

ADAPT-PT/RC 2019

Getting Started Guide

ADAPT PT Mode – Two-way Slab

Indian Code of Practice

December 2019

Copyright © ADAPT Corporation all rights reserved

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.



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



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



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



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

IIIDI	
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, $f_{pu} = 1860 \text{ N/mm}^2$
	• Effective (long term) stress, $f_{se} = 1200 \text{ N/mm}^2$
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

TABLE 2-1 DESIGN SPECIFICATIONS

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.

er General Settings	
General Title: THREE SPAN TWO-WAY SLAB INDIAN PT	
Specific Title: Getting Started Example 1	
Structural System:	Geometry input
	Conventional C Segmental
	Drop Panel / Drop Cap / Transverse Beam
	Include Drops & C Yes I No
	Recommend Slab Thickness
I wo-Way Slab Une- <u>Way Slab</u> Beam	C Yes 📀 No
<u>C</u> ancel <u>D</u> K <u>Next>></u>	

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.

🗝 Criteria - Design Code		×
Design codes]
C American-ACI318 (1999)	C Brazilian-NBR6118 (2014)	C European-EC2 (2004)
C American-ACI318 (2005) / IBC 2006	 British-BS8110 (1997) 	Indian-IS1343 (2004)
C American-ACI318 (2008) / IBC 2009	C Canadian-A23.3 (1994)	Indian-IS1343 (2017)
C American-ACI318 (2011) / IBC 2012	C Canadian-A23.3 (2004)	C Hong Kong-CoP (2007)
C American-ACI318 (2014) / IBC 2015	C Canadian-A23.3 (2014)	C Hong Kong-CoP (2013)
C Australian-AS3600 (2001)		C Chinese GB 50010 (2002)
C Australian-AS3600 (2009)		
Design Code Annex		
None		_
<< <u>B</u> ack	<u>O</u> K <u>C</u> ancel	<u>N</u> ext >>

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.

The Design Cattings		
- Design Settings		
Execution mode: Reduce moments to Face-of-Support : Redistribute moments (post-elastic) Use Equivalent Frame Method	C ≜utomatic	Cellulations of Precompression (P/A) Calculations of Precompression (P/A) Calculations of Precompression (P/A) Consider all sections (including drop panels/caps/TB's)
Contribution to unbalanced moment Top Isolated Bars: 100 2	Bottom Isolated Bars: 100	p x Post-Tensioning: 100 x

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.

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

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

"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.

=- Span Geo	metry													
$ \begin{array}{ c c c c c c c c c c c c c c c c c c c$														
Legend L-Cant = Lef R-Cant = Rig	t Cantiles ght Cantil	ver ever	NP = PR =	Non-Prismatic Prismatic	s Sec. Seg.	= Section = Segments	0-0 L	= Referenci = Span Ler	e plane Igth	? Rh= refe	Distance from rence plane	<- N M -3	/ = Left Multipl > = Right Multip	ier olier
Label	PF	1	Sec.	Seg.	L	Ь	h	bf	hf	bm	hm	Bh	<- M =	M -> =
Typical	PR	-	Ø		0.000	0	200					200.00	0.50	0.50
L-Cant		-					(
SPAN 1	PR	-	đ		6.083	6500	200					200.00	0.62	0.38
SPAN 2	PB	-	đ		9.265	7100	200					200.00	0.34	0.66
SPAN 3	PR	-	0		7.102	6600	200					200.00	0.55	0.45
R-Cant	PR	-	Ø		2.000	6000	200					200.00	0.50	0.50
<pre></pre>														

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.



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.





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.

≓∸ Material - Concrete	
Concrete strength at 28 days	⊂ Cylinder € Cube
Slab / Beam Weight : @ Normal © Semi Lightweight Strength at 28 days (fck) 40, Modulus of Elasticity at 28 Days : 31623, Ultimate Creep Coefficient : 2, Concrete strength at stressing (initial condition)[fcki] : 30,	Column Strength at 28 days (fck) 40. N/mm² Modulus of Elasticity at 28 Days : 31623. N/mm²
Cancel	<u>D</u> K <u>N</u> ext>>

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.

=- Material - Reinforcement			×
Congitudinal reinforcement Yield strength (fy) main bars : Modulus of elasticity : Preferred bar size for top bars : Preferred bar size for bottom bars :	500 N/mm² 200000. N/mm² 12 v 12 v	Shear reinforcement C Stud (headed bar) Preferred stirrup bar size : Yield strength (fy) shear reinforcement:	10 v 415. N/mm²
	<< <u>B</u> ack <u>D</u> K	<u>Cancel</u> <u>N</u> ext >>	

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.

