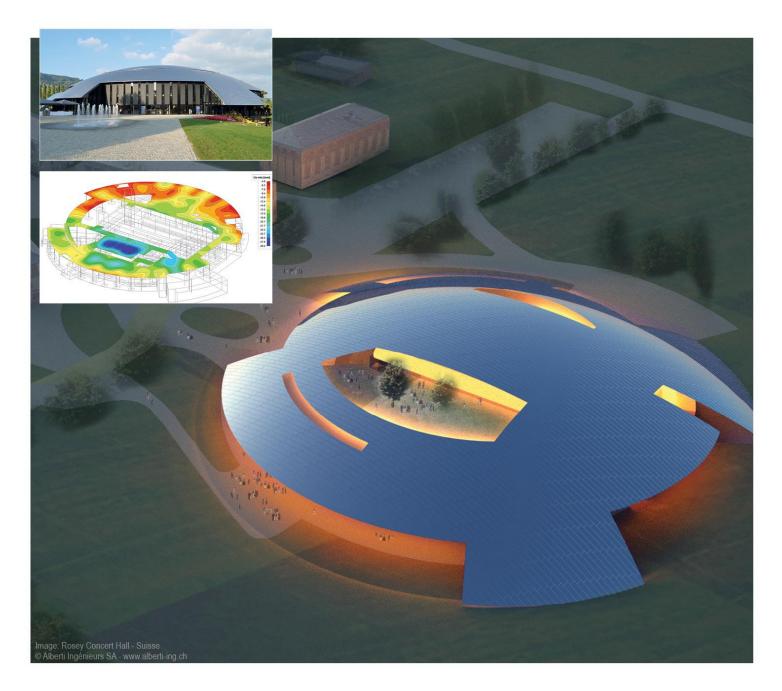
SCIAENGINEER



Tutorial

Post-Tension Concrete Modeling & Analysis (IBC)

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. Scia is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 Nemetschek Scia nv. All rights reserved.

Release:Scia Engineer 15Manual:US Post Tension TutorialRevision:07/2015

Nemetschek Scia Offices

Latest version is available on: http://nemetschek-scia.com/en/contact/offices

Belgium Headquarters

Nemetschek Scia nv Industrieweg 1007 B-3540 Herk-de-Stad Tel.: +32 13 55 17 75 Fax: +32 13 55 41 75 E-mail: info@scia-online.com

Support Phone CAE (Scia Engineer) Tel: +32 13 55 09 90

CAD (Allplan) Tel: +32 13 55 09 80

Support e-mail: support@scia-online.com

Austria

Scia Datenservice Ges.m.b.H Dresdnerstrasse 68/2/6/9 A-1200 Wien Tel.: +43 1 7433232-11 Fax: +43 1 7433232-20 info@scia.at

Support

Tel: +43 1 7433232-12 E-mail: support@scia-online.com

Brazil

Nemetschek Scia do Brasil Rua Dr. Luiz Migliano, 1896 – sala 702, CEP SP -05711-001 São Paulo Brasil Tel: +55 11 4314-5880 E-mail: brasil@scia-online.com

Czech Republic

Nemetschek Scia s.r.o. Evropská 2591/33d 160 00 Praha 6 Tel.: +420 226 205 600 Fax: +420 226 201 673 E-mail: info.praha@scia.cz

Nemetschek Scia s.r.o. Slavickova 827/1a 638 00 Brno Tel.: +420 530 501 570 Fax: +420 226 201 673 E-mail: info.brno@scia.cz

France

Nemetschek Scia sarl Centre d'Affaires 29, Grand' Rue FR-59100 Roubaix France Tel.: +33 3.28.33.28.67 Fax: +33 3.28.33.28.69 france@scia-online.com

Germany

Scia Software GmbH Emil-Figge-Strasse 76-80 D-44227 Dortmund Tel.: +49 231/9742586 Fax: +49 231/9742587 info@scia.de

Netherlands

Nemetschek Scia bv Wassenaarweg 40 NL- 6843 NW Arnhem Tel.: +31 26 320 12 30 Fax: +31 26 320 12 39 info@scia.nl

Slovakia

Nemetschek Scia s.r.o. Murgašova 1298/16 SK - 010 01 Žilina Tel.: +421 415 003 070-1 Fax: +421 415 003 072 info@scia.sk

