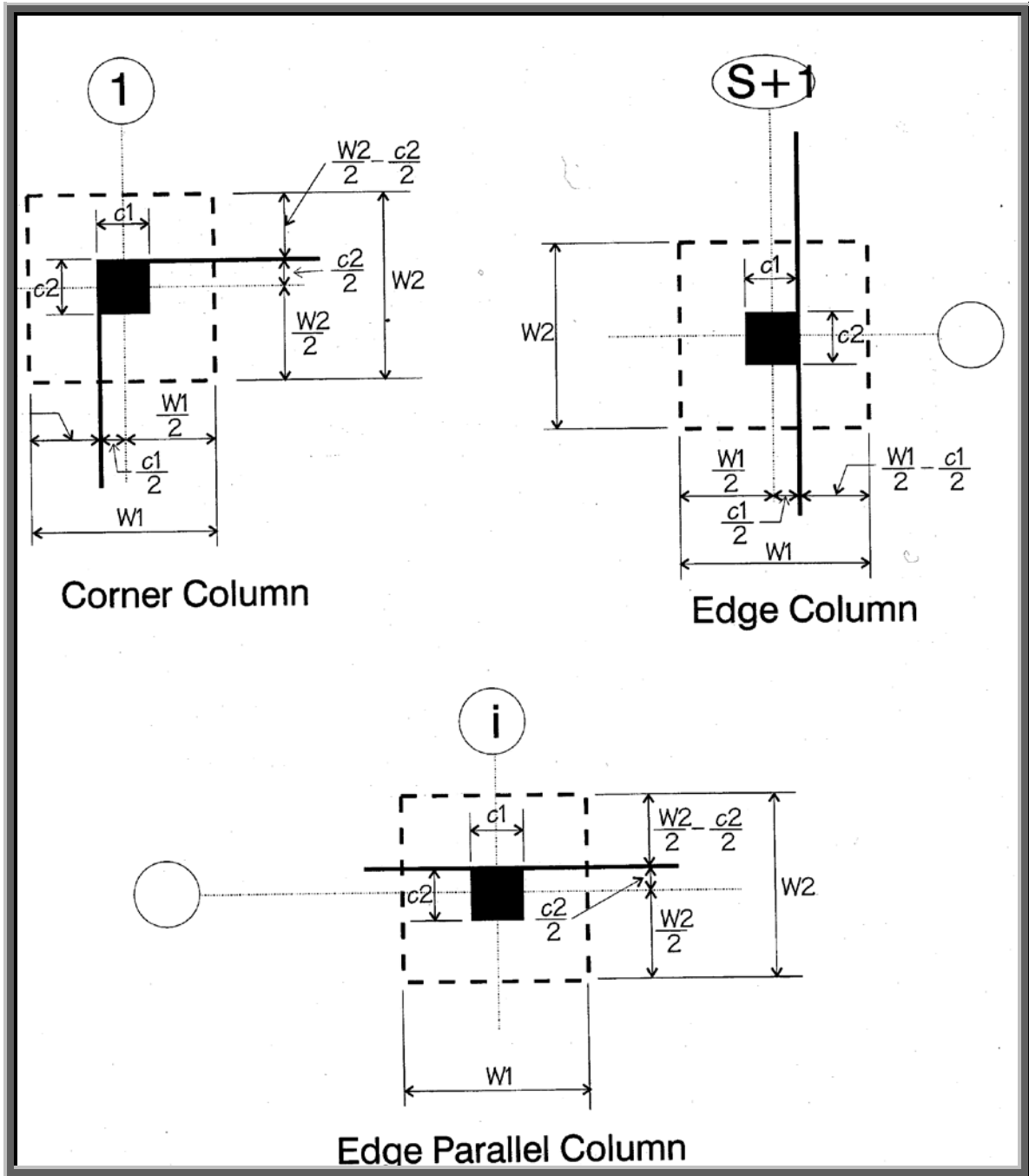


Figure 3.10 – Edge, Corner, & Edge Parallel Drop Caps

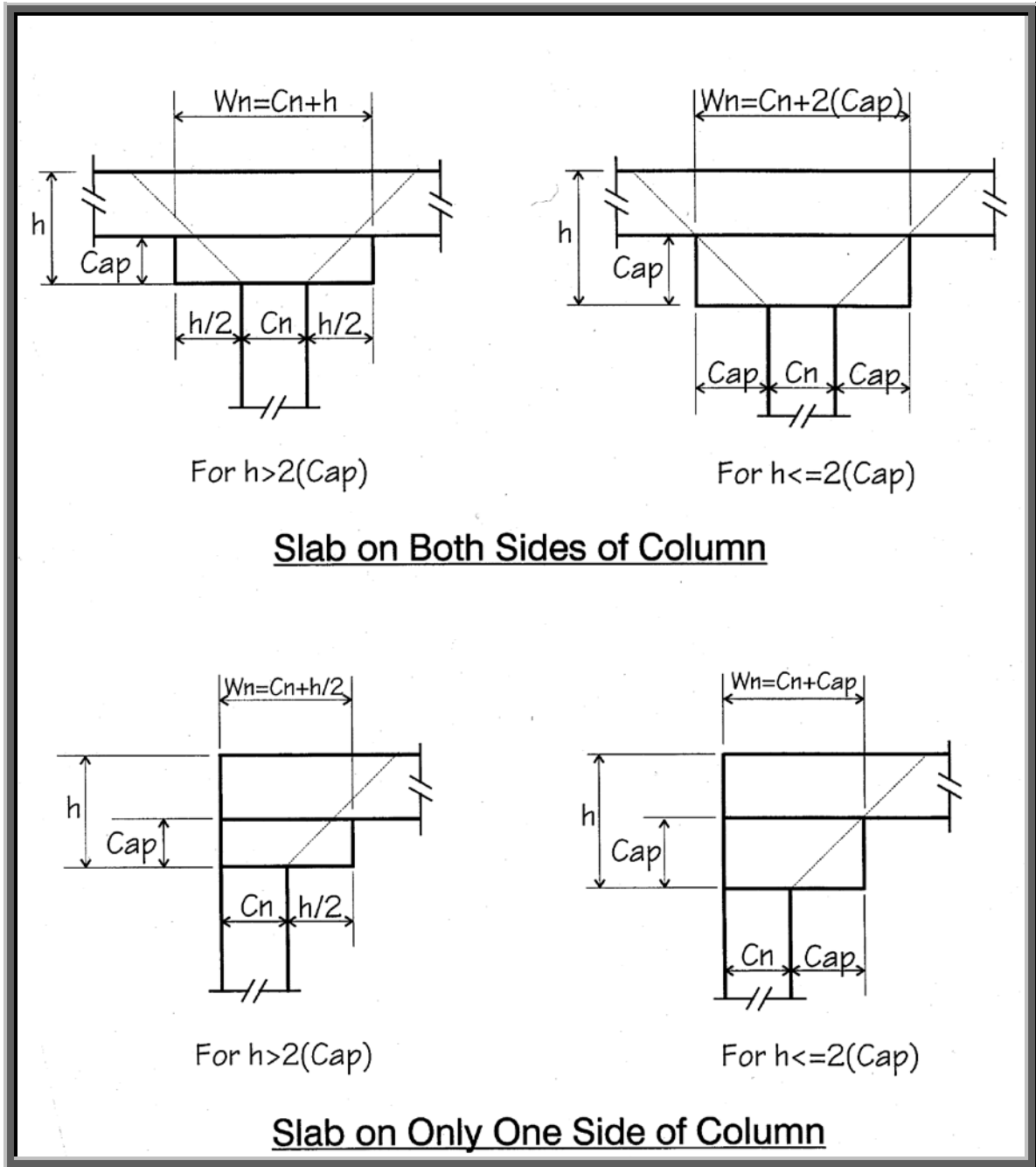


Figure 3.11 – Minimum Cap Plan Dimensions – W_n

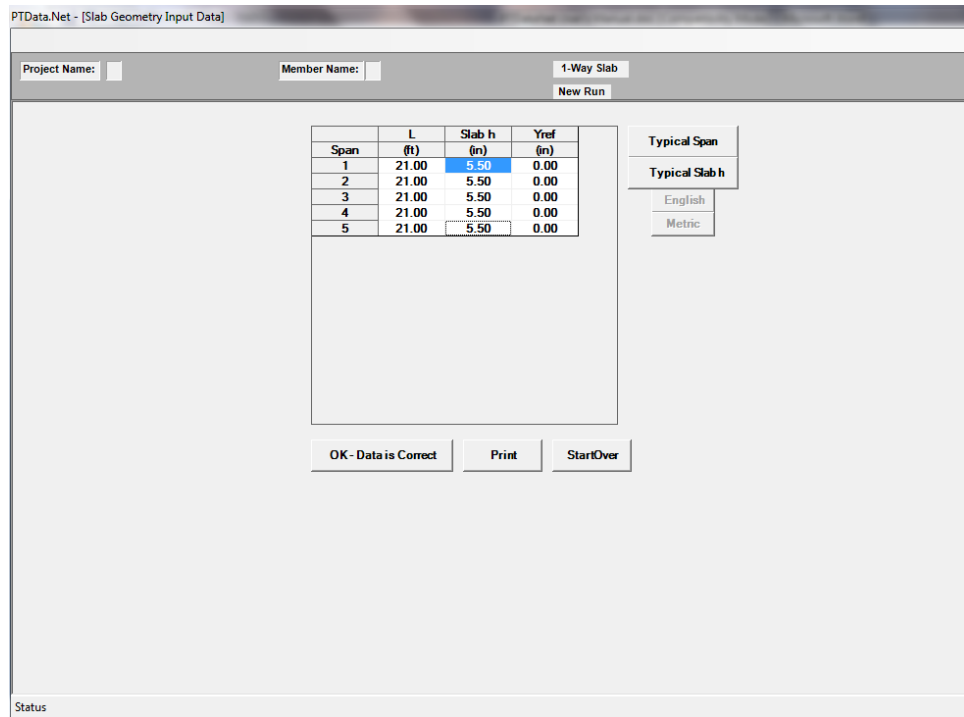


Figure 3.12 – One-Way Slab Geometry Input Screen

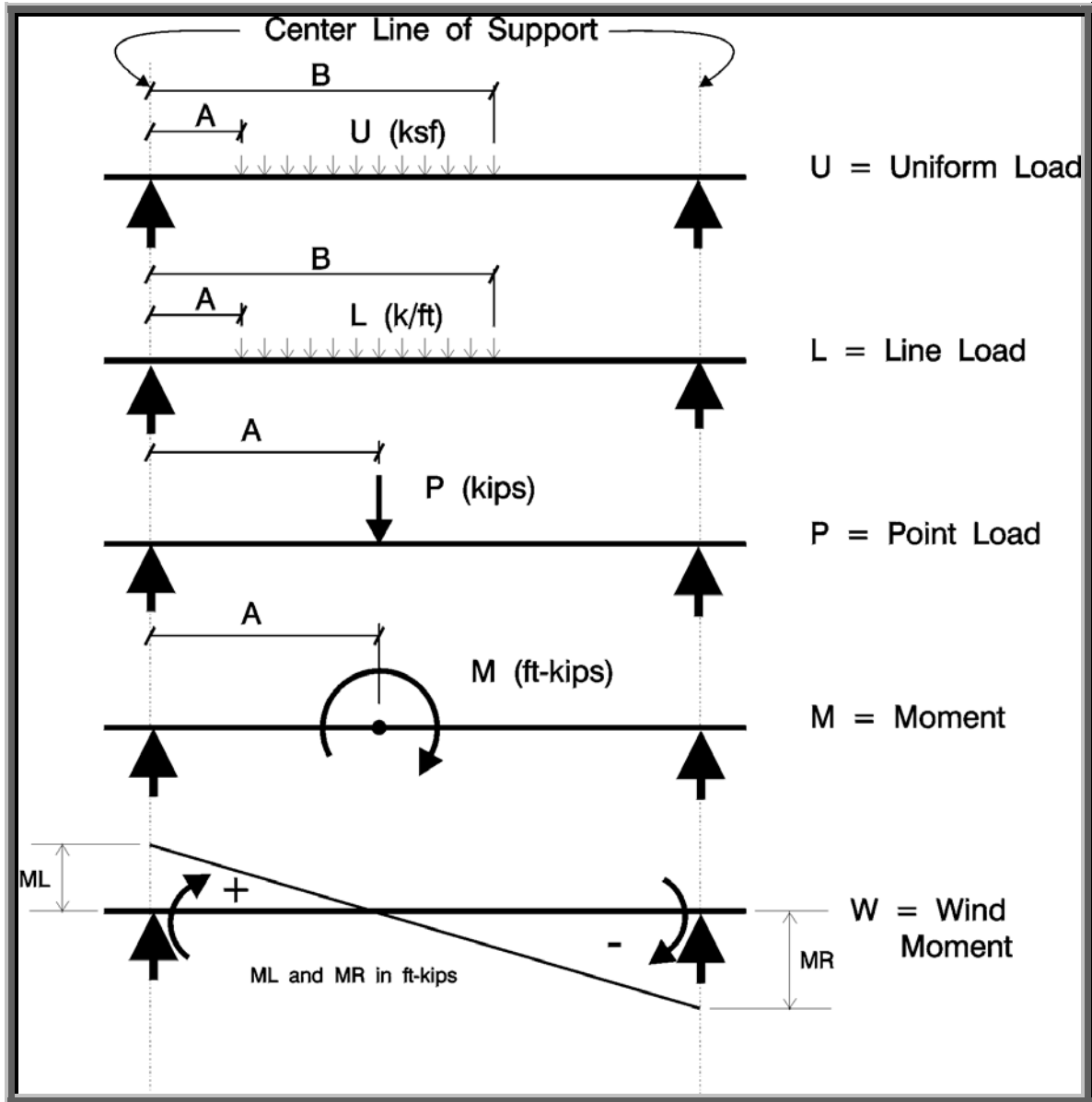


Figure 3.13 - Applied Loads

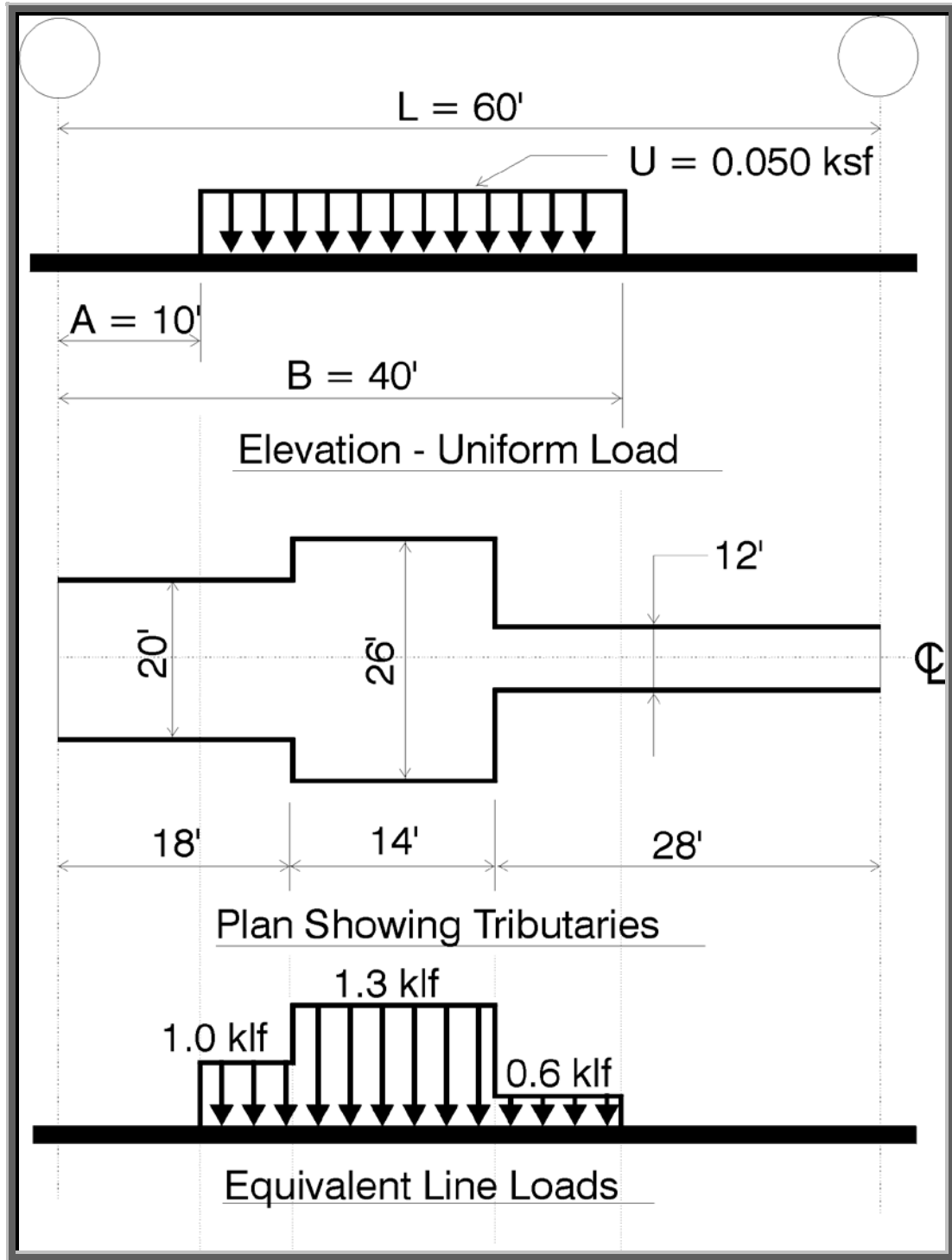


Figure 3.14 – Uniform Load (U) Example

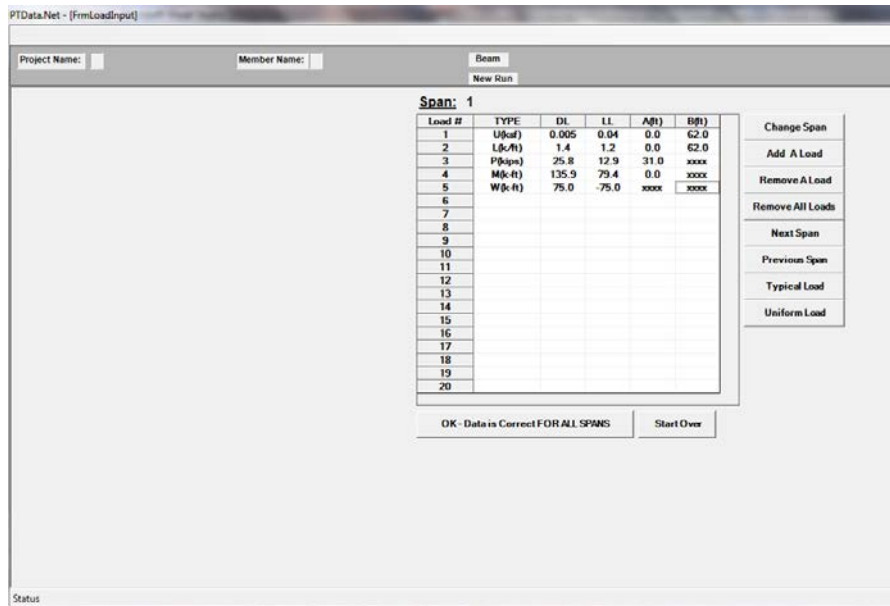


Figure 3.15 – Superimposed Load Input Screen

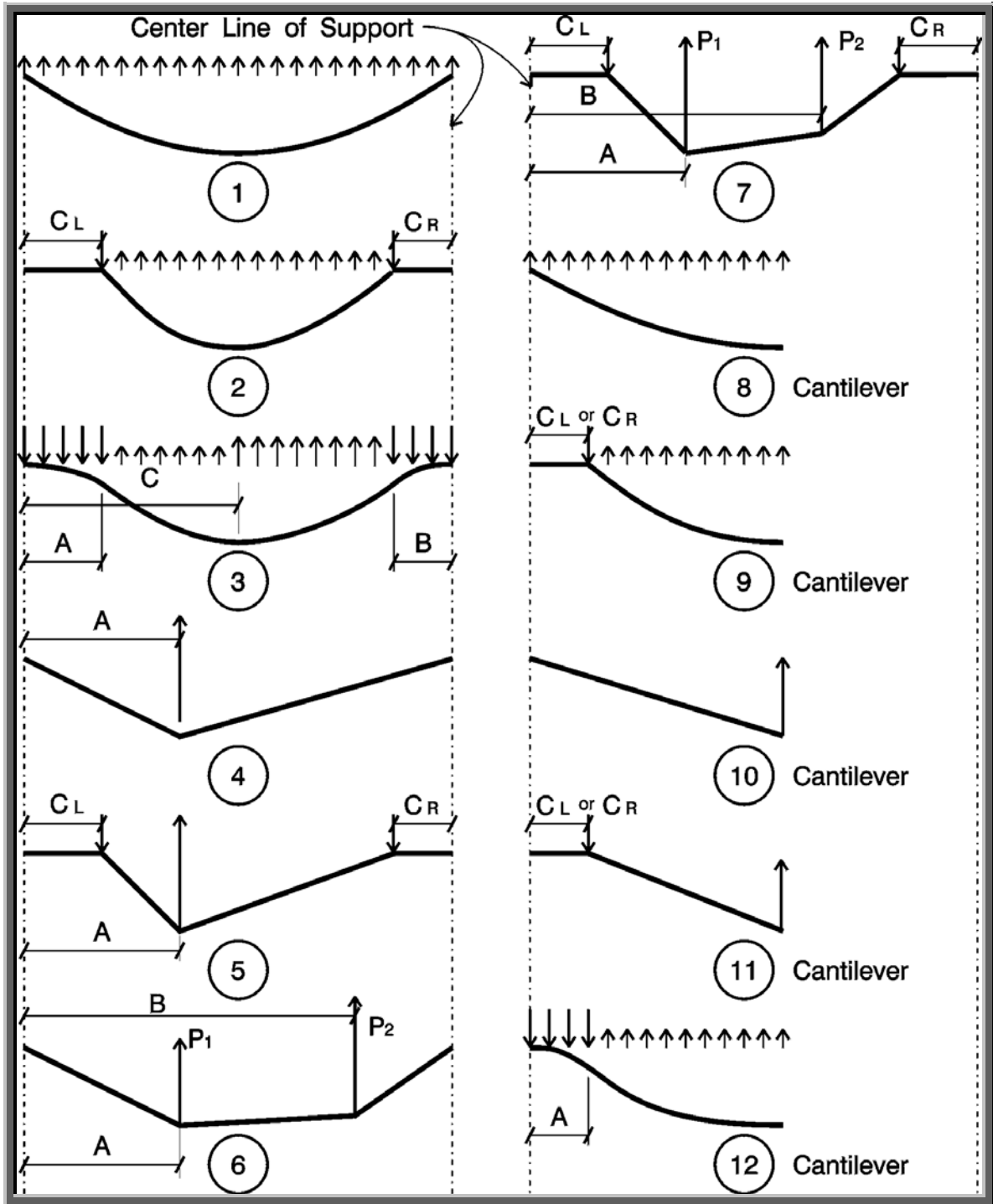


Figure 3.16 – Tendon Profile Types

PTData.Net - [Tendon Profile Input Data]

Project Name: Application Manual Member Name: Sample Beam

New Run

Tendon Profile Types

| Span | Type | cL(in) | cR(in) | A(ft) | B(ft) | C(ft) | P2/P1 |
|-------|------|--------|--------|-------|-------|-------|-------|
| LCANT | 9 | xxxx | 12.00 | xxxx | xxxx | xxxx | xxxx |
| 1 | 1 | xxxx | xxxx | xxxx | xxxx | xxxx | xxxx |
| 2 | 2 | 12.00 | 12.00 | xxxx | xxxx | xxxx | xxxx |
| 3 | 3 | xxxx | xxxx | 6.00 | 6.00 | 30.00 | xxxx |
| 4 | 4 | xxxx | xxxx | 25.00 | xxxx | xxxx | xxxx |
| 5 | 5 | 12.00 | 12.00 | 28.00 | xxxx | xxxx | xxxx |
| 6 | 6 | xxxx | xxxx | 28.00 | 40.00 | xxxx | 1.00 |
| 7 | 7 | 18.00 | 24.00 | 16.00 | 42.00 | xxxx | 0.50 |
| RCANT | 12 | xxxx | xxxx | 5.00 | xxxx | xxxx | xxxx |

OK - Data is Correct Start Over

Status

Figure 3.17 – Tendon Profile Input Screen

THE REVIEW MENU

After finalizing of the tendon forces and profiles is complete, the **REVIEW MENU** is the next available screen. The **REVIEW MENU** can also be accessed directly from the **MAIN MENU** if valid output data exists to be reviewed for the current set of input data. Valid output data includes data which has been calculated for the current set of input data, or a recalled “PTD” file which has not been modified. If such valid data does not currently exist, the **REVIEW MENU** command button in the **MAIN MENU** will be disabled. The **REVIEW MENU** is used to review *on the screen* values which were calculated for the current run (tendon forces and profiles, moments, shears, flexural stresses, deflections, unstressed reinforcing, etc.), and to modify the calculated tendon forces and profiles as desired and see the effects those modifications have on the various frame parameters. The **REVIEW MENU** is shown in Figure 4.1.

