ADAPT-PT/RC 2019

Getting Started Guide

ADAPT PT Mode – Two-way Slab

Indian Code of Practice

December 2019

1. INTRODUCTION

This ADAPT-PT/RC 2019 Getting Started Guide is intended to be used as a practical example and guide for modelling a 2D post-tensioned two-way slab frame in the PT mode of the program.

While the example is related to a specific system type (two-way slab) the workflow applies to input of one-way slabs and beams.

For additional information, refer to the ADAPT-PT/RC 2019 Getting Started Guide for RC mode and the ADAPTPT/RC 2019 User Manual. Both documents can be accessed from the HELP menu of the program.

After installation of PTRC 2019 is completed, user should have a desktop icon of the same.

It is *recommended* to right click on the desktop icon and choose Run as administrator option to launch the program in administrator mode. The opening screen will be as shown below.

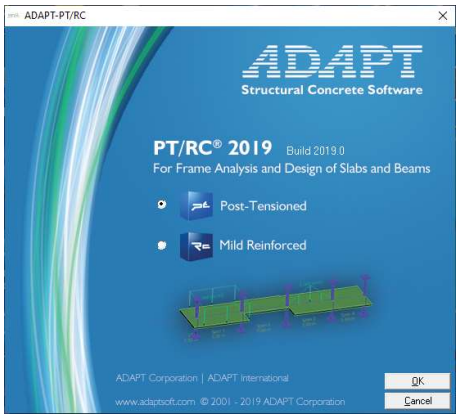


FIGURE 1-1 PTRC 2019 OPENING SCREEN

This screen gives the option to select a design mode. Select the “Post-Tensioned” option and click on “OK” to open ADAPT-PT/RC 2019 basic screen.

ADAPT PTRC is a 2D frame analysis solution. The GUI consists of a main window with menu bars and toolbars which are used to feed input data into wizard-based *Input Forms* to create the model.

This model is then solved, and result & reports generated from the solution in different user-friendly formats. The typical GUI of PTRC basic screen are as shown in the nest figure.

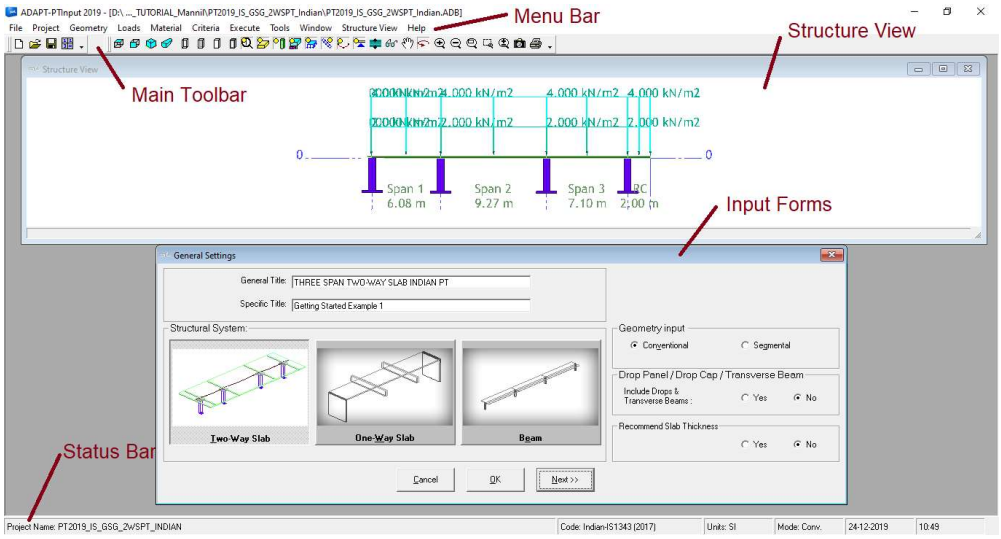


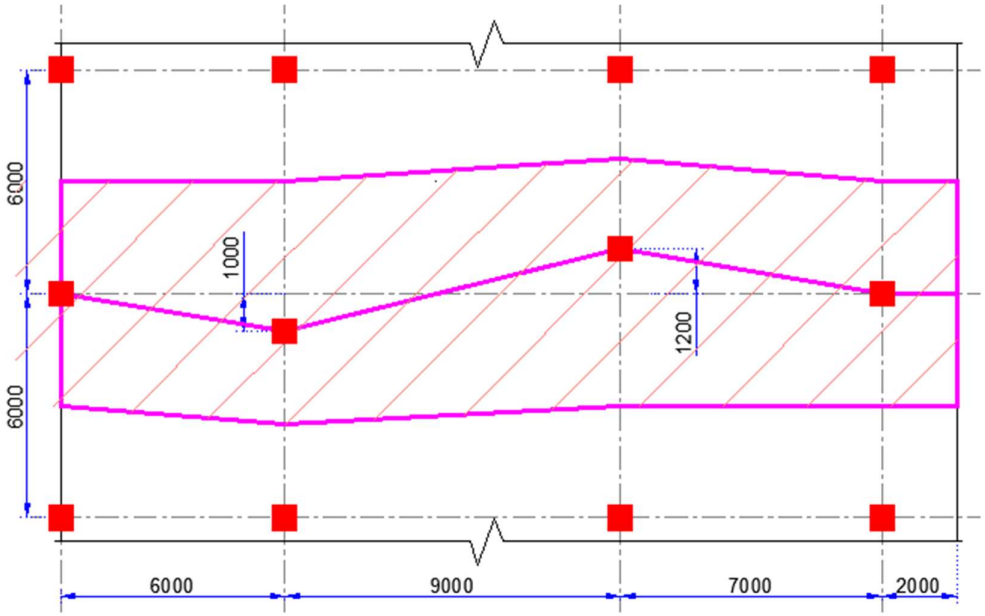
FIGURE 1-2 ADAPT PTRC USER INTERFACE

2. TUTORIAL PROBLEM

The objective of this tutorial is to explain how a floor strip or frame line is idealized from a complete floor system and modelled as a slab or beam-frame ADAPT-PT/RC 2019.

This tutorial will demonstrate the step-by-step procedure in the PT mode of ADAPTPT/RC 2019 to generate data, analyse and design a column-supported slab which is a part of a floor system.

The following figure shows a typical floor system with the shaded part of the strip selected for present design. This is a 3-span column supported slab with cantilever extensions on both sides.



NOTE : ALL COLUMN SIZE 650mm X 650mm.

FIGURE 2-1 PART PLAN OF SLAB SHOWING STRIP

The centre line lengths and tributary widths are identified from the plan which is as shown in following figure.

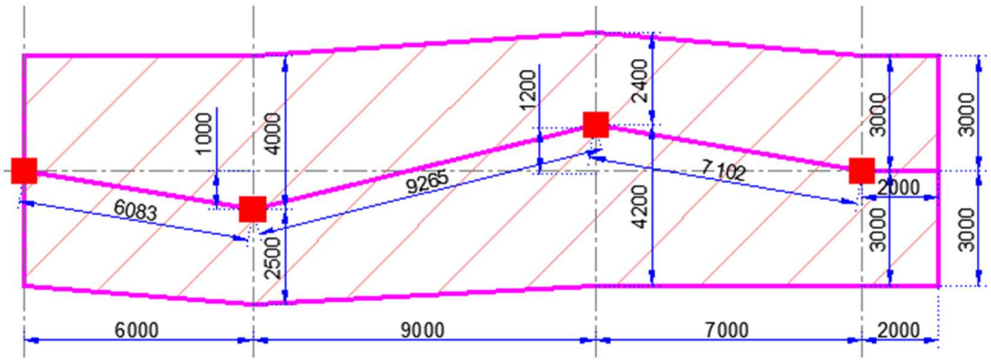


FIGURE 2-2 STRIP DETAILS

Next, user needs to idealize the design strip widths along each span on both sides as shown in the following figure.

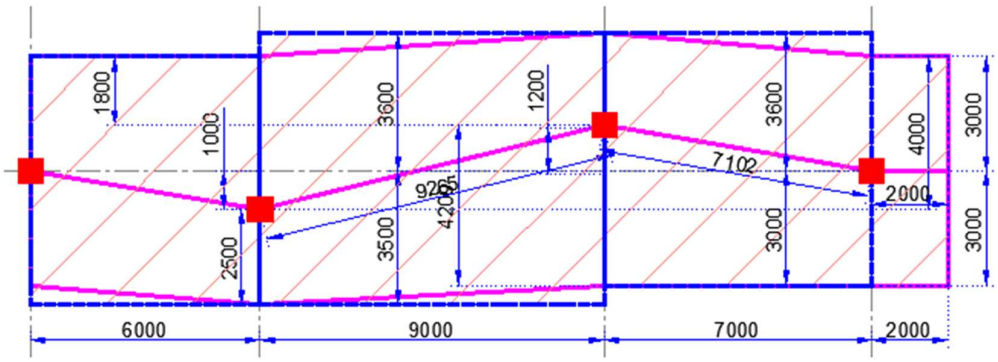
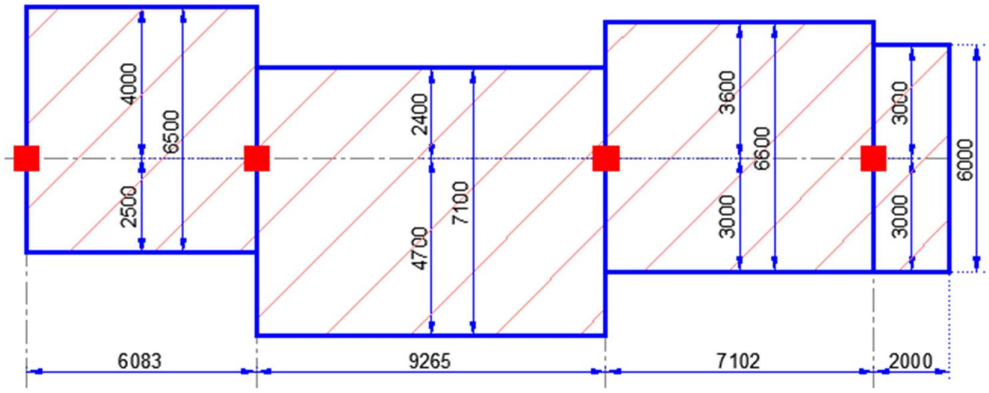


FIGURE 2-3 SLAB IDEALIZED STRIP DETAILS

Finally, inclined spans need to be straightened as shown in figure hereafter.