Switzerland

Nemetschek Scia Switzerland Dürenbergstr. 24 CH-3212 Gurmels Tel.: +41 26 341 74 11 Fax: +41 26 341 74 13 info@scia.ch

USA

Nemetschek Scia North America 7150 Riverwood Drive Columbia, MD (USA) Tel.: +1 443-542-0638 Fax: +1 410-290-8050 usa@scia-online.com

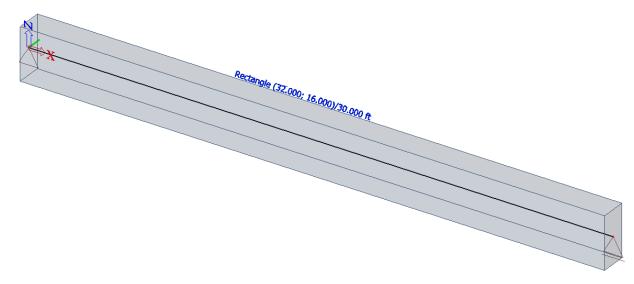
Table of contents

Table	of contents4
Post-T	ensioned Concrete Beam5
1.	Input Geometry and Post-Tensioning5
2.	Calculation9
3.	Results9
Post-T	ension Concrete Slab11
1.	Input Geometry and Post-Tensioning11
2.	Input of Loads15
3.	Calculation16
4.	Results16

Post-Tensioned Concrete Beam

1. Input Geometry and Post-Tensioning

Before modeling can start, the project data must first be input. For this particular project, a **frame XZ** structure type with **3000 psi concrete** will be selected. Additionally, in the functionalities tab the option for prestressing should be checked (along with the Advanced setting which enables anchorage losses).



The beam is type and size is first selected using the cross section 3 service. Once the service is open, select the **rectangle** from the **concrete** group. Next, the size of the beam will be assigned as H = 32" and B = 16" with concrete material C3000. Once the cross section is created, the 3 Beam input tool from the Structure service is used to input the **30**' beam.

Finally, the shortcut buttons on the command line for supports $4 \le 1$ are used to add a **sliding** support at one end and a **hinged** support at the other.

Before a tendon can be input, a load case of the type **Prestress** needs to be created. This is required in order to store the loads that are caused by the tensioning. It is possible to create multiple load cases of the type "prestress", thus allowing a user to model the sequentionnaly tensioning of the cables and the complementary losses. The sequentional post-tensioning can also be modeled with linear construction stages. In that case, the various "prestress" load cases will be added to each construction stage.

For this example, three linear load cases will be created: self-weight, prestress-1 & prestress-2

Each individual load case will be assigned to a specific **linear construction stage.** In order to enable the model for construction stages access the **Project** settings and change the model type to **Construction Stages.** The H Construction stages icon will now appear in the main service tree. Create 3 individual construction stages as follows:

- ST1 with load case: self-weight
- ST2 with load case: prestess-1
- ST3 with load case: prestess-2

Construction stages		×
🚚 🕃 🗶 🔛 🗠	AI	• 7
ST1	Name	ST2
ST2 ST3	Order of stage	2
515	Description	
	Last construction stage	
	□ Load case permanent or I	
	Load case	Empty Load Case - 2 🔹 🔹
	Gamma min [-]	0.00
	Gamma max [-]	1.00
	Load case prestress	
	Load case	Prestress-1 🔹
	Gamma min [-]	1.00
	Gamma max [-]	1.00
	Actions	
	Variable load cases	>>>
New Insert Edit	Delete	Close

When creating ST2 and ST3, add an empty permanent load case so that only the prestressing of the tendon is accounted for in the stage.

Once the cross section has been modeled and the self-weight load case is input, the $\frac{1}{1000}$ tendons can also be added thru use of the Structure service. The input of tendons on 1D members can be done in three ways:

- Through direct input, where the user inserts the cable using 3D modeling
- Through direct input, where the user imports the cable path using a CAD drawing and converts it into a cable.
- Through source geometry where the user inputs coordinates for the cable and the program will interpolate between these point through geometric methods (which the user also indicates).

Once the input of the post-tension tendon is selected, the properties for the tendon will become available. The properties for this strand should be defined as shown in the image below.

