



TUTORIAL CABLES

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 2021 SCIA nv. All rights reserved.

Table of Contents

Table of Cont	ents3
Introduction.	
Chapter 1:	Theoretical background5
Chapter 2:	Modelling6
2.1.	Cross-sections7
2.2.	Geometry7
2.3.	Boundary conditions10
•	Loads and combinations13
-	Analysis15
Chapter 5:	Results16

Introduction

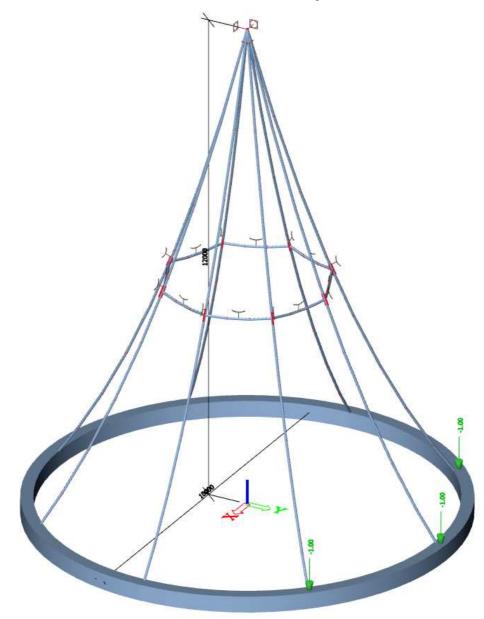
SCIA Engineer allows you to model a structure with cables.

First, we will discuss the theoretical background.

Next, we will explain how to model and load a chandelier structure.

Finally, we will run the (nonlinear) analysis and evaluate some results.

The figure below shows the model of the structure that we will design.



Chapter 1: Theoretical background

Since SCIA Engineer version 21.1, cables are fully supported in the 64bit version in the default postprocessing environment. For cable analysis the module sens.03 is necessary. This module is included in the Expert and Ultimate edition.

A cable element is an element without bending stiffness (Iy and Iz \approx 0). During the solving of the equations this behavior is taken into account, so only axial forces (in tension only) will occur and you will obtain zero values for the internal forces M_x, M_y, M_z, V_y and V_z. The displacements (in the intermediate nodes) have thus been calculated without bending stiffness.

The cable type is a special type of a beam nonlinearity, which can be applied on 1D members in a model.

You will need to create nonlinear combinations to take into account this nonlinear property during the analysis. A geometrical nonlinear analysis should be executed by calculating with a 3rd order analysis with the Newton-Raphson calculation method.

A curve formed by a hanging cable is called a "Catenary". With relatively small bending (height - sag roughly 10% of its length) the curve can be approximated by a parabola (this approach was implemented previously in the 32bit version of SCIA Engineer in the post-processing environment 'v16 and older'), but such approximation however becomes imprecise with greater sag / length ratio (and for different heights of the begin and end nodes of the cable).

The catenary curve is linearly approximated into a polyline. The approximation points (nodes) are calculated from the catenary equation defined by two edge points of the beam and its parameter a, which is defined as a quotient (normal force / cable weight per unit length). The normal force is defined via the nonlinearity parameters and the cable weight per unit length is calculated from the cross-section and material parameters. Formula of the general catenary equation:

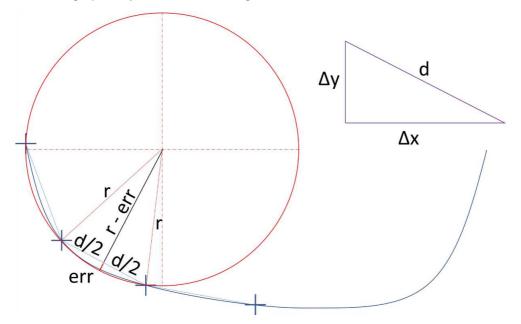
$$y = a \cdot \cosh\left(\frac{x - k}{a}\right) + c$$

Parameters k and c represent the horizontal and vertical shifts of the curve, respectively.

The relation between catenary length **s**, vertical span of two definition points **v** and their horizontal span **H** is:

$$\sqrt{s^2 - v^2} = 2 \cdot a \cdot \sinh\left(\frac{H}{2a}\right)$$

You have full control of the catenary curve approximation precision. A general curvature approach has been utilized. Using the curvature (parameter defining how much the curve bends at the given point) the radius of the circles (of the same curvature as the curve in the touching point) is calculated. Concept of the approximation error is graphically described in the figure below:



The detailed background (including formula to calculate the value of sag) can be found on our help-pages: https://help.scia.net/webhelplatest/en/#analysis/nonlinear_analysis/cables/cable.htm

Chapter 2: Modelling

In the Project settings dialog, activate the Concrete and Steel material.

