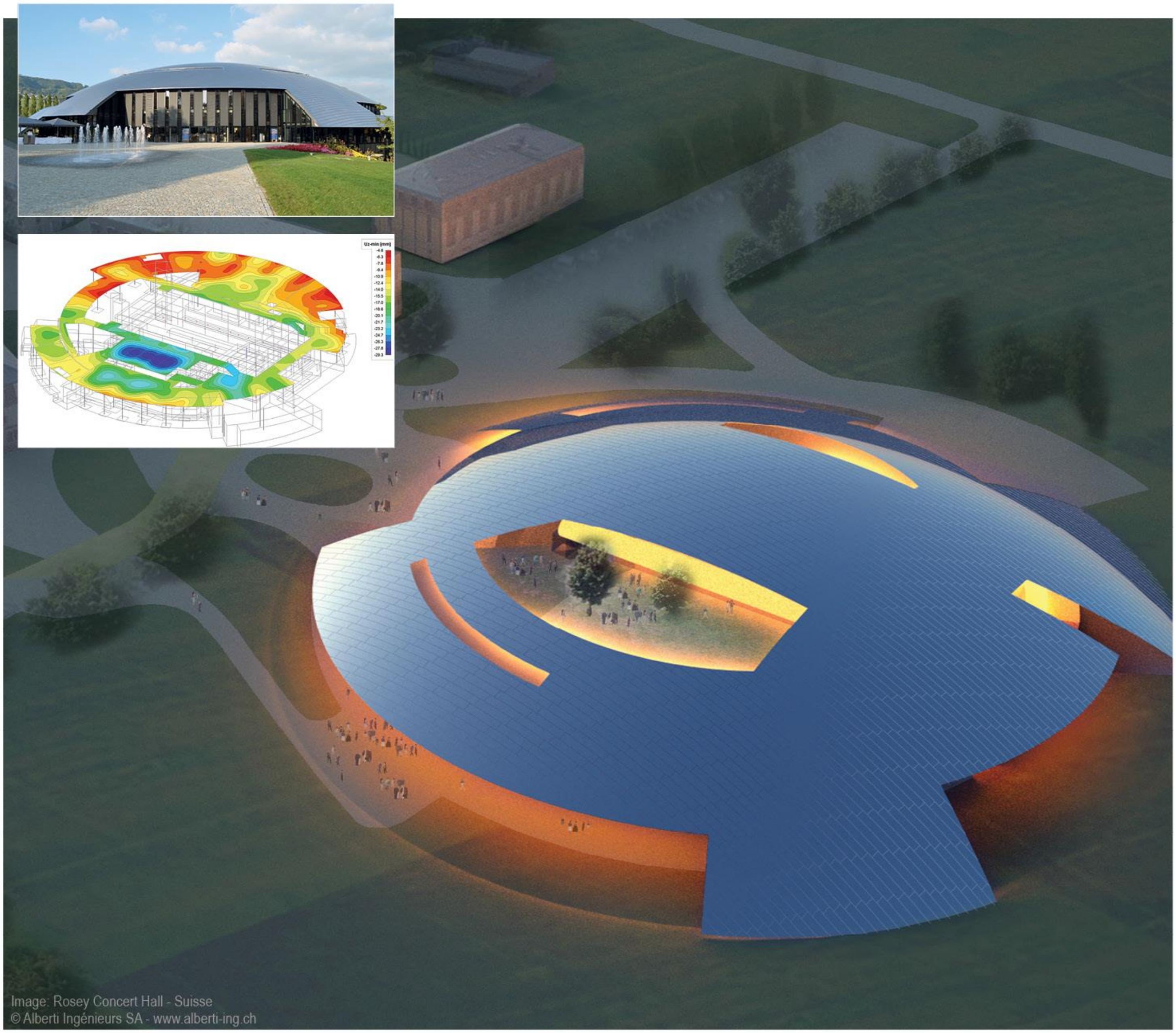
SCIA ENGINEER



Tutorial

Post-Tension Concrete Modeling & Analysis (IBC)

Release: Scia Engineer 15
Manual: US Post Tension Tutorial
Revision: 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

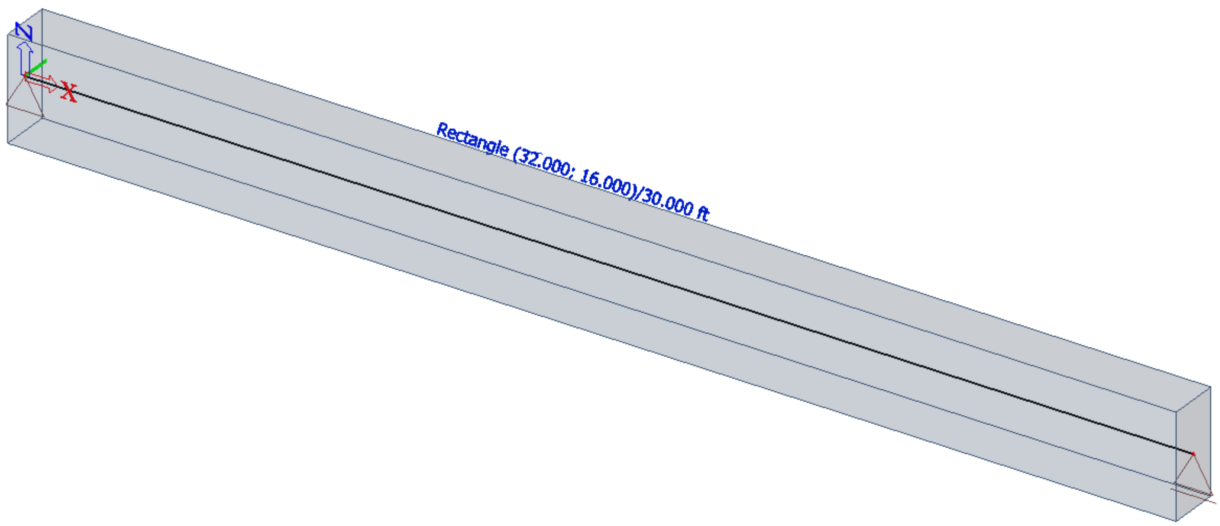
USA
Nemetschek Scia North America
7150 Riverwood Drive
Columbia, MD (USA)
Tel.: +1 443-542-0638
Fax: +1 410-290-8050
usa@scia-online.com

Table of contents

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 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 Beam input tool from the Structure service is used to input the **30'** beam.