The **REVIEW MENU** contains various Options which can be selected by clicking on the appropriate command button. **REVIEW MENU** 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 and return to the **MAIN MENU**.

All **REVIEW MENU** 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.

Following is a detailed description of each of the various **REVIEW MENU** Options:

4.1 - Forces and Tendon Profiles - Review 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 **PTData.Net** in its automatic design. The screen used to display the forces and profiles in **PTData.Net** is shown in Figure 4.2. This is the only **REVIEW MENU** screen 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, **PTData.Net** recalculates all other frame values which are a function of the tendon force and profile immediately upon leaving the screen on its way back to the **REVIEW MENU**. Output data accessed from the **REVIEW 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, **PTData.Net** uses the cross-sectional area of one tendon (A_{ps}) which is input in the **General Input Data Screen** (Item 16, Section 3.3) and the value f_{se} for effective tendon stress which is input in the **Configuration Screen**. 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 **PTData.Net** 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 F_e ” or “Typical No. of Strands” command buttons.

In the automatic design procedure **PTData.Net** 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 **REVIEW MENU** 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 **PTData.Net** will adjust the force accordingly. In the example shown in Figures 4.2 the number of strands has not been adjusted to an integer value.

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 Review Screen** only for Tendon Types 6 or 7. See Figure 3.16 for a graphical description of the tendon profile dimensions.

PTData.Net 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. In the example shown in Figure 4.2 Span 3 has a single constant prestress force (420.00 kips) but two different cross-sections, thus it has different minimum and maximum average compressions. Spans 2 and 4 have two cross-sections, and they both have an added tendon “tail” from Spans 1 and 5 respectively. Thus there are also two different prestress forces present in Spans 2 and

4 (the through force of 420.00 kips and, in the portion of the span with the added tendon tail, a total force of 532.27 kips (an "added" force of 112.27 kips). Spans 2 and 4 also have different minimum and maximum compression values as shown.

To edit the data in the **Forces and Tendon Profiles Review Screen** see the procedure described in Section 3.1. If calculations for this run have been made previously, **PTData.Net** 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 REVIEW MENU windows and/or you would like to have PTData.Net 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.

To return to the **REVIEW MENU** and accept the force and profile data as shown press the "Review Menu" command button.

If any value has been changed in the Forces and Tendon Profiles Review Screen, (this includes pressing the "Previous Force and Profile" command button) PTData.Net 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 REVIEW 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 General Input Data Screen (Section 3.3) a flag will appear with a corresponding asterisk (*) in the flexural stress chart at the location of exceedence. 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.

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 F_e ; 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 in²; the top and bottom section moduli *S_t* and *S_b* in in³; and the average compressive stress *F/A* in ksi.

To leave the screen and return to the **Review Menu** press the “Review Menu” command button.

4.3 - Flexural Stress Summary Screen

If you select this Option **PTData.Net** 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 for **General Input Data** Item #6 (see Section 3.3). The heading for Left End Spans identifies those spans as the left cantilever and Span 1 (**CL - 1**) and the heading for Right End Spans identifies them as Span 5 and the Right Cantilever (**5 - CR**). The remaining Spans 2 through 4 are identified as interior spans in the heading for Interior Through Spans (**2 - 4**). The maximum tensile stress in the top beam fiber in the left end spans is 0.214 ksi and it occurs in Span 1 at a distance of 28.75 feet from the left support centerline (Joint 2). The maximum top fiber tensile stress in the interior Spans 2 through 4 is 0.275 ksi and it occurs in Span 3 at a distance of 1.25 feet from the left support centerline of Span 3 (also at Joint 3). 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 -0.465 ksi and it occurs in Span 5, 16.77 feet from the left support centerline (in Span 5 between Joints 5 and 6). At the bottom beam fiber the maximum compressive stress is -0.799 ksi and it occurs in Span 5 at a distance of 0.83 feet from the left support centerline (Joint 5).

Users must pay particular attention to the critical compressive stresses shown in this summary. The **PTData.Net** 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.

To exit the **Flexural Stress Summary Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.4 - Unfactored Beam Moments Review Screen

The **Unfactored Beam Moments Review 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 $x = 32$ feet (the midpoint of the span) the most positive live load moment possible (Max M+) is +201.43 foot-kips; the most negative live load moment possible (Max M-) is -17.82 foot-kips. For non-skipped live loads (see Section 3.3, Item #5) the moments shown in the two live load columns will be equal.

To exit the **Unfactored Beam Moments Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**

4.5 - Unfactored Beam Shear Review Screen

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.

To exit the **Unfactored Beam Shear Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.6 – Flexural Concrete Stresses Review Screen

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 **PTData.Net** 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 $x = 32.02$ feet the stress at the **top** beam fiber caused by the **most negative moment** (Max M-) which can occur at that design point is -0.470 ksi. The stress at the top beam fiber caused by the **most positive moment** (Max M+) which can occur at that design point is -708 ksi. Thus the **range** of flexural stresses which can occur at the top beam fiber at $x = 32.02$ feet is from -0.470 ksi in compression to -708 ksi in compression. Similarly at the **bottom** of the beam at $x = 32.02$ feet the flexural stresses can range between -245 ksi under the **most negative moment** possible and $+0.370$ 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 **PTData.Net** 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**. **PTData.Net** 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. **PTData.Net** 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 1.8 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 **PTData.Net** tensile stresses are positive in sign and compressive stresses are negative in sign.

In the example shown in Figure 4.7, at $x = 63.11$ feet (the face of the right support) the flexural stress caused by transfer loads at the **top** beam fiber is -598 ksi. At the **bottom** of the beam at $x = 63.11$ feet the flexural stress is -0.158 ksi. The **PTData.Net** automatic design procedure is based upon limiting **service load**, not transfer stresses. The user must verify that the transfer stresses, as calculated by **PTData.Net** and displayed in this screen, are acceptable.

To exit the **Flexural Concrete Stresses Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.7 - Deflection and Cracking Moment Review 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 M_{cr} Top and M_{cr} Bot. The cracking moment is the applied moment at the design point which, when combined with the effects of prestressing ($-F/A \pm M_{ba}/S$), 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 $7.5\sqrt{f'_c}$.

To exit the **Deflection and Cracking Moment Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.8 - Tendon Balanced Load & Concrete Dead Loads Review 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 Review 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 **PTDdata.Net**. 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), PTDData.Net 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.

To exit the **Tendon Balance Load & Concrete Dead Load Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.9 - Factored Load Rebar Review 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. **PTData.Net** 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 M_{design} possible and the required areas of tensile reinforcement (A_s) and compression reinforcement (A'_s) at the top and bottom of the beam, and the ultimate tendon stress f_{ps} used in the strength calculation at that design point. Note that M_{design} 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 **General Input Data Screen**.

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.

To exit the **Factored Rebar Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

4.10 - DL+LL/4 Rebar Review Screen

This screen, shown in Figure 4.15 for a case with $\%R=15\%$, is similar to the **Factored Load Rebar Review 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 **General Input Data Screen**.

4.11 - Beam Shear Design Review Screen

This screen, shown in Figure 4.16, reviews the major parameters in the beam shear design. The **Beam Shear Design Review 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 **General Input Data Screen**, 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 **PTData.Net** 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.

To exit the **Beam Shear Design Review Screen** press the "Review Menu" command button and **PTData.Net** will return to the **REVIEW MENU**.

4.12 - Punching Shear Stress Review 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:

- A_c - Area of the critical section.
- J_c - 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 **PTData.Net** 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 ϕ factor depending upon which Code is specified (1997 UBC or ACI 318-08+). If an applied stress exceeds the allowable stress at any joint, **PTData.Net** will alert the user to this condition with a warning banner at the bottom of the screen.

PTData.Net 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 Review 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 **PTData.Net**, if any applied stress exceeds the allowable stress, not only will the warning banner appear but the “Design Capitals” command button will be enabled.

Press this button and **PTData.Net** 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, **PTData.Net** 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 Capitals” 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 **PTData.Net** 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, **PTData.NET** 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. **PTData.Net** 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 **PTData.Net** will automatically modify the capital dimensions to occur on the side of the column without the slab band. Without any capital geometry being entered, **PTData.Net** 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 **PTData.Net** 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.

To exit the **Punching Shear Analysis Review Screen** press the “Review Menu” command button and **PTData.Net** will return to the **REVIEW MENU**.

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.

4.15 - Return to the Main Menu Option

Use this Option to return to the **MAIN MENU** for changing input data in the current run or starting a new run.