	Name TND
	Description
	Number 1
ng 🛞 ng	Type Internal
iis 🐷 🐨 lig	Layer Layer1
·	Geometry
	Geometry input Direct input
	Projection of intermediate points Perpendicularly
	LCC
	Material
	Material Grade 250 LR ASTM A910 - 2 x 0.114 x
	Number of tendon elements in ten 2
	Number of tendons in group (ng) 1
	Area [inch^2] 0.040
	Diameter of duct finch] 0.228
	Load Case President
	2 Channing
	Type of stressing Type 3 💌
	Prestressing from Both ends, anchored at the end, restresssed 💌
	Coefficient of friction in curved pa 0.3
	Unintentional angular displaceme 0.001
	Anchorage set - begin [inch] 0.250
	Anchorage set - end [inch] 0.250
	Stress during correcting - begin [ksi] 200.00
	Stress during correcting - end [ksi] 200.00
	Duration of keeping stress [s] 300
	Duration of keeping stress [s] 300 Initial stress - begin [ksi] 200.00
	Initial stress - begin [ksi] 200.00
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00 Overhang of tendon not included i 0.000
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00 Overhang of tendon not included i 0.000
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00 Overhang of tendon not included i 0.000 Output 0.000 Distance between sections for out 0.500
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00 Overhang of tendon not included i 0.000 Overhang of tendon not included i 0.000 Distance between sections for out 0.500
	Initial stress - begin [ksi] 200.00 Initial stress - end [ksi] 200.00 Overhang of tendon ot included i 0.000 Overhang of tendon ot included i 0.000 Distance between sections for out 0.500 Arc Curve type Circle + radius

- 1. Material and number of tendons (ns) elements in a tendon
- 2. Load Case for prestressing
- 3. Type of prestressing (Types 1 thru 5, for more information see pg. 13)
- 4. Properties of the prestressing

To effectively draw the tendons on the beam, enable the following snaps from the cursor snap settings menu \aleph located on the command line. Using the snap points the tendons should be entered at the 1/6 point from the edge and bottom.

It is also possible to utilize the **Edit Table Geometry** to modify the tendons position. Once the first tendon has been input, the 2^{nd} can be created by copying it to the same location on the other side of the beam.

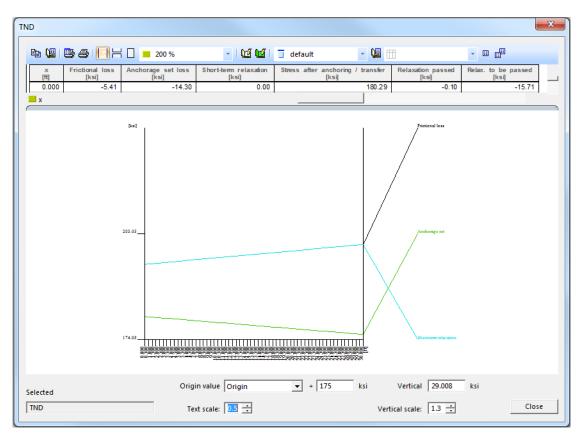
Lastly, make sure that the tendons utilize separate prestressing load cases.

h) 🔽 Points on line	e-curve - N-ths	6
i) 🔲 Points on line	e-curve - % of leng	gth 10.00 %
j) 🔽 Surface edge	s	
k) 🔲 General solid	s	2000
	1	
	ОК	Cancel

	Coord X [ft]	Coord Y [ft]	Coord Z [ft
1	0.000	-0.444	-0.889
2	30.000	-0.444	-0.889
*	0.000	0.000	0.000
	_		

After entering the tendons, the losses (frictional, anchorage and short term relaxation) found in the beam can be reviewed. If tensioning is done at the beginning of a tendon, there is a large anchorage loss which becomes smaller towards the end of the member. Additionally, the loss due to friction increases towards the end of the cable.

Lastly, the stress after transfer (SAT) is calculated at each location defined in the tendon properties. This stress is determined by subtracting the anchorage and friction losses from the initial stress. This is the stress that is sent to the solver during the analysis.



The last step before the staged calculation is assigning various model elements to the different construction stages. For this example, add the **member B1** and the two supports to stage ST1. Additionally, stage ST2 will have the left most tendon (TND2) in it, while stage ST3 will add the right most tendon (TND3). Once the model elements have been added, the calculation is ready to be run.

2. Calculation

Before the calculation is performed, it is important that the tendons are divided into sufficient finite elements to ensure an accurate analysis. These settings are found in the **Mesh Setup**. Additionally, the number of sections on an average member can be set. This setting is found in the programs **Solver Setup**.

