Finite element analysis of a steel conoid shell

Install SCIA Engineer on your computer. You can download the software for free from https://www.scia.net/en/forms/free-engineering-software-student-licence-subscription

🈻 S	CIA Engineer	
	SCiA 21.1.3	026.64 - student version
) ⅔ ⊗	₽ a
	Project browser	
	New	Ctrl+N
	Open	Ctrl+O
	Open from	•
	Recent projects	•
	Savo	Ctrl+S
	Save	Gurra

Project data	\frown				×
Basic data Fu	unctionality Action	ns Unit Set Protec	tion		
	DATA			MATERIAL	
	Name:	-		Concrete	
-				Steel 🔽	
	Part:	-		Material S 235	·
	Description:	-		Masorry	
	· ·			Aluminium	
	Author:	-		Timber	
	Date:	28.05.2022		Steel fibre concrete	
				Other	
		\frown			
E	Structure:	🛃 General XYZ	~	CODE	
				National Code:	
	Post processing environment	🐗 default	~	C-EN 🗸	
	Model	8 0no		National annex:	
The second second	Model.	⊠ One	· · ·	Standard EN 💌	
		<u>64bit ve</u>	ersion info		2
				OK Car	ncel



Choosing the functionality makes extra menu items visible.

Every finite element program consists of three parts.

- 1) Drawing the structure (pre-processing)
- 2) Performing the analysis
- 3) Displaying the results (post-processing)

We first enter 5 points for drawing a conoid shell.



	. fund	i fini	Z [m]	Member	2D member			
11	5.000	0.000	0.000					
12	5.000	10.000	0.000					
13	-5.000	10.000	0.000					
14	-5.000	0.000	0.000					
15	0.000	0.000	5.000					



The shell is a 12 mm curved steel plate.

2D member				\times
	Name	\$1		^
	Element type	Standard	~	
	Element behaviour	Standard FEM		
	Туре	shell (98)	~	
	Material	S 235	×	
	FEM model	Isotropic	~	
	FEM nonlinear model	none	~	
ez	Thickness type	eenstant		
	Thickness [mm]	12		
	Member system-plane at	Centre	~	
	Eccentricity z [mm]	0		
Z	LCS type	Standard	~	
	Swap orientation	no		-
x y	LCS angle [deg]	0.00		
		ОК	Cance	el

Click on the nodes N1, N2, N3, N4. Click on "New circular arch".

Es	al.esa		Esal	2.esa		Esa3.	esa	+				
New shell member - New po	lygon -	Start	poi N	IEW	CIRCO	AL AF	C					
	4	đ	4	X	C	\mathbf{Y}	\cap	S	\sim	\odot	0	S

Click on the nodes N5 and N1.

Click right mouse button and click Confirm action.



The shell is supported by fixing the straight edges.

INPUT PANEL	💼 Structure 🗸				
Boundary conditions 🗸	🥔 All tags 🗸				
🛦 🐨 🗊 📾 😹	🌢 🤗 🕨 🦓 💾 📥 🔶				
≠ ≝ ∞ = ∞ ∞					

Line support on 2D member edge			×
	Name	Sle1	
	Constraint	Fixed	*
	х	Rigid	*
AD-2	Y	Rigid	*
	Z	Rigid	*
↓Z	Rx	Rigid	*
	Ry	Rigid	*
	Rz	Rigid	*
x1	Geometry		
x2	System	GCS	*
	Coord. definition	Rela	*
Z	Position x1	0.000	
	Position x2	1.000	
X Y	Origin	From start	*
			DK Cancel

Click on the straight edges.



Click right mouse button and click Confirm action.

One load case is already present: Self-weight. We add a load case: Snow.



Load cases			×	K
≓ -: 🖸 🕨 🔒 🐘 🐟 🗢 🔲 🕒	All		• T	
LC1 - Self weight	Name	LC1		
	Description	Selfweight		
	Action type	Permanent		۷
	Load group	LG1	۷.	
	Load type	Selfweight		۷
	Direction	-Z		۷
New Insert Edit Delete			Close	•

Load cases					×
📑 📲 🗹 🕩 🛢 🖷	۵ 🖈 🗖	🕒 All		¥]	ľ
LC1 - Self weight		Name	LC2		
LC2 - Snow		Description	Snow		
		Action type	Permanent		٣
		Load group	LG1	~	
		Load type	Standard		٣
	Actions				
			Delete all loads	>>	·>
		Copy all lo	ads to another loadcase	-	~
New Insert Edit	Delete			Clo	se

Now we can enter the snow load: 2 kN/m^2





We make a load combination in which both Self-weight and Snow occur at the same time. Safety factors are not used.