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

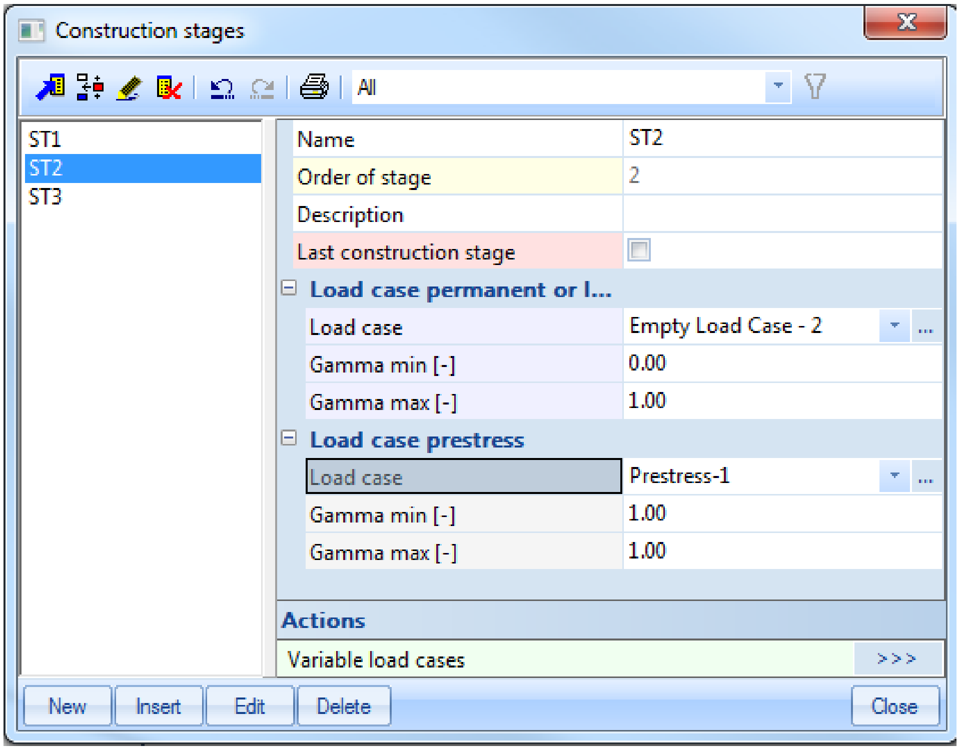
Before a tendon can be input, a load case of the type **Prestress** needs to be created. This is required in order to store the loads that are caused by the tensioning. It is possible to create multiple load cases of the type "prestress", thus allowing a user to model the sequentionnaly tensioning of 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 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

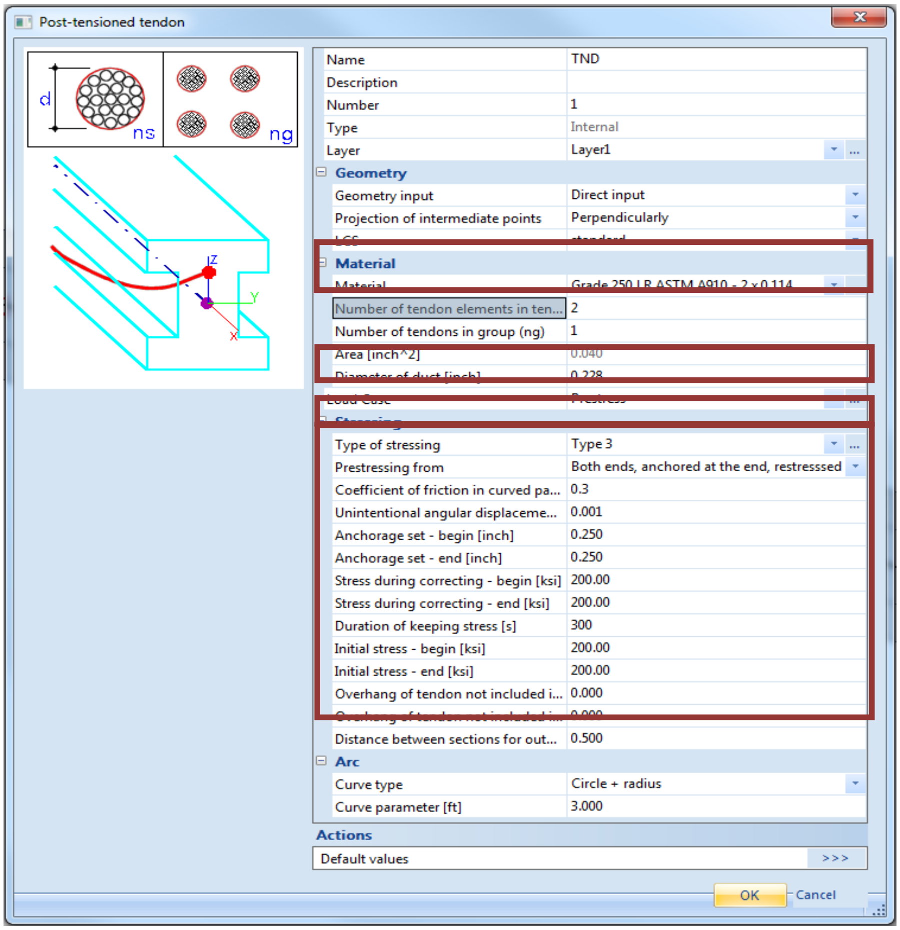
When creating ST2 and ST3, add an empty permanent load case so that only the prestressing of the tendon is accounted for in the stage.



