# Structural Concrete Software

# ADAPT-PT/RC 2019

# **Getting Started Tutorial**

ADAPT-PT mode

Update:

December 2019

Copyright © ADAPT Corporation all rights reserved

ADAPT-PT/RC® 2019-Tutorial- 1

This ADAPT-PT/RC 2019 Getting Started Tutorial is intended to be used as a practical example and guide for modeling 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 Tutorial for RC mode and the ADAPT-PT/RC 2019 User Manual. Both documents can be accessed from the HELP menu of the program.

The example model is created with the help of a wizard which consists of different *Input Forms*. Each *Input Form* can be accessed at any time through the *Menu Bar*. The input that you provide on the *Input Forms* is displayed real-time in the *Structure View*. The view of the structure can be modified with the help of the *View Toolbar* which contains *View Tools* with which you can change the perspective, turn on and off components and zoom. The *Main Toolbar* contains *Common Tools* such as *New Project*, *Open Project* and *Save Project*.



FIGURE 1: ADAPT-PT User Interface

#### 1 COLUMN-SUPPORTED SLAB (TWO-WAY SYSTEM)

The objective of this tutorial is to explain how a floor strip or frame line is idealized from a complete floor system and modeled as a slab- or beam-frame ADAPT-PT/RC 2019. This tutorial will demonstrate the step-by-step procedure in the PT mode of ADAPT-PT/RC to generate data, analyze and design a column-supported slab which is a part of a floor system. A column-supported slab is generally considered as a two-way system. The tutorial covers the following features of the program:

- Generation of input data, using the simple "Conventional" option of the program.
- Design based on the post-tensioning "effective force" method, as opposed to selection of number of tendons.

The structure selected is a typical design strip from a floor system. The geometry, material, loading and other particulars of the structure are given below. The geometry of the whole floor is shown in **Figure 1-1**. The design strip for this tutorial is shown hatched in **Figure 1-2**.









The lengths and tributary widths of the spans of the design strip in orthogonal direction are shown in **Figure 1-3**.



FIGURE 1-3 The elevation of the design strip is shown in Figure 1-4.





The length of the spans of the design strip along support line 2 is shown in **Figure 1-5**.



FIGURE 1-5

The idealized design strip is shown in Figure 1-6.



#### FIGURE 1-6

#### MATERIAL PROPERTIES AND LOADING

Thickness of slab = 10 inch	
(i) Material Properties	
• Concrete:	
Compressive strength, f'c	= 4000 psi
Weight	= 150 pcf
Modulus of Elasticity	= 3605 psi
• Prestressing:	
Low Relaxation, Unbonded System	
Strand Diameter	= 0.5 inch
Strand Area	= 0.153 inch <sup>2</sup>
Modulus of Elasticity	= 29000 ksi
Ultimate strength of strand, f <sub>pu</sub>	= 270 ksi
Minimum strand cover	
From top fiber	= 1.5 inch
From bottom fiber	
Interior spans	= 1.5 inch
Exterior spans	= 2 inch
Nonnrestressed Reinforcement <sup>*</sup>	
Vield stress f.	= 60 ksi
Modulus of Elasticity	= 29000  ksi
Minimum Rebar Cover	= 1 inch Top and Bottom
(ii) Loading	
Superimposed Dead load	- 30 nsf (uniform)

Superimposed Dead load Live load = 30 psf (uniform) = 50 psf (uniform)

# 1.1 GENERATE THE STRUCTURAL MODEL

When the program is launched from the desktop shortcut or from the computer START menu, the opening screen will appear as below. This screen gives you the option to select a design mode of **Post-Tensioned or Mild Reinforced**. For this tutorial, select the **Post-Tensioned** option and **OK**. This will open the PT mode of the program where the PT input forms are active and loaded.



In the ADAPT-PT 2019 input screen, click the *Options* menu and set the *Default Code* as **American-ACI318 (2014) / IBC 2015** and *Default Units* as **American**.

# 1.1.1 Edit the Project Information

#### 1.1.1.1 General Settings

Open the new project by clicking either **New** on the *File* menu or the **New Project** button on the toolbar. This automatically opens the *General Settings* input screen, as in **Figure 1.1-1**. You can enter the *General Title* and /or *Specific Title* of the project in that window. For the purpose of this tutorial, enter the *General Title* as **Support Line 2**. This will appear at the top of the first page of the output. Enter *Specific Title* as **Two Way Slab**. This will appear at the top of each subsequent page of the output.

ADAPT-PT/RC<sup>®</sup> 2019-Tutorial- 7

Next, select Geometry Input as Conventional.

Next, select the *Structural System* as **Two-Way slab**. Then there is an option to include drop caps, transverse beam and/or drop panels. In this case select **No**. Select **No** for the option to *Recommend Slab Thickness*.

Click **Next** at the bottom right of this screen to open the next input screen, *Design Settings*.

L		General Settings		<u> </u>
General Title:				
Structural System:			Geometry input Conventional Drop Panel / Drop Cap / T Include Drops & Transverse Beams : Becommend Stab Thickness	C Segmental ransverse Beam C Yes ← No
<u>I</u> wo-Way Slab	One- <u>₩</u> ay Slab	B <u>e</u> am	Recommend Stab Trickness	⊂ Yes ເ⊂ No
	Cancel	<u>O</u> K		

FIGURE 1.1-1

#### 1.1.1.2 Design Code

In the Design Code screen, set the code as American-ACI318 (2014) / IBC 2015.

	Criteria - Design Code	
Design codes		
C American-ACI318 (1999)	C Brazilian-NBR6118 (2014)	€ European-EC2 (2004)
C American-ACI318 (2005) / IBC 2006	C British-BS8110 (1997)	C Indian-IS1343 (2004)
American-ACI318 (2008) / IBC 2009	C Canadian-A23.3 (1994)	C Hong Kong-CoP (2007)
C American-ACI318 (2011) / IBC 2012	C Canadian-A23.3 (2004)	C Chinese GB 50010 (2002)
American-ACI318 (2014) / IBC 2015	C Canadian-A23.3 (2014)	
🔿 Australian-AS3600 (2001)		
tesign Code Annex		
None		¥
<< Back	OK Cancel	Next >>

FIGURE 1.1-2

ADAPT-PT/RC<sup>®</sup> 2019-Tutorial- 8

#### 1.1.1.3 Design Settings

This screen is divided into three parts: *Analysis options, Design options,* and *Contribution to unbalanced moment*.

In *Analysis options*, you can select various calculation settings. First, select the *Execution Mode* as **Interactive**.

Next, select **Yes** for *Reduce Moments to Face-of-Support* option.

Select **No** for the option to *Redistribute moments*.

Select **Yes** for the Equivalent Frame Modeling.

In *Design options*, **check** *Use all provisions of the code* that you have selected in the previous step.

In Generate moment capacity based on, check Design Values.

In Calculation of Precompression (P/A), leave unchecked.

In *Contribution to unbalanced moment*, leave the contribution of *Top isolated bars*, and *Bottom isolated bars*, and *Post-tensioning* as default values (**100** percent).

nalysis options		⊤ Design options
Execution mode: Reduce moments to Face-of-Support : Redistribute moments (post-elastic) Use Equivalent Frame Method	C Automatic	Calculations of Precompression (P/A)     Consider all sections (ncluding drop panels/caps/TB's)
Contribution to unbalanced moment Top Isolated Bars: 100 <b>%</b>	Bottom Isolated Bars: 100	z Post-Tensioning: 100 z
	CC Back OK	Cancel Next >>

**FIGURE 1.1-3** 

ADAPT-PT/RC<sup>®</sup> 2019-Tutorial- 9

Click **Next** at the bottom right of the *Design Settings* screen to open the *Span Geometry* input screen.