NOTE : ALL COLUMN SIZE 650mm X 650mm.

FIGURE 2-4 SLAB IDEALIZED STRIP FOR PT/RC INPUT

The following table provides design specifications for this model. Other data if required will be assumed during the design process.

TABLE 2-1 DESIGN SPECIFICATIONS

Items	Descriptions
Design Code	• IS 456:2000 (Reaffirmed in February 2016) • IS 1343:2012 (Reaffirmed in November 2017)
PT Design Classification	• Type 3 • Allowable Crack Width = 0.2 mm
Geometry Details	• All Slab Depth = 200 mm • All slab end conditions = Interior
Story Heights	• Both Top & Bottom Story Height = 3 m
Loading Details	• Uniform loads : SDL = 2 kN/m^2, LL = 4 kN/m^2
Material Design Grades	• Concrete = M40 for all members • Reinforcement = Fe500 • Post Tensioning = Low Relaxation Bonded System • Strand Area = 98 mm^2 • Ultimate strength, f_{pu} = 1860 N/mm^2 • Effective (long term) stress, f_{se} = 1200 N/mm^2
Reinforcement Details :	• Mesh Reinforcement = 8mm dia. @ 300 mm c/c Both-ways at Bottom only • Main Reinforcement = 12 mm dia. Top & Bottom • Shear Reinforcement = 10 mm dia. Stirrups
Minimum Cover :	• 25 mm clear cover for reinforcements • 40 mm cover to CGS for tendons

2.1. INPUT AND MODEL GENERATION

2.1.1. General Settings & Design Code

Open a new project by clicking either the "New Project" icon on the Main Toolbar or on the menu **File | New**. This automatically opens the General Settings input screen. Select "***Two-Way Slab***" as structural system and other details as shown in figure hereafter.

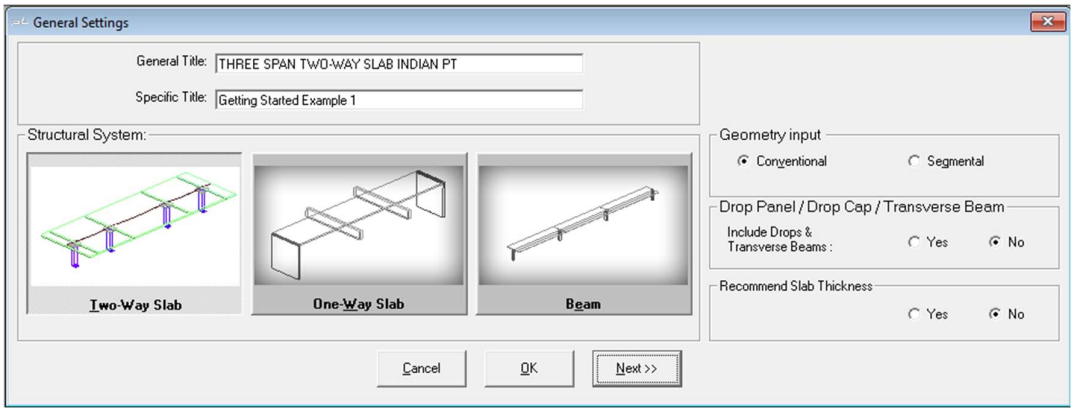


FIGURE 2-5 GENERAL SETTINGS INPUT SCREEN

Note that the "Title" inputs are not mandatory and are merely for documentation purpose only. Click on **Next >>** button to continue.

This opens the Design Code selection option screen. Select **Indian – IS1343 (2017)** option as shown in figure hereafter and click on **Next >>** button to continue.

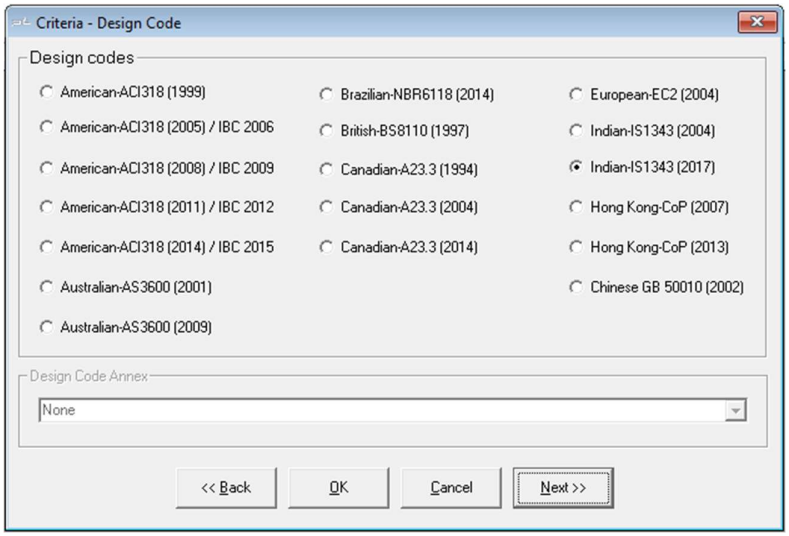


FIGURE 2-6 DESIGN CODE SELECTION

2.1.2. Design Settings

This opens the "Design Settings" input screen. Select the analysis and design options through the Design Settings dialog box as shown in figure hereafter. Note that user can disregard certain design options of the code here. Click on **Next >>** button to continue.

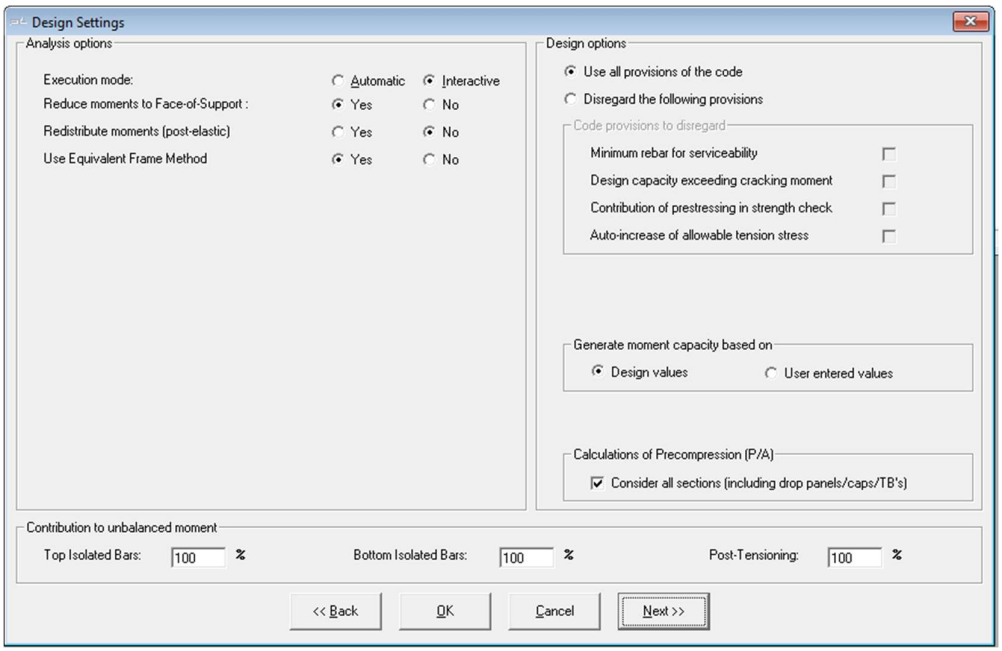


FIGURE 2-7 DESIGN SETTINGS INPUT

2.1.3. Span Geometry

This opens the "Span Geometry" input screen. This screen is used to enter the span geometry of the slab which can be modelled as prismatic (uniform) or non-prismatic (non-uniform). Note that the design strips idealized in previous figure would require the input of right and left multiplier values ("<- M" and "M ->"). The table below shows simple calculation required for the same based on the ratio of the tributary width to the total width of each span.

TABLE 2-2 CALCULATION OF LEFT (<- M) & RIGHT MULTIPLIER (M - >)

Particulars	SPAN 1	SPAN 2	SPAN 3	R-CANT	Units
Span (L)	6.083	9.265	7.102	2.000	m
Overall Depth (h)	200	200	200	200	mm
Left Tributary Width	4000	2400	3600	3000	mm
Right Tributary Width	2500	4700	3000	3000	mm
Total Width	6500	7100	6600	6000	mm
Left Multiplier (<- M)	0.62	0.34	0.55	0.50	Factor
Right Multiplier (M ->)	0.38	0.66	0.45	0.50	Factor

"Rh" value should be kept equals to the slab depth which shall move the reference plane to the bottom of the slab. Note that all legends are already provided in the screen. Enter the data as given in following figure in the input screen.

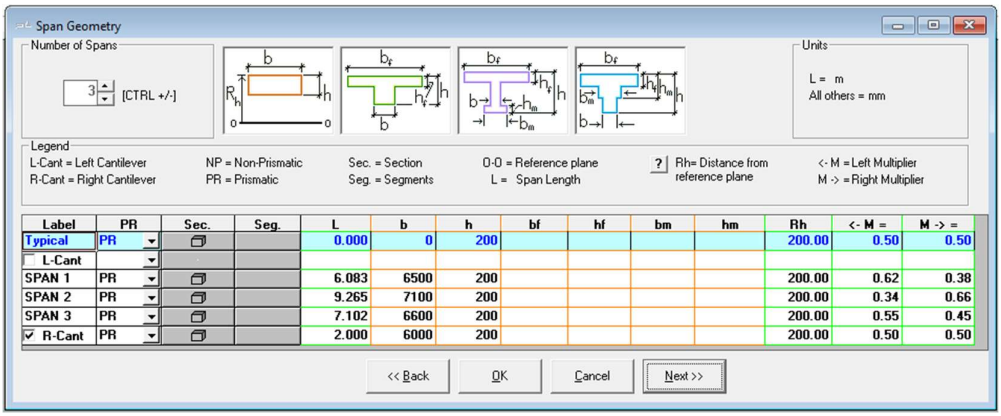


FIGURE 2-8 SPAN GEOMETRY INPUT

The graphical views of the structure are shown form the "Structure View" window in plan and elevation view as shown hereafter. Note that the graphical view is for viewing only and no data can be changed or edited from the same.