	DATA			MATERIAL			
199	Name:	J		Concrete	191		
	Name.	1		Contraction of the second s	C25/30	v	
	Part	-		Reinforcement m			
	100 ALIAN			Steel	2		
1	Description:			Material		v	
	Author:	-		Masonry			
				Aluminium			
	Date:	ċ		Timber			
				Steel fibre concre			
	Structure:	🛃 General XYZ	*	CODE			
				National Code:			
	Post processing environment	🤌 default	14. S	EC - EN		۰.	.)
finan in	Model:	関 One	¥	National annex:			
N STRACT		64bit versi	on into	Standard I	EN	۰	

On the Functionality tab activate the Cables functionality:

	GENERAL	DE	TAILED	
-	Property modifiers	2	Nonlinearity	
	Model modifiers		Beam local nonlinearity 🚽	
	Parametric input		Support nonlinearity/basic soil s 💀	
	Climatic loads 🔀		Initial imperfections	
	Mobile loads		Geometrical nonlinearity 🚽	
	Dynamics		General plasticity	
	Stability 📈		Compression-only 2D members	7
	Nonlinearity 🔽		Cables 💡	
	Structural model 🛃		Friction support/Soil spring	J
	IFC properties		Membrane elements	
	Prestressing		Subsoil	
	Bridge design		Soil interaction	
and the second second	Excel checks		Pad foundation check	
			Steel	
A DECORPORATION			Plastic hinge analysis	
-			Fire resistance checks	,

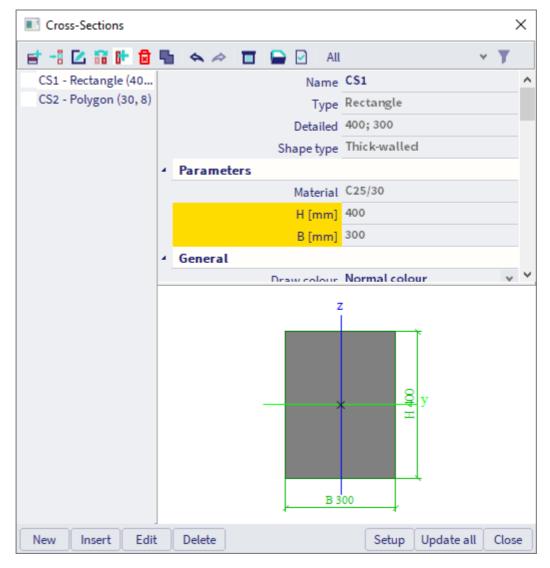
Note that for cable analysis the **module sens.03** is necessary. This module is included in the **Expert** and **Ultimate** edition.

When the Cables functionality is turned on, automatically the functionality **Geometrical nonlinearity** is turned on as well.

2.1. **Cross-sections**

Go to Menu bar > Libraries > Cross sections (or with the icon in the workstation Structure).

Add a **rectangular** concrete cross-section of **400 x 300 mm** for the circular ring at the bottom of the chandelier. Add a **circular** shape or **polygon** of **diameter 30 mm** for the cable elements.



2.2. **Geometry**

You can open the **Marking menu** by holding the **Alt** button together with the **right mouse click**. In the subgroup **Model** you can choose to add a **1D member**:



Draw the ring as two circular arcs with cross-section CS1.

Then you can use the Marking menu again to add 8 Column elements with cross-section CS2 and length 12 m.

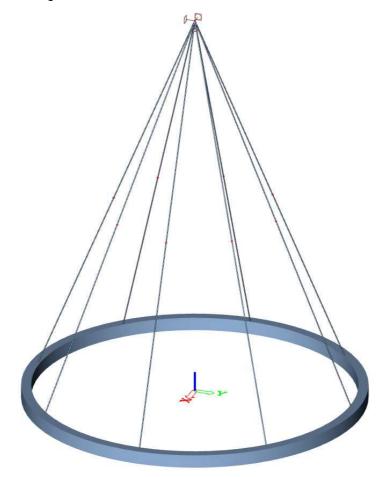
Then select the top nodes of the column elements. Change the X and Y coordinate of these nodes to 0 m via the property panel or via the **input table** (Tools > Input table):