# 1.1.2 Edit the Geometry of the Structure

# 1.1.2.1 Enter Span Geometry (Figure 1.1-4)





This screen is used to enter the cross-sectional geometry of the slab as per Figure 1-6.

Set the Number of Spans as 4 either by clicking the up arrow or using CTRL +.

Select the section, *Sec*, as **Rectangular** and edit **29.20 ft** for length, *L*, **310.00 in** for width, *b*, and **10 in** for height, *h*, for SPAN 1. Similarly enter details for SPAN 2, 3, 4 and R-Cant as shown in Figure 1.1-5. The widths (*b*) of each span are average tributary width for that span.

As you enter the values, the span is displayed in real-time in the 3D window.

The reference height (Rh) identifies the position of a reference line that is used to specify the location of the tendon. Typically, the reference height is set equal to the slab depth. Edit reference height, *Rh* as **10 in**, i.e., slab depth, for all spans.

The left and right multiplier columns (<-M and M->) are used to specify the tributary width to indicate how much of the tributary falls on either side of the support line. For this tutorial, tributary method is used, i.e., tributary widths are entered as width, *b*, and

the ratio of the tributary width on either side of the support line is entered as the left and right multipliers. For SPAN 1, enter <-M and M-> as **0.47** and **0.53** respectively. Similarly enter details for SPAN 2, 3, 4 and R-Cant as shown in Figure 1.1-5.

Number of 5	ipans 1 + [C	TRL +/	1				b→ ←	<sup>j</sup> ‡h <sub>f</sub> ⊱h <sub>m</sub> _h				L = f	it ners = in	
_egend _Cant = Lef	t Cantilev	/er	NP =	Non-Prismatic	Sec. =	= Section	0-0 =	Reference	plane	<u>?</u> Rh=	Distance fror	n <-M	1 = Left Multipli	er
n•∪ant = Hig	yni Cariui	evei	Fn =	Flishauc	Seg	- Segments	L =	opan ceni	gan.		ioneo piano	- 191	* = mignit wraitiip	oller
Label	PR		Sec.	Seg.	Sey	b b	L=	bf	hí	bm	hm	Rh	<- M =	M -> =
Label	PR PR		Sec.	Seg.	L 0.00	b 0.00	L = h 10.00	bf	hf	bm	hm	Bh 10.00	<- M = 0.50	M -> = 0.50
Label vpical L-Cant	PR PR	evel •	Sec.	Seg.	L 0.00	b 0.00	h 10.00	bf	hf	bm	hm	Rh 10.00	<- M = 0.50	M -> = 0.50
Label pical L-Cant SPAN 1	PR PR		Sec.	Seg.	L 0.00 29.20	b 0.00 310.00	h 10.00	bf	hf	bm	hm	Rh 10.00	<- M = 0.50 0.47	M -> = 0.50 0.53
Label pical L-Cant SPAN 1 SPAN 2	PR PR PR PR		Sec.	Seg.	L 0.00 29.20 32.81	b 0.00 310.00 372.00	h 10.00 10.00 10.00	bf	hf	bm	hm	Rh 10.00 10.00 10.00	<- M = 0.50 0.47 0.44	M -> = 0.50 0.53 0.56
Label ypical L-Cant SPAN 1 SPAN 2 SPAN 3	PR PR PR PR PR PR		Sec.	Seg.	L 0.00 29.20 32.81 34.74	b 0.00 310.00 372.00 417.00	L = h 10.00 10.00 10.00 10.00	bf	hf	bm	hm	Rh 10.00 10.00 10.00 10.00	<- M = 0.50 0.47 0.44 0.50	M -> = 0.50 0.53 0.56 0.50
Label ypical L-Cant SPAN 1 SPAN 2 SPAN 3 SPAN 4	PR PR PR PR PR PR PR		Sec.	Seg.	L 0.00 29.20 32.81 34.74 32.28	b 0.00 310.00 372.00 417.00 360.00	h 10.00 10.00 10.00 10.00 10.00	bf	hf	bm	hm	Rh 10.00 10.00 10.00 10.00 10.00	<- M = 0.50 0.47 0.44 0.50 0.57	M -> = 0.50 0.53 0.56 0.50 0.43

FIGURE 1.1-5

Click **Next** on the bottom line to open the next input screen.

# 1.1.2.2 Enter Support Geometry

This screen is used to input column/wall heights, widths and depths. You may enter dimensions for columns/walls above and/or below the slab.

Select the **Both Columns** from the support selection. Enter **9.02 ft** for *H1* and **9.84 ft** for *H2* in the typical row and press **ENTER**, since all the supports are the same height.

Next, enter the dimensions of the supports. B is the dimension of the column/wall crosssection normal to the direction of the frame. D is the column/wall dimension parallel to the frame.

Enter the given column/wall dimensions as in Figure 1.1-6.

On this input screen, you can select for each support whether the left edge and the right edge of that support is interior or exterior.

In this case, all supports are interior as the span is an interior span.

C	Lower Colur	nn	⊙ Both C	Columns	į	C No Colum	ns		↓ B		¢	ļ		
Legend H1 = Lower I H2 = Upper I % = Percent	Column Lengt Column Lengt age of colum	h h n stiffness	D D B	= Dimension i c = Diameter ( = Dimension r	in Span [ of circula normal to	Direction ar column ) span	Left ed Right e	lge = Interior :dge = Interio	or exterior or or exteri	 Dr		Jnits F 2	H = ft All others = ir	n
						o al service								
Support	H1	B	D	Dc	%	H2	B	D	Dc	%	Left ed	ge	Right ea	lge
Support	H1 9.02	B 47.24	D 47.24	Dc 1	% 00.00	H2 9.84	B 0.00	D 0.00	Dc	% 100.00	Left ed Exterior	ge •	Right ea	lge 
Support	H1 9.02 9.02	B 47.24 295.20	D 47.24 19.69	Dc 1	% 00.00 00.00	H2 <u>9.84</u> 9.84	B 0.00 295.20	D 0.00 19.69	Dc	<b>%</b> 100.00 100.00	Left ed Exterior Exterior	ge ▼	Right ec Exterior Exterior	ige ▼
Support	H1 9.02 9.02 9.02	B 47.24 295.20 47.24	D 47.24 19.69 47.24	Dc 1	% 00.00 00.00 00.00	H2 9.84 9.84 9.84	B 0.00 295.20 47.24	D 0.00 19.69 47.24	Dc	<b>%</b> 100.00 100.00 100.00	Left ed Exterior Exterior Exterior	ge •	Right eo Exterior Exterior Exterior	lge v
Support	H1 9.02 9.02 9.02 9.02 9.02	B 47.24 295.20 47.24 47.24	D 47.24 19.69 47.24 47.24	Dc 1 1 1 1	% 00.00 00.00 00.00 00.00	H2 9.84 9.84 9.84 9.84	B 0.00 295.20 47.24 47.24	D 0.00 19.69 47.24 47.24	Dc	<b>%</b> 100.00 100.00 100.00 100.00	Left ed Exterior Exterior Exterior Exterior	ge •	Right ec Exterior Exterior Exterior Exterior	
Support	H1 9.02 9.02 9.02 9.02 9.02 9.02	B 47.24 295.20 47.24 47.24 47.24	D 47.24 19.69 47.24 47.24 47.24	Dc 1 1 1 1 1	%       00.00       00.00       00.00       00.00       00.00       00.00	H2 9.84 9.84 9.84 9.84 9.84 9.84	B 0.00 295.20 47.24 47.24 47.24	D 0.00 19.69 47.24 47.24 47.24	Dc	<b>%</b> 100.00 100.00 100.00 100.00 100.00	Left ed Exterior Exterior Exterior Exterior Exterior	ge + + + + + + + + + + + + + + + + + + +	Right ec Exterior Exterior Exterior Exterior Exterior	lge



Click **Next** on the bottom line to open the *Supports Boundary Conditions* input screen.