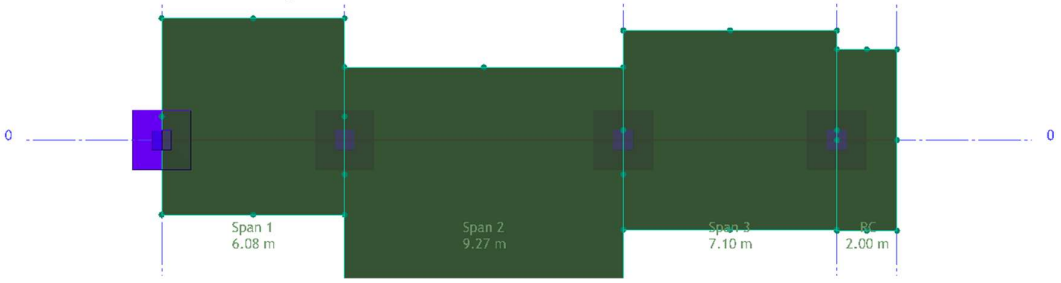


FIGURE 2-9 PLAN VIEW

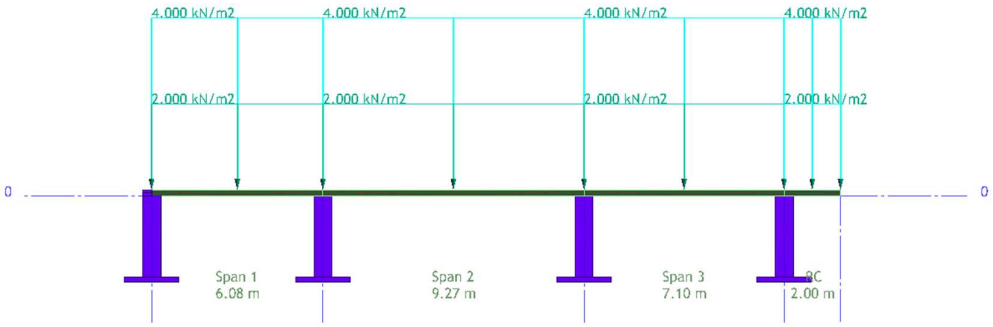


FIGURE 2-10 ELEVATION VIEW

This page can also be accessed in an existing project from menu **Geometry | Span Geometry**, Verify the input data carefully and click on **Next >>** button to continue.

2.1.4. Support Geometry and Stiffness

This opens the "Support Geometry and Stiffness" input screen. This screen is used to input column heights, widths and depths. This page also provides users the options:

- To consider Lower and/or Upper Columns
- To enter some percentage of column stiffness for analysis purpose
- To enter the position of tributaries to be interior/exterior condition for both edges

- To consider mathematical point support using “No Column” option.

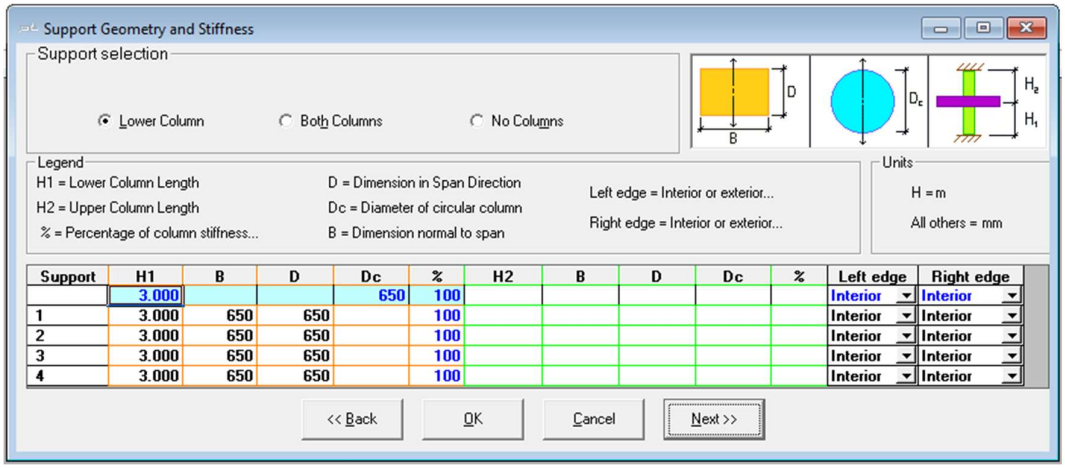


FIGURE 2-11 SUPPORT GEOMETRY & STIFFNESS INPUT

This page can also be accessed in an existing project from menu **Geometry | Support Geometry**. Enter the data as shown above and click on **Next >>** button to continue.

2.1.5. Support Boundary Conditions

This opens the “Supports – Boundary Conditions” input screen as shown in following figure. This screen is used to enter support widths and column boundary conditions. Support widths can be entered if the “Reduce Moments to face of- support” option in the Design Settings screen was selected as “Yes”. However, if it was selected as “No”, no values can be entered in the SW column. This input value will be used to calculate the reduced moments. Turn on the “SW=Actual Width of Support” options to automatically fill up the support width (SW) values.

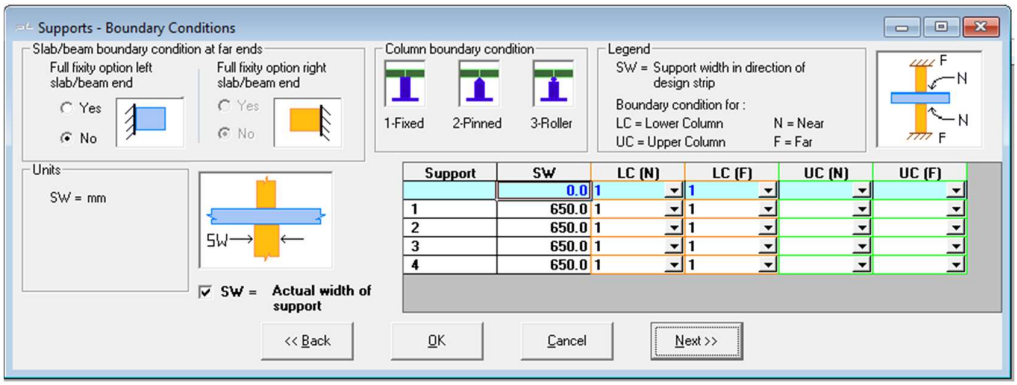


FIGURE 2-12 SUPPORT BOUNDARY CONDITIONS

This page can also be accessed in an existing project from menu **Geometry | Boundary Conditions**. Enter the options / data as required and click on **Next >>** button to continue.

2.1.6. Loads

This opens the Loads input screen. This page allows users to specify a variety of load types. To enter a load for all spans, enter 'all' or 'ALL' as the span number and select load "Class" from the drop-down menu. Enter "U" in the "L/T-?" column as shown in the legends at the top. Next enter the load values in uniform load (i.e. "w" column) as shown below.

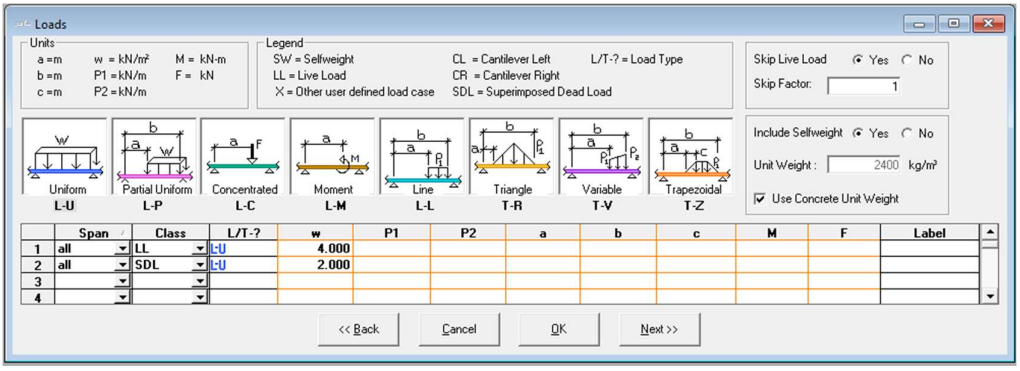


FIGURE 2-13 ALL SPAN LOAD INPUT

Click on **Next >>** button to go to the next screen and **<< Back** button to check that all loads are entered in all spans automatically as shown hereafter.

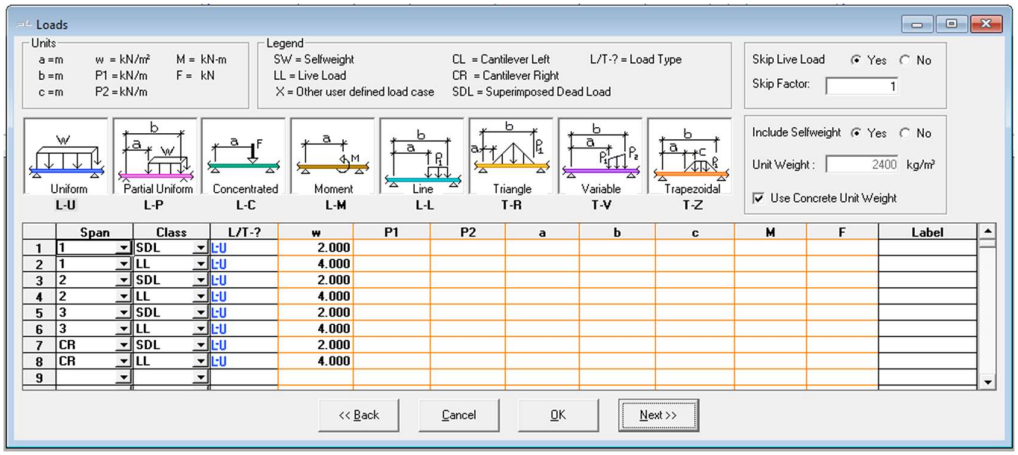


FIGURE 2-14 LOAD INPUT FORM (FINAL)

Also note that following points in this context –

- The load class SW is for self-weight of the structure which will be available only if user selects "No" to the "Include self-weight option.