Name		
Solver		
Run one nonlinear combination		
Neglect shear force deformation (Ay, Az >> A)		
Type of solver	Direct	-
Number of sections on average member	1	
Maximal acceptable translation [mm]	1000,0	
Maximal acceptable rotation [mrad]	100,0	
Print time in Calculation Protocol	M	

When the solver settings are established, select the calculation 🛱 button and enable the checkboxes for Linear Calculation and Construction Stage Analysis – Linear.

3. Results

In the results menu the **internal forces** and **tendon stresses** can both be viewed for the concrete beam. Since individual prestressing load cases were established, the results can also be viewed for the separate load cases as well. When comparing the results for the axial stress on the beam due to the two prestresses.

Axial stress on Beam (Prestress – 1):

Detailed stress in section	× .	Properties	
Picture Information	Type of stress © Nomal-stress © Shear stress	Stress (1)	▼ Vi V⁄ ▼ Stress
-70.3 	C von Mieses C von Mieses C Fatique psi Member: B1 Section : 0 000 ▼ t Setup of print V Text V Text V Nomal stress V Shear stress Shear st	Selection Type of loads Load cases Filter Cross-section parts Fibres Drawing Values Extreme Drawing setup 1D Section	All Load cases Prestress - 1 No All All Standard Normal + Member All All All
	OK Cancel Apply		

ιx

* * * *

•

× Detailed stress in section Properties <mark>4</mark> х 🖃 🖬 🏹 🖉 Picture Information Stress (1) Type of stress × 69.8 Normal- stress C Shear stress Stress Name C von Mise C Fatique von Mises All Selection Type of loads Load cases psi Prestress - 2 Load cases B1 Member Section Filter No Ŧ 0.000 ▼ ft Setup of print All * Cross-section parts 69.8 🔽 Text All Ŧ Fibres Picture -Standard Drawing Vormal stress Shear stress -Values Normal + von Mises Ŧ Extreme Member ✓ Fatique Drawing setup 1D ... To Document All Ŧ Section Scale • 209.4 1 + Text ок Cancel Apply

Axial stress on Beam (Prestress – 2):

The above axial stress results are logical considering the application of the construction stages. Since tendon TND2 is tensioned in the first stage (ST1), there will be some loss of tendon stress in TND3 occurring when it is tensioned in the 2nd stage (ST2). Additionally, these results can be seen in the Endon Stresses found in the Structure tree menu.

<u>Note:</u> There is an option for **prestress** in the property menu of **internal forces**. If this is ticked on, then the section will be calculated as if it was possibly already prestressed. For **prestress-1** this will not make a difference, but for **prestress-2** this can give different results, because at the time of tensioning for tendon TND3, there is already stress in the section due to tendon TND2. This will make the equivalent section (and the moment of inertia) bigger, which causes the results for **prestress-2** not to be the same as those of **prestress-1** for a regular section.

Tendon stresses

Linear calculation, Extreme : Member Selection: All Tendons: All by selection Class : ST1 (ULS)

lifeload (min)

Case	Tendon	x [ft]	Stress after anchoring / transfer [ksi]			Lmin [ksi]		MinStress [ksi]	MaxStress [ksi]
ST1 (ULS)	TND2	30.000	174.93	0.00	0.00	0.00	0.00	174.93	174.93
ST1 (ULS)	TND2	0.000	180.26	0.00	0.00	0.00	0.00	180.26	180.26

Explanations of symbols		Explanations of symbols			
Stress after anchoring / transfer	Stress of prestressingreinforcement after anchoring (post-tensioned	Lmax	Loss (change of) prestressing cause life load (max) Minimal stress in phase		
	tendon)/aftertransfer of prestress (pre-tensionedtendon)	MinStress			
LED	Loss due to sequential prestressing+ loss caused by the elastic deformation of concrete	MaxStress	Maximal stress in phase		
LCS	Loss due to creep and shrinkage of concrete + loss due to long-term steel relaxation				
Lmin	Loss (change of) prestressing caused by				

Post-Tension Concrete Slab

In this example post-tensioned cables are introduced on a concrete slab. This slab will be modeled as a 2D-element, allowing the analysis to be run in a general XYZ environment. In order to take into account long-term losses, the user would estimate these losses and simply include them by giving the prestressed cables a lower initial stress. It is also possible to tension the cables sequentially and calculate these losses by using linear construction stages.