#### **1.1.2.3** Enter Support Boundary Conditions

This screen is used to enter support widths and column boundary conditions.

Support widths can be entered if you answered "Yes" to the "Reduce Moments to faceof- support" question on the *Design Settings* screen, i.e., if you answered "No", you cannot input values in the *SW* column. This input value will be used to calculate the reduced moments.

Since the support width, *SW*, is set to the column/wall dimension (D) as a default, the SW values will be automatically determined from the support geometry and cannot be modified by the user. If you want to input the *SW* values, **uncheck** the *SW=Column Dimension box*.

Select the boundary conditions for *lower* and *upper* columns as **1**(fixed) from the drop down list.

Leave the *End Support Fixity* for both the left and right supports as default **No**. This will be used when the slab or beam is attached to a stiff member.

ab/beam boundary con Full fixity option left slab/beam end	dition at far ends Full fixity option right slab/beam end	- Column boundary con	dition	- Legend SW = Support design :	width in direction o	h	F
C Yes	C Yes	1-Fixed 2-Pinned	3-Roller	Boundary con LC = Lower Co UC = Upper Co	dition for : olumn N = I olumn F = F	Near <sup>:</sup> ar	F
nits		Support	SW	LC (N)	LC (F)	UC (N)	UC (F)
$\Omega(t) = in$			0.00	<u> </u>	<u> </u>	- 1	-
3 W = m		1	19.69	· _ 1	_ 1	<b>•</b> 1	
		2	47.24	<b>–</b> 1	- 1	- 1	-
		3	47.24	<b>–</b> 1	- 1	- 1	
	la contraction de la contracti	4	47.24	<b>–</b> 1	- 1	- 1	-
	G Sut - Actual width of	5	47.24	· _ 1	<b>v</b> 1	• 1	
	support						
	1	1		1			



Click **Next** at the bottom of the screen to open the input screen *Loading*.

#### 1.1.3 Enter Data

#### 1.1.3.1 Edit the Loading Information

Enter the span number as **1** in the *Span* column. If the loads are the same for all the spans, you can type **ALL** or **all** in the *Span* column. This will copy the data to all of the spans.

If you choose not to include Self-weight, you now have the option to define the selfweight (**SW**) as a *Class*. In any case, you can choose to specify additional dead load as superimposed dead load (**SDL**) as a *Class*.

PT/RC 2104 gives you the option to specify any load as an **X** Class.

Select the *Class* as **SDL** from the drop-down list and specify the load type as uniform either by typing **U** in *L*-? or by **dragging the icon** from the graphics of the uniform loading.

The default of the load type when you select the load class is L-U; so leave it as is for this tutorial.

Type **0.03**  $k/ft^2$  (=30 psf) for superimposed dead load in the *w* column. You can enter DL with or without self-weight, since the program can calculate self-weight automatically. In order to calculate the self-weight automatically, you must answer **Yes** to *Include Self-*

*Weight* question at the top right of the screen and enter **150 pcf** as unit weight of concrete.

Repeat the procedure for live load by entering the **span number** and changing the *Class* to **LL** and *w* value to **0.05**  $k/ft^2$  (=50 psf) for all spans.

Answer **Yes** to *Skip Live Load*? at the top left of the screen and enter the *Skip Factor* as 1 (Figure 1.1-8).



FIGURE 1.1-8

If you go to any other form and navigate back to the Loads input form, you will see that the loading information is now entered in the table for each span (**Figure 1.1-9**).

Loa	ıds			nend-							-	×
a= b= c=	, ft w = k/ ft P1 = k/ ft P2 = k/	ft² M = k ft F = k ft	-ft S	W = Selfweight L = Live Load < = Other user def	ined load case	CL = Cant CR = Can SDL = Sup	ilever Left tilever Right perimposed Dea	L/T-? = Load T d Load	ype Skip	) Live Load ( ) Factor:	•Yes C M	ło
, Ţ	Jniform	artial Uniform	Concentrated	Moment L-M	Line		iangle	b a Pitte Pitte Variable T-V	Trapezoidal T-Z	Include Se Ye Unit Weigh	fweight es C No t: 150 pcf	
	Span	Class	L/T-?	w	P1	P2	а	Ь	с	м	F	
1	1 💌	SDL 👱	EU	0.030							1	
2	1 💌	LL 👱	ĿU	0.050				[]				
3	2 💌	SDL 👱	ĿU	0.030	1			[]				
4	2 💌	և 👱	ĿU	0.050				[]			[]	
5	3 💌	SDL 👱	ĿU	0.030	1	i		[]			11	
6	3 👱	Ш 👱	ĿU	0.050	- 11	·		[]			[]	
7	4 🗸	SDL 👱	ĿU	0.030				[]			()	
8	4 💌	և 👱	ĿU	0.050				[]			[]	
9	CR 👤	SDL 👱	ĿU	0.030				[]			()	
10	CR 💌	LL 👱	ĿU	0.050				[]			i	
11	<b>_</b>	<u> </u>	1					[]			()	
12	<u> </u>	<u> </u>	A		1	<u> </u>		[]			[]	
13	-	<u> </u>	a		- 1			[]				
1.14				1				· ·			· · · · · · · · · · · · · · · · · · ·	
				<< <u>B</u> ack	Cancel		<u>o</u> k	<u>N</u> ext>>				

#### FIGURE 1.1-9

Click **Next** at the bottom of the screen to open the *Material - Concrete* input screen.

#### 1.1.4 Edit the Material Properties

#### 1.1.4.1 Enter the Properties of Concrete

Select **Cylinder** concrete strength at 28 days. Select the **Normal** *weight* and enter the *strength at 28 days* for slab/beam and column as **4000 psi**. When you press **Enter** from the strength input value, the *Modulus of Elasticity* will be calculated automatically based on the concrete strength and the appropriate code formula.

For this tutorial, keep the value of creep coefficient as **2**. The creep coefficient will be used in the calculation of long-term deflection. Consider *Concrete strength at stressing* as **3000 psi**.

Slah / Ream		
Weight :       Image: Second Control of Seco	Strength at 28 days (f'c) Modulus of Elasticity at 28 Days :	4000. psi 3605. ksi
<< <u>B</u> ack Cancel	<u>D</u> K <u>N</u> ext >>	

**FIGURE 1.1-10** 

#### 1.1.4.2 Enter the Properties of Reinforcement

In the section *Longitudinal reinforcement*, change the values for *Yield Strength* and *Modulus of Elasticity* to **60 ksi** and **29000 ksi** respectively. Change the *Preferred Bar Sizes* 

*for Top and Bottom* to **5** and **8** respectively. These will be used when calculating the number of bars required.