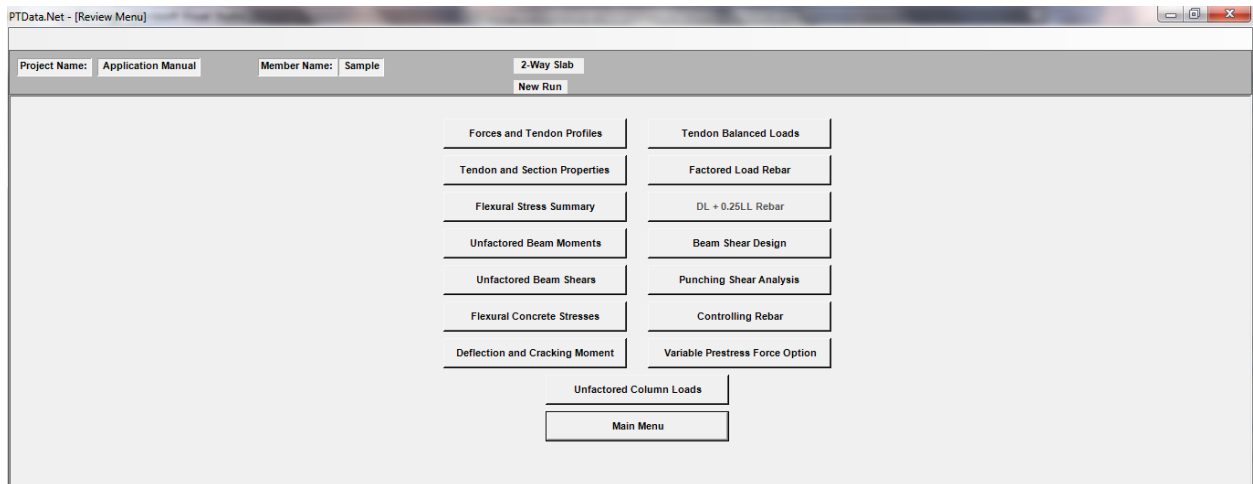


Figure 4.1 – The Review Menu

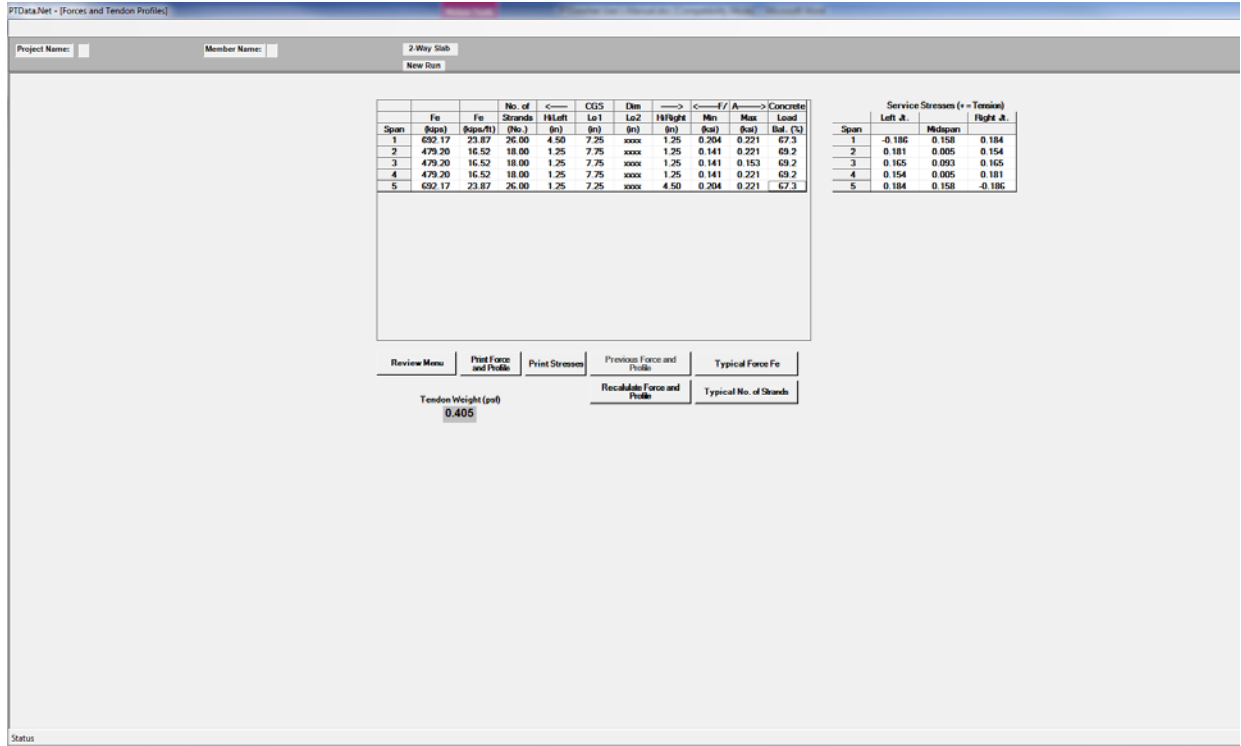


Figure 4.2 – Forces & Tendon Profiles Review Screen

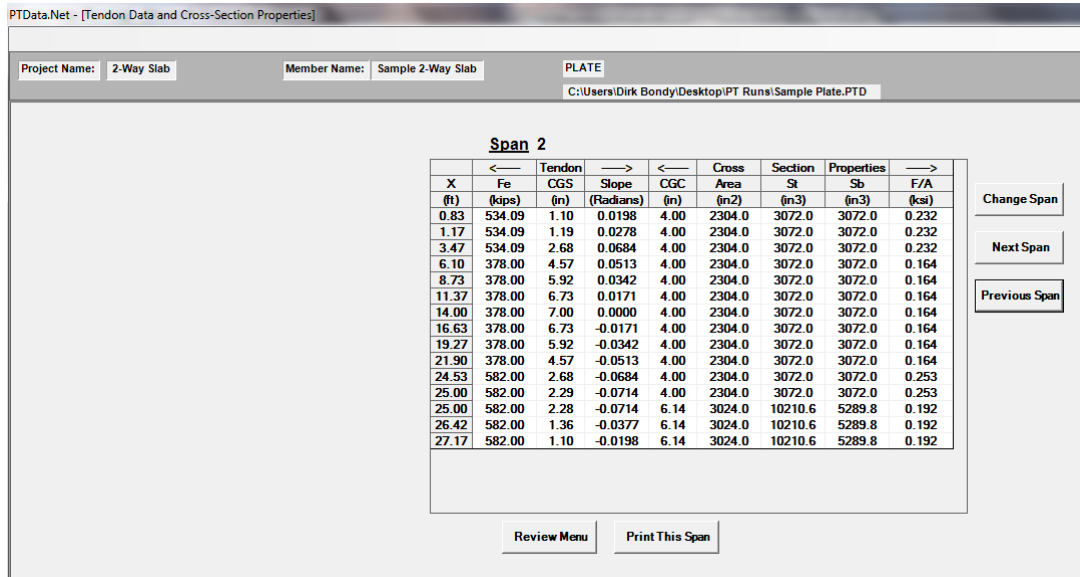


Figure 4.3 – Tendon Data & Section Properties Review Screen

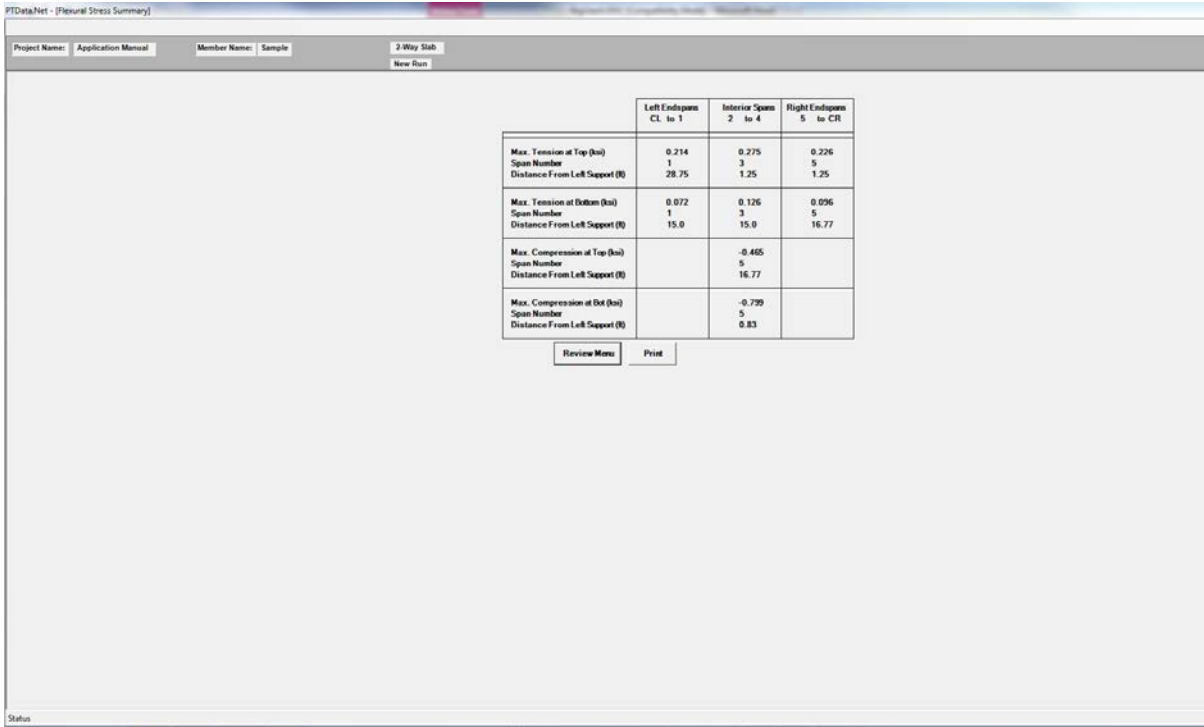


Figure 4.4 – Flexural Stress Summary Review Screen

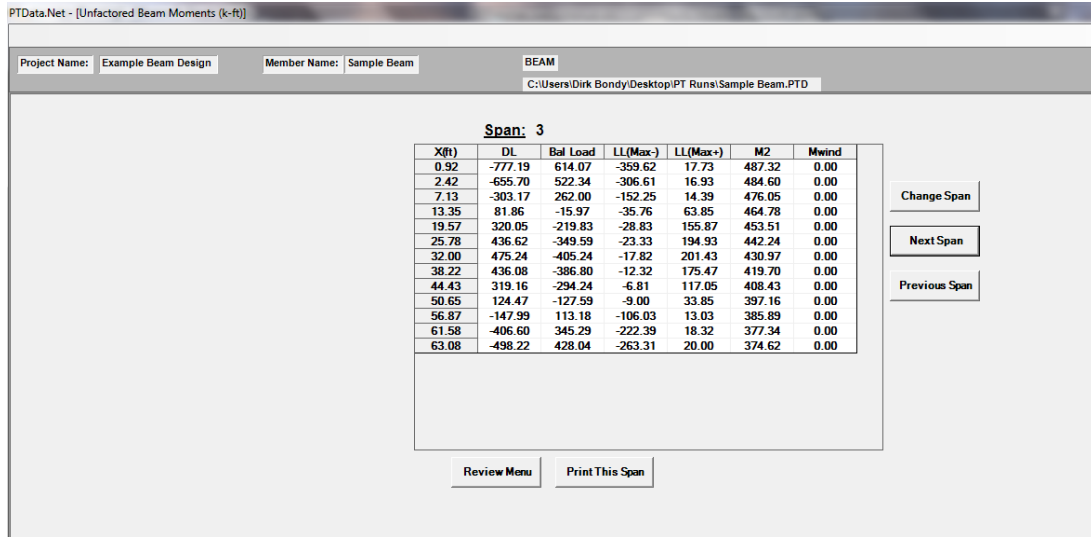


Figure 4.5 – Unfactored Beam Moments Review Screen