This example will be limited to a linear analysis of a post-tensioned slab without consideration for construction staging.

1. Input Geometry and Post-Tensioning

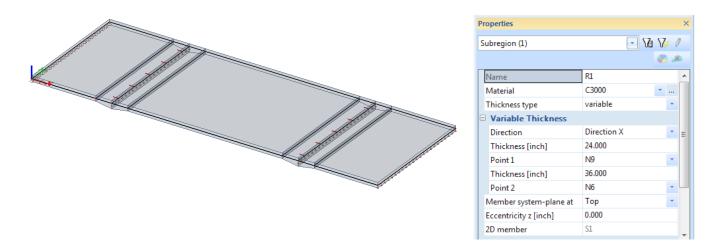
Before modeling can start, the project data must first be input. For this particular project, a **general XYZ** structure type with **3000 psi concrete** will be selected. Additionally, in the functionalities tab the option for prestressing should be checked (along with the Advanced setting which enables anchorage losses).

data						2	S	Pro	ject data					x
data Fu	nctionality Loa	ds Code Setup Protection						в	asic data Fu	inctionality Loads Code !	Setup Protection			
ia III	Data			Material					Scia	Dynamics		Prestressing		
leer	Name:	Post-Tension Concrete Slab		Concrete					Engineer	Initial stress		Advanced		
				Material		•				Subsoil		Concrete		
	Part:	-		Steel						Nonlinearity		Fire resistance		
				Timber						Stability		Hollow core slab		
	Description:	•		Other						Climatic loads				
				Aluminium						Prestressing	V			
	Author:	BLF												
	Date:	22.05.2015												
				Code										
	Structure:			National Code:										
	General XYZ		-	IBC		.								
	Project Level:	Model:												
	Advanced	One												
	The second second second			and the setting										
	Thick-Walled Co	pricrete cross-sections: the advance	ed 20 FEM r											
					ок	Cancel								
		data Functionality Loa Data Data Name: Part: Description: Author: Date: Structure: General XYZ Project Level: Advanced	data Functionality Loads Code Setup Protection Data Name: Post-Tension Concrete Slab Part: Description: Author: BLF Date: 22.05.2015 Structure: General XYZ Project Level: Model: Advanced One	data Functionality Loads Code Setup Protection Data Name: Post-Tension Concrete Slab Pat: Description: Author: BLF Date: 22.05.2015 Structure: General XVZ Project Level: Model: Advanced One	data Functionality Loads Code Setup Protection ia Name: Post-Tension Concrete Slab Material Part: - Steel Timber Description: - Other Aluminium Author: BLF Other Aluminium Date: 22.05.2015 Code National Code: Structure: General XYZ IEC IEC Project Lovel: Model: Advanced One w Thick-walled' Concrete cross-sections: the advanced 2D FEM method is off! Code IEC	data Functionality Loads Code Setup Protection Data Material Name: Post-Tension Concrete Slab Part: - Description: - Author: BLF Date: 22.05.2015 Structure: Code General XXZ * Project Level: Model: Advanced One	State Functionality Loads Code Setup Protection Image: Post-Tension Concrete Slab Material Part: - Image: Post-Tension Concrete Slab Description: - Author: BLF Date: 22.05.2015 Image: Post-Tension Concrete Slab Structure: Code General XVIZ Image: Project Level: Model: Advanced Image: One Image: Post-Tension Concrete cross-sections: the advanced 2D FEM method is off!	data Functionality Loads Code Setup Protection Name: Post-Tension Concrete Slab Part: . Description: . Author: BLF Date: 22.05.2015 Structure: General XYI2 Project Level: Model: Advanced Code Thick-walled' Concrete cross-sections: the advanced 2D FEM method is off!	Stata Functionality Loads Code Setup Protection Image: Data Data Name: Post-Tension Concrete Slab Material Part: - Other Description: - Athor: BLF Date: 22.05.2015 Other Structure: General XYZ Project Level: Model: Advanced Advanced One Thick-walled Concrete cross-sections: the advanced 2D FEM method is off!	State Functionality Loads Code Setup Protection Basic Outs Outs State State<	State Functionality Loads Code Setup Protection Image: Post-Tension Concrete Slab Image: Post-Tension Concrete Slab Image: Post-Tension Concrete Slab Image: Post-Tension Concrete Slab Part: - Image: Post-Tension Concrete Slab Image: Post-Tensio	Jata Data Material Concrete Image: Post-Tension Concrete Slab Image: Post-Tens	Initial State Initial St	The structure: Structure: Structure: Structure: General XVZ Thick-walled' Concrete Orse-sectors: the advanced 2D FEM method is off!