In Shear reinforcement, select *Stud (headed bar)* and change *Preferred Stud diameter, Yield strength shear reinforcement* and the *Number of rails per side* to **0.5 inch, 60 ksi** and **2** respectively. Set the number of rails per side to **2.** Do not select the two options at the bottom-right of the input window. These are new options for v2019 and allow the user to check two-way shear with respect to seismic drift requirements as well as consideration of the least critical setion ,bo, at d/2 from the edge of reinforcement. Refer to the **ADAPT-PT/RC 2019 User Manual** for more information.

ongitudinal reinforcement			Shear reinforcement		
Yield strength (ly) main bars : Modulus of elasticity : Preferred bar size for top bars : Preferred bar size for bottom bars :	66.72 29007.37 5 5 5	ksi	Preferred stud diameter : Yield strength (fy) shear reinforcement: Number of rails per side "b": Number of rails per side "d":	0.5	▼ in 60 ksi 2 2
Column Strip Allocation In spans : Over interior columns : Over exterior columns :	60. 75. 100.	% % %	<ul> <li>Include minimum shear reinforcement</li> <li>Consider octagonal critical sections p</li> <li>Edge Distance of Rails:</li> </ul>	for drift ast shear reinfo 1	in in
	<< Back	<u>o</u> k	Cancel <u>N</u> ext>>		

**FIGURE 1.1-11** 

Click **Next** at the bottom of the screen to open the next input screen.

# **1.1.4.3** Enter the Post-Tensioning System Parameters

Select the *Post-tensioning system* as **Unbonded** and leave the default values of the other properties as they are as in **Figure 1.1-12**.

Post-tensioning system : C Bonded C Unbonded Area of Tendon (one or more strands) : 0.153 in <sup>2</sup>
Area of Tendon (one or more strands): 0.153 inf
Ultimate Strength of Tendon (fpu): 270. ksi
Effective (long-term) Stress (fse) : 175. ksi



Click **Next** at the bottom of the screen to open the input screen, *Base Non-Prestressed Reinforcement.* 

#### 1.1.4.4 Edit Base Reinforcement

The program allows you to specify a base reinforcement that is taken into consideration when designing the structure. Select **Yes** in the *Base Reinforcement* section.

You have the choice between defining a mesh or isolated rebar. For this example choose **Isolated** from the drop down box.

Next specify the span where your base reinforcement starts. For this example, let the rebar start at the beginning of span 1. Therefore, enter a  $\mathbf{1}$  in *First end location* and a  $\mathbf{0}$  in *X*1/*L*.

If you want to specify the end of the reinforcement at the end of span number 4, define **4** for *Second end location* and **1** for X2/L.

Furthermore, you specify **4** bars (*Number*) with *Bar Size* of **6** as **Bottom** bars with a *Cover* of **2 inch**.

Ba	se Non-Prestressed F	Reinforcement	t								×
			Ba	se Reinforcement	φY	es 🤇 No					
Leg	end										
Ty (Is	pe = Mesh reinforceme olated)	ent or single stra	ight bar(s)	L = spar	ı length asso	ciated to X1 and X2	2	Spacing and	cover are in ir		
Fir wł X1 rei	st end location, Secon nich reinforcement start , X2 = distances of the nforcement to its immed	d end location = s and terminate first and secon diate left suppor	= the spans s id end of a t	in Number Spacing	= number of = distance b	isolated bars between the mesh b	ars				
	Туре	First end location	X1/L	Second end	X2/L	Bar Size	Number	Spacing	Top/Botto	n Cover	
1	Isolated 💌	1 💌	0.00	4 🗾	1.00	6 💻	4		Bottom	- 2	
2	<u> </u>								-	<u>-</u>	
3	<u> </u>									<b>1</b>	-1
4									-	1	
E .		<u> </u>		<u> </u>						<b>_</b>	
5										. 1	•

# **FIGURE 1.1-13**

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Allowable Stresses*.

# 1.1.5 Edit the design criteria

#### 1.1.5.1 Enter the Initial and Final Allowable Stresses

Tensile stresses are input as a multiple of the square root of  $f'_c$ , and compressive stresses are input as multiple of  $f'_c$ .

The default values given in **Figure 1.1-14** are according to **ACI 318 (2014)**. Leave the default values as they are.

ension stress	es		
	Initial Stress / f'ci^½	Sustained Stress / f'c^½	Total Stress / f'c^½
Top Fiber :	3.	6.	6.
Bottom Fiber :	3.	6.	6.
	0.0	0.45	0.0
	1	1 0.43	1

**FIGURE 1.1-14** 

Click **Next** at the bottom of the screen to open the next input screen, *Criteria* – *Recommended Post-Tensioning Values*.

# **1.1.5.2** Enter the Recommended Post-Tensioning Values

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*.

The values shown in **Figure 1.1-15** are according to the selected code and the experience of economical design.

Average Precompre Minimum	ssion 125.	psi	Maximum :	300.	psi
Percentage of Dead	Load to E	Balance			
Minimum :	50	%	Maximum :	125	%
	Maximur	n Spacing E (multiple of	Between Tendons : slab depth)	8	
	- 1		1		

**FIGURE 1.1-15** 

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Calculation Options*.

# 1.1.5.3 Select the Post-Tensioning Design Option

The two design options are Force Selection" and "Force/Tendon Selection", as in **Figure 1.1-16**. **Force Selection** is the default option. Keep the default option.

nalysis and design met	nod	C Calci	ulate force/number
	m	of ter	ndons
<< <u>B</u> ack	<u>о</u> к	<u>C</u> ancel	Next >>

# **FIGURE 1.1-16**

In this 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. For more information on the Calculated Force option, please refer to the **ADAPT-PT/RC 2019 User Manual**.

Click **Next** at the bottom of the screen to open the next input screen, *Criteria – Tendon Profile*.

# 1.1.5.4 Specify the Tendon Profiles

The program allows you to specify up to three tendon paths per span. You can define one profile for each of the three tendons.

In the section *Option for tendons* you can define *the Default extension of terminated tendon as fraction of span*.

Also, you can specify the *Shape of tendon extension* from the *Left end* and the *Right end*.

Criteria - Tendon Legend	Profile X3 4 L sed Parabola		2 = Partia	X3 +		X1 X2 3 = Harped Parabola X1 X2 4 = Straight X1 5 = Extended Reversed Parabola
Tendon A prol Span Typical	file Type	Tendon B p X1/L 0.100	vrofile X2/L 0.500	Tendon X3/L 0.100	C profile	Option for tendons Default extension of terminated tendon as fraction of span: Left end 0.2 Right end 0.2
Span 1 Span 2 Span 3 Span 4 Right cantilever	1 v 1 v 1 v 1 v 1 v	0.100 0.100 0.100 0.100	0.500 0.500 0.500 0.500	0.100 0.100 0.100 0.100		Shape of tendon extensions:         Tendon C (Left end)         Tendon B (Right end)                © Downward parabola; anchor at centroid              © Downward parabola; anchor at centroid              © Downward parabola; anchor at centroid              © Townward parabola; anchor at centroid              © Downward parabola; anchor at centroid
	_			< <u>B</u> ack	<u>o</u> ĸ	Cancel

For this example, leave the default values as shown in **Figure 1.1-17**.