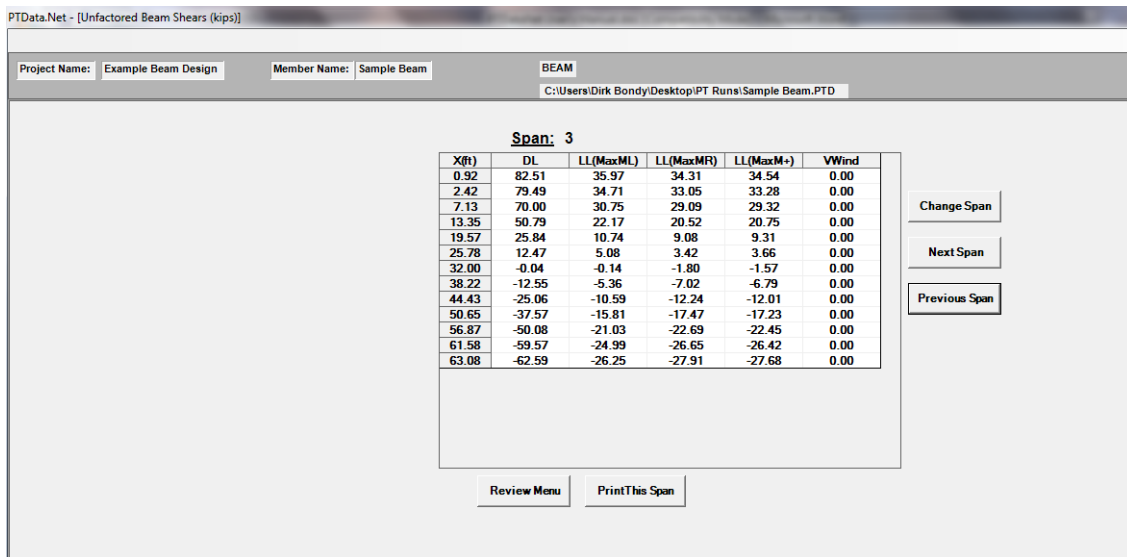


Figure 4.6 – Unfactored Beam Shears Review Screen

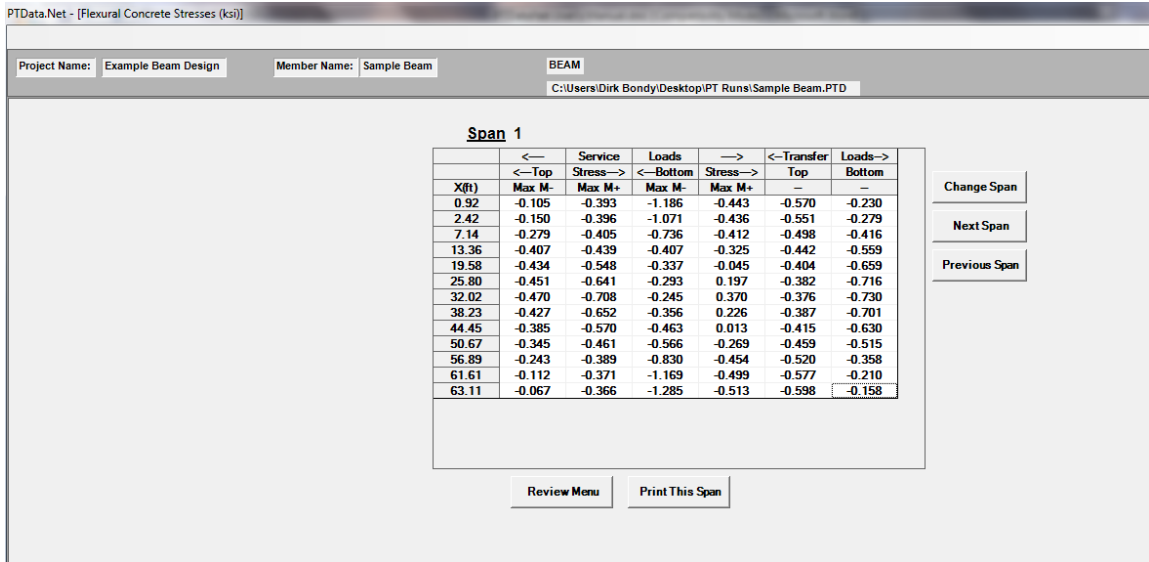


Figure 4.7 – Flexural Concrete Stresses Review Screen

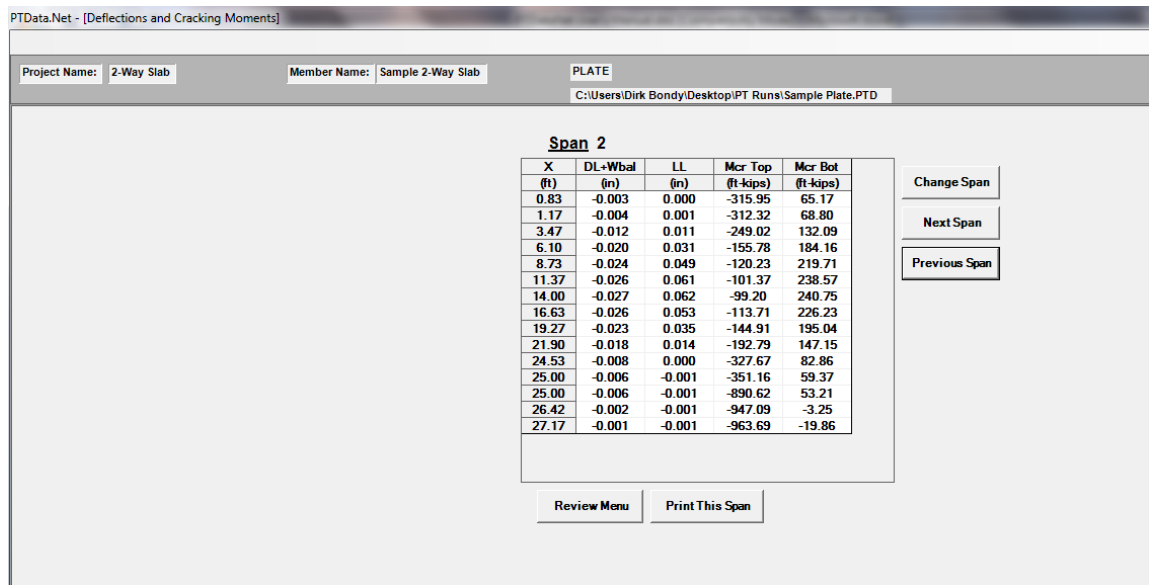


Figure 4.8 – Deflections and Cracking Moments Review Screen

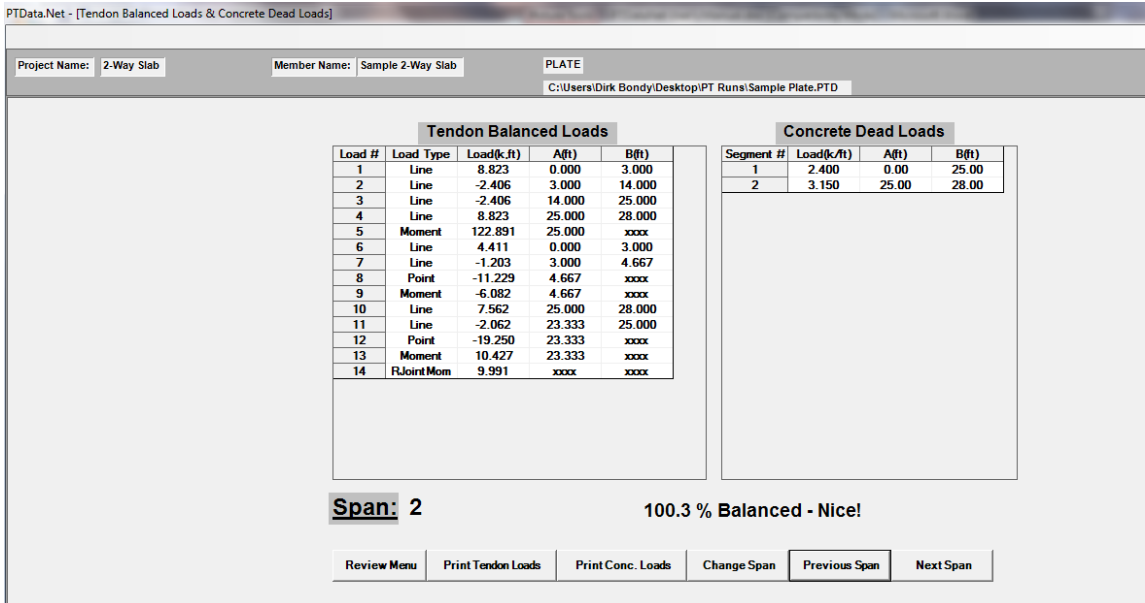


Figure 4.9 – Tendon Balanced Load & Concrete DL Review Screen

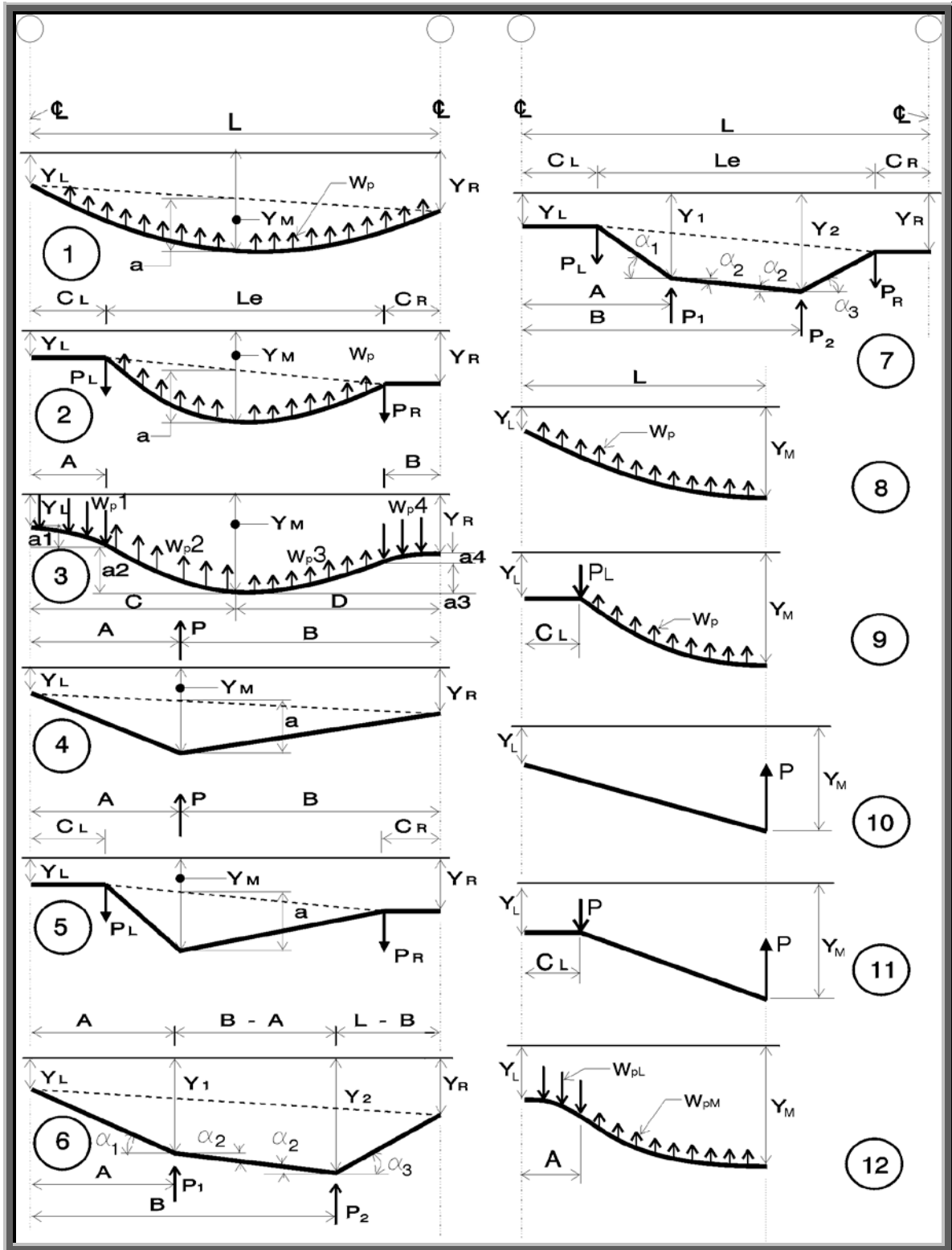


Figure 4.10 – Tendon Curvature Loads

