\ SCIA ENGINEER

TUTORIAL

STEEL PLATE DESIGN

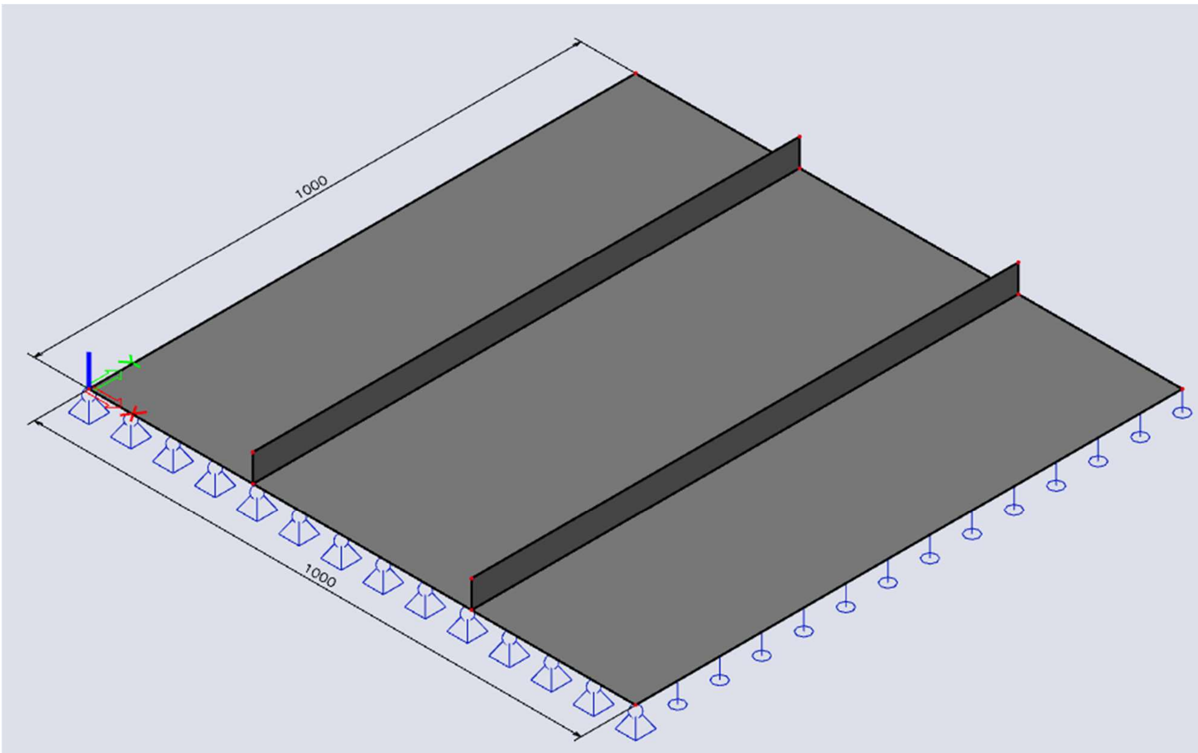
Table of Contents

Introduction

SCIA Engineer enables you to use the functionalities ‘Stability’ and ‘General Plasticity’ to evaluate steel plates.

This tutorial describes:

- How to activate the functionality ‘General plasticity’;
- How to perform a stability analysis;
- How to apply the buckling shape from the stability analysis in the nonlinear analysis;
- How to evaluate the results of the analysis.