# **FIGURE 1.1-17**

Click **Next** at the bottom of the screen to open the next input screen, *Criteria* – *Minimum Covers*.

# 1.1.5.5 Specify Minimum Covers for Post-Tensioning Tendons and Mild Steel Reinforcement

The cover for the prestressing steel is specified to the center of gravity of the strand (cgs). Edit *CGS* of the tendon as **1.5 inch** for both the top fiber and the interior spans of bottom fiber and **2.0 inch** for the exterior spans for the bottom fiber.

For nonprestressed reinforcement, edit **1.0 inch** *Cover* for both the top and the bottom (as shown in **Figure 1.1-18)**.

finimum CGS of tendon from the top fiber:	1.5 in
finimum CGS of tendon from the bottom fiber	
Interior Spans :	1.5 in
Exterior Spans :	2 in
Non-prestressed Reinforcement	
Clear Bar Cover (Top) :	🚺 in
Clear Bar Cover (Bottom) :	1.0 in
Clear Bar Cover (Side) :	1.0 in

**FIGURE 1.1-18** 

Click **Next** at the bottom of the screen to open the input screen, *Criteria* – *Minimum Bar Extension.* 

# 1.1.5.6 Specify Minimum Bar Length and Bar Extension of Mild Steel Reinforcement

The values given as default for "minimum bar lengths" are according to ACI-318 (2014). Use the default values (as shown in **Figure 1.1-19**). Note that the "development length" is user-defined and is the bar extension beyond point of zero moment where reinforcement is no longer required.

The values entered for cut-off lengths are used to calculate top and bottom bar lengths when minimum reinforcement requirements govern. Note that where Rebar Curtailment is input, those settings will take priority over Minimum bar length settings.

Criteria - Minimum Bar Extension	×
- Minimum bar lengths Cut off length of minimum reinforcement over suppo Cut off length of minimum reinforcement in span (len	rt (length/clear span) : 0.17 ngth/clear span) : 0.33
-Development length of reinforcement required for str Top Bar Extension:	ength
Bottom Bar Extension:	12. in
<< Back <u>D</u> K <u>C</u> a	ancel <u>N</u> ext >>

**FIGURE 1.1-19** 

Click **Next** at the bottom of the screen to open the input screen, *Criteria – Rebar Curtailment*.

# 1.1.5.7 Specify Top and Bottom Rebar Curtailment Input for Spans

The default values given for Long and Short Bars relative for length equal to fraction of span and % of area of required steel are according to Chapter 13 of ACI-318 (2014). Keep the default values (as shown in **Figure 1.1-20 and Figure 1.1-21**). Note that curtailment rules relative to the selected code are applied to top and bottom bars and are conservatively adjusted such that the bar lengths are equal on both sides of the support or equal at both sides of the center of span. Also, 2 bars are set as a minimum requirement.

The reinforcement result output for this example will be based on the curtailment rules which take priority over reinforcement rules relative to the previous section for bar extensions. In the case where curtailment rules are not sufficient to the default solution (that solution related to the bar extension input) the reinforcement arrangement and length output will be given as that taken from the bar extension input.

			Crit	teria - Rei	nforceme	nt Curtailm	ient			
escription										
Top Bars:	ACI-318 0	Curtailment. Tv	vo-W	/ay Slab. Top	) Bars.			_		
Bottom Bars:	ACI-318 0	Curtailment. Tv	vo-W	/ay Slab. Bot	tom Bars.			_		
	Top Bars		Υ		Bottom Ba	ars				
	Car	ntilever	ij		Exterio	or Span	1	Ĩ,		Interior Span
	1	×L	Ì	0.3	×L	0.3	×L	Ì	0.3	×L
Longer Bars:	0.5	*Ast		0.5	* Ast	0.5	* Ast	1	0.5	*Ast
	0	min bars		0	min bars	0	min bars	1	0	min bars
	0.5	×L	1	0.2	×L	0.2	×L	1	0.2	*L
Shorter Bars:	0.5	* Ast		0.5	* Ast	0.5	* Ast	1	0.5	* Ast
	0	min bars		0	min bars	0	min bars	a Te	0	min bars
Note: Top bare urtail rules base 'None / Off 'User Defined	extensions ri	Curtail lengt	hs m gths Span	easured from based on	supports	Curtailment Ru Curtailment Ru	ules for Top Bar	s Bar	s	Curtailment File
ADAPT ACI-318		C Suppo	rt Sp	oan Length	I Equa	Extensions of	Top Bars over S	Sup	port	File Save
Eurocode EC	2	<< Back	<	0	<	Cancel	Next>>			

**FIGURE 1.1-20** 

		C	riteria - Rei	nforcemer	nt Curtailm	ient		[
Description								
Top Bars:	ACI-318	Curtailment. Two	o-Way Slab. Top	Bars.				
Bottom Bars:	ACI-318	Curtailment. Two	o-Way Slab. Bott	tom Bars.				
	Top Bars		<u>}</u>	Bottom Ba	IS			
	Ca	antilever	<u> </u>	Exterio	r Span	_ 1	<u> </u>	Interior Span
	0	- <sub>*L</sub>	0	×L	0	×L	0	×L
Longer Bars:	0.5	*Ast	0.5	* Ast	0.5	*Ast	0.5	* Ast
<i></i>	2	min bars	2	min bars	2	min bars	2	min bars
	0	- *L		×L	0	= <sub>*L</sub>	0	×L
Shorter Bars:	0.5	* Ast	0.5	* Ast	0.5	* Ast	0.5	* Ast
	0	min bars		min bars	0	min bars	0	min bars
Note: Bottom b Curtail rules base None / Off User Defined ADAPT	ar extensio	Curtail lengt Curtail lengt Clear Sp C Support	is measured from hs based on ban Length Span Length	Options Options Apply Apply Equal	Curtailment Ru Curtailment Ru Extensions of	iles for Top Bars iles for Bottom B Top Bars over S	ars upport	Curtailment File - File Open File Save
• ACI-318	_		1		- 1		1	
<ul> <li>Eurocode EC</li> </ul>	2	<< <u>B</u> ack	4 <u>0</u>	\$	Lancel	<u>N</u> ext >>		

FIGURE 1.1-2

# 1.1.5.8 Input Load Combinations

**Figure 1.1-20** shows the screen which is used to input the load combination factors for service and strength (ultimate) load conditions. It is also used to enter any applicable strength reduction factors. The default values are according to the American-ACI318 (2014)/IBC 2015.

The program allows you to specify four strength load combinations and four service load combinations. For American-ACI318 (2014)/IBC 2015, two of the service load combinations are reserved for sustained load and two for total load.