	Name	X [m]	Y [m]	Z [m]	Member	2D me				
1	N9	0.000	5.000	12.000	B11					
2	N11	-3.536	3.536	12.000	B12					
3	N12	-5.000	0.000	12.000	B13					
\$	N14	-3.536	-3.536	12.000	B14					
5	N15	0.000	-5.000	12.000	B15					
5	N17	3.536	-3.536	12.000	B16					
r	N18	5.000	0.000	12.000	817					
8	N20	3.536	3.536	12.000	B18					

Run the Check structure command to eliminate the duplicate nodes at the top:

Jearchide	plicate nodes	Ign	ore parameters	
HECK OF ME	MBERS			
Check m	Result of check of nodes			×
earch nut	These problems ha	ve been found	d with nodes:	
earch dup				
-	Members with undefined nodes:	-	Correct it	
	Free nodes	0		
HECK OF D	Duplicate nodes: Nodes not in slab:	7	_ 🗹 Correct it	
Check da		0	-	
	Incorrect coord.	0	-	
HECK OF A	Do vou w	ant to correct	it?	
Check ad				0%
Check fre	ОК	Cancel		
Спеск пе		V70		
	EEL CONNECTIONS			
HECK OF ST				

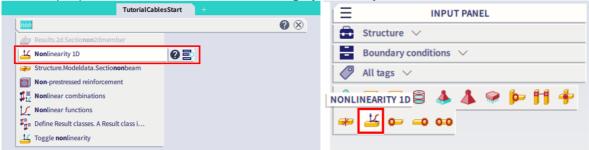
Now we have already following structure:



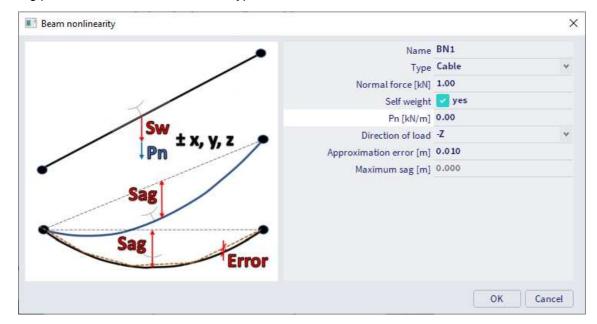
2.3. **Boundary conditions**

You can use the marking menu to quickly add a fixed support at the top.

Now you can apply the Nonlinearity 1D property to the elements. You can search for it via the SCIA Spotlight or find it via Input panel > workstation Structure > category Boundary conditions.



Following parameters are available for the type cable:



- Normal force [kN]: value of the pre-stressing axial force (positive) •
- Self-weight:
- Pn [kN/m]:
- Direction of load:

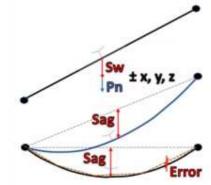
Maximum sag [m]

specifies the load direction (Pn and self-weight) Approximation error [m]: determines the maximal allowed distance between point on the analytically defined catenary curve and the projection of this point (in the direction of load) on the linearly approximated polyline of the cable.

specifies the value of the additional load (so other than self-weight)

determines, whether the cable is subjected to self-weight

automatically calculated value, it is the maximal distance (in the direction of load) between a node of the catenary curve and its projection on the straight line connecting both ends of the cable



Set 1 kN as normal force, activate self-weight and set 0 kN/m as Pn.

When applying the nonlinearity to the elements automatically the geometry is deformed according to the catenary curve which is calculated with the inputted parameters.

Add an element with cross-section CS1 in the middle of two vertical elements.

Via the subgroup Modify of the Marking menu you can select the Multicopy command and copy the horizontal element 8 times.

Aulticopy			>
Number o	f copies	8 + opy	Connect selected nodes with new beams Copy additional data
DISTANCE \	/ECTOR		HOW TO DEFINE THE DISTANCE ?
Define dist	ance by curs	ior 🔽	 between two copies
x V	0.000	m m	from original to the last copy HOW TO DEFINE THE ROTATION ?
z	0.000	m	between two copies from original to the last copy
ROTATION			ROTATION AROUND
rx ry	0.00	deg deg	current UCS distance vector
rz	45	deg	OK Cancel

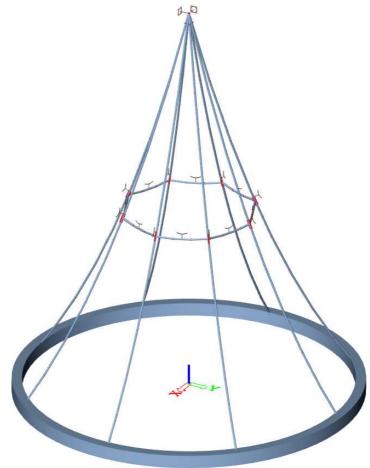
Connect the new elements with the existing vertical elements. When you now would change the nonlinearities (e.g. change the value of the normal force), it will have an effect on the elements with the nonlinearity, but as well on the connected elements.