Steel plate design according to EN1993-1-5

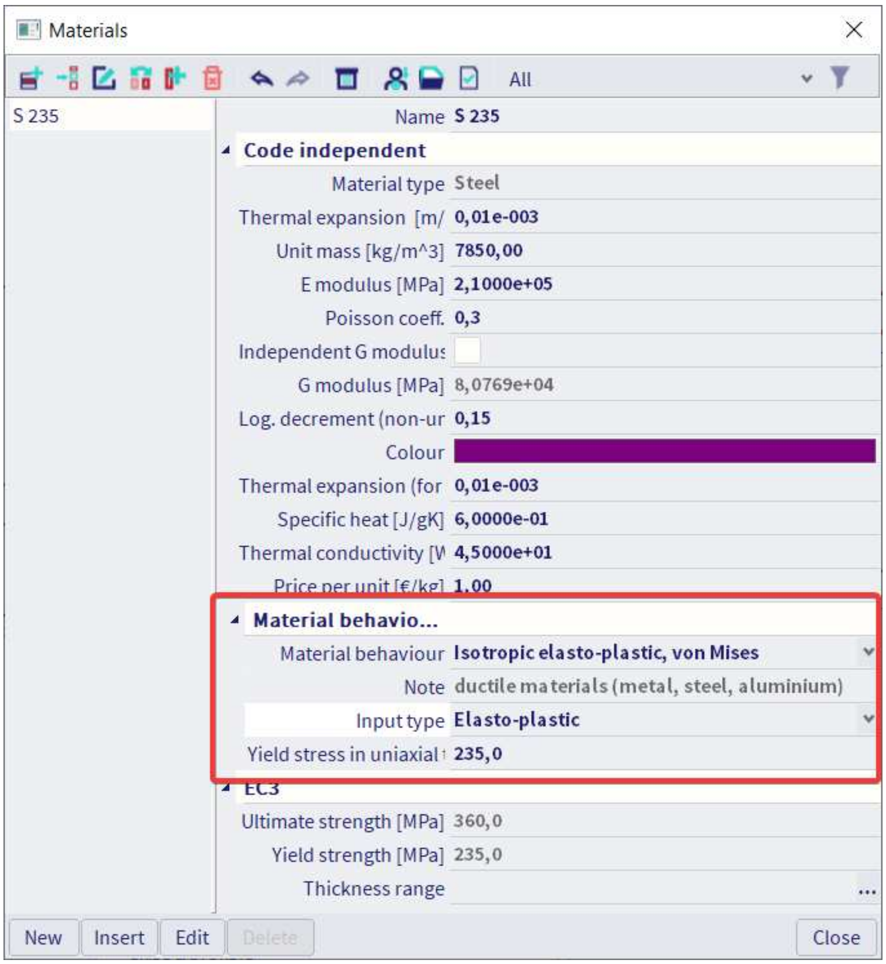
1. Model the steel structure within SCIA Engineer using finite elements

Use plastic material properties for the steel plates. Doing so, the nonlinear analysis will automatically consider the plastic material behaviour and afterwards the plastic results can be verified in the 'Results' workstation of the process toolbar (plastic stresses and strains).

More info about the module 'General Plasticity' can be found in our online help:

https://help.scia.net/webhelplatest/en/#analysis/nonlinear_analysis/general_plasticity/general_plasticity.htm

First activate the functionality 'General Plasticity' in the 'Project settings' (Main menu > File > Project settings). Afterwards you can activate the plasticity behavior in the material properties. Go to 'Libraries' in the main menu and chose 'Materials from the dropdown menu:



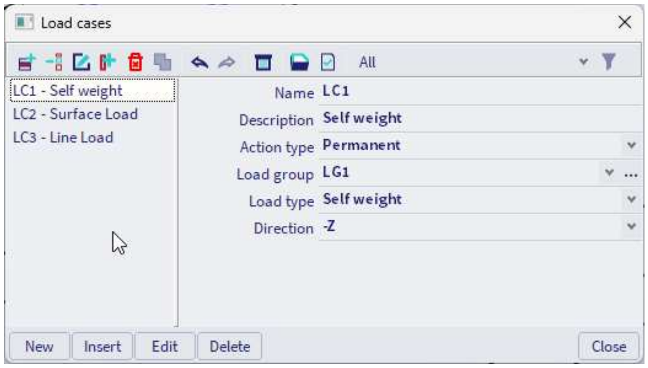
You can model a plate of 1 m by 1 m in material quality S 235 with a thickness of 10 mm.

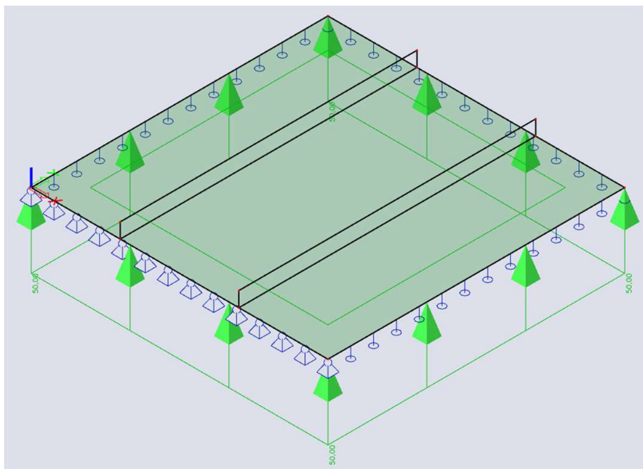
On top of it you can model two stiffeners of 0.05 m height in the same material quality and with the same thickness of 10 mm.