Criteria - Lo	oad Combinat	ions																	×
Strength lo	ad combinat	ion facto	ors						Service	oad con	nbinati	on facto	ors —						1
1: 1.2	SW + 1.6	LL +	1.2	SDL +	1.6	×	. 1	HYP	1: 1	SW +	0.3	LL +	1	SDL +	0.3	× +	1	PT	Sustained
2: 0	sw + 0	LL +	0	SDL +	0	×	. 0	HYP	2: 0	sw +	0	 LL +	0	SDL +	0	×+	0	PT	Load
3: 0	sw + 0	LL +	0	SDL +	0	×	. 0	HYP	3: 1	sw +	1	LL +	1	SDL +	1	×+	1	PT	Total
4: 0	sw + 0	LL +	0	SDL +	0	×	0	HYP	4: 0	sw +	0	LL +	0	SDL +	0	×+	0	PT	Load
Maximum	strength redu	ction fac	ctors						Initial loa	d combi	nation	factors							
Bending: (max value) :	0.9	One-way shear :	0.75	T w she	o-way [ ar :	0.75			1: 1	SW +	0	LL +	0	SDL +	0	×+	1.15	PT	
			-La	teral Loa - Include Lateral L	ad con .oads	nbinati	on fact et Value	ors	Legend SW = S LL = I	ielfweight ive Load	SDL X	= Superi = Other I	mposed Loading	IDL					
							[	<< <u>B</u> ack	<u>0</u> K	<u><u> </u></u>	ncel								

For this example, do not include lateral loads.

FIGURE 1.1-22

# 1.2 SAVE AND EXECUTE THE INPUT DATA

To save the input data and execute the analysis, either select **Execute Analysis** on the menu bar or click on the **Save & Execute Analysis** button **1**. Then, give a **file name** and **directory** in which to save the file. Once the file is saved, the program will automatically execute the analysis by reading the data files and performing a number of preliminary data checks.

Once the execution gets completed, the "PT Recycling" window, as shown in **Figure 1.2-1** opens. If an error is detected, the program will stop and display a message box indicating the most likely source of the error.

cycl	e Recall s Exit	Bi St	eration No: 3 T Force alanced DL tresses (servi tresses (initial	8   W M M ce) Tr	eight of PT in OK in OK ens OK ens OK	0.48 Ib/ Max Max Comp Comp	7ft2 OK OK OK	PT selection Force sel Lendon :	method ection	Status of da	Iata displayed         NG = No Good (does not meet speci requirements)           OK = meets specified requirements							
		Extreme fib	er stresses [ ;	<u>4</u> ]		Ĭ	Tendon	selection and	extents [ 5	1								
	Te	ndon force a	and height	[1]	Ĩ		Required ar	nd provided P	T force [ <u>2</u> ]				Require	d PT forc	e[ <u>3</u> ]			
Force selection method     Tendon B     Tendon C     Tendon C     Current Text						1 - Specify a constant or variable force along a single tendon path identified by tendon profile selected         Left: face of supp.           2 - Specify a constant force for each of the tendon profiles selected         PA: average prec 3DL balanced: pre- balanced by tendon									port at left of span inport at right of span ecompression at midspan percentage of total dead load don			
				Cu	rrent i enut <-	- Tendon	Control Poin	t Height —	>			< Re	equired F	Force —	->			
	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	28	28.9	746.2	241	69	5.00	2.00	8.50	28	28.9	746.2	539	539	539	241	69		
	28	24.1	746.2	201	67	8.50	1.50	8.50	28	24.1	746.2	554	554	554	201	67		
	28	21.5	746.2	179	54	8.50	1.50	8.50	28	21.5	746.2	697	697	697	179	54		
	28	24.9	746.2	207	50	8.50	2.00	5.25	28	24.9	746.2	745	745	745	207	50		
	28	27.5	746.2	229	108	5.25		5.00	28	27.5	746.2	408			229	108		

# FIGURE 1.2-1

Here you can optimize the design by changing the tendon forces and tendon heights. Select **1-Single tendon path** for the *Force selection method*.

'% DL balanced' is close to the max limit of 125 for CL. For the 'Change the PT force to **500 kips** for CL. The status indicator at the top right of the Recycle window will begin to flash as 'Recycle'.

Since we selected the "Force Selection" option during data entry, the program will only allow the "Force Selection" mode for execution.

Once all of the changes are made as shown in **Figure 1.2-2**, click on the **Recycle** button to update all of the tabs, the Design Indicator box and the Recycle Graphs.

State         Iteration No: 3         Weight of PT           recycle         Recall         PT Force         Min            Balanced DL         Min          Stresses (service)         Tens            Graphs         Egit         Stresses (service)         Tens          Stresses (initial)         Tens						0.48     b/t2       Max        Comp        Comp						NG = No Good (does not meet specified requirements) OK = meets specified requirements					
		Extreme fib	er stresses [	4]		Ì	Tend	on selection and	i extents [ 5	1	]						
	Te	endon force a	and height	[1]	Ĩ		Required	and provided F	PT force [2	1			Require	d PT forc	e[ <u>3</u> ]		
Tendon A     Force selection method						1 - Specify path identil 2 - Specify selected	a constant or ied by tendor a constant fo	r variable force and profile selected profile selected prce for each of	Left: face of support at left of span Center: midspan Right: face of support at right of span P/A: average precompression at midspan %DL balanced: percentage of total dead load balanced by tendon All Tendons								
	<			Cu	rrent Tend	on ———		×	» <			All Ten	dons —				
	<	PT Force per	PT Force	— Cu  P/A	rrent Tendo K SDL balanced	on — — Tendor Left	Control Po	oint Height — Right	-> <	Total PT force	Total PT	All Ten < Re Left	dons — equired F Center	orce – Right	> Total P/A	Total %DI	
1	<	PT Force per unit width 28.9	PT Force 746.2	Cu P/A 241	%DL balanced	on — — Tendor Left 5.00	Control Po Center	oint Height — Right 8.50	Total strands	Total PT force per unit width 28.9	Total PT force 746.2	All Ten < Re Left 539	dons — equired F Center 539	Force — Right 539	-> Total P/A 241	Total %DL	
1 2	<	PT Force per unit width 28.9 24.1	PT Force 746.2 746.2	Cu P/A 241 201	Rirrent Tendi C %DL balanced 69 67	on — — Tendor Left 5.00 8.50	Control Po Center 2.00 1.50	pint Height — Right 8.50 8.50	Total strands	Total PT force per unit width 28.9 24.1	Total PT force 746.2 746.2	All Ten < Re Left 539 554	dons — equired F Center 539 554	Force	> Total P/A 241 201	Total %DL 69 67	
1 2 3	Number of strands 28 28 28 28 28	PT Force per unit width 28.9 24.1 21.5	PT Force 746.2 746.2 746.2	P/A 241 201 179	%DL balanced 69 67 54	on — — Tendor Left 5.00 8.50 8.50	Control Po Center 2.00 1.50 1.50	Dint Height	Total strands 28 28 28 28 28 28 28 28 28 28 28 28 28	Total PT force per unit width 28.9 24.1 21.5	Total PT force 746.2 746.2 746.2	All Ten < Re Left 539 554 697	dons — equired F Center 539 554 697	Force — Right 539 554 697	-> Total P/A 241 201 179	Total %DL 69 67 54	
1 2 3 4	Number of strands          28         28         28         28         28         28         28         28         28         28         28         28         28         28	PT Force per unit width 28.9 24.1 21.5 24.9	PT Force 746.2 746.2 746.2 746.2	P/A 241 201 179 207	%DL balanced 69 67 54 50	on — Tendor Left 5.00 8.50 8.50 8.50	Control Po Center 2.00 1.50 1.50 2.00	Dint Height	> Control Strands 28 28 28 28 28 28 28 28	Total PT force per unit width 28.9 24.1 21.5 24.9	Total PT force 746.2 746.2 746.2 746.2 746.2	All Ten < Re Left 539 554 697 745	dons — equired F Center 539 554 697 745	Force	-> Total P/A 241 201 179 207	Total %DL 69 67 54 50	