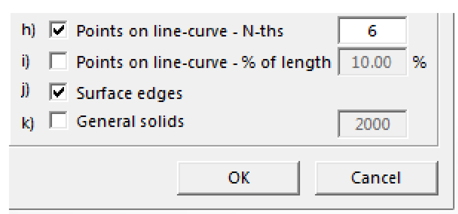
Once the cross section has been modeled and the self-weight load case is input, the Tendons can also be added thru use of the Structure service. The input of tendons on 1D members can be done in three ways:

- 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.

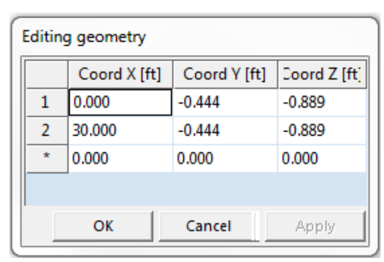


1. Material and number of tendons (ns) elements in a tendon
2. Load Case for prestressing
3. Type of prestressing (Types 1 thru 5, for more information see pg. 13)
4. Properties of the prestressing



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

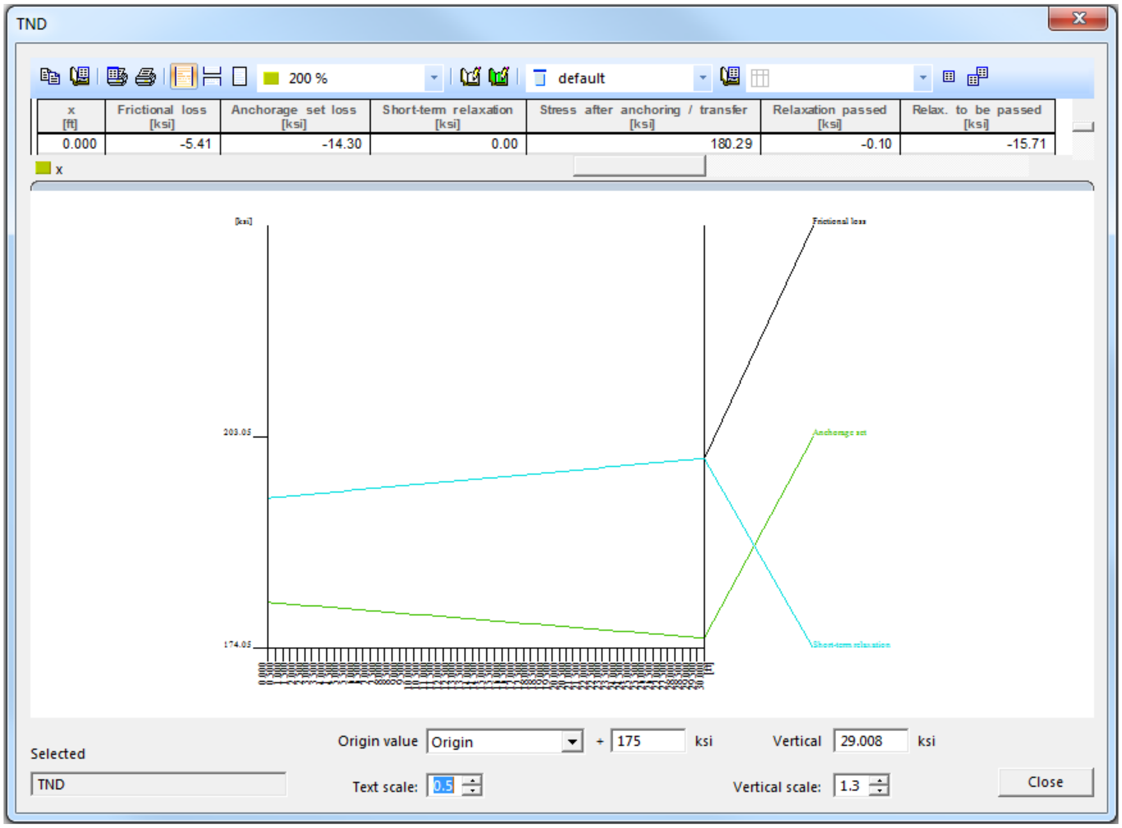
It is also possible to utilize the **Edit Table Geometry** to modify the tendons position. Once the first tendon has been input, the 2nd can be created by copying it to the same location on the other side of the beam.



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

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.

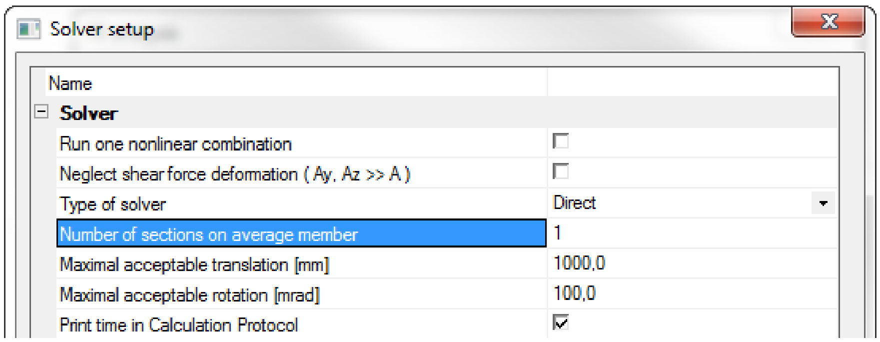
Lastly, the stress after transfer (SAT) is calculated at each location defined in the tendon properties. This stress is determined by subtracting the anchorage and friction losses from the initial stress. This is the stress that is sent to the solver during the analysis.