Post-tensioning system : Bonded Unbonded Area of Tendon (one or more strands) :
Area of Tendon (one or more strands) : 99. mm²
Ultimate Strength of Tendon (fpu): 1860. N/m
Effective (long-term) Stress (fse) : 1200. N/mr

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.

Base N	Ion-Prestressed R	einforcemen	:							- 0	x
			Ba	se Reinforceme	nt (© Y	'es C No					
Legend											
First end location, Second end location = the spans in which reinforcement starts and terminates Number = number of isolated bars or shear legs X1.X2 = distances of the first and second end of a Spacing = distance between the mesh bars or stirrups											
reinford	Cement to its immed	First end	t X1/L	Second end location	X2/L	Bar Size	Number	Spacing	Top/Bottom	Cover	
1 M	esh 🗾	1 🚽	0.00	3 🗸	1.00	8 🚽		300.00	Bottom 💌	25	
2	<u> </u>	<u> </u>		<u> </u>		<u>-</u>			<u> </u>		
4	-	-		•		-			-		
5	<u>•</u>	<u> </u>		<u> </u>		<u> </u>			<u> </u>	1	
		-						-			
			<< <u>B</u>	ack	<u>D</u> K	Cancel	Next>>				

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.

Criteria - Allował	ole Stresses		×
Design classific	ation		
C Type 1	C Type 2 (v1)	C Type 2 (v2)	🗭 Туре 3
Tension stresse	9S		
	Initial Stress / fcki^½	Sustained Stress / fck^½	Total Stress / fck^½
Top Fiber :	0.22	0.791	0.791
Bottom Fiber :	0.22	0.791	0.791
Complession st	Initial Stress / fcki	Sustained Stress / fck	Total Stress / fck
Allowable crack	width		
	0.2 mm		
	<< Back OK	<u>C</u> ancel	ext >>

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.

Туре	Tens	ile Stress (MI	Pa)	Remarks
	Initial	Service Service		
	(Transfer)	Sustained	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 √f _{cki}	As calculated from		Design is based on cracked
		Table 10 and Figure 6		section.
		of IS 1343	: 2012	

 TABLE 2-3 DESIGN CLASSIFICATION AS PER INDIAN CODE

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".

Minimum (0.86	N/mm²	Maximum :	2.1	N/mm²
Percentage of Dea	id Load to E	alance			
Minimum :	25.	%	Maximum :	150.	%
	Maximun (n Spacing Be (multiple of sla	tween Tendons : ab depth)	8	

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.

Analysis and design methor	d					
Force selection		C Calculate force/number of tendons				
<< Back	ок	Cancel	Next>>			

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.

Criteria - Calculation Options	
Analysis and design method	
C Eorce selection	 Calculate force/number of tendons
Friction stress losses	
Ratio of jacking stress to ultimate strength :	0.8
Strand's Modulus of Elasticity :	200000. N/mm²
Angular Coefficient of Friction (Mu) :	0.07
Wobble Coefficient of Friction (K):	0.046 rad/m
Anchor set (Draw-in of wedges)	6.0040 mm
	6.55
Cong - term stress losses Perform Long-term Loss Calculations :	C No 💿 Yes
Long - Term stress loss estimate	0, N/mm²
Long - term stress loss parameters Type of Strand :	C Stress-Relieved
Age of Concrete at Stressing :	5 days
Concrete's Modulus of Elasticity at Stressing :	10500.78 N/mm²
Relative Ambient Humidity (RH) :	80 %
Volume to Surface Ratio (V/S) :	101.6 mm
Ratio of Superimposed Dead Load to Total Dead Load :	1
Are all tendons stressed at one time:	
<< Back DK Canc	cel Next >>

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.

- Criteria - Tendon Prof	ile						
	×3 +		×1 X2	×3 +		- ×1	
1 = Reversed	Parabola		2 = Parti	al Parabola		3 =	= Harped Parabola 4 = Straight 5 = Extended Reversed Parabola
Tendon A profile		Tendon B p	rofile	Tendor	n C profile		Option for tendons
Span	Туре	X1/L	X2/L	X3/L	A/L	11	Delauk extension of terminated terdon as iracitor of span.
Typical 1	_	0.100	0.500	0.100	0.000	н.	Left end 0.2 Right end 0.2
Span 1 1	-	0.100	0.500	0.100	_	н.	Shape of tendon extensions:
Span 2 1		0.100	0.500	0.100	_	н.	Tendon C (Left end) Tendon B (Right end)
Bight cantilever 1	÷	0.100	0.000	0.100	_	н.	Ownward parabola; anchor at centroid Ownward parabola; anchor at centroid
				1			C Follow shape specified in above table C Follow shape specified in above table
				<< Back	<u></u> K		<u>Cancel</u>

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.