FIGURE 1.2-2

After the recalculation of the stresses and required forces along the member, based on the current values, the window, as shown in **Figure 1.2-3**, with the "OK" status for all items in the design indicator box opens.

			resses (initial	) T 41	ens OK	Comp	OK Tendon	selection and	evtents [5	1	ן					
	Te	ndon force a	and height	<u>[1]</u>	Y	<u> </u>	Required ar	nd provided P	T force [ 2 ]		_L		Require	d PT forc	∍[ <u>3]</u>	
(° 1	Fendon A	Force 1 C 2	e selection m - Single tenc - Multiple ter	ethod Ion path Idon path		1 - Specify path identifi 2 - Specify selected	a constant or v ied by tendon p a constant force	ariable force a rofile selected e for each of f	llong a sing he tendon p	e tendon profiles	Left f Cente Right P/A: a %DL t balani	ace of su r: midspar face of s average p balanced: ced by ter	pport at lef n upport at ri percentag ndon	t of span ght of spa sion at mi je of total	an dspan dead load	
	<			Ci	rrent Tendo ->	on — — Tendon	Control Poir	nt Height —	> < >			All Ten <— Re	dons — equired F	Force —	->	
1	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
	28	28.9	746.2	241	69	5.00	2.00	8.50	28	28.9	746.2	539	539	539	241	69
	28	24.1	746.2	201	67	8.50	1.50	8.50	28	24.1	746.2	554	554	554	201	67
	28	21.5	746.2	179	54	8.50	1.50	8.50	28	21.5	746.2	697	697	697	179	54
6	28	24.9	746.2	207	50	8.50	2.00	5.25	28	24.9	746.2	745	745	745	207	50
	Contraction of the Contraction o	18.4	500.0	153	72	5.25		5.00	19	18.4	500.0	408			153	72



You can check the final stresses either by clicking **Extreme fiber stresses [4]** tab in the *PT Recycling* window (**Figure 1.2-4**).

PT Recycling									
Becycle Regall	Iteration No: <b>4</b> PT Force Balanced DL Stresses (service) Stresses (initial)	Weight of F Min OK Min OK Tens OK Tens OK	PT 0.49 K Max K Max K Comp K Comp	Ib/ft2 OK OK OK	PT selection r	nethod S ction lection	Status of data d	isplayed —	NG = No Good (does not meet specified requirements) OK = meets specified requirements
Tendon for	Tendon force and height [1] Required and provided PT force [2] Required PT force [3]								Required PT force [3]
Extreme fibe	er stresses [ 4 ]		γ'	Tendo	n selection and ex	ents [5]	—		
C       Sustained load condition         C       Initial load condition         Tension Stress / (fc)^1/2       Compression Stress / (fc)       <- Allowable/suggested values ->         Left       Center       Right       Left       Center       Right       Tens (bp)       Tens (bot)       Comp         1       2.523       0.000       0.160       0.102       0.114       6.000       6.000       0.600         2       0.933       0.438       2.983       0.116       0.107       0.147       6.000       6.000       0.600         3       4.091       2.822       4.537       0.154       0.134       0.161       6.000       6.000       0.600         CR       0.000       0.160       0.125       6.000       6.000       0.600       0.600         CR       0.000       0.633       0.33       0.125       6.000       6.000       0.600									

# FIGURE 1.2-4

**Graphs** displays a set of three graphs which provide detailed information on the tendon profile, the tension and compression stresses and the required versus provided post-tensioning forces at 1/20<sup>th</sup> points along the spans **(Figure 1.2-5).** 





The top diagram, the **Tendon Height Diagram** shows the elevation of tendon profile selected. Tendon profile can be viewed either with concrete outline or without concrete outline by checking the option at the left of the screen.

The second diagram, **Stress Diagrams**, plots the maximum compressive and tensile stresses at the top and bottom face of the member. You can view the stresses due to e.g. *Self-weight, Superimposed Dead Load, Live Load, Post-tensioning* and *Sustained* each separately, or in combination, by selecting the options at the screen. Also you can verify the top and bottom stresses due to the service combination with the allowable values. In **Figure 1.2-5**, it shows the final top fiber stresses with the allowable stresses.

In which, *gray color* represents the allowable value, *top curve* represents the tensile stress and *bottom curve* represents the compressive stress. If the calculated stress is not within the limit, i.e., the top or bottom curve is outside the gray portion; you need to modify the forces to optimize the design.

The third diagram, **Post-Tensioning Diagrams** shows the required and provided posttensioning force at 1/20th points along each span. The *vertical line* represents the *required* post-tensioning and the *horizontal line* represents the *provided* post-tensioning at that section. At each design section along a span, the program performs an analysis based on the post-tensioning force at that section

**Required and provided PT force [2]** tab in the *PT Recycling* window shows PT forces provided in the left, center and right region of each span as well as the forces required in each region for minimum P/A, Wbal %DL and allowable tensile stresses (**Figure 1.2-6**).

PT Recycling Recycle Recall Graphs Exit	Iteration No: 4 PT Force Balanced DL Stresses (service) Stresses (initial)	Weight of Min O Min O Tens O Tens O	PT 0.49 I K Max K Max K Comp K Comp	b/ft2 OK OK OK	PT se © Eo C Le	lection method rce selection ndon selection	CURRENT	NG = No Good (does not meet specified requirements) OK = meets specified requirements
Extreme	e fiber stresses [ <u>4</u> ]		Ì	Т	endon selecti	on and extent	:[5]	
Tendon for	ce and height [ <u>1</u> ]			Require	ed and prov	ided PT for	2	Required PT force [3]
	1 2 3 4 CR	539.1 554.2 696.5 745.4 407.5	539.1 554.2 696.5 745.4	539.1 554.2 696.5 745.4	746.2 746.2 746.2 746.2 746.2 746.2	746.2 746.2 746.2 746.2 746.2	746.2 746.2 746.2 746.2	
					All f	Forces are in	n Kips	

# **FIGURE 1.2-6**

**Required PT force [3]** tab in the *PT Recycling* window shows the required PT forces for only the most recently calculated profile. (Figure 1.2-7).

PT Recycling Recycle Graphs Exit	Iteration PT For Balanc Stresse Stresse	n No: <b>4</b> ce ed DL :s (service) :s (initial)	Weight of PT Min OK Min OK Tens OK Tens OK	0.49 Max Max Comp Comp	Ib/ft2 OK OK OK OK	PT select © Eorce © Iende	tion method – selection on selection	Status o	if data displayed —	NG = No Good (does not meet specified requirements)
Extr	reme fiber str n force and h	esses [ <u>4</u> ] eight [ <u>1</u> ]		Ì	Te Requ	endon selection ired and provide	and extents [	51		Required PT force [3]
	1 2 3 4 CR	< Rec 539.1 554.2 696.5 745.4 407.5	Quired PT Fc Center 539.1 554.2 696.5 745.4	Prce> Right 539.1 554.2 696.5 745.4	<- Based 474.8 374.7 596.4 616.6 0.0	d on 1 ensile 3 Center 40.4 250.8 429.0 332.0	Stresses -3 Right 293.8 533.8 626.2 236.7	Other Cons P/A (mid) 387.5 465.0 521.3 450.0 407.5	Wbal %DL 539.1 554.2 696.5 745.4 346.9	
					All Force	s are in Kips				

# FIGURE 1.2-7

Note that in this example, **Tendon selection and extents [5]** tab in the *PT Recycling* window is inactive, as single-tendon path has been chosen.