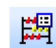


The last step before the staged calculation is assigning various model elements to the different construction stages. For this example, add the **member B1** and the two supports to stage ST1. Additionally, stage ST2 will have the left most tendon (TND2) in it, while stage ST3 will add the right most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

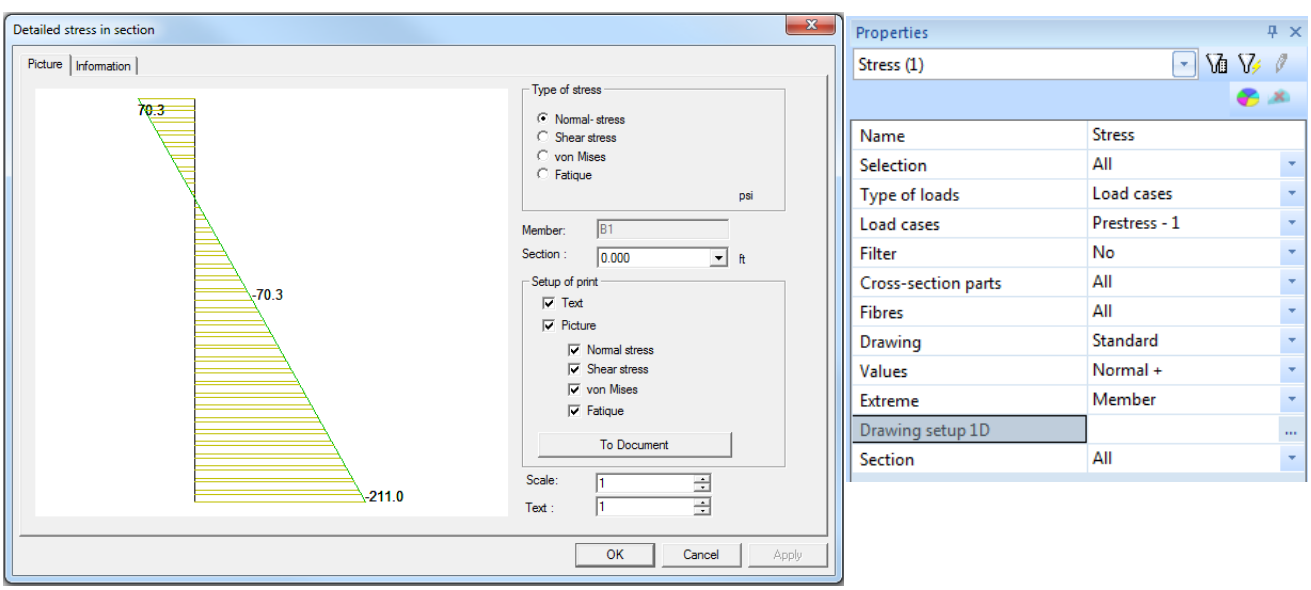


When the solver settings are established, select the calculation  button and enable the checkboxes for **Linear Calculation** and **Construction Stage Analysis – Linear.**

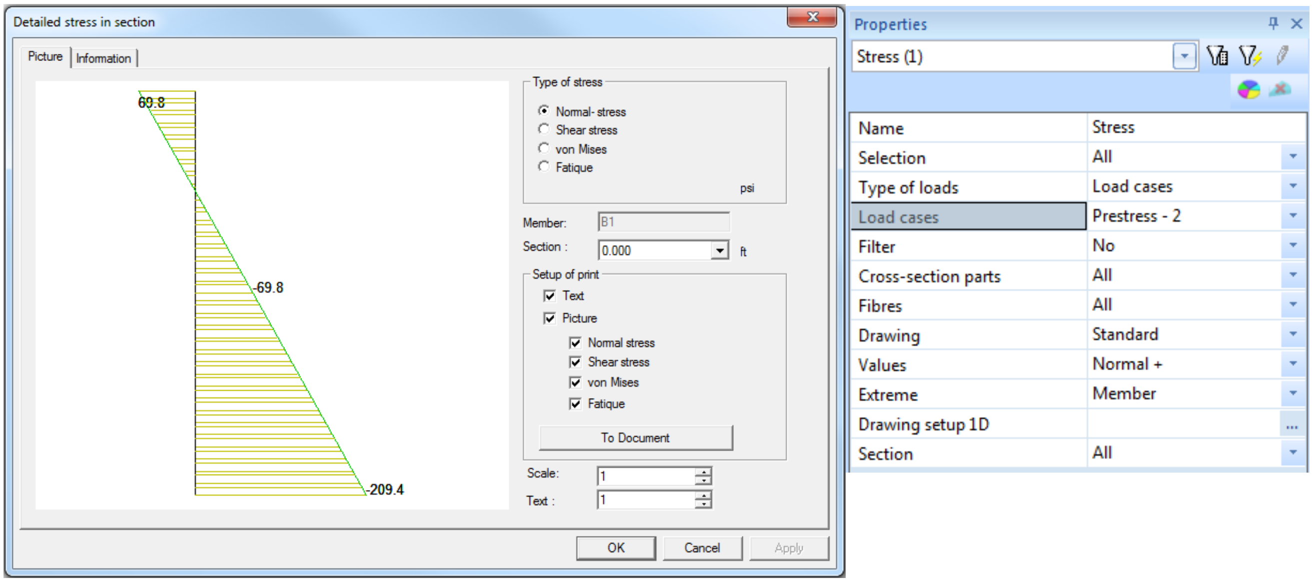
3. Results

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):



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 Tendon Stresses found in the Structure tree menu.

Note: 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)

Case	Tendon	x [ft]	Stress after anchoring / transfer [ksi]	LED [ksi]	LCS [ksi]	Lmin [ksi]	Lmax [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	
Stress after anchoring / transfer	Stress of prestressing reinforcement after anchoring (post-tensioned tendon)/after transfer of prestress (pre-tensioned tendon)
LED	Loss due to sequential prestressing + loss caused by the elastic deformation of concrete
LCS	Loss due to creep and shrinkage of concrete + loss due to long-term steel relaxation
Lmin	Loss (change of) prestressing caused by life load (min)

Explanations of symbols	
Lmax	Loss (change of) prestressing caused by life load (max)
MinStress	Minimal stress in phase
MaxStress	Maximal stress in phase