Combinations			×
et -: 🖸 🕩 🗟 🐟 🖉 🗖	Input combinations	*	
ULS-Set B (auto)	Name	SLS-Char (auto)	
SLS-Char (auto)	Description		
	Туре	EN-SLS Characteristic	
	Updated automatically	~	
	Structure	Building	
	Active coefficients		
4 Conte	ents of combination		
	LC1 - Self weight [-]	1.000	
	LC2 - Snow [-]	1.000	
<			>
Actions			
		Explode to envelopes	>>>
		Explode to linear	>>>
	Show Decom	posed EN combinations	
New Insert Edit Delete	e		Close

We first perform a linear elastic analysis.

î <u>†</u> î	 ‡≡	₽ ≣	\$₹	$\mathfrak{A}_{\mathbb{T}_{=}}^{\mathbb{T}_{=}}$	<u>.</u>	<u></u>

The influence length is 2.4*sqrt(5*0.012) = 0.6 m. The element size should be 1/6 of this. Let's set the element size to 0.3 m. This is a bit large but we can always make it smaller later.

FE analysis		×
Calculations	Mesh setup	^
 Linear analysis Load cases: 2 Other processes 	Average number of 1D mesh element: 1 Average size of 1D mesh element on c 0.300 Average size of 2D mesh element [m] 0.300	
Test input of data	Setup for connection of structural ent	
Save project after analysis	 Advanced mesh settings Solver setup 	
	Specify load cases for linear calculatie	
	General Neglect shear force deformation (A) Bending theory of plate/shell analys Kirchhoff Type of solver	* *

SCIA Engineer: End of analysis	×
Mesh generation: OK Calculation of static load cases: OK Linear analysis: OK	
Maximal translation -1.2 mm in node 1269 [.0.000,8.766,0.606] (load case LC2) Maximal rotation 1.8 mrad in pode 1398 [-0.000,9.697,0.152] (load case LC2) Sum of loads and reactions is OK	
ОК	

Note that the software has checked equilibrium of the loads and the support reactions. If this would not be OK the arithmetic accuracy is not OK.

Let's look at the results. Deflection due to the load combination.



The deflection is largest in the flat part of the shell. Nonetheless it is just 2.1 mm which is surely acceptable.

The mesh is shown too. The elements may be too large to describe the red peak accurately. For now we leave the element size as it is because we do not want to increase the computation time.

Note that the mesh consists of rectangles and triangles. Triangles are less accurate than rectangles. Why does the program choose triangles?

Let's look at the Von Mises stresses.



In the picture below the mesh is not displayed. This can be selected in "Drawing setup 2D".



In most of the shell the stress is almost zero. The largest stresses occur at the edges (edge disturbance). A largest stress of 25.7 N/mm² is not much. But this will increase when the mesh is refined. (Not to infinity because it is not a singularity.) Nonetheless, perhaps the shell can be designed much thinner. This view is at the outside surface of the shell. If you turn the shell around you can see the stresses at the inside surface.

Let's reduce the element size to check accuracy.



The new linear elastic calculation takes a minute.

The difference between the two computations is just 0.4 N/mm^2 . So an estimate of the error that we make with the 0.150 mm mesh is 0.4 N/mm^2 or 2% which is sufficiently small for most applications. In the following we will work with 0.300 m elements.

Let's check buckling. Now we need to define stability load combinations.



(The teacher does not understand why the software does not use the normal load combinations for stability analysis too. Perhaps this extra step will be removed in future software versions.)

Stability comb	inations - S1		×
Contents of co	ombination case C1 - Self weight C2 - Snow	List of load cases	
Name : Coeff :	S1 Correct	Delete All Add All	
		OK Cancel	

Now we can perform the linear buckling analysis. Let's compute 6 buckling modes.

Note that first a linear analysis is performed. This is to compute the membrane forces, which cause buckling.





		^
lculations	Mesh setup	
Linear analysis	Average number of 1D mesh elements (1	
Load cases: 2	Average size of 1D mesh element on cu 0.200	
Linear stability	Average size of 2D mesh element [m] 0.300	
Buckling modes: 6	Connect members/nodes 🗹	
Nonlinear stability	Setup for connection of structural entit	
Buckling modes: 6	Advanced mesh settings	
ier processes	Solver setup	
Test input of data	 Advanced solver settings 	
lest input of data	4 General	
Save project after analysis	Neglect shear force deformation (Ay, 🔽	
Save project arter anatysis	Bending theory of plate/shell analysis Kirchhoff	
	Type of solver Direct	
	Number of sections on average memt 10	
	Warning when maximal translation is 1000.0	
	Warning when maximal rotation is gr 100.0	
	Coefficient for reinforcement 1	
	Effective width of plate ribs	
	Nonlinearity	
	Initial stress	
	▷ Dynamics	
	4 Stability	
	Type of eigen value solver Lanctos	
	Number of buckling modes 6	
	4 Soil	
Calculate	SCIA Engineer: End of analysis X	
	Mesh generation: OK Calculation of static load cases: OK Linear analysis: OK 	
	ОК	12

Let's look at the results.