If the solutions are not acceptable, you can change post-tensioning layout and recycle until an acceptable solution is reached. Once you are satisfied with the solution, select **Exit** at the top left of the *PT Recycling* screen to continue with the calculations.

The program continues with the calculations based on the most recent tendon forces and profile selection. Once successfully finished, you return to the main program window. The Results can be viewed and/or printed as Reports, Graphs and PT Summary as shown in **Figure 1.2-8**.



**FIGURE 1.2-8** 

Close the above window by clicking **X** at the top right corner.

# **1.3 CREATE REPORTS**

ADAPT-PT/RC 2019 includes the **Report Generator**. To setup the report, select the

**Report Setup** item on the *Options* menu or click the **Report Setup** button **E** on the main toolbar. The *Report Generator* screen shown in **Figure 1.3-1** will open.

The program allows you to generate reports in an MS-Word<sup>®</sup> editable format. You have the following options:

- Report cover: Select this option to generate a report cover with your logo and company information. To update your company information, click on Update Company Info on the *Report Generator* and you will see the screen Company Information shown in Figure 1.3-2.
- Table of Contents
- Concise Report: This report includes Project Design Parameters and Load Combinations as well as a Design Strip Report containing Geometry, Applied Loads, Design Moments, Tendon Profile, Stress check / Code check, Rebar Report, Punching Shear, Deflection and Quantities.
- Tabular Reports Compact
- Tabular Reports Detailed
- Graphical Reports

#### • Legend

	List of all Sections		List of Selected Sections		
Rep Tabl Conc Tab Tab Grap Lege	ort Cover e of Contents cise Report ular Reports - Compact ular Reports - Detailed whical Reports and				
	User Selections				
		Cours Coloction	Cause as Default	Province Departs	11.1.0
	Remove Selection	Save Selection	Save as Default	browse nepoirs	Update Company Info

#### FIGURE 1.3-1

Simply check any item in the *List of all Sections* to include it in the report. The item will then appear in the *List of Selected Sections* on the right hand side of the *Report Generator*.

To generate and view the report, click on **Generate/View Report** on the bottom of the *Report Generator*.

The program allows you to open and view existing reports by clicking on **Open Reports**.

The Report Generator allows you to save report content as either a default template or as a user defined template. This enables you to quickly select content for any project by either using the default content or any other user defined content.

To define content as the default template, select report content from the List of all Sections and click on **Save as Default**.

To define content as a user defined template, select report content from the List of all Sections and click on Save Selection. You are asked to enter a name for your selection. This name appears then in the drop down box in the **User Selections** frame.

Company Information	
L.	
1	
This information will appear at the bottom of each page	1
Logo	
	Browse
This logo will appear at the cover page of the report	

FIGURE 1.3-2

To open the "PT Summary Report" (Figure 1.3-3) either click the BuilderSum button on the tool bar or select the PT summary item on the *View* menu.

1 - PROJECT TITLE: "Sup	ADAPT - STR ADAPT-PT Version 2017 port Line 2"	UCTURA L CONCI Date: 09- 25 - 2017	RETE SOFTWARE Ime: 18:13 File: PTRO	SYSTEM 2017_PT_GBG_M1	4
1.1 Design Strip: Two-Way Slap * 1.2 Load Case: Envelope 2 - MEMBER ELEVATION	0	<u> </u>			~
[11]		9 SPUN 2 52.91			SPON 4 R-Earc
		$\searrow$	$\wedge$	$\wedge$	
3 - TOP REBAR				0	2
3.1 ADAPT selected 3.2 ADAPT selected 3.3 Num. of layers		**************************************	6 7 1 1	C Separe	(2) CESTON (2) CESTON (1) CESTON
4 - TENDON PROFILE	~ /	$\wedge$	$\wedge$	$\wedge$	
4.1 Datum Line					
4.2 CGS Distance A (in) 4.3 Force A (kips)	5.00 2.00 997.194	6.50 1.50 667.164	6.20 607	150 8.50 1194	2.00 535.00 007.104 007.104
4.6 CGS Distance B (in) 4.7 Force B (kips)					50
4.10 CGS Distance C (h) 4.11 Force C (kips)					
4.14 For ce/Width (kipsiti) 4.15 Precompression PIA (cs)	26.69 224.90	22.49 197.42	20 161	109 7.19	25.62 25.66 215.16 213.66
5 - BOTTOM REBAR		100 B 100			
5.1 ADAPT selected 5.2 ADAPT selected	Bersmann"				
5.3 Num. of layers		• •	1 1		1 1 111
6 - REQUIRED & PROVID	ED BARS				
6.1 TopBers	2.0				
provided					
6.2 Editor Ears	13	4	<u> </u>		
7 - PUNCHING SHEAR					
RE-Reinforce NG-Exceeds code NA-not applicable or not performed					
7.1 Stress Ratio Shear Force (kips) Bending Moment (kips*11)	 - 0.00 - 0.00	0.78 - 24278 129.18	0.98 - 302.84 118.18	0.95	0.87 -1126.05 -1360.14
7.2 Status	NA	OK	OK	OK	dĸ
8 - LEGEND	- Stressing End	4	Dead End		
9 - DESIGN PARAMETER 9.1 Code: American ACO18 (2014)/6 9.2 Reber Cover: Top = 1 in Bottom	S sc (2015) r. = 4000 pair, = 6 - 1 in Rebar Table.	10 kai (longitudinai)    f <sub>2</sub> =  i	50 kai(a hear) f <sub>2*</sub> = 270 k		
10 - MATERIAL QUANTIT	ES			20020000000000	
CONCRETE Total volume of concrete = 3267.5 m <sup>2</sup>	MILO S Total w	eight of reber = 2065.2 lb		PRESTRESSING STEEL Total weight of landon = 17	87.116
Area covered = 3921.4 ft <sup>3</sup>	Averag	e rebar usage = 0.527 lb/t	1, 0.632 (bm <sup>3</sup>	Average tendon usage = 0.	450 b/m <sup>2</sup> , 0.547 ib/m <sup>2</sup>
11 - DESIGNER'S NOTES					

FIGURE 1.3-3

To view the graphs, either click the **Show Graphs** button  $rac{1}{2}$  from the toolbar or select **Graphs** in the menu.

**Forces Diagram** button **I** displays forces for selected load combinations or envelop as shown in **Figure 1.3-4**.



**FIGURE 1.3-4** 

**Moment Diagram** button with displays forces for selected load combinations or envelop as shown in **Figure 1.3-5**.



ADAPT-PT/RC® 2019-Tutorial- 36







FIGURE 1.3-6

**Rebar Diagram** button kiplays forces for selected load combinations or envelop as shown in **Figure 1.3-7**.



FIGURE 1.3-7

# Additional references to be consulted with this manual:

- 1. ADAPT-PT/RC 2019 User Manual
- 2. ADAPT-PT/RC 2019 Getting Started Tutorial RC mode
- 3. ADAPT-PT/RC 2019 Verification Manual