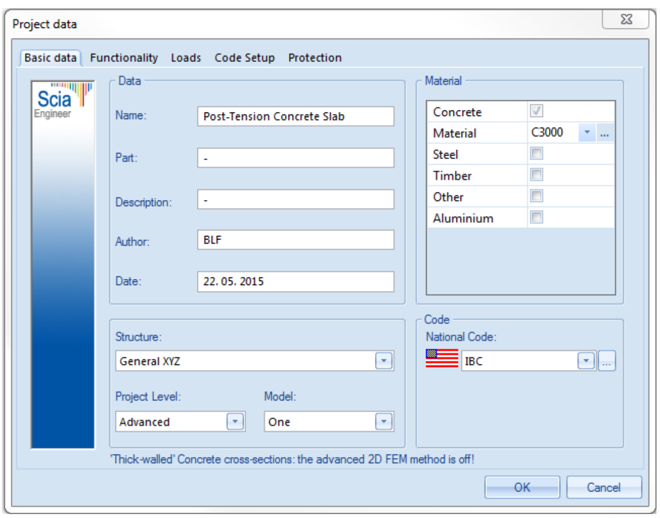
Post-Tension Concrete Slab

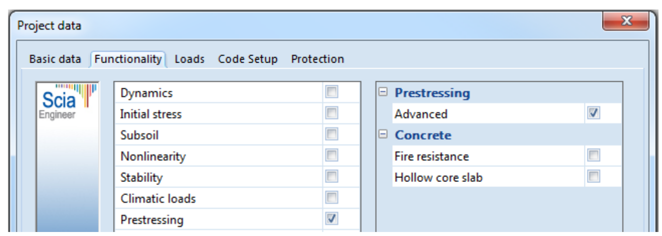
In this example post-tensioned cables are introduced on a concrete slab. This slab will be modeled as a 2D-element, allowing the analysis to be run in a general XYZ environment. In order to take into account long-term losses, the user would estimate these losses and simply include them by giving the 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).

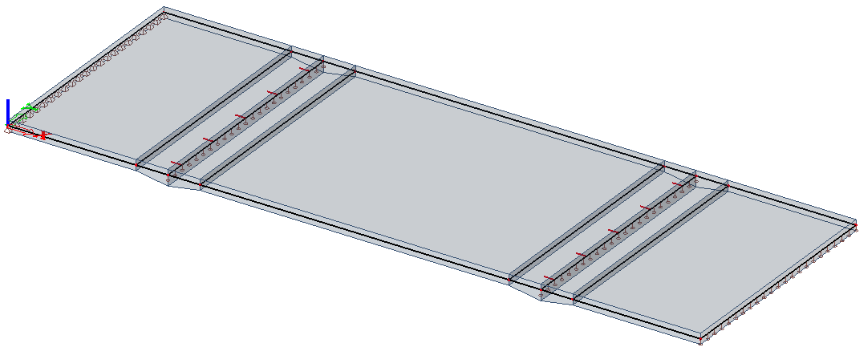


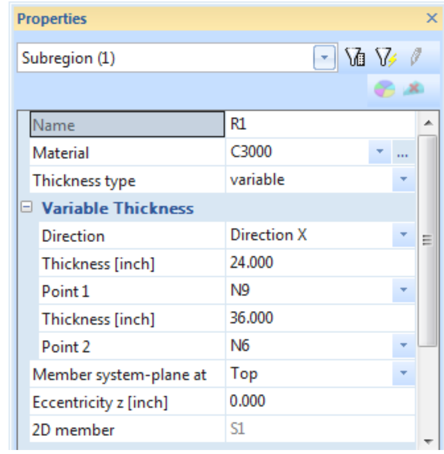


For this example, the plate is modeled as **120' x 45'** with a thickness of **24".** The slab is supported in four separate locations:

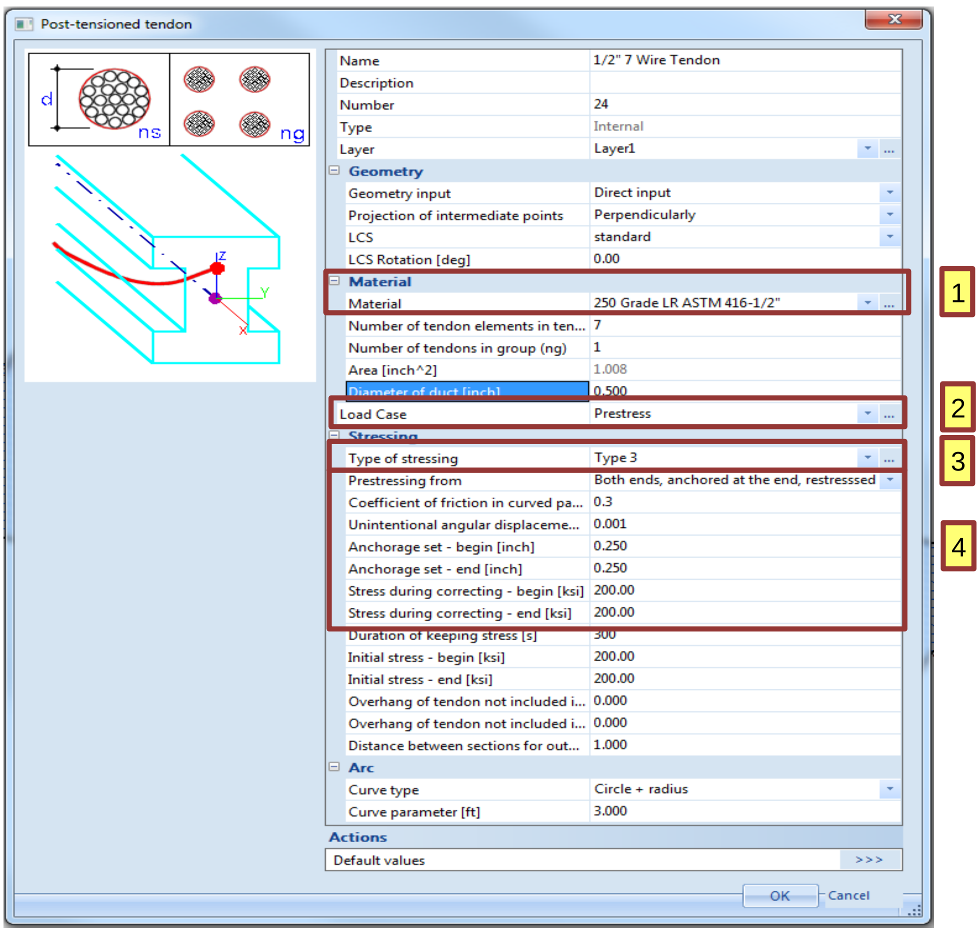
- Each end (one pinned member edge support & one sliding member edge support)
- **30'** from each end with sliding member edge supports
    - In order to support the slab internally, Internal edge are added from the 2D member components service. Once the edges are added, the edge supports can be added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