Minimum CGS of tendon from the top fiber :	40.0 mm
Minimum CGS of tendon from the bottom fiber	
Interior Spans :	40.0 mm
Exterior Spans :	40.0 mm
Non-prestressed Reinforcement	
Clear Bar Cover (Top) :	25.0 mm
Clear Bar Cover (Bottom) :	25.0 mm
Clear Bar Cover (Side) :	25.0 mm

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.

=+ Criteria - Minimum Bar Extension	×							
Minimum bar lengths Cut off length of minimum reinforcement over support (length/clear span) : 0.17 Cut off length of minimum reinforcement in span (length/clear span) : 0.33								
Development length of reinforcement required for strength								
Top Bar Extension: 300. mm								
Bottom Bar Extension: 300. mm								
<< <u>Back DK Cancel Next>></u>								

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 ...**.

escription —									
Top Bars:	ACI-318 Curtailment. 1	wo-\	√ay Slab. To	p Bars.			_		
Bottom Bars:	ACI-318 Curtailment.	wo-	√ay Slab. Bo	ttom Bars.					
	Top Bars			Bottom Ba	IIS				
	Cantilever	11	[Exterio	or Span		μ		Interior Span
	1 ×L	i	0.3	×L	0.3	×L	ļ	0.3	×L
Longer Bars:	0.5 *Ast	Ì	0.5	* Ast	0.5	* Ast	ļį.	0.5	* Ast
	0 min bars	1	0	min bars	0	min bars	1	0	min bars
	0.5 ×L	li	0.2	×L	0.2	×L	li	0.2	×L
Shorter Bars:	0.5 *Ast	ľ	0.5	* Ast	0.5	* Ast	l:	0.5	* Ast
	0 min bars	li	0	min bars	0	min bars	li	0	min bars
Noto: Tao bara	uton since component los	at la a							
Note. Top bare	extensions represent ien	juns i	neasured from	n supports					
urtail rules base	d on Curtail le	ngth	s based on —	Options-	o				Curtailment File
None / Ulf	Clear	Spa	n Length		Curtaiment Rul	es for Top Ba	ns D		File Open
	C Supp	ort S	pan Length		Curtaiment Rul	es for Bottom	ьar	2	File Save.
ADAFT					Extensions of 1	op bais over	Sup	pon	
ACI-STO			1	. 1				1	

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.

=+ Criteria - Lo	oad Combinati	ons														×
Strength lo	Strength load combination factors						Service lo	ad combinat	combination factors							
1: 1.5	SW + 1.5	ш.+	1.5 SDL	+ 1.5	× + 1	HYP	1: 1	SW + 0.3	LL +	1	SDL +	0.3	×+	1	PT	Sustained
2: 0	SW + 0	LL +	D SDL	+ 0	× + 0	HYP	2: 0	SW + 0	LL +	0	SDL +	0	×+	0	PT	Load
3: 0	SW + 0	LL +	D SDL	+ 0	× + 0	HYP	3: 1	SW + 1	LL +	1	SDL +	1	×+	1	PT	Total
4: 0	SW + 0	LL +	D SDL	+ 0	× + 0	HYP	4: 0	SW +	LL +	0	SDL +	0	×+	0	PT	Load
Material I	Factors						Initial load	combination	factors							
Concrete	: 1.5 F	Reinforcen	ient 1.1	5 Po	st-Tensioning	: 1.15	1: 1	SW +	LL +	0	SDL +	0	×+	1.15	PT	
	Lateral Load combination factors							lfweight SDL e Load X	= Super = Other	imposed Loading	DL					
							<u>0</u> K	Cancel								

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.

P1 Recycling P1 Recycling Image: Second Stresses (service) Tens OK Comp OK PT selection method Status of data displayed Image: Stresses (service) Tens OK Comp OK																
	Extreme fiber stresses [4] Tendon selection and extents [5]															
Tendon force and height [1] Required and provided PT force [2] Required PT force [3]																
Image: Tendon A Force selection method 1 - Specify a constant or variable force along a single tendon path identified by tendon profile selected Left: face of support at left of span Image: Tendon B C 1 - Single tendon path 1 - Specify a constant or variable force along a single tendon profile selected High: face of support at light of span Image: Tendon C Image: Processing a constant force for each of the tendon profile selected Processing a constant force for each of the tendon profile selected Left: face of support at light of span Image: Tendon C Image: Processing a constant force for each of the tendon profile selected Processing a constant force for each of the tendon profile selected Description																
<				Cu	rent Tendo >	on — — Tendor	Control Poin	t Height —:	· <			All Ten < Re	dons — equired F	Force —	->	>
Nu	umber of P strands	T Force per unit width	PT Force	P/A	%DL balanced	Left	Center	Right	T otal 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
<u>CR 11 203.53 1221.20 1.02 51 160 100 11 203.53 1221.20 1032 1.02 91</u>																