	Tutori	ialCablesStart	
	Connect members/nodes		?
	– Con nect members/nodes		
1	 Disconnect members/nodes 		
	Restore default user con figuration		
	Ç ^{III} Define the load case. A load case	Ctrl+L	
	rt Connection forces - input		

Select all the new elements via the Expand selection function in the Property panel and add the Cable nonlinearity as well to the elements.

	ø					
	≡			1D ME	MBER (1)	
	ø	₿	IJ			
				Name	B12	
				Layer	$\texttt{Layer1} \lor$	
				Туре	beam (80) 🗸	
			Anal	ysis model	Standard \checkmark	
				FEM type	standard \checkmark	
			Cro	oss-section	CS2 - Polvgon (30, %)	
ls Is	olate s	electio	n			
E	xpand	selecti	on			-1
S S	ubtrac	t from	select	ion		

Now the modelling is finished:



Chapter 3: Loads and combinations

Go to the load cases and create a new load case LC2:

Load cases			Delete all loads >>>	
et -: 🗹 🕩 🖬 🖷	* / 🖬 🖨 🗹	All	Y	
LC1 - Self weight	Name	LC2		
LC2	Description			
	Action type	Permanent		*
	Load group	LG1	۷	
	Load type	Standard		٣
	Stage for composite a	Final stage, long term		٣
	Actions			
		Delete all loads	>>	>
		Copy all loads to another loadcase	>>	>
New Insert Edit	Delete		Clos	e

Via the subgroup Load of the Marking menu you can select the Point Load in Node command and add some point loads of 1 kN to the structure:

E Point force in node		×
Name	F1	
RZ F Direction		¥
	Force	v
Angle [deg]		
Value - F [kN]	-1.00	
✓ Geometry		
Fx i i Fy Fz x i i	GCS	*
	OK Cance	:
	015	

Open the nonlinear combinations:



And create a nonlinear combination NC1 with load cases LC1 and LC2:

📑 📲 🗹 📴 🗧 🗸	1	* Y
NC1	Name	NC1
	Description	
		Ultimate v
	Stage for composite analysis m	Automatic v
	4 Contents of combination	
	LC1 - Self weight [-]	1.000
	LC2 [-]	1.000
New from combinati	New Insert Edit Delete	e Close

Chapter 4: Analysis

As the calculation model is completely ready, you now can start the calculation.

Use the **Calculate** button in the middle of the process wheel in the **process toolbar** to start the analysis (or via **Menu bar > Tools > Calculation & Mesh > Calculate**).



When starting the analysis, check that for the **Mesh setup** the **Average number of 1D mesh elements on** straight 1D members (linear approximation between the nodes of the catenary curve) and in the **Advanced** solver settings Geometrical nonlinearity is set to 3rd order (large deformations). If there are problems with convergence, increase the values of **Number of increments**, or **Maximum iterations**. Additionally, the Solver precision ratio or Solver robustness ratio might be altered to have an influence on the convergence criteria as well. Select **Nonlinear analysis** and press **Calculate**.

Calculations	 Mesh setup 	
Calculations	Average number of 1D mesh element: 4	
Linear analysis	Average size of 1D mesh element on c 0.200	
Load cases: 2	Average size of 2D mesh element [m] 1.000	
Nonlinear analysis Nonlinear combinations: 1	Connect members/nodes	
	Setup for connection of structural ent	
Other processes	Advanced mesh settings	
Test input of data	 Solver setup 	
	Specify load cases for linear calculati	
Save project after analysis	Specify combinations for nonlinear ca	
	 Advanced solver settings 	
	▷ General	
	Effective width of plate ribs	
	Nonlinearity	
	Geometrical nonlinearity 3rd order (large deformation)	
	Method of calculation Newton-Raphson	•
	Number of increments 5	
	Maximum iterations 20	_
	Solver precision ratio 1	
	Solver robustness ratio 1	
	 Initial stress 	
Calculate	Initial stress 🗸	

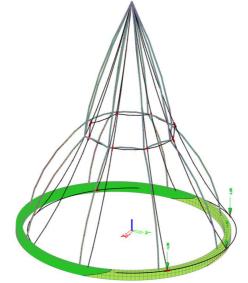
Chapter 5: Results

When the calculation is completed, results can be viewed. The relevant commands in the **workstation Results** become active now.