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.**

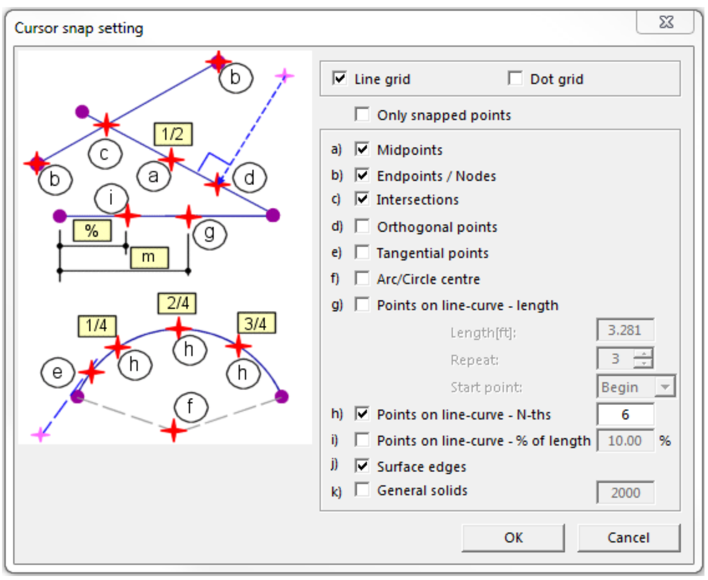
2. Identifies the load case for which the “prestress” of the tendon will be added. The load type for this load case should be set as type **Prestress.**

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 dependant analysis must be done. For this particular example, we will use **Type 3.**

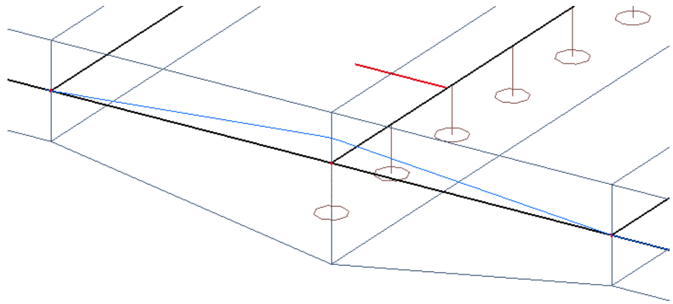
4. Other parameters are used to further define the type of post-tensioning. These parameters are indicated in the dialogue above as separate properties. For example, it is possible to set the initial stress in the tendon, for this example **200 ksi** will be used. Additionally, since the loss due to 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 . Then navigate to the **Structure** service to input Tendons - Post-tensioned internal tendon .

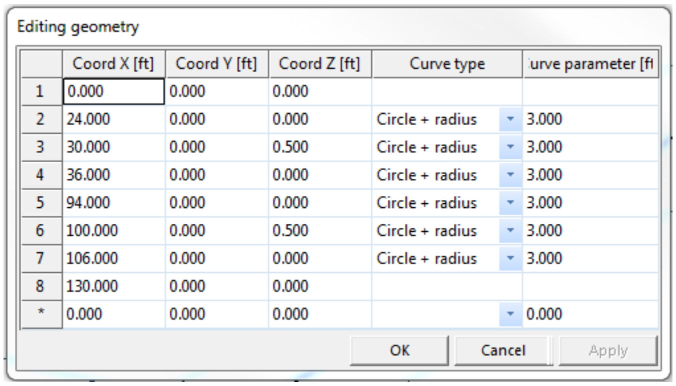
To effectively draw the tendon on the slab, enable the following snaps from the cursor snap settings menu located on the command line.



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.



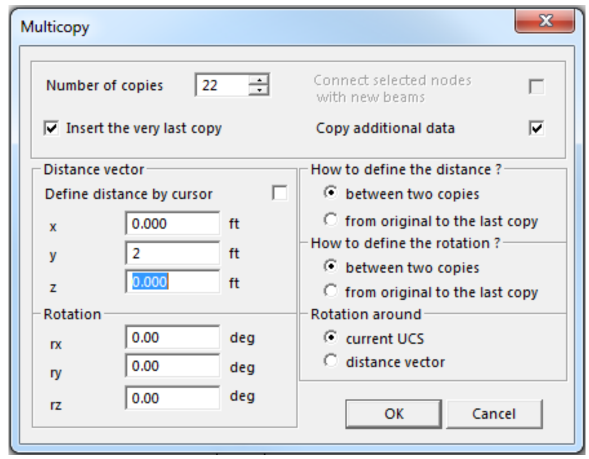
The coordinates of the tendon may also be adjusted using the **Table Edit Geometry** function in the properties of a selected cable.