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



In this case all design status is Ok, hence click on $\boxed{\mathbb{R}}$ 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.

ecution Successfully Completed	×
- To display text Report use toolbar icon:	S E
To display detailed Graphs use toolbar icon:	\ominus
To display graphical Summary use toolbar icon:	-
- Other actions - select from program menu above.	

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 is on the main toolbar. The **Report Generator** screen shown in following figure will open.

📇 Report G	enerator				-						
	List of all Sections			List of S	ist of Selected Sections						
Report Table Concis Concis	t Cover of Contents se Report - Project Design Parameters an - Design Strip Report Getting St ar Reports - Compact ar Reports - Detailed ical Reports d	d Load Combinations arted Example 1	Report Cover Table of Conter Concise Report Tabular Report Graphical Report Legend	nts s - Compact rtts							
	User Selections										
	Remove Selection	Save Selection	Save as Default	Browse Reports	Update Comp	bany Info					
	Default		~	Create New Report	Į.						
	Create Optional Spreads	neet Report (XLS)									

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.





2.2.5. Summary Report

To open the **PT Summary Report** either click the open **BuilderSum** subtrom 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** $\overset{\text{DXF}}{\overset{\text{rest}}}$ 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.

Export Tendon Profiles and Reinfor	cement to DXF Drawing	– 🗆 X
Drawing Title	N PT - Getting Started Example 1	
Drawing Font G Arial C Arial C Times New Roman Font Scaling Large Font Scale: I Drawing Scaling Vertical Scale: I Tendon Profile Settings Tendon Diameter [mm]: I denter 2 Offset [mm]: I Height Roundup [mm]:	Tendon Profiles Selection Tendon A Tendon A Tendon B Tendon C Tendon Height Reference Level Definition Tendon Height Location Level Tendon Height Location Level Tendon Support Heights Tendon Drawing Arrangement Overlaid Tendon Profiles Stacked Tendon Profiles	Tendon Visualization Details Image: Tendon Solid Profile Image: Tendon Anchor Points Image: Tendon Control Points Image: Tendon Height Details Image: Tendon Height Details Image: Tendon Height Details Image: Tendon Height Stable Image: Tendon Heights at Extreme Points Image: Heights at 20th Points Image: Heights at Interval Points Image: Image: Tendon Heights Image: Tendon Heights at Interval Points Image: Tendon Heights at Interval Points
Reinforcement Layout Selection Longutudinal Reinforcement Stirrup Reinforcement	Reinf. Drawing Arrangement C Overlaid Reinf. Drawing Stacked Reinf. Drawing	Reinf. Visualization Details P Bar Solid Drawing Development Points Change File

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.



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

List of Contents

1.	INT	RODU	CTION	1
2.	TUT	ORIA	L PROBLEM	2
	2.1.	INPUT	AND MODEL GENERATION	5
		2.1.1.	General Settings & Design Code	5
		2.1.2.	Design Settings	5
		2.1.3.	Span Geometry	6
		2.1.4.	Support Geometry and Stiffness	7
		2.1.5.	Support Boundary Conditions	8
		2.1.6.	Loads	9
		2.1.7.	Concrete	10
		2.1.8.	Reinforcement	10
		2.1.9.	Post-Tensioning	11
		2.1.10.	Non-Prestressed Base Reinforcement	11
		2.1.11.	Allowable Stresses	12
		2.1.12.	Recommended Post-Tensioning Values	13
		2.1.13.	Calculation Options	14
		,	2.1.13.1. Force Selection	14
		-	2.1.13.2. Tendon Selection	14
		2.1.14.	Tendon Profile	15
		2.1.15.	Cover for Reinforcement and Post-Tensioning	16
		2.1.16.	Minimum Bar Extension	16
		2.1.17.	Reinforcement Curtailment	17
		2.1.18.	Load Combinations	17
	2.2.	EXECU	JTION AND OUTPUT	19
		2.2.1.	Recycling Window	19
		2.2.2.	Execution Completed	20
		2.2.3.	Show Graphs	21
		2.2.4.	Report Generation	22
		2.2.5.	Summary Report	22
		2.2.6.	DXF Export	23