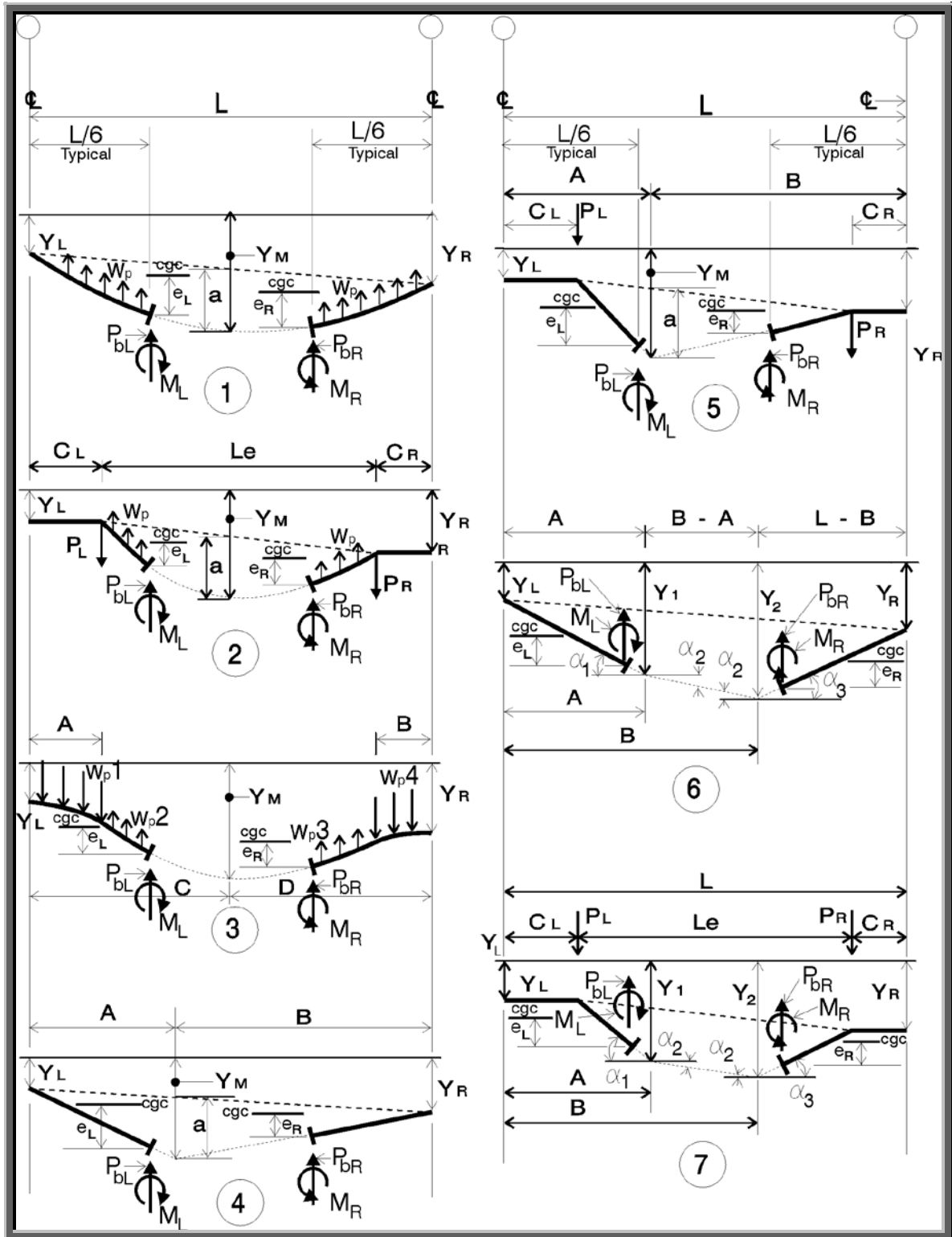


Figure 4.11 – Added Tendon Curvature Loads - Spans

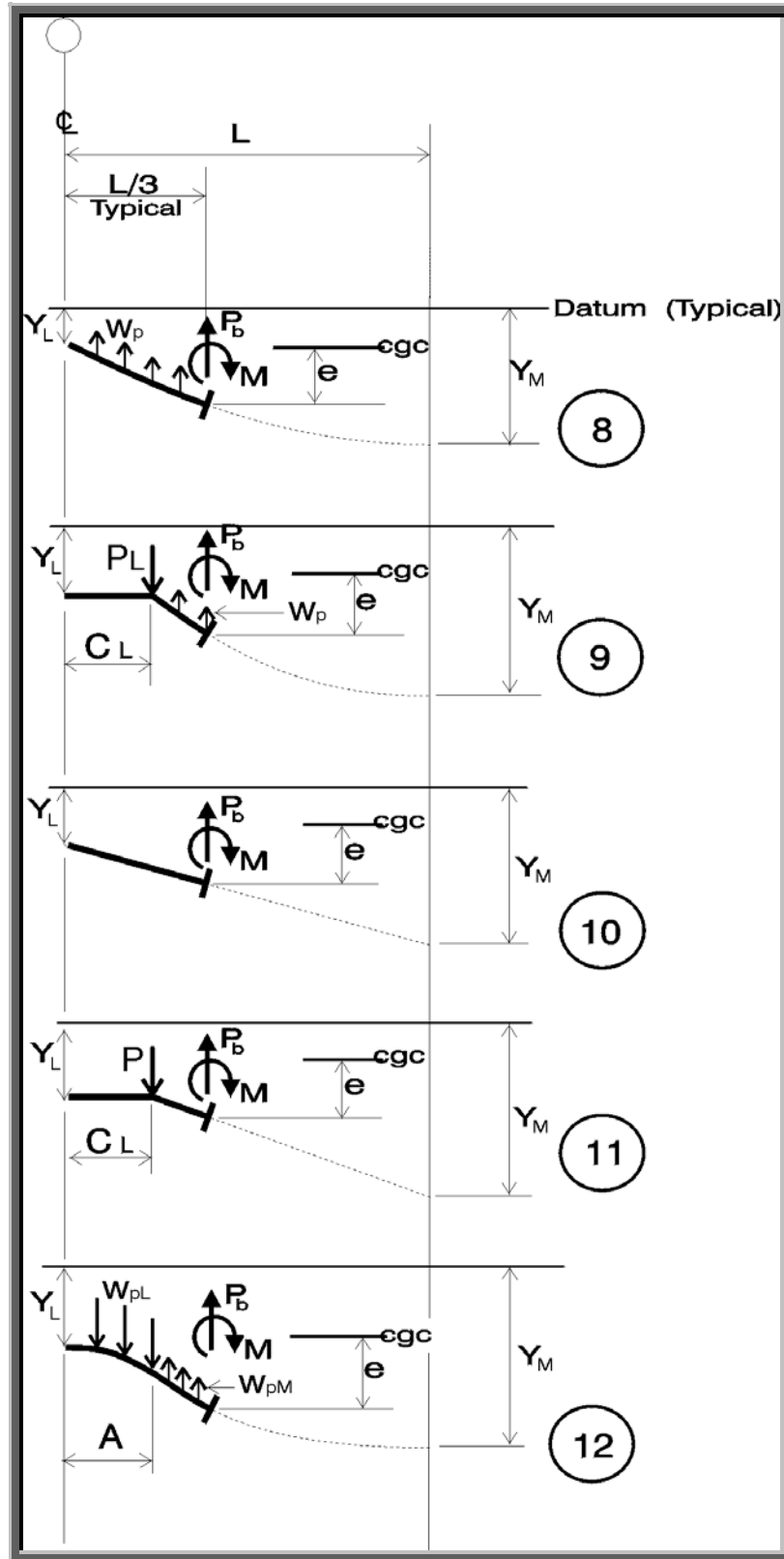


Figure 4.12 – Added Tendon Balanced Loads - Cantilevers

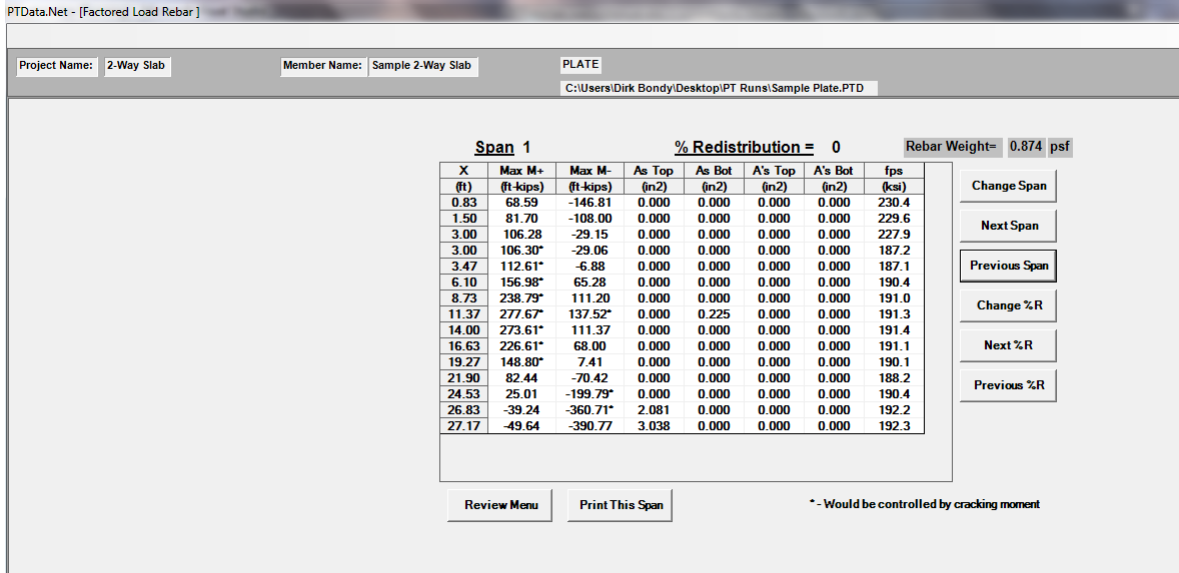


Figure 4.13 – Factored Load Rebar Review Screen (%R=0)

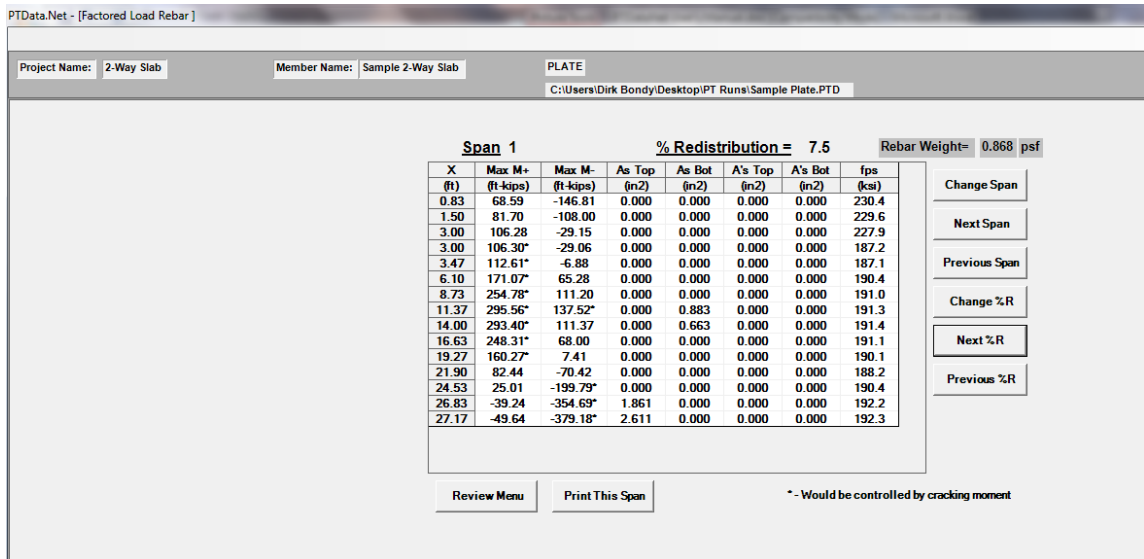


Figure 4.14 – Factored Load Rebar Review Screen (%R=7.5)

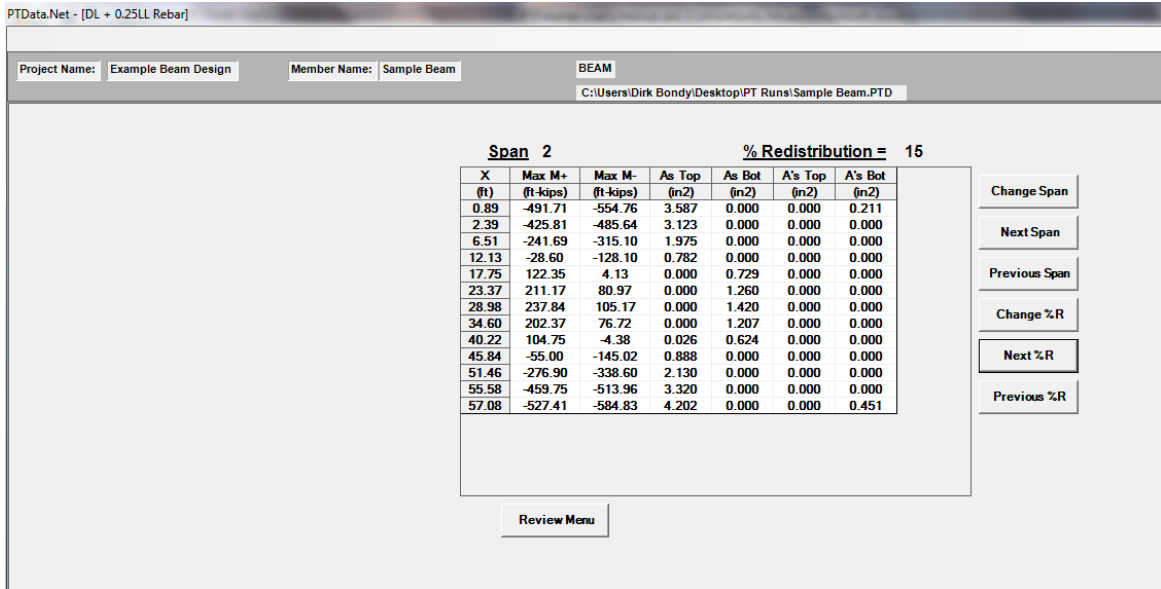


Figure 4.15 – DL+0.25LL Rebar Review Screen (%R=15)