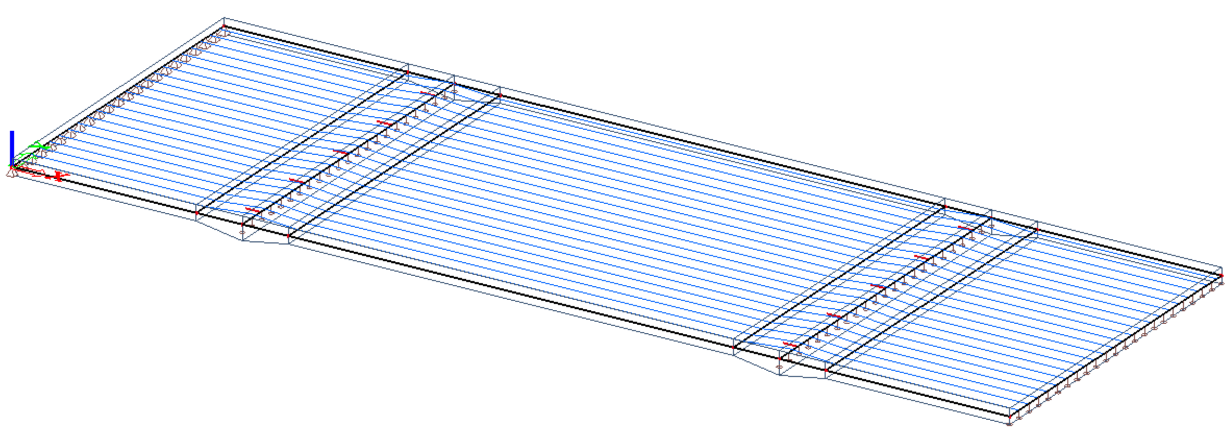
Editing geometry

	Coord X [ft]	Coord Y [ft]	Coord Z [ft]	Curve type	urve parameter [ft
1	0.000	0.000	0.000		
2	24.000	0.000	0.000	Circle + radius	3.000
3	30.000	0.000	0.500	Circle + radius	3.000
4	36.000	0.000	0.000	Circle + radius	3.000
5	94.000	0.000	0.000	Circle + radius	3.000
6	100.000	0.000	0.500	Circle + radius	3.000
7	106.000	0.000	0.000	Circle + radius	3.000
8	130.000	0.000	0.000		
*	0.000	0.000	0.000		0.000

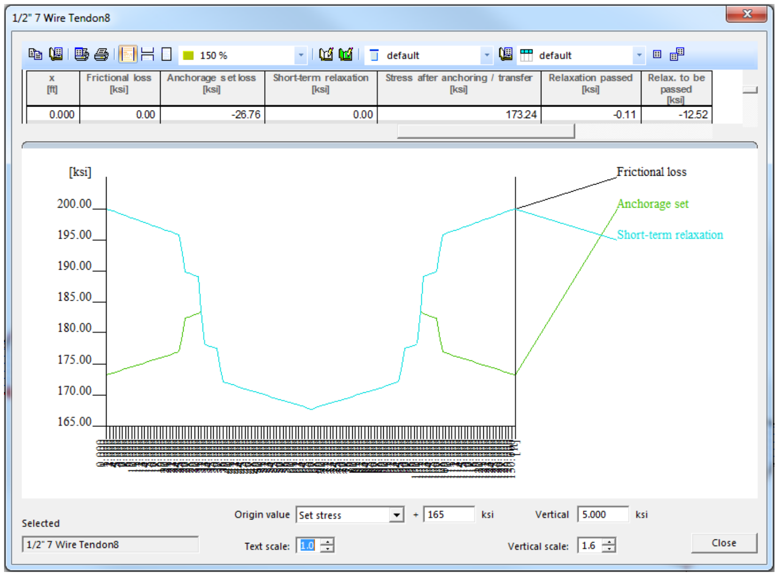
OK Cancel Apply



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.

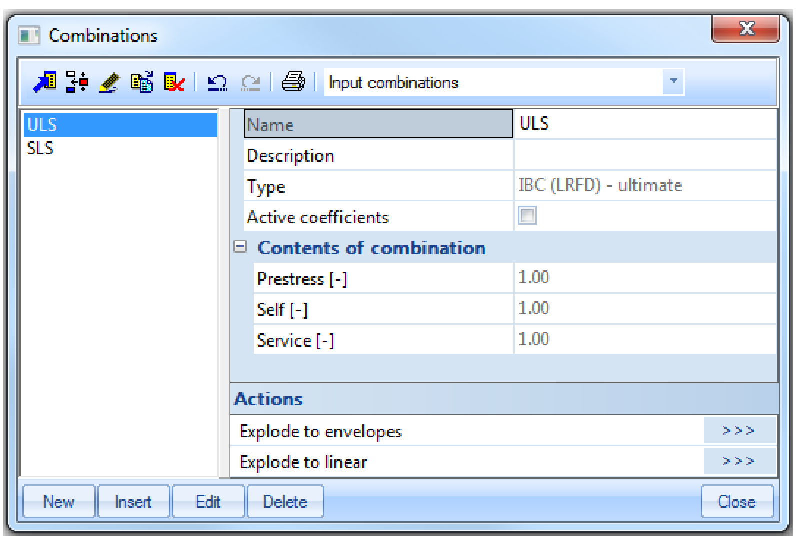


2. Input of loads

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

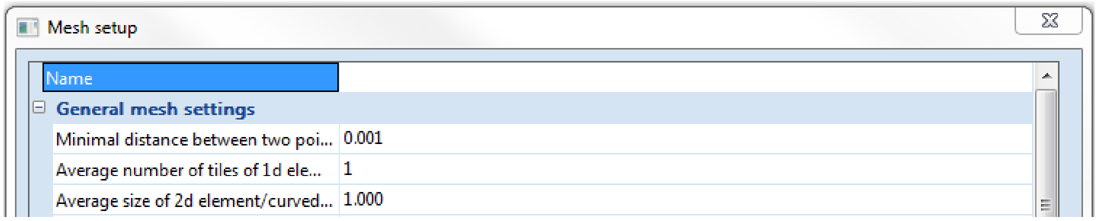
LC2: Self Weight
LC3: Service Load ➔ a surface load of -40 lb/ft^2 applied to the entire slab

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



3. Calculation

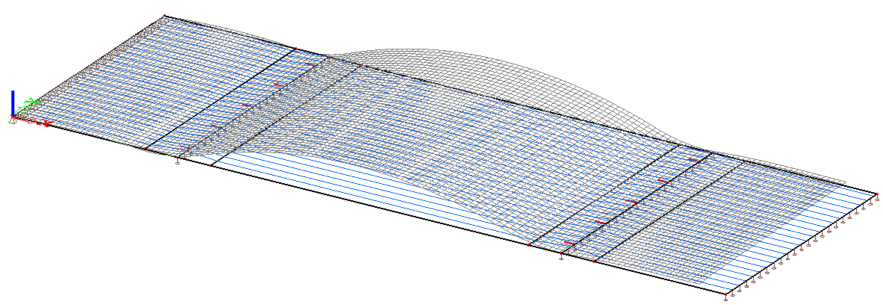
Before the calculation is performed, the size of the 2D mesh is set. For this example, the mesh size is set at 1'. After changing the mesh setting, the linear calculation is executed.