- When the Skip Loading function is used, the program considers those 6 patterns shown in figure below. The program reports deflections at all 1/20th points along spans for all skip patterns and enveloped Max and Min results in report Section 36.

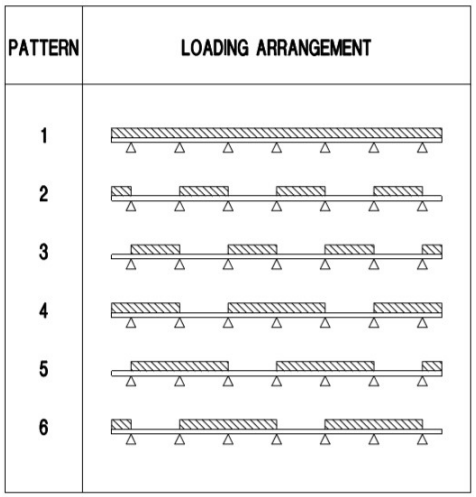


FIGURE 2-15 SKIP PATTERNS

This page can also be accessed in an existing project from menu **Loads**. Check the data as said before and click on **Next >>** button to continue.

2.1.7. Concrete

This opens the "Material – Concrete" input page as shown in following figure. This screen is used to enter concrete properties. The value of "Modulus of Elasticity …" changes automatically based on standard formulae as specified in the user selected codes on clicking on that field. Refer user manual for details.

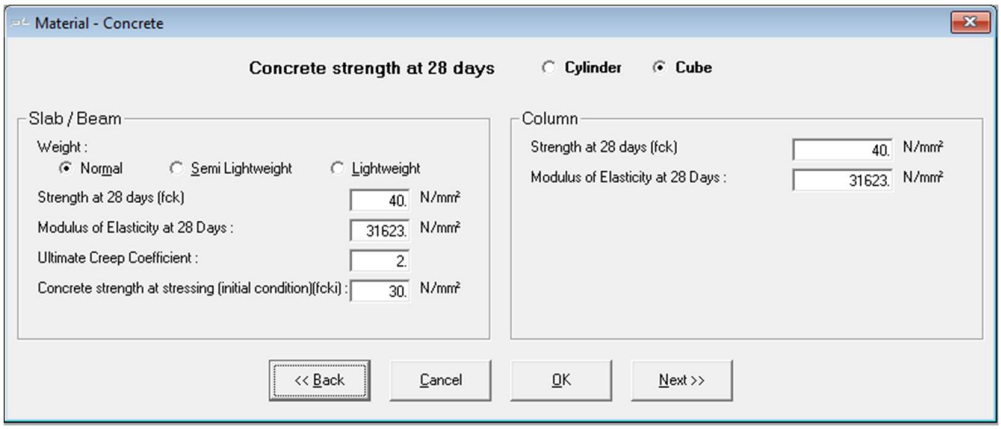


FIGURE 2-16 MATERIAL - CONCRETE INPUT

This page can also be accessed in an existing project from menu **Material | Concrete**. Enter the options / data as required and click on **Next >>** button to continue.

2.1.8. Reinforcement

This opens the "Material – Reinforcement" input page. This screen is used to specify mild reinforcing bar sizes and properties for longitudinal and shear reinforcement.

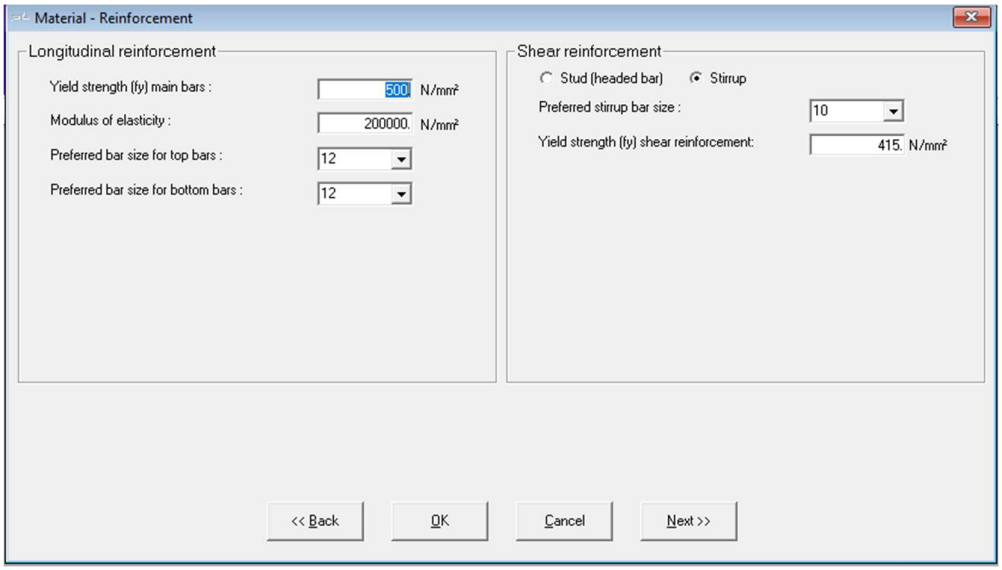


FIGURE 2-17 MATERIAL - REINFORCEMENT INPUT

This page can also be accessed in an existing project from menu **Material | Reinforcement**. Enter the options / data as shown in figure and click on **Next >>** button to continue.

2.1.9. Post-Tensioning

This opens the "Material – Post-Tensioning" input page. This screen is used to input the post-tensioning system parameters. This option is available in PT mode only.

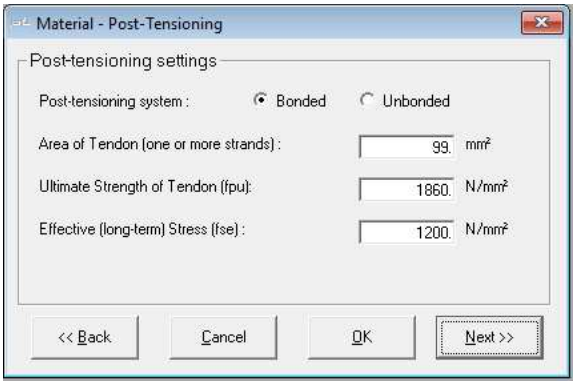


FIGURE 2-18 MATERIAL - POST-TENSIONING

This page can also be accessed in an existing project from menu **Material | Post-Tensioning**. Enter the options / data as shown in figure and click on **Next >>** button to continue.

2.1.10. Non-Prestressed Base Reinforcement

This opens the "Base Non-Prestressed Reinforcement" input page. If user selects "Yes" then remaining part of the page is displayed. The program allows user to specify a base reinforcement that is taken into consideration when designing the structure. User has an

option to define reinforcement as mesh or isolated longitudinal bars for all system types and stirrups for one-way slabs and beams.

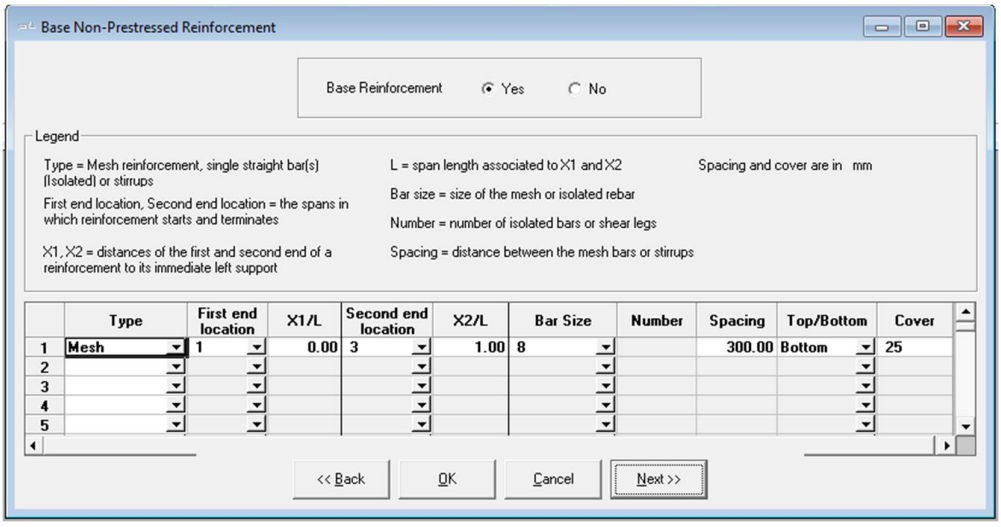


FIGURE 2-19 BASE NON-PRESTRESSED REINFORCEMENT

This page can also be accessed in an existing project from menu **Criteria | Base Non-Prestressed Reinforcement ….** Enter the options / data as shown before and click on **Next >>** button to continue.

2.1.11. Allowable Stresses

This opens the Criteria – Allowable Stresses input page as shown in following figure.

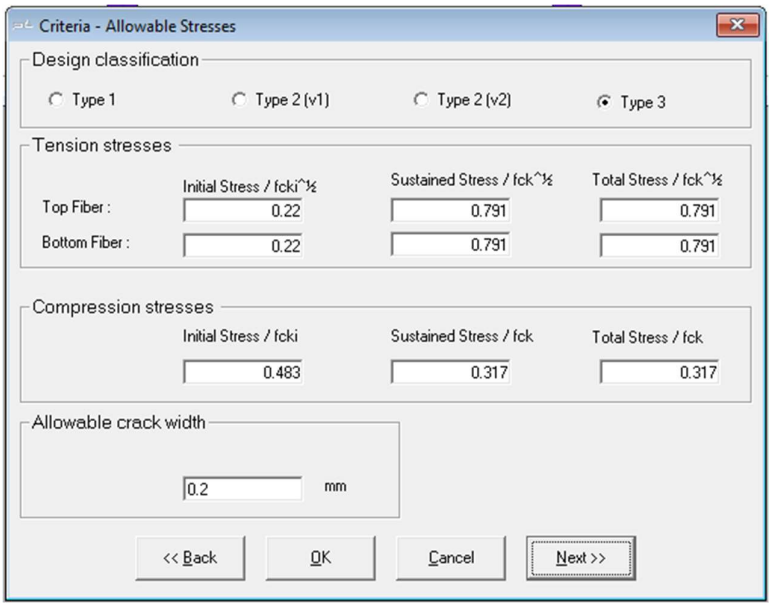


FIGURE 2-20 CRITERIA - ALLOWABLE STRESSES

This screen is used to enter/select allowable stresses for different design classifications. The general architecture of the “Design Classification” are enumerated in the following table.

TABLE 2-3 DESIGN CLASSIFICATION AS PER INDIAN CODE

Type	Tensile Stress (MPa)			Remarks
	Initial (Transfer)	Service Sustained	Service Total	
Type 1	0.22 $\sqrt{f_{cki}}$	0.0	0.0	No tensile stress is allowed in service condition.
Type 2 (v1)	0.22 $\sqrt{f_{cki}}$	3.0	3.0	If tensile stress occurs in service sustained condition.
Type 2 (v2)	0.22 $\sqrt{f_{cki}}$	0.0	4.5	If no tensile stress occurs in service sustained condition.
Type 3	0.22 $\sqrt{f_{cki}}$	As calculated from Table 10 and Figure 6 of IS 1343 : 2012		Design is based on cracked section.

Please refer "ADAPT Indian Code Implementation" document for details on the same. Please note the following points from the above table as a special mention:

- Design classification of Type 2 as per code is broken down into two sub options, viz. "Type 2(v1)" and "Type 2(v2)" based on whether Tensile stress occurs in Service Sustained condition or not.
- "Crack Width" is allowed only in "Type 3" till a maximum value of 0.3mm.
- Only Design "Type 3" values are editable so that user can enter their own values as deemed fit. The defaults are automatically loaded when the type is shifted to any other type.
- User has flexibility to control the way this option works from "Design Settings" window by disregarding relevant and available options of the code based on their specific requirements.
- These options are a new-feature for ADAPT PTRC 2019 version and is available only when user selects the latest Indian code of design, i.e. "Indian-IS1343 (2017)".

This option is applicable to PT mode only. This page can also be accessed in an existing project from menu **Criteria | Allowable Stresses**. Click on **Type 3** option and leave the default values here. Click on **Next >>** button to continue.

2.1.12. Recommended Post-Tensioning Values

This opens the **Criteria – Recommended Post-Tensioning Values** input page as shown in following figure. This screen is used to specify minimum and maximum values for average precompression (P/A: total prestressing divided by gross cross-sectional area) and percentage of dead load to balance (Wbal).

These values are used by the program to determine the post-tensioning requirements and the status of the "Pmin/Pmax" and "WBAL Min/ Max" indicators on the "Recycle window".

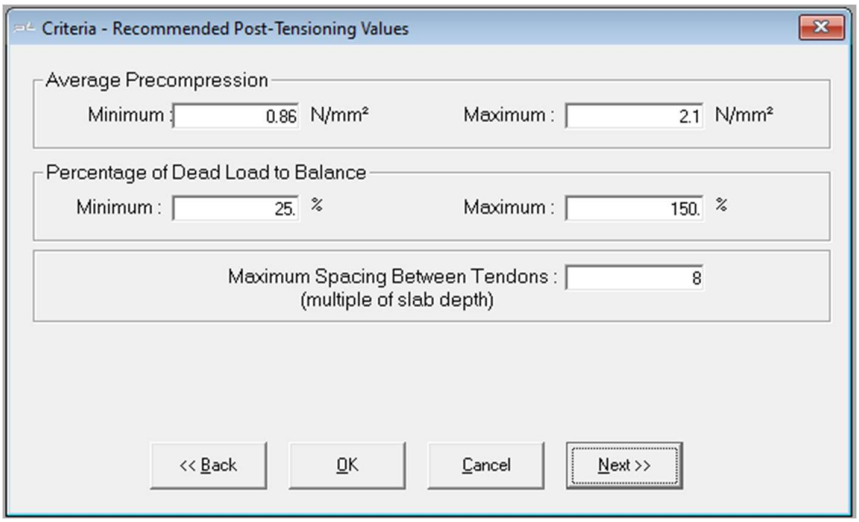


FIGURE 2-21 CRITERIA - RECOMMENDED POST-TENSIONING VALUES

This option is applicable to PT mode only. This page can also be accessed in an existing project from menu **Criteria | Post-Tensioning…**. Enter the options / data as shown before and click on **Next >>** button to continue.

2.1.13. Calculation Options

This opens the **Criteria – Calculation Options** input page as shown in following figures. This screen is used to select the post-tensioning design options, viz., **Force Selection** and **Force/Tendon Selection**. The same is described in following sections.

2.1.13.1. Force Selection

In **Force Selection** option, a tendon will be assigned a final and constant effective force, equal to the jacking force minus all stress losses, expressed as a single value as shown in following figure. Select **Calculate force/number of tendons** option to continue.

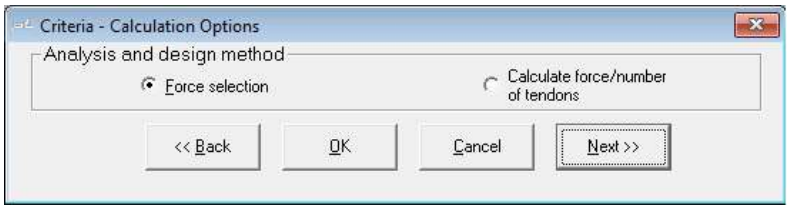


FIGURE 2-22 CRITERIA - CALCULATION OPTIONS: FORCE SELECTION

2.1.13.2. Tendon Selection

If **Calculate force/number of tendons** option is specified, the screen shall expand to present additional inputs required to calculate the detailed prestressed losses.

Long-term losses may either be entered as a lump sum value, or the information required to calculate them may be entered. These values of prestressing material constants and

shall be generally found PT strand manufacturer's document or may also be entered based on industry standard. The same is shown in figure hereafter.

This option is applicable to PT mode only. This page can also be accessed in an existing project from menu **Criteria | Calculation Options …**. Enter the options / data as shown hereafter and click on **Next >>** button to continue.

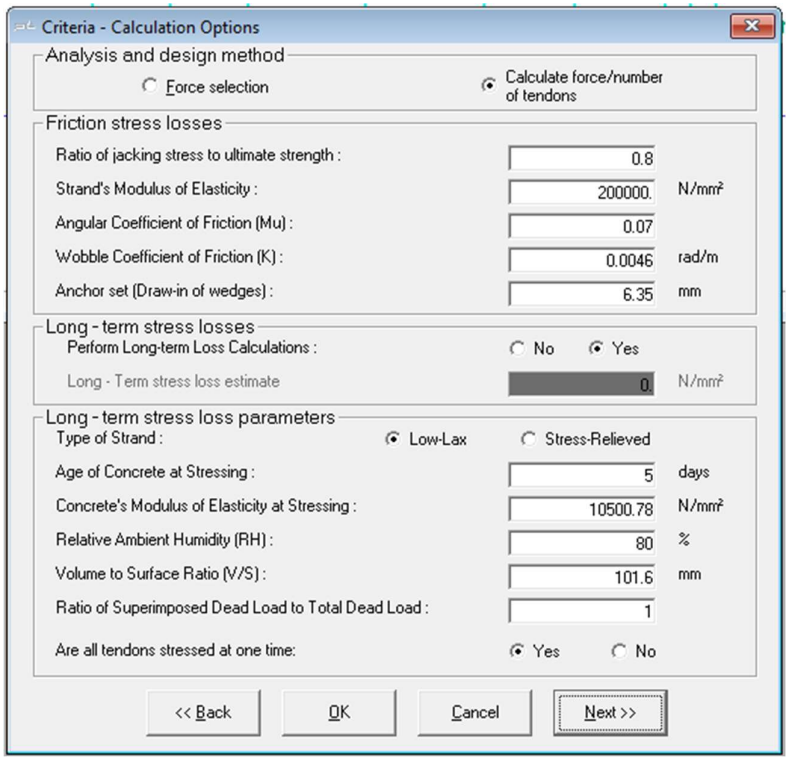


FIGURE 2-23 CRITERIA - CALCULATION OPTIONS: TENDON SELECTION

2.1.14. Tendon Profile

This opens the **Criteria – Tendon Profile** input page as shown in figure hereafter.

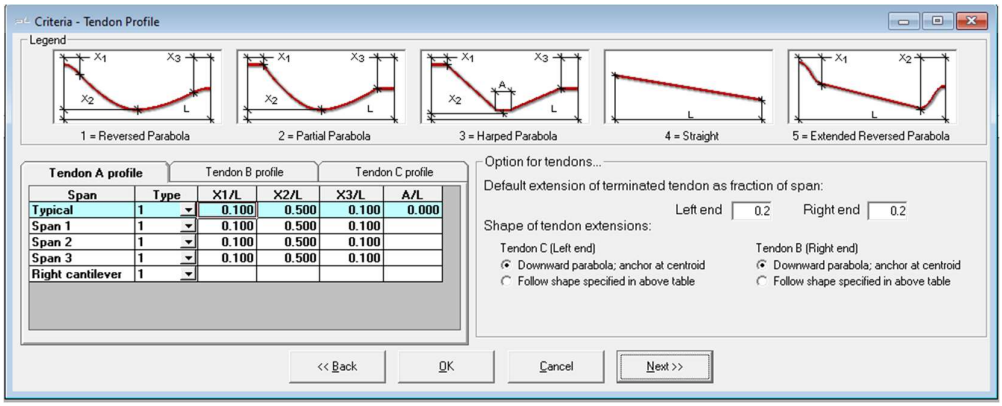


FIGURE 2-24 CRITERIA - TENDON PROFILE

The program allows user to specify up to three tendon paths per span. User can define one profile for each of the three tendons. In the section **Option for tendons**, user can

define the Default extension of terminated tendon as fraction of span. Also, User can also specify the Shape of tendon extension from the **Left end** and the **Right end**.

This option is applicable to PT mode only. This page can also be accessed in an existing project from menu **Criteria | Tendon Profile**. Enter the options / data as required and click on **Next >>** button to continue.

2.1.15. Cover for Reinforcement and Post-Tensioning

This opens the **Criteria – Cover / CGS** input page as shown in following figure. Note that the cover for the pre-stressing steel is specified to the centre of gravity of the strand (CGS) whereas, for mild steel, it is clear cover.