First buckling mode:



It buckles in this shape when the load combination is multiplied with a load factor of 9.09. The subsequent buckling load factors are 9.13, 12.37, 12.70, 13.70, 14.24 et cetera.

The figure shows that this buckling mode moves mostly inwards. However, the exact same mode is sometimes displayed as moving mostly outwards. Does this make sense?

Clearly, an edge beam would supress most buckling modes. With an edge beam the shell could be even thinner.

Let's look at the natural frequencies. First we need to add mass to the load cases.

(The program already knows about the mass of the materials but it does not know whether other loads have a mass. For example snow load has a mass and wind load has no mass.)

Let's study the vibrations without snow. The mass of the materials is in something called MG1.

/ 📾 🔊						MASS GRONPS
	í¹ì	‡≡	‡≡	₽ ≣	#₹	H. 2

Mass groups		>	×
📑 📲 🔀 📴 🐟 🛷 🛅 🕞 🗛	~	Ţ	r
MG1 Name MG1			
Description			
Bound to load case Yes			۷
Load case LC1 - Self weight	t	٣	
Keep masses up-to-date with loads			
Actions			
Create masses from load	d case	>>>	•
Delete all m	asses	>>>	
New Insert Edit Delete		Close	•

We make a mass combination of just this mass.

			C	MBIN	ΙΑΤΙΟ	N OF M	ASS	GROUPS
í <u>‡</u> ì.	‡≡	 ₽ ≣	₽ ₹	µ≡ µ≡	4)	

Combinations of	mass groups - CM1	×
Contents of com	nbination oup I	List of mass groups
Name : Description : Coeff :	CM1 1 Correct	Delete All Add All
		OK Cancel

Combinations of mass groups		×
et -: 🖸 🕩 🖬 🐟 🖉 🕻	All	× T
CM1	Name CM1	
	Description	
4 Con	ents of combination	
	MG1 [-] 1.000	
New Insert Edit Dele	te	Close

Now we can compute the natural frequencies.



	Average size of 1D much element on a 0 200		
Calculations	Average size of 1D mesh element on c 0.200	Average size of 1D mesh element on c 0.200	
Z Linear analysis	Average size of 2D mesh element [m] 0.300		
Load cases: 2	Connect members/nodes 🗹		
🗸 Modal analysis	Setup for connection of structural ent		
Eigenmodes: 10	Advanced mesh settings		
Other processor	Solver setup		
other processes	 Advanced solver settings 		
Test input of data	✓ General		
	Neglect shear force deformation (Ay 🔽		
Save project after analysis	Bending theory of plate/shell analys Kirchhoff		
	Type of solver Direct		
	Number of sections on average men 10		
	Warning when maximal translation 1000.0		
	Warning when maximal rotation is g 100.0		
	Coefficient for reinforcement 1		
	Effective width of plate ribs		
	Nonlinearity		
	Initial strong		
	- Dynamics		
	Type of eigen value solver Lanczos		
Calculate	Number of eigenmodes 10		
Calculate	Use IRS (Improved Reduced System)		

Let's look at the results. First vibration mode: The natural frequency is 8.71 Hz.



The subsequent natural frequencies are 8.71, 12.61, 12.78, 15.37, 15.81, 16.34, 17.01, 18.18, 20.01 et cetera. Note that there is little space between the natural frequencies. This is typical for shell structures.

Frame structures are sometimes designed such that the frequency of the load is no problem because it is in between two natural frequencies. Is this possible for shells too?

The program has computed the 10 smallest natural frequencies. If you want to see more you can set this just before starting the computation.

This shell is not sensitive to wind load because wind gusts have a frequency of approximately 1 Hz or smaller.

This shell is sensitive to earthquakes. It would not be sensitive to earthquakes, if all natural frequencies were larger than 10 Hz.

Vibration mode shapes often look like buckling mode shapes. There is a situation in which they are exactly the same. This is when the load is so large that the structure almost buckles. If you give the structure is little push it moves away slowly and comes back slowly. In this situation the natural frequency is almost 0 Hz and the vibration mode shape is the same as the buckling mode shape.

Why does the program always perform linear analyses before computing natural frequencies?

Let's do a nonlinear analysis. First we need to specify nonlinear load combinations. (The teacher thinks that there is no difference between linear and nonlinear load combinations.)