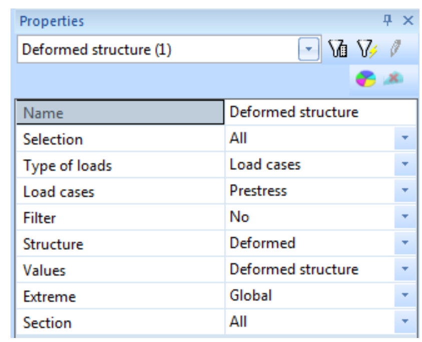


4. Results

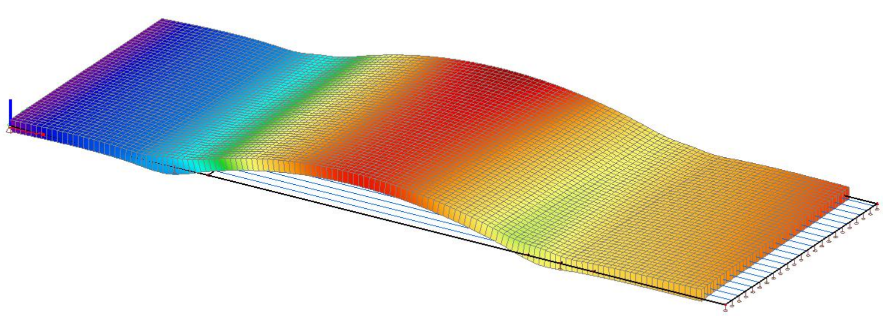
Once the model has been built and the analysis run, it is possible to evaluate results for whether or not the optimal post-tensioning has been applied. Scia Engineer provides a complex analysis of the slab and tendons while taking into account the short term losses from various sources. For the IBC code, 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.

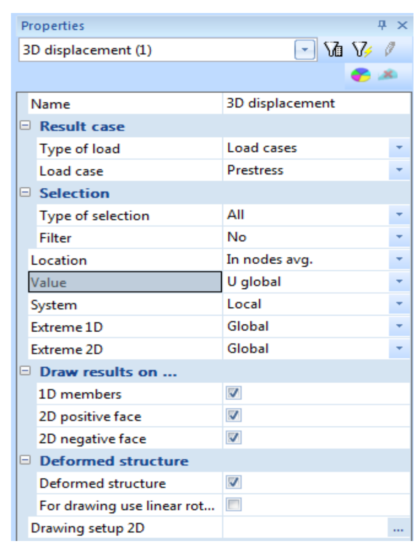
Deformed Structure:





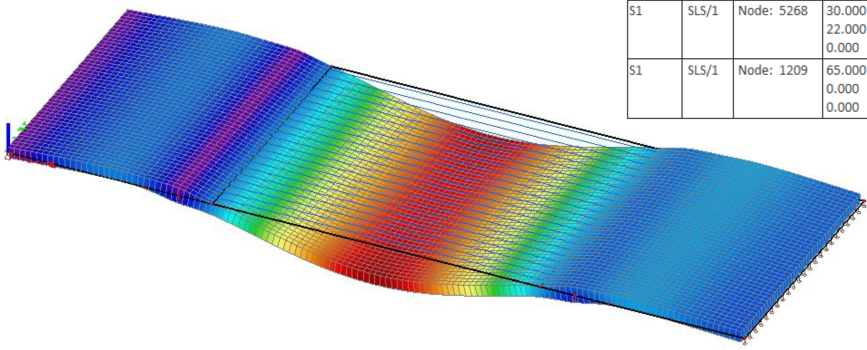
3D Displacement:





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

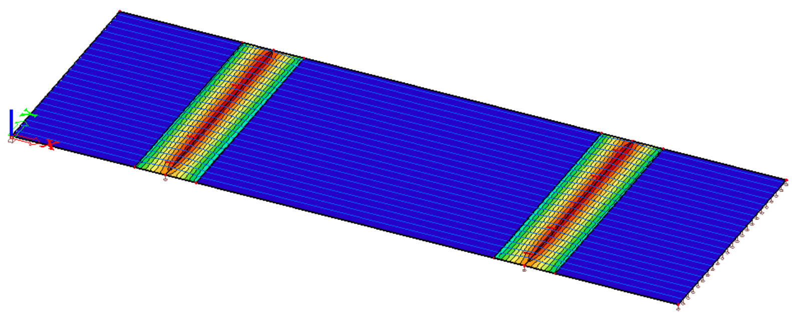
Name	Case	Mesh	Position [ft]	ux+ [inch] ux- [inch]	uy+ [inch] uy- [inch]	uz+ [inch] uz- [inch]	ϕ_x [deg]	ϕ_y [deg]	ϕ_z [deg]	U global+ [inch] U global- [inch]
S1	SLS/1	Node: 5268	30.000 22.000 0.000	0.002 -0.051	0.001 0.001	0.000 0.000	0.000	0.087	0.000	**0.002** 0.051
S1	SLS/1	Node: 1209	65.000 0.000 0.000	-0.049 -0.049	-0.002 0.007	-0.842 -0.842	0.021	0.000	0.001	0.844 **0.844**



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.

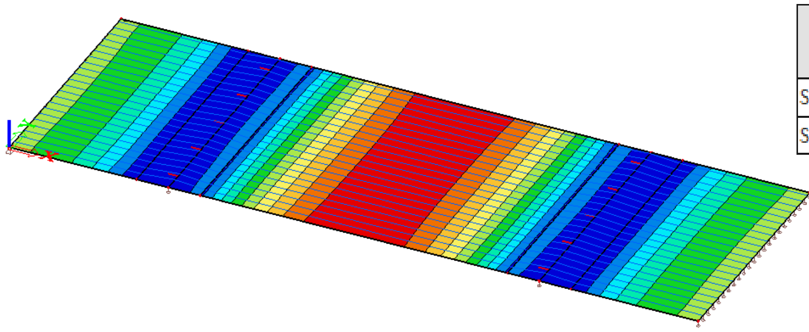
In addition to deformation, the layout of the tendons can also be verified using the internal forces.

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):



Member	elem	Case	mx [kipft/ft]
S1	5584	1.2D+1.6L+1.0PT	**-113.00**
S1	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.