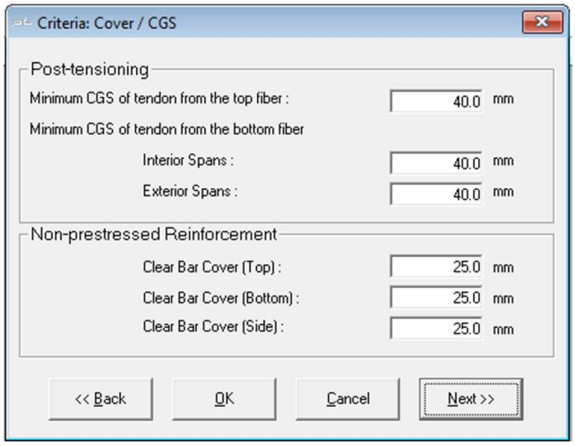


FIGURE 2-25 CRITERIA - COVER / CGS

This page can also be accessed in an existing project from menu **Criteria | Minimum Covers**. Enter the options / data as mentioned and click on **Next >>** button to continue.

2.1.16. Minimum Bar Extension

This opens the **Criteria – Minimum Bar Extension** input page as shown in following figure.

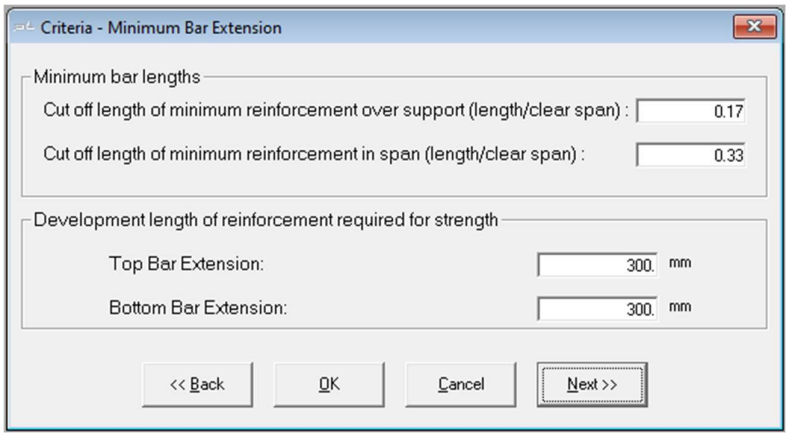


FIGURE 2-26 CRITERIA - MINIMUM BAR EXTENSION

This screen is used to define how mild steel reinforcement bar lengths are calculated. This page can also be accessed in an existing project from menu **Criteria | Minimum Bar Extension ….** Enter the options / data as required and click on **Next >>** button to continue.

2.1.17. Reinforcement Curtailment

This opens the **Criteria – Reinforcement Curtailment** input page as shown in following figure. This page can also be accessed in an existing project from menu **Criteria | Reinforcement Curtailment ….**

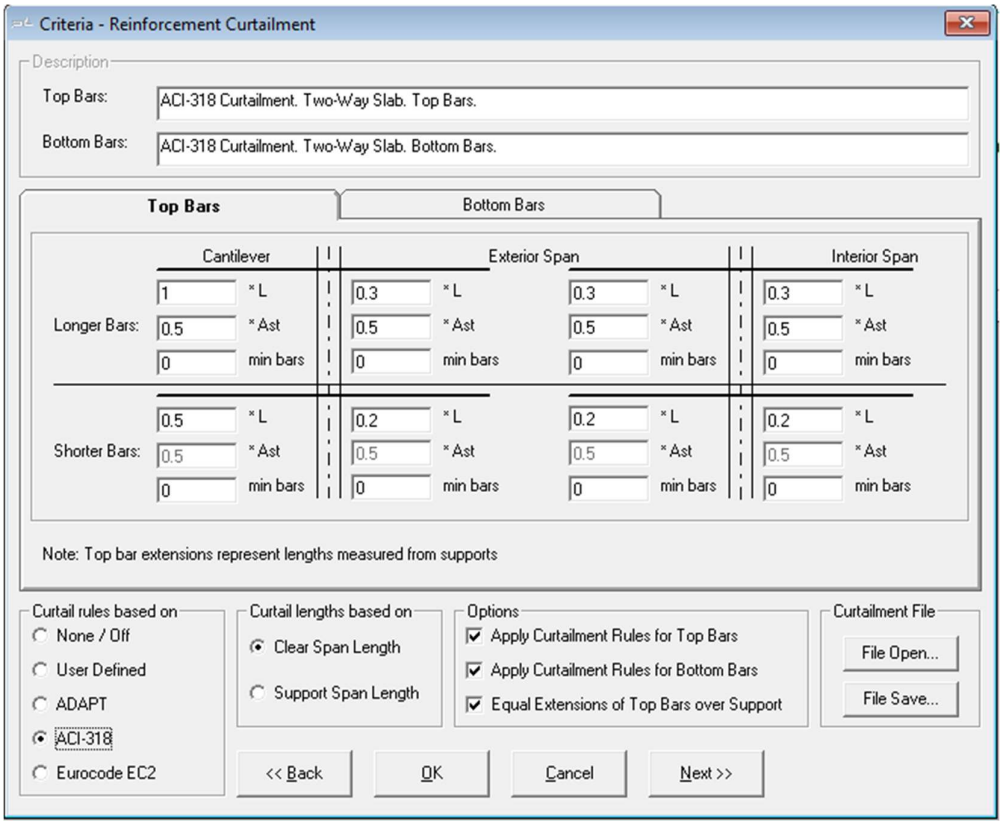


FIGURE 2-27 CRITERIA - REINFORCEMENT CURTAILMENT

This screen is used to define how mild steel reinforcement bar lengths are calculated. Enter the options / data as required and click on **Next >>** button to continue.

2.1.18. Load Combinations

This opens the **Criteria – Load Combinations** input page as shown in following figure. The program allows users to specify four strength, two service sustained and two service total load combinations.

The default load combinations are already loaded for the current code selection here. The Initial load combination is only active for the PT mode.

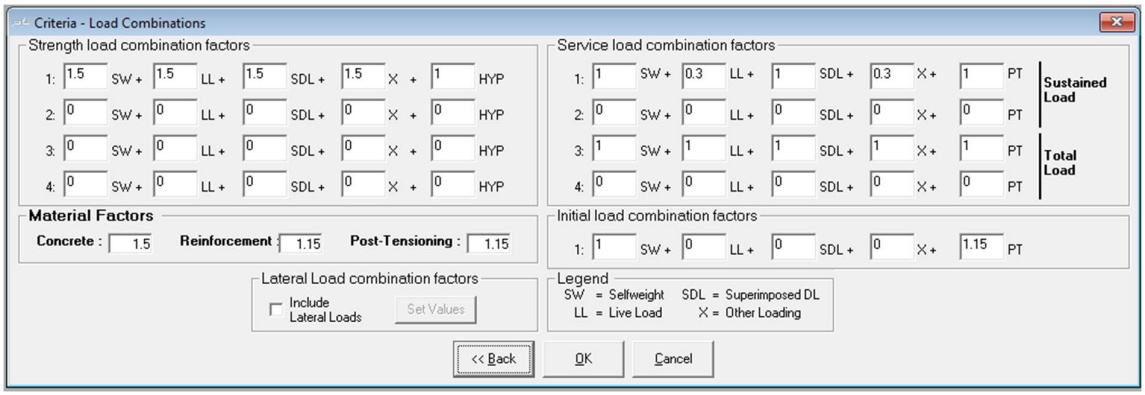


FIGURE 2-28 CRITERIA - LOAD COMBINATIONS

This page can also be accessed in an existing project from menu **Criteria | Load Combinations …**. Enter the options / data as shown before.

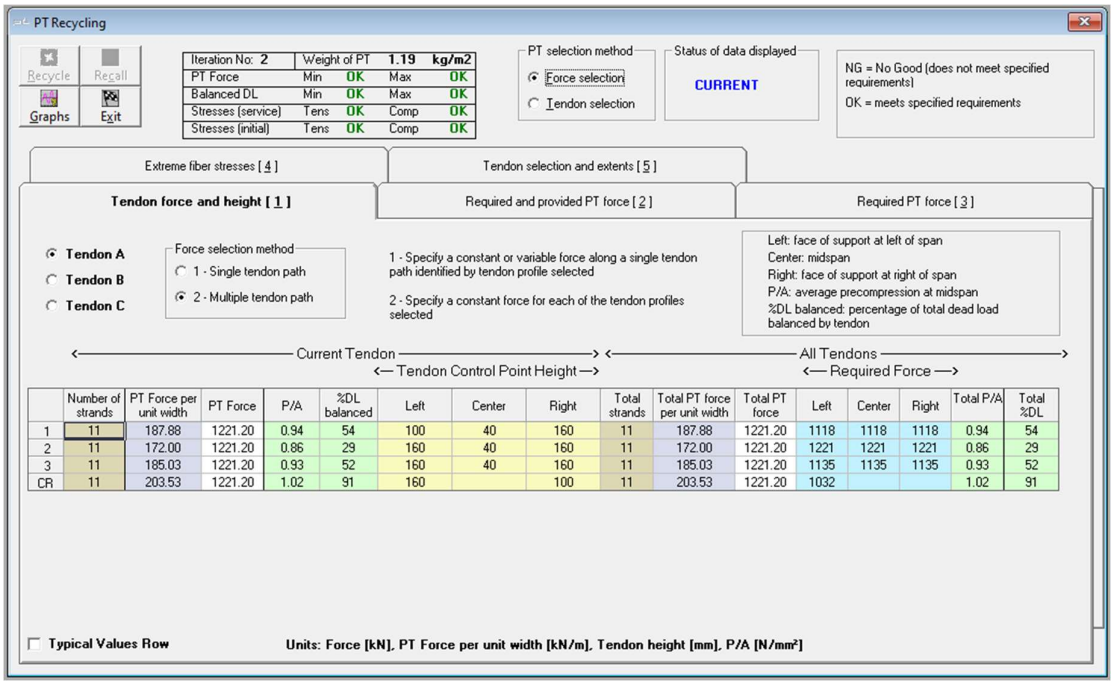
This completes the input definition of the project data. To save the input data and execute the analysis, either select **Execute** on the menu bar or click on the **Save & Execute Analysis** button.