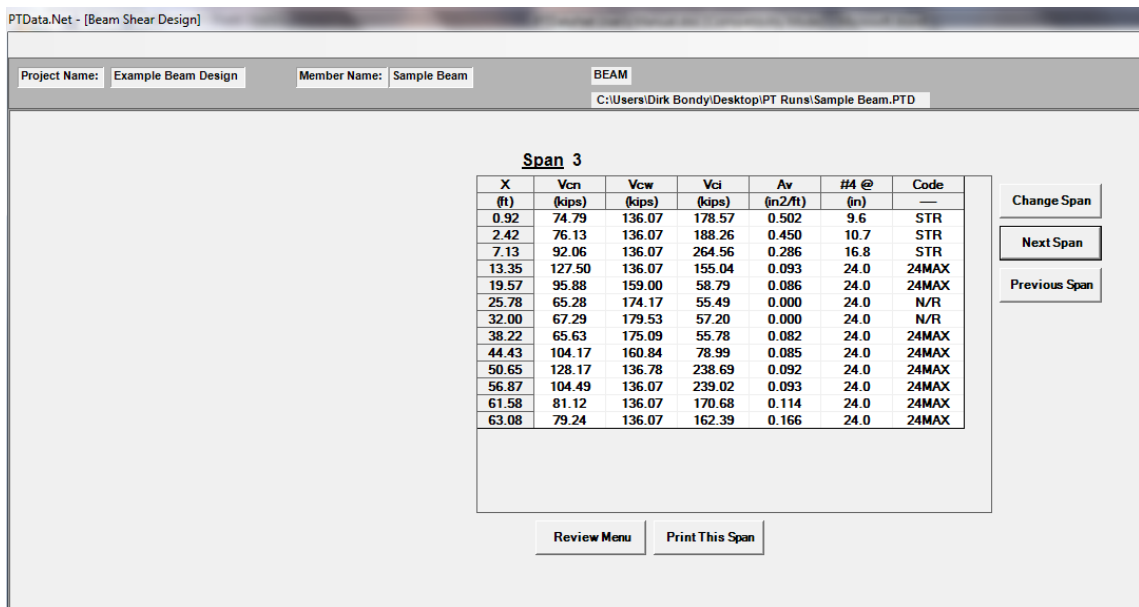


Figure 4.16 – Beam Shear Design Review Screen

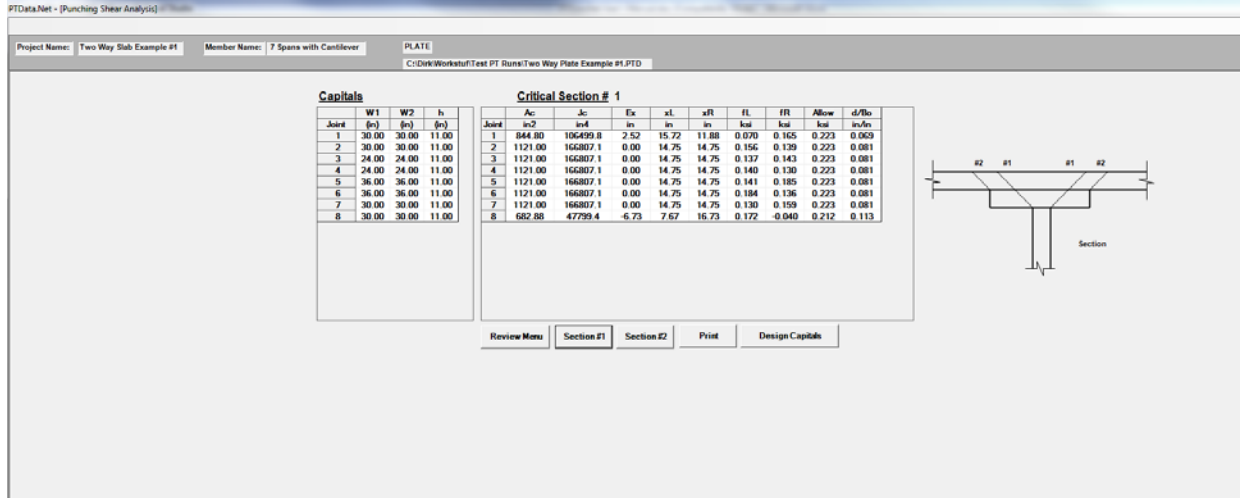


Figure 4.17 – Punching Shear Analysis Review Screen – Sect #1

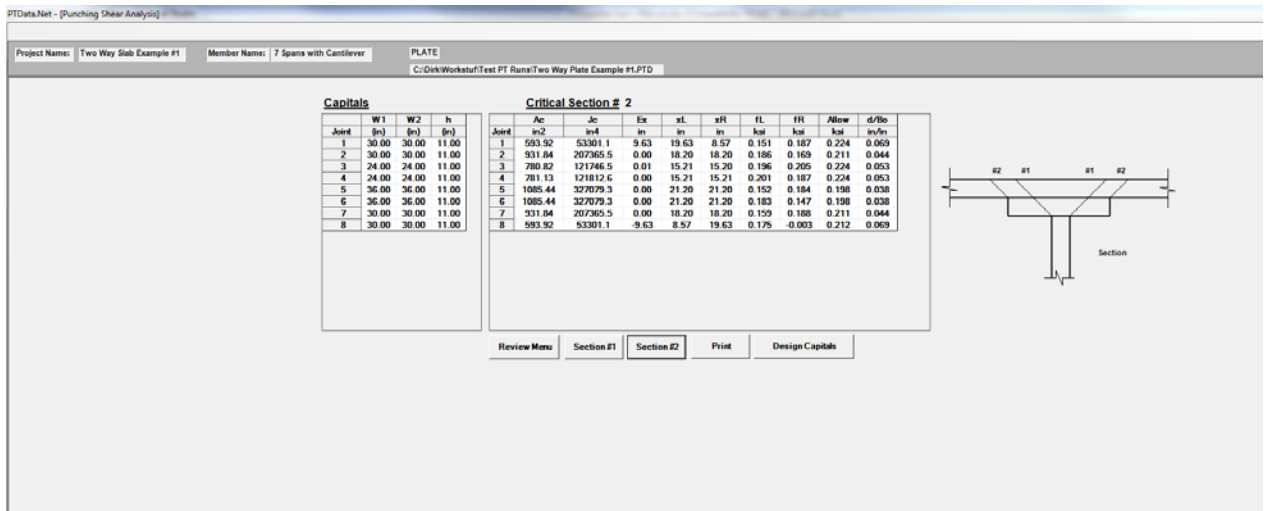


Figure 4.18 – Punching Shear Analysis Review Screen – Sect #2

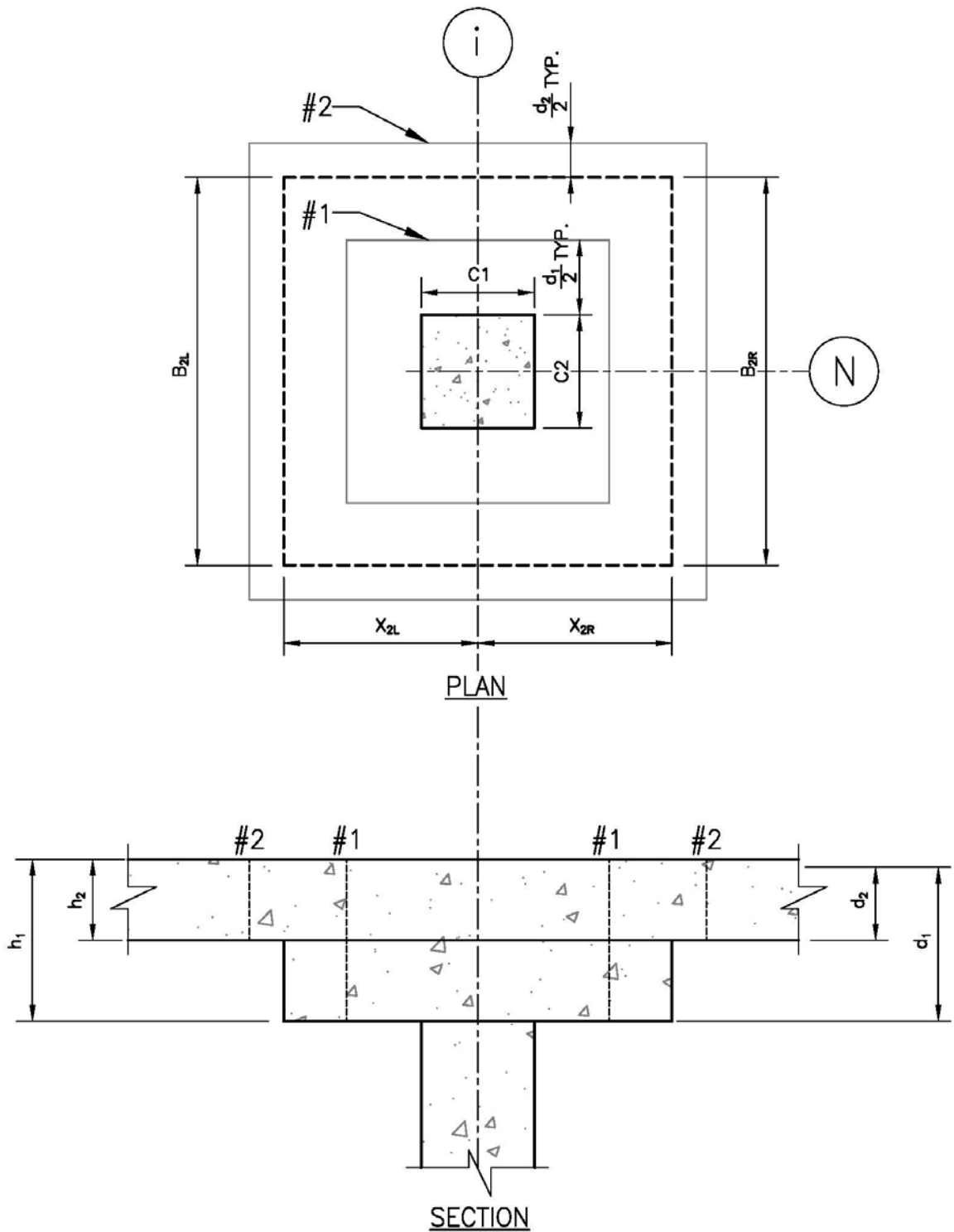


Figure 4.19 – Critical Punching Shear Sections

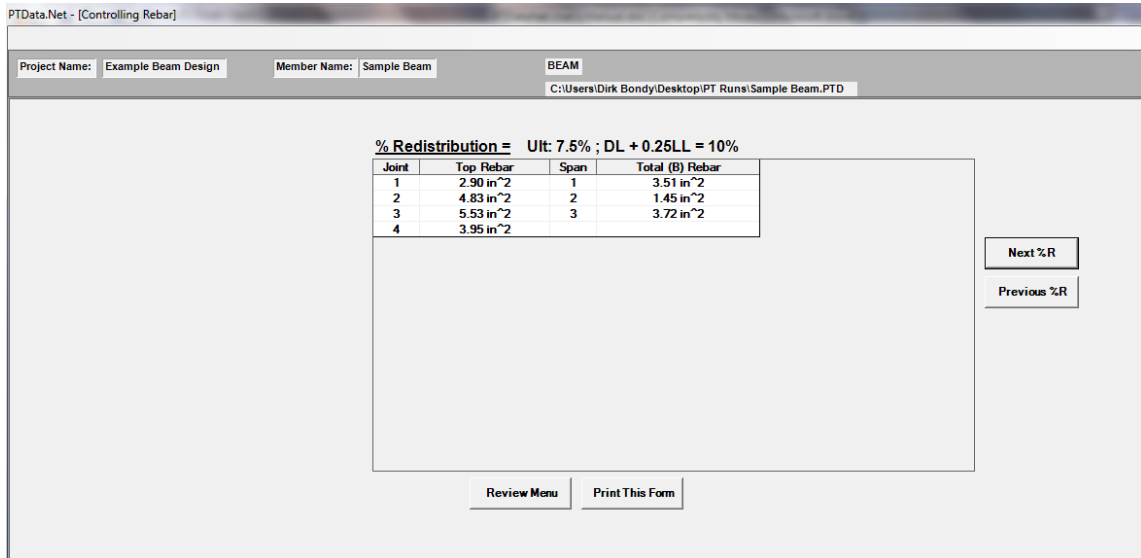


Figure 4.20(a) – The Controlling Rebar Screen for Beams

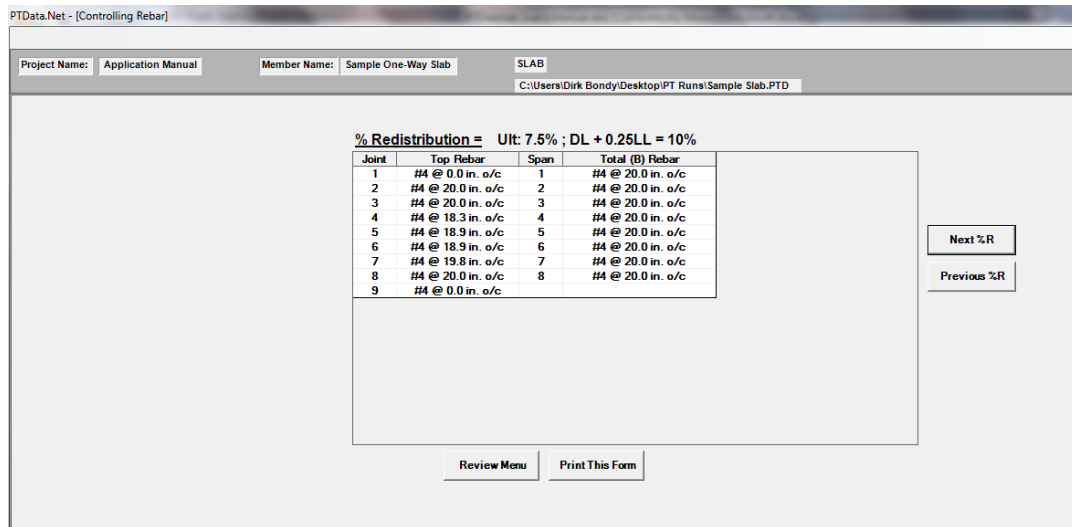


Figure 4.20(b) – The Controlling Rebar Screen for One-Way Slabs

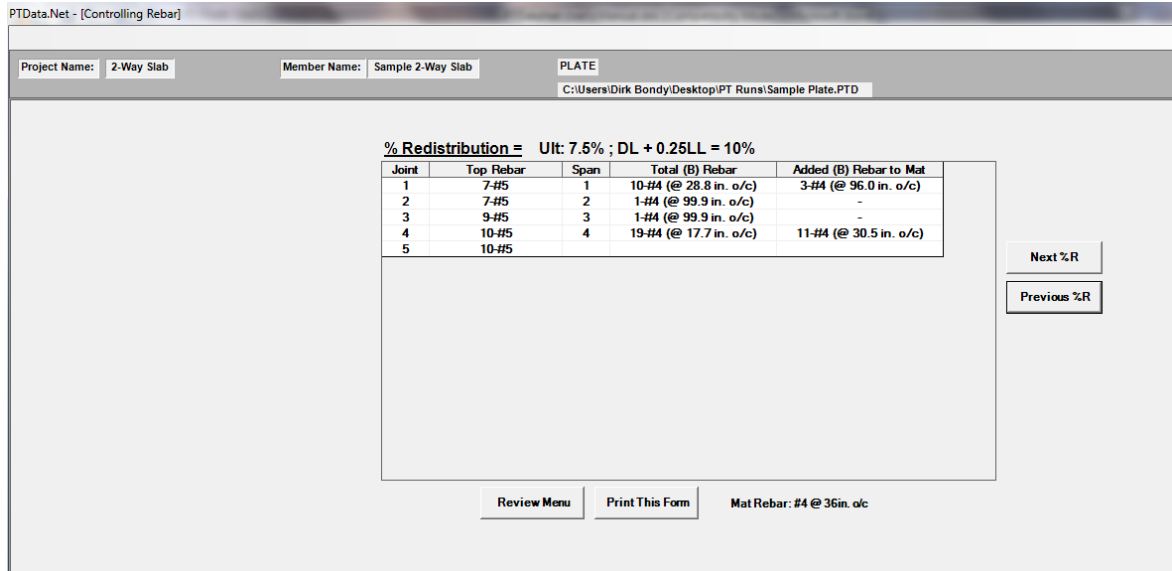


Figure 4.20(c) – The Controlling Rebar Screen for Two-Way Slabs

THE VARIABLE PRESTRESS FORCE OPTION

The **Variable Prestress Force** Option is selected from the **REVIEW 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 **PTData.Net** 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

PTData.Net 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. **PTData.Net** 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 μ** The value μ in the friction loss equation shown in Item #1 above. The curvature coefficient μ is a multiplier of α , 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. **PTData.Net** 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. **PTData.Net** 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. **PTData.Net** 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. **PTData.Net** 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 Review Menu

Once all of the data required for the **Variable Prestress Force** mode is entered, **PTData.Net** proceeds to the **Variable Prestress Force Review 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 **REVIEW MENU**, and when all data is correctly input, start the Variable Prestress Force Calculations. When the calculations are completed, the **REVIEW MENU** will appear and the output data can be reviewed.

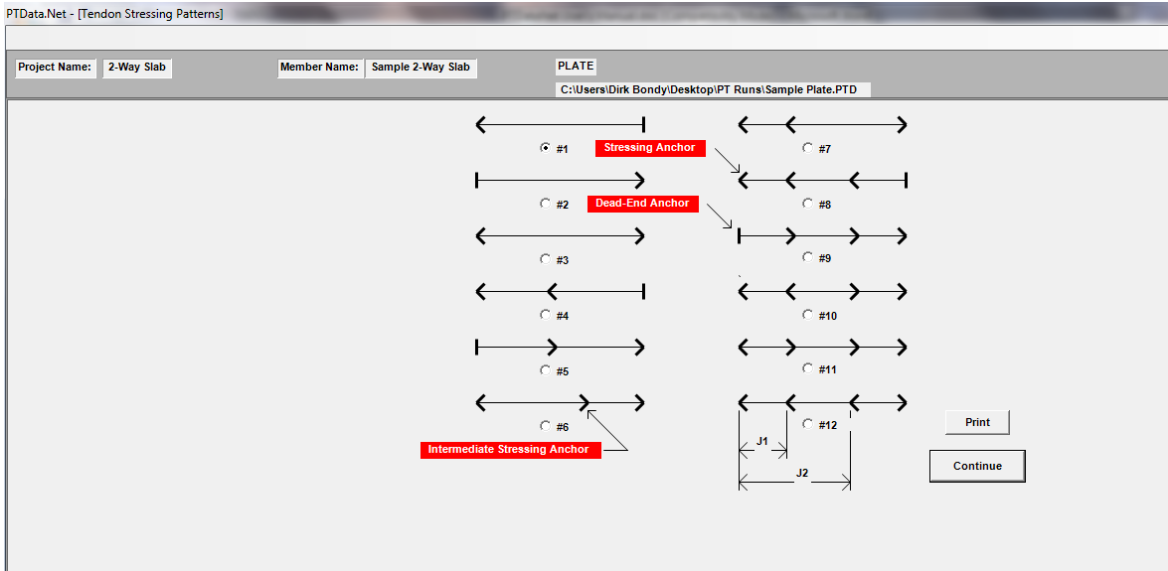


Figure 5.1 – Tendon Stressing Patterns Input Screen

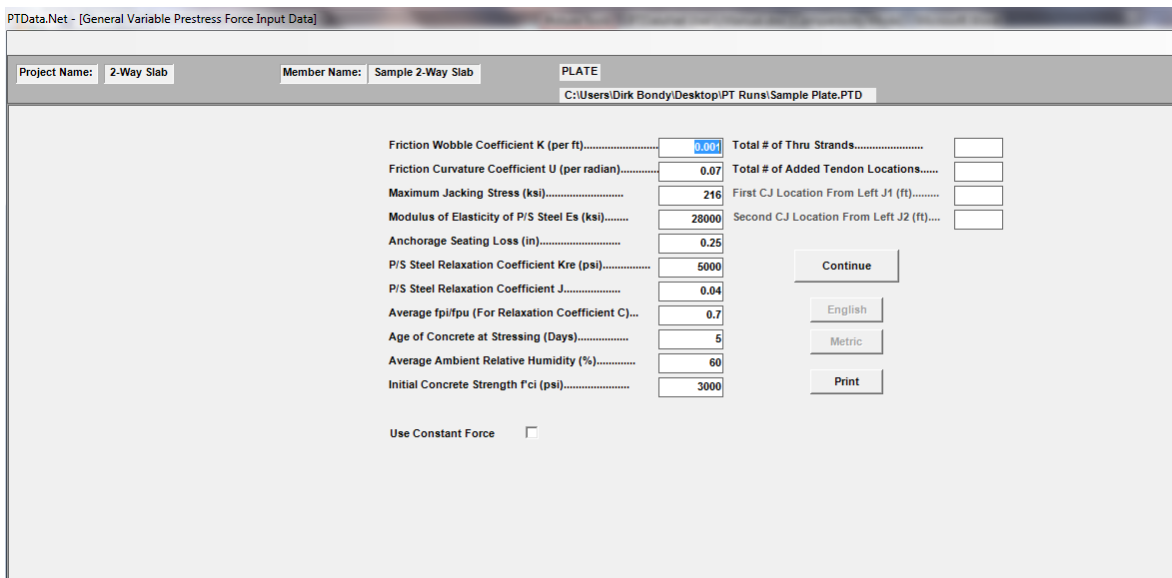


Figure 5.2 – General Variable Prestress Force Input Data Screen

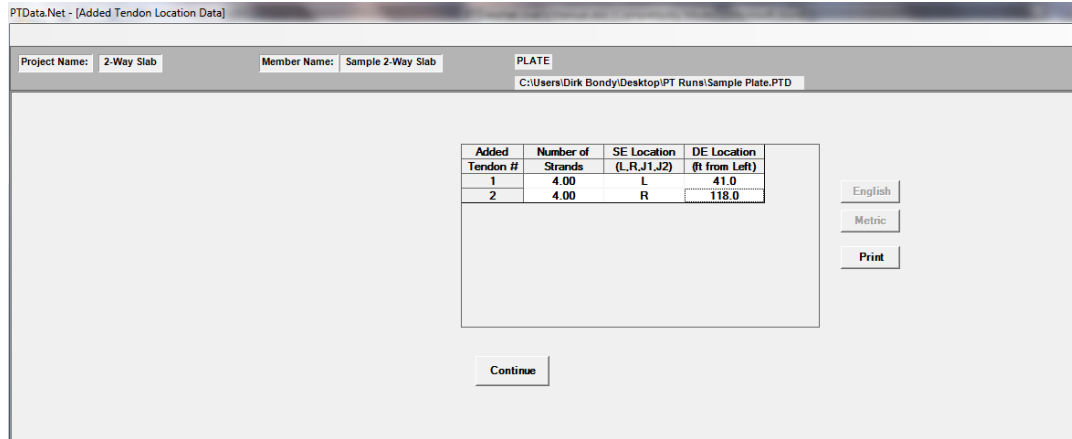


Figure 5.3 – The Added Tendon Location Input Screen

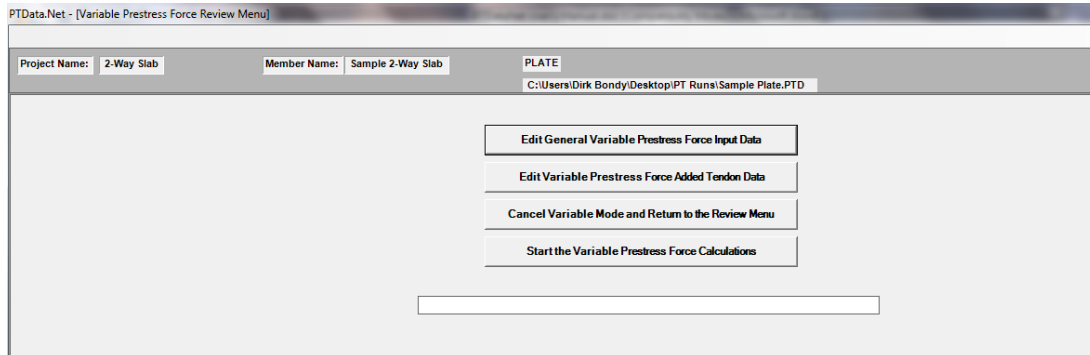


Figure 5.4 – Variable Prestress Force Review Menu