For this example, the plate is modeled as **120' x 45'** with a thickness of **24"**. The slab is supported in four separate locations:

- Each end (one pinned member edge support & one sliding member edge support)
- 30' from each end with sliding member edge supports
 - In order to support the slab internally, ¹ Internal edge are added from the 2D member components service. Once the edges are added, the edge supports can be added as well.

The next step in modeling the slab is to add **subregions** located near the internal edges in order to thicken the slab. Using the 2D member component, **Geometry Subregion** it is possible to add 6' wide portions to the slab that vary in thickness from **24**" to **36**". Additionally, the alignment is set to "top" so that the top surfaces of each subregion are aligned. When the subregions have been added, the structure will look as follows:



After creating the concrete slab, the post-tension cables can then be input. Tendons can be input directly or using geometry brought in from a CAD file. For this example, a single tendon will be created using the direct input method.

	Name	1/2" 7 Wire Tendon	
	Description		
d 00000	Number	24	
	Туре	Internal	
	Layer	Layer1 🔹	
	Geometry		
	Geometry input	Direct input	
	Projection of intermediate points	Perpendicularly 🔹	
	LCS	standard 👻	
	LCS Rotation [deg]	0.00	
	Material		וו
	Material	250 Grade LR ASTM 416-1/2" 🔹	
	Number of tendon elements in ten	7	[]
	Number of tendons in group (ng)	1	
	Area [inch^2]	1.008	
	Diameter of duct (inch)	0.500	
	Load Case	Prestress 💌	
	- Stressing		1
	Type of stressing	Туре 3 💌	
	Prestressing from	Both ends, anchored at the end, restresssed 🝸	
	Coefficient of friction in curved pa	0.3	
	Unintentional angular displaceme	0.001	
	Anchorage set - begin [inch]	0.250	
	Anchorage set - end [inch]	0.250	
	Stress during correcting - begin [ksi]	200.00	
	Stress during correcting - end [ksi]	200.00	
	Duration of keeping stress [s]	300	
	Initial stress - begin [ksi]	200.00	
	Initial stress - end [ksi]	200.00	
	Overhang of tendon not included i	0.000	
	Overhang of tendon not included i	0.000	
	Distance between sections for out	1.000	
	- Arc		
	Curve type	Circle + radius	
	Curve parameter [ft]	3.000	
			1
	Actions		

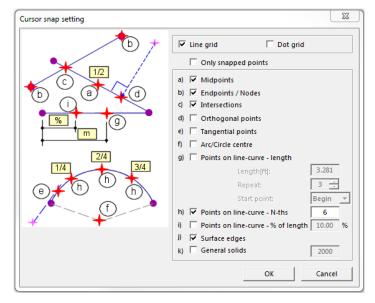
1. When using direct input, the material used can be selected from a pre-defined library. The library includes materials for various sizes of 2, 3 and 7 wire strands. The material properties of these cables are defined by ASTM 416 and ASTM A910. To access additional materials from the library, select the ellipsis — next to the defined material. The material library for the project will open and the specific properties (code independent and IBC) will be displayed. To choose a different material, select the system database 😰 where the additional strand types

are located. To add a material to the project database, select the desired material from the system database and click **Copy to Project.**

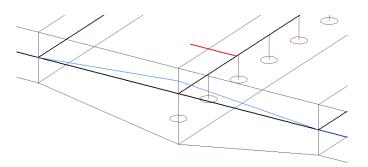
- 2. Identifies the load case for which the "prestress" of the tendon will be added. The load type for this load case should be set as type **Prestress.**
- 3. In Scia Engineer there are 5 different types of stressing. The types can be chosen based on the desired method of calculation for immediate losses. For all 5 types, anchorage losses are included (in type 4 anchorage losses are the only immediate loss). In types 1, 2, 3 & 5 other losses are included (short term relaxation, etc.). For long term losses to be included, a time dependent analysis must be done. For this particular example, we will use **Type 3**.
- 4. Other parameters are used to further define the type of post-tensioning. These parameters are indicated in the dialogue above as separate properties. For example, it is possible to set the initial stress in the tendon, for this example **200 ksi** will be used. Additionally, since the loss due to friction is being accounted for, it is important to denote whether the cable is being stresses from the beginning or the end.