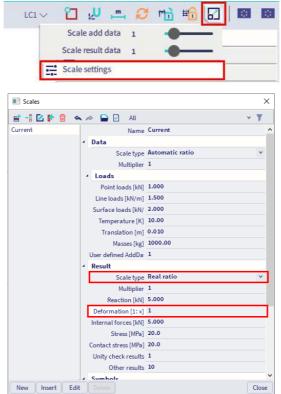
Select the workstation Results in the process toolbar and click on the command 3D deformations.

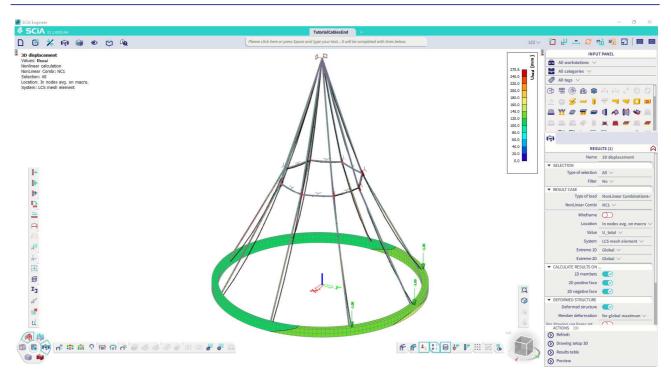


With the 3D deformations you can evaluate the behavior of the cables for the nonlinear combination NC1 and you will see that the cables go upwards. The graphical results look a bit strange because SCIA Engineer presents the biggest deformation as 1 m:



This makes sense for general structures, but for cables it's more convenient to look at the real ratio, which can be edited via the **Scale settings** in the **View bar**. Set **Scale type** to **Real ratio** and **Deformation [1:x]** to **1**.





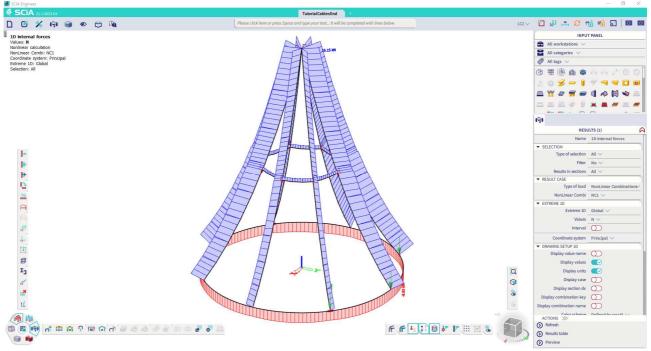
You can notice that the load straightens the curved elements.

You can also have a look at the 1D internal forces such as normal forces N or bending moments My. Select the **workstation Results** in the **process toolbar** and click on the command **1D internal forces**.

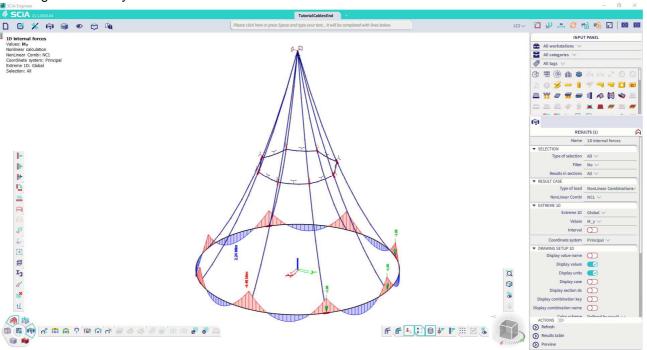


For these results you can put the Scale type back to Automatic ratio in the Scale settings.

Normal forces N:



Bending moments My:



So in the cables, the only non-zero values of internal forces are for the normal forces N (tension). On the second picture we can see that there are no bending forces present in the cables

Notes:

- The input of the cable element only defines the initial shape / sag. Afterwards the cable can be loaded by real loads. The normal force after calculation will therefore be different to the inputted initial normal force N.
- Adding a cable into the model will adapt the mesh also for the linear calculations. This means that the linear calculation may become unstable when a cable is added, and a nonlinear calculation becomes necessary to obtain results.