In PT mode the program can be executed in **Automatic** or **Interactive** mode from **Design Settings** screen. *In RC mode the program is run in automatic mode only. Refer user manual for details.*

2.2. EXECUTION AND OUTPUT

2.2.1. Recycling Window

Once the execution gets completed, the **PT Recycling** window, as shown in figure hereafter opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error. The data consistency checks are not exhaustive, however, which means that the user is ultimately responsible for ensuring that the data is entered correctly.



	Number of strands	PT Force per unit width	PT Force	P/A	%DL balanced	Left	Center	Right	Total strands	Total PT force per unit width	Total PT force	Left	Center	Right	Total P/A	Total %DL
1	11	187.88	1221.20	0.94	54	100	40	160	11	187.88	1221.20	1118	1118	1118	0.94	54
2	11	172.00	1221.20	0.86	29	160	40	160	11	172.00	1221.20	1221	1221	1221	0.86	29
3	11	185.03	1221.20	0.93	52	160	40	160	11	185.03	1221.20	1135	1135	1135	0.93	52
CR	11	203.53	1221.20	1.02	91	160		100	11	203.53	1221.20	1032			1.02	91

FIGURE 2-29 PT RECYCLING WINDOW

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

User can revise the design data using editable data in the tabs to modify tendon profile, force, number and extents to obtain desirable outcome mainly from the tabs namely, Tendon force and height (1) and Tendon selection and extents (5). Each revision should be followed by a click in the **Recycle** button to update the results with the revised data. In an existing design project with a solution available user can also click on **Recall** button to recall the previously edited design data saved in the solution folder. Once an acceptable solution is reached click on **Exit** button to accept the calculations and proceed to report preparation.

For example, click on the 

button on the **Recycle** screen to get the graphs as shown in following figure.

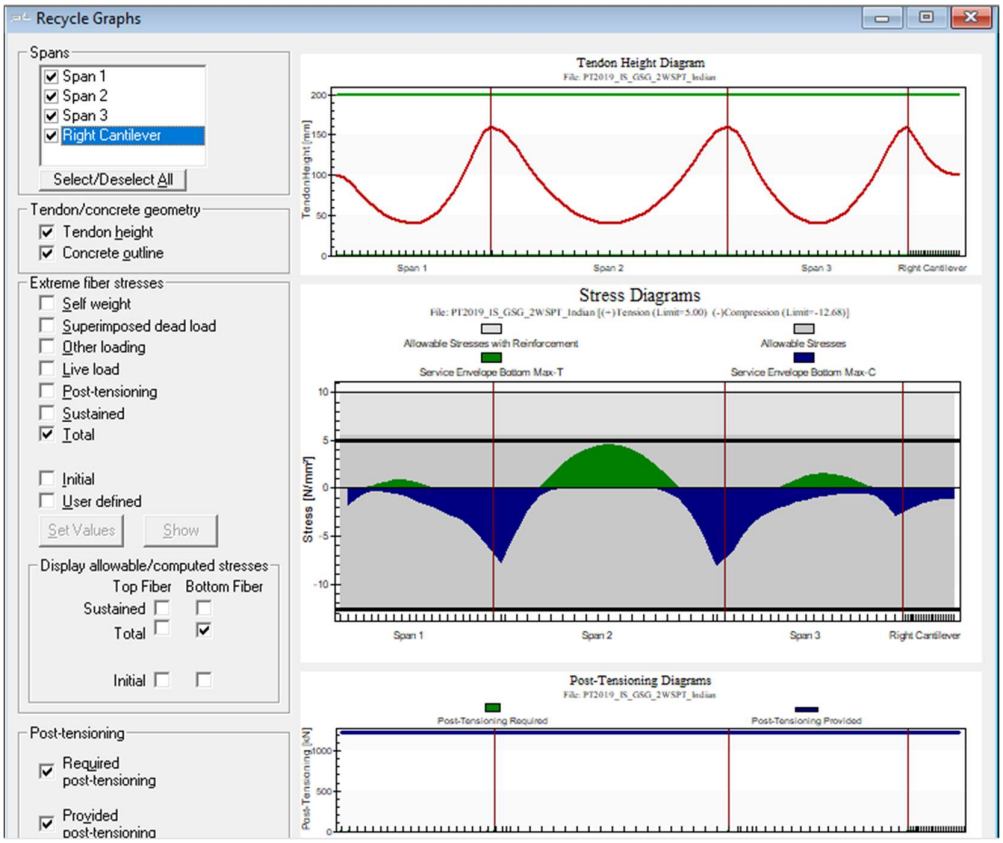


FIGURE 2-30 RECYCLE GRAPH

In this case all design status is **Ok**, hence click on Exit button on the Recycle Screen to accept the solution and move ahead to prepare reports.

2.2.2. Execution Completed

Once successfully finished, program returns to the main program window. The Results can be viewed and/or printed as Reports, Graphs and PT Summary as shown in following figure.

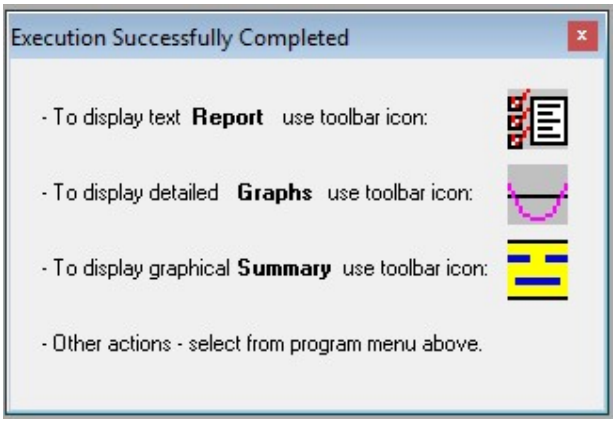


FIGURE 2-31 EXECUTION COMPLETED

2.2.3. Show Graphs

The user can now select the **Show Graphs** button to view all the results graphs. This option can also be invoked from the menu **View | Graphs** and selecting the appropriate items from the sub-menu. Several graphs are shown to depict all design conditions like Moment, Shear, Deflection, Stress etc.

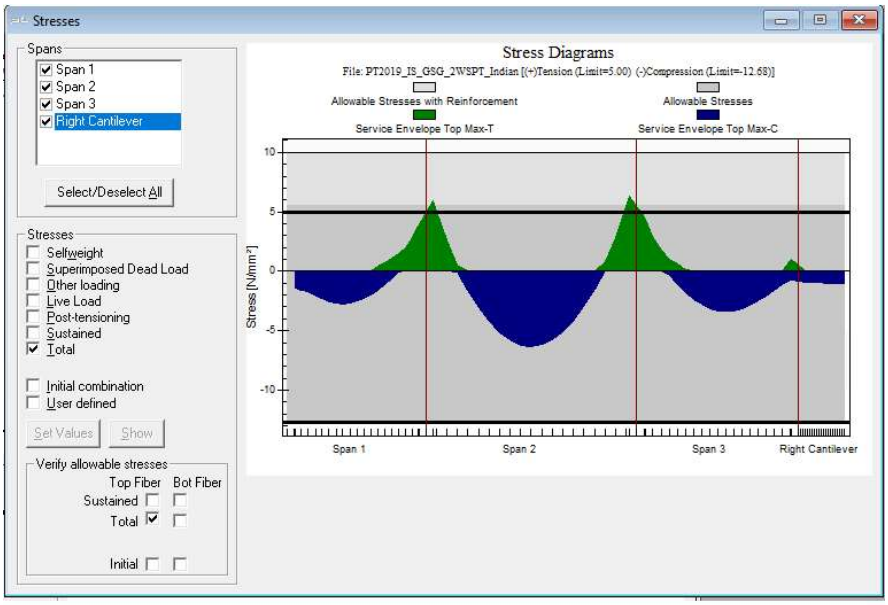


FIGURE 2-32 SERVICE TOTAL - TOP STRESS DIAGRAM

These graphs clearly demonstrate allowable and actual design data. User can right click on the graphs and modify these graphs to show different data labels including options to Export the graph to different formats as shown in the following figure.

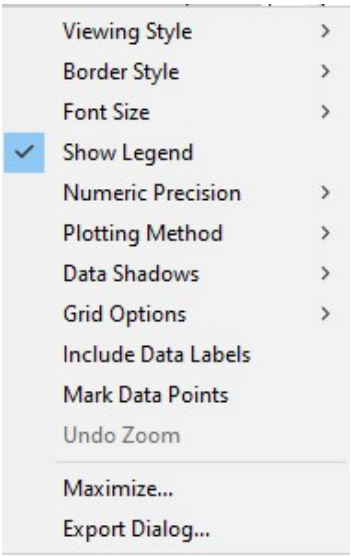


FIGURE 2-33 ANNOTATION OPTIONS FOR GRAPHS

2.2.4. Report Generation

ADAPT-PT/RC 2019 has a powerful Report Generator. To generate the report, select the **Report Setup** item on the **Options** menu or click the **Report Setup** button on the main toolbar. The **Report Generator** screen shown in following figure will open.

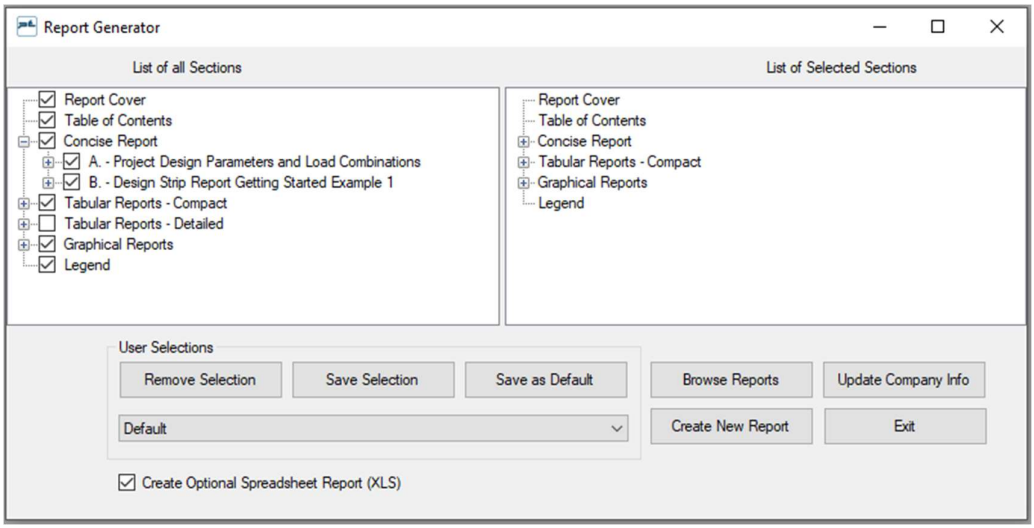


FIGURE 2-34 REPORT GENERATOR

Select the data as given here and click on **Create New Report**, save the location and file name as required and click on **Save**. Software would run all the reports automatically and generate the report in RTF and XLS format. Please wait until the report in compiled and the RTF and XLS file is opened for review. The following figure shows an excerpt from the report generated.