After the material and properties of the cable are defined, it is possible to insert the cable into the slab. For this example, one cable will be drawn and the Multicopy command will be used to insert cables on the remainder of the slab. In order to input the cable, switch the model view to the Y direction 3. Then navigate to the **Structure** service to input 7. Tendons - Post-tensioned internal tendon.

To effectively draw the tendon on the slab, enable the following snaps from the cursor snap settings menu 😤 located on the command line.



Then using the sketching tool draw in a tendon on the slab. The tendon resides at the centerline of the slab at all points along the slab except for the locations of the internal supports where the tendon is **6**" above the slab centerline.

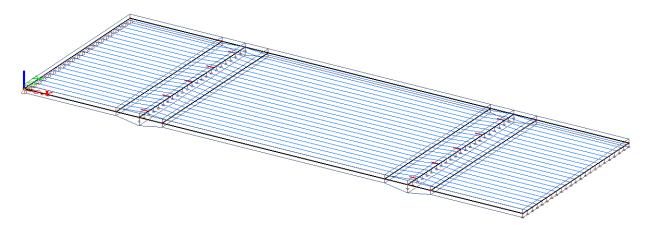


The coordinates of the tendon may also be adjusted using the **Table Edit Geometry** function in the properties of a selected cable.

	Coord X [ft]	Coord Y [ft]	Coord Z [ft]	Curve type		urve parameter [fl
	0.000	0.000	0.000			
2	24.000	0.000	0.000	Circle + radius	+	3.000
;	30.000	0.000	0.500	Circle + radius	*	3.000
Ļ	36.000	0.000	0.000	Circle + radius		3.000
;	94.000	0.000	0.000	Circle + radius	*	3.000
;	100.000	0.000	0.500	Circle + radius	+	3.000
	106.000	0.000	0.000	Circle + radius	-	3.000
}	130.000	0.000	0.000			
ł	0.000	0.000	0.000		-	0.000

Number	of copies	22 ÷	Connect selected nodes with new beams	Γ
🔽 Insert	the very last co	ру	Copy additional data	$\overline{}$
Distance	vector		How to define the distance ?	
Define di	stance by curso	r 🗆	between two copies	
x y	0.000	ft ft	C from original to the last How to define the rotation ?	сору
z Rotation	0.000	ft	 between two copies from original to the last Rotation around 	сору
rx ry	0.00	deg deg	C current UCS	

Once the first cable has been placed, the Multicopy tool can be used to copy the element **22** times at a spacing of **2'**. Once the cable has been copied, the initial element can be deleted since it would not exist in reality. The result is a concrete slab with **22** total cables.



After entering the post-tensioned cables, they still need to be assigned to the plate. This can be done for all the cables simultaneously through the use of the Action **allocate automatically**. Once the cables are allocated the tendon losses (frictional, anchorage and short term relaxation) can be viewed.

[ft]	Frictional loss [ksi]	Anchorage set loss [ksi]	Short-term relaxation [ksi]	Stress after anchoring / [ksi]	transfer	Relaxation passed [ksi]	Relax. to be passed	
0.000	0.00	-26.76	0.00		173.24	-0.11	[ksi] -12.52	
[ks	a]				I.	Fric	tional loss	
200.0	000				1	Anc	horage set	
195.0		7				Sho	rt-term relaxatio	n
190.0				J				
185.0								
180.0		1		h				
175.0		77						
170.0	000				\neg			
165.0								
165.0								

2. Input of loads

In addition to the Prestress loading, two additional load cases will be added:

LC2: Self Weight

LC3: Service Load \rightarrow a surface load of -40 lb/ft² applied to the entire slab

After the load cases have been created and the load applied, ULS & SLS combinations can be set up.