Here we can specify a shape imperfection. We choose an imperfection shaped as the first buckling mode with an amplitude of 24 mm (2 times the shell thickness).

Nonlinear combinations	×
📑 🗄 🔀 🕪 🖬 🐟 🗢 🔲 All	* T
NC1 Na	me NC1
NC2 Descript	ion
Ту	ype Ultimate 🗸 🗸
Contents of combination	l
LC1 - Self weight	t[-] 1.000
LC2 - Snow	/[-] 1.000
Bow imperfecti	on None Y
Global imperfecti	on Buckling shape 🗸 🗸
Stati	lity S1 🗸
Eigen sh	ape 1
Max deformation [m	n] 24.0
New from combinati New Insert Edit Dele	te Close

We also apply the imperfection in the other direction. So an amplitude of -24 mm.

Nonlinear combination	ons	×
et -: 🖸 🕩 🖬 🔶	🗙 🎓 🛅 All	• T
NC1	Name	NC2
NC2	Description	
	Туре	Ultimate v
	Contents of combination	
	LC1 - Self weight [-]	1.000
	LC2 - Snow [-]	1.000
	Bow imperfection	None 👻
	Global imperfection	Buckling shape 🗸 🗸
	Stability	\$1 v
	Eigen shape	\sim
	Max deformation [mm	-24.0
]	\checkmark \land
New from combinati	New Insert Edit Delete	Close

The software applies the load in 10 equal steps (increments). After each step it uses the Newton-Raphson method to compute the displacements. This involves a few iterations until sufficient accuracy is obtained. The program would stop prematurely if more than 20 iterations have been performed without finding sufficient accuracy (no convergence).





The imperfection with an amplitude of 24 mm gives the largest deflection.

Compare the deflection of the linear analysis to that of the nonlinear analysis. What causes the differences?

The Von Mises stress is largest for the imperfection with an amplitude of 24 mm. However, it is still very small.



Let's see if the load factor of the linear buckling analysis is correct. We increase the loads with a factor 9.09.

This is the buckling load factor computed in the linear buckling analysis.

Nonlinear combinations X					
📑 📲 🗹 🕩 🛢 🐟	🗙 🗢 🔲 All	× T			
NC1	Name	NC1			
NC2	Description				
	Туре	Ultimate 🗸			
	Contents of combination	\frown			
	LC1 - Self weight [-]	9.090			
	LC2 - Snow[-]	9.090			
	Bow imperfection	None Y			
	Global imperfection	Buckling shape 👻			
	Stability	S1 ¥			
	Eigen shape	1			
	Max deformation [mm]	24.0			
New from combinati	New Insert Edit Delete	Close			

We apply the load in 90 increments.

FE analysis			>
Calculations	4 General		
Calculations Linear analysis Load cases: 2 Nonlinear analysis Nonlinear combinations: 2 Modal analysis Eigenmodes: 10 Linear stability Buckling modes: 6	Neglect shear force deformation (Bending theory of plate/shell ana Type of solv Number of sections on average m Warning when maximal translation Warning when maximal rotation i Coefficient for reinforceme	A) V Kirchhoff Direct 10 1000.0 s g 10.0 1	*
Nonlinear stability Buckling modes: 6	Nonlinearity Geometrical nonlinear	ty 3rd order (large deformation)	*
Other processes Test input of data	Method of calculati Number of increme Maximum iterati	nts 90 ns 100	Y
Save project after analysis	Solver precision ra Solver robustness ra	io 1	
	 Initial stress Dynamics Stability 		
	Type of eigen value solv	er Lanczos es 6	۷
Calculate	▷ Soil		

EM dialog			
Calculation Nonlinear			
0.25 u [m]	f	ì 10 [deg]	
0 1	00	200	
Increment	48 / 90		
Nonlinear combination	1		
Show Paramete Node	Value	Unit	
	1 33E-001	fml	
✓ ti 5	6.78E+000	[dea]	
	FE	-Calculation 64 -	- Warning X
 Stop after this nonlinear of Redraw graph 	ombination No.	The stiff The struct in FE-no 1.452450	fness matrix is singular! ucture is unstable. Instability found ode No. 1007 [-1.1115286684493; 6.97167688220049; i02825973] (macro S1), direction phi_Y, increment 48 .
Break	Pa		OK

For the amplitude of 24 mm the Newton-Raphson procedure diverges at load increment 48. The load factor is then $48/90 \times 9.09 = 4.85$. The deflection is then 133 mm. Often, this is the maximum load.

We need to assume that the shell buckles at this load. What is the knockdown factor?

This shell is sensitive to shape imperfections but not very sensitive.

This design can be improved: The thickness can be reduced. If an edge beam is added, the thickness can be reduced even further.