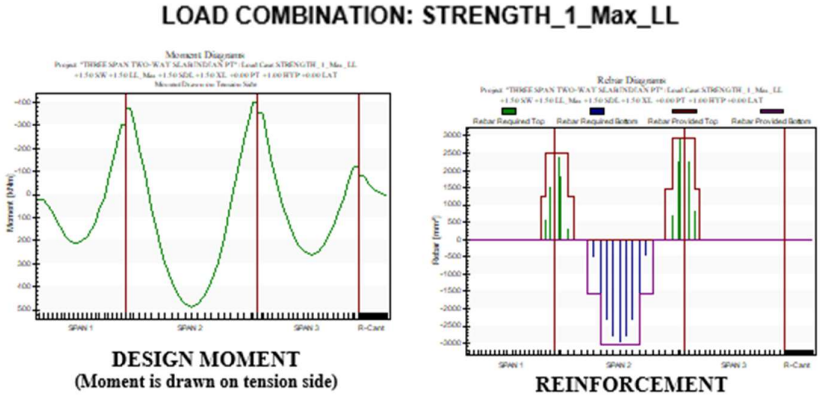


FIGURE 2-35 EXCERPT FROM RTF FORMAT REPORT GENERATED

2.2.5. Summary Report

To open the **PT Summary Report** either click the open **BuilderSum** button on the tool bar or select the PT summary item on the **View** menu.

A new window named **ADAPT-BuilderSum** is opened. Click on the icon to open a summary report as shown in figure below. There are other tools which are self-explanatory.

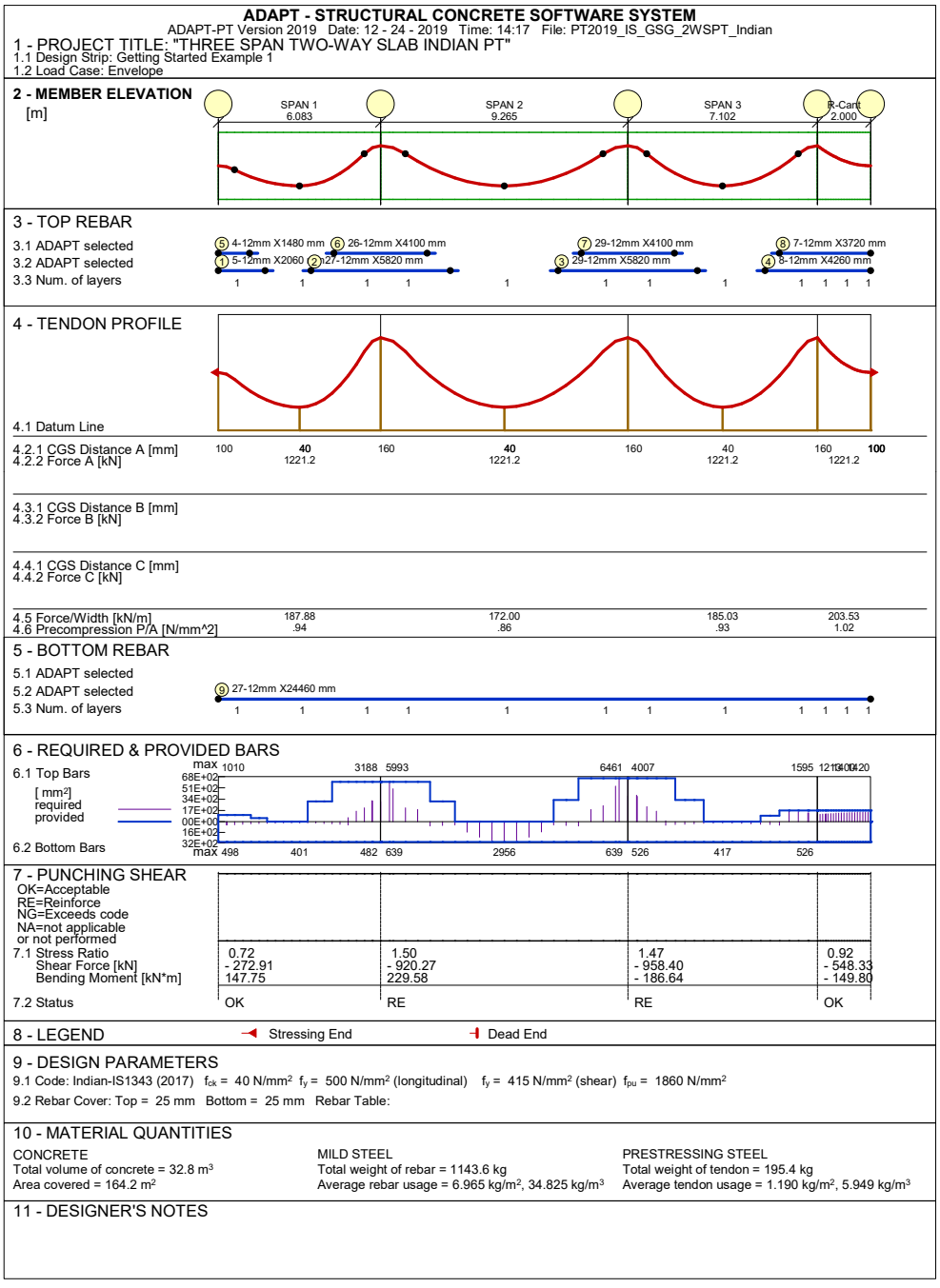


FIGURE 2-36 SUMMARY REPORT

2.2.6. DXF Export

To export the tendon profiles to DXF format, click on the **Export to DXF file** button on the tool bar or select the **Export to DXF file** option from the **File** menu. It opens the

Export Tendon Profiles and Reinforcement to DXF Drawing dialogue as shown in following figure.

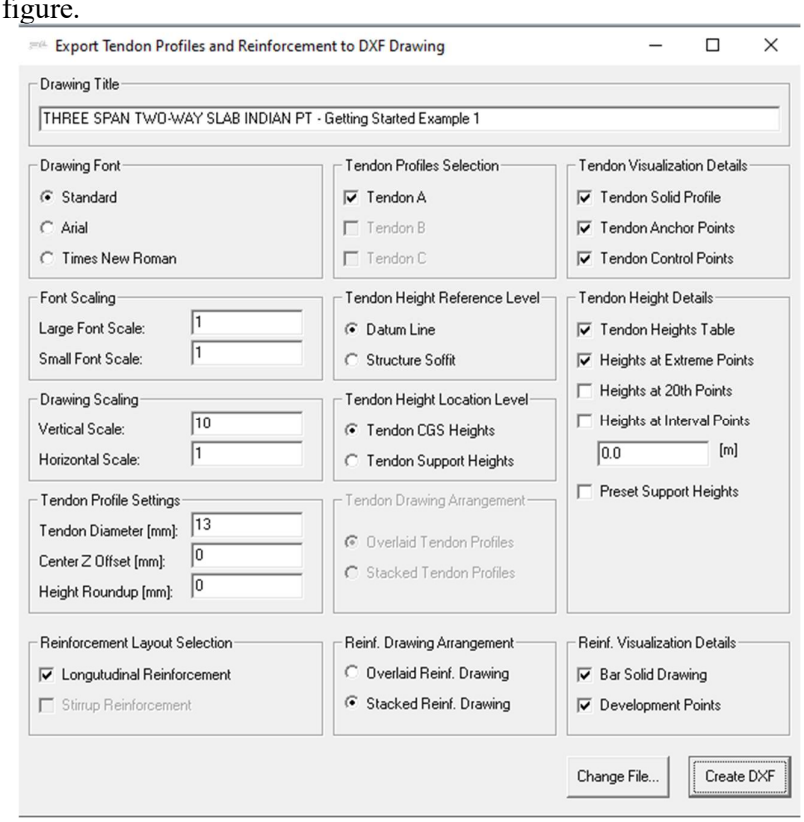


FIGURE 2-37 EXPORT TO DXF DRAWING DIALOUGE

Type the drawing title in the area provided and click on **Create DXF** button to save the DXF file. The DXF file is saved within the project data folder as shown in following figure.

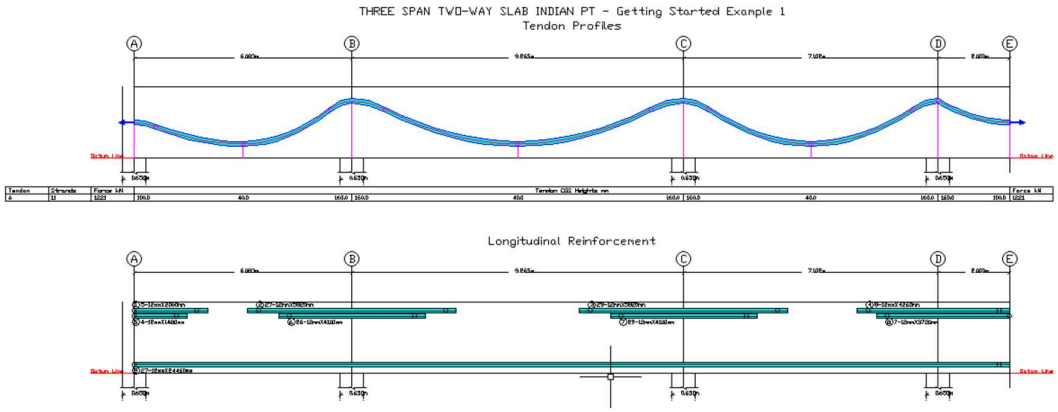


FIGURE 2-38 SAMPLE DXF OUTPUT

This completes this tutorial. Please contact ADAPT Technical Support at support@adaptsoft.com for any questions.

List of Contents