Combinations		X	
A 💱 🖋 📸 💽 🖆	🔉 🖂 🚭 Input combinations	Υ	
ULS	Name	ULS	
SLS	Description		
	Туре	IBC (LRFD) - ultimate	
	Active coefficients		
	Contents of combination		
	Prestress [-]	1.00	
	Self [-]	1.00	
	Service [-]	1.00	
Actions			
	>>>		
	Explode to linear		
New Insert Edit	Delete	Close	

3. Calculation

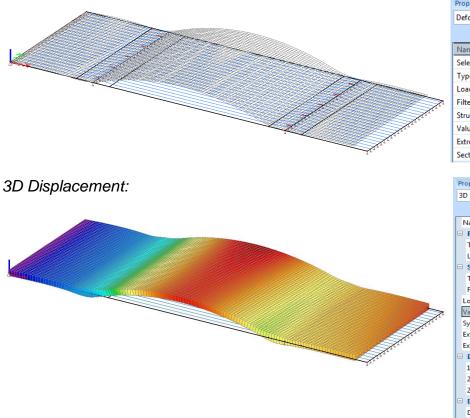
Before the calculation is performed, the size of the 2D mesh is set. For this example, the mesh size is set at 1'. After changing the mesh setting, the linear calculation is executed.

Mesh setup		X
Name		<u> </u>
General mesh settings		
Minimal distance between two poi	.001	
Average number of tiles of 1d ele		
Average size of 2d element/curved	.000	=

4. Results

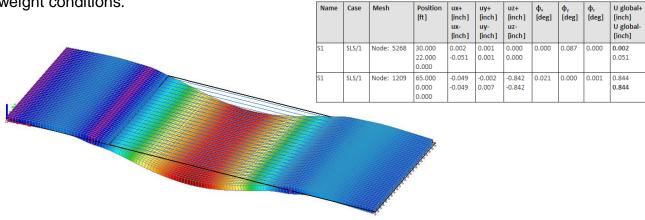
Once the model has been built and the analysis run, it is possible to evaluate results for whether or not the optimal post-tensioning has been applied. Scia Engineer provides a complex analysis of the slab and tendons while taking into account the short term losses from various sources. For the IBC code, the program does not design the necessary amount of post-tensioning required. Rather, the user may evaluate the results (deformations and internal forces) in order to determine if the applied tendon layout is sufficient.

Deformed Structure:



Properties	ų ×		
Deformed structure (1)	🖃 Va V/ 🖉		
	😤 🍂		
Name	Deformed structure		
Selection	All		
Type of loads	Load cases 🔹		
Load cases	Prestress 🔹		
Filter	No 🔻		
Structure	Deformed 🔹		
Values	Deformed structure		
Extreme	Global 🔹		
Section	All		

Properties	Ф	×
3D displacement (1)	- Va V/ /	7
	6 3	•
Name	3D displacement	
Result case		
Type of load	Load cases	
Load case	Prestress	*
Selection		
Type of selection	All	*
Filter	No	*
Location	In nodes avg.	*
Value	U global	*
System	Local	*
Extreme 1D	Global	*
Extreme 2D	Global	*
Draw results on		
Unaw results of		
1D members	V	
	V V	
1D members		
1D members 2D positive face	V	
1D members 2D positive face 2D negative face	V	
1D members 2D positive face 2D negative face Deformed structure		

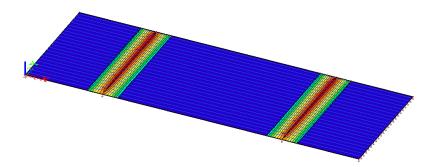


The displacement results (for pre-stressing) are the opposite of those found under self weight conditions.

The layout of the cables in the slab is sufficient because the displacement of the prestess counteracts the displacement of the self-weight and service load.

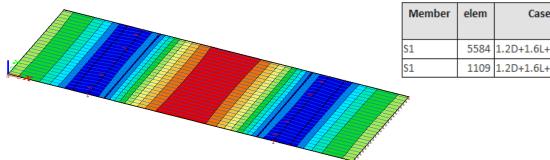
In addition to deformation, the layout of the tendons can also be verified using the internal forces.

Mx Moment due to prestressing load case:



Member	elem	Case	mx [kipft/ft]
S1	3688	Prestress	-7.87
S1	5576	Prestress	61.89

Mx Moment due to ultimate load combination (1.2D+1.6L+1.0PT):



Case mx [kipft/ft] 5584 1.2D+1.6L+1.0PT -113.00 1109 1.2D+1.6L+1.0PT 100.33

The mx moment from the added post-tension cables is opposite of those found from the self-weight and service loading.

For a design to be complete, the user would need to continue to evaluate whether or not the forces on the slab can be taken by the concrete + prestressing. If this is not the case, the prestressing force can be increased and the linear analysis can be performed again.