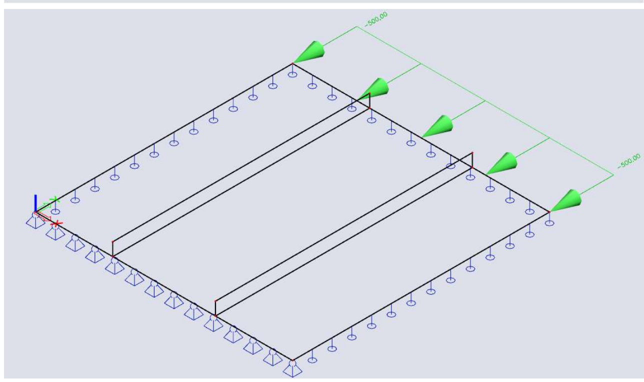
Add line supports to the edges of the 2D member with Z direction set as Rigid and all other degrees of freedom set as Free. Put X and Y direction for 1 edge as Fixed to avoid that the model becomes unstable.

2. Insert loads

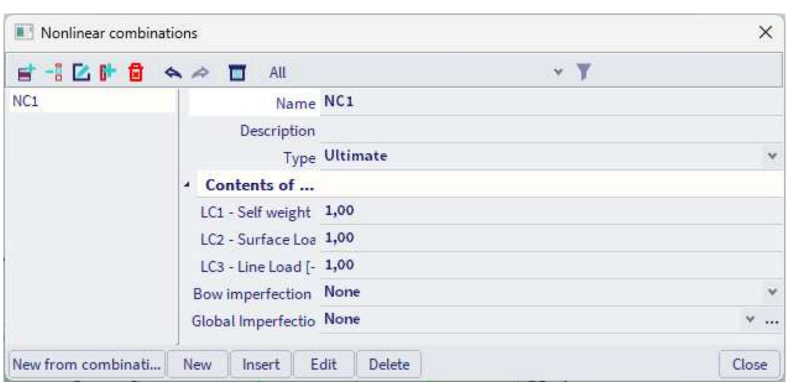
You can create a variable load case with a surface load of 50 kN/m² and another variable load case with a line load of 500 kN/m on the edge of the plate element. You can define a linear combination.







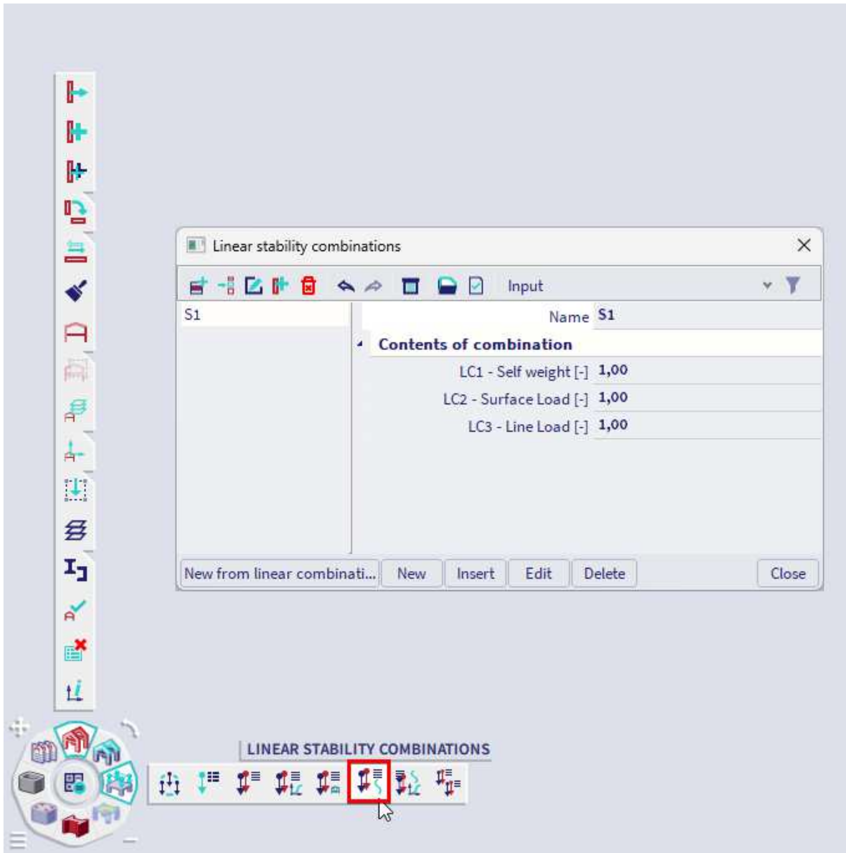
Put all load cases also in a nonlinear combination (to take into account plasticity of the material). For a real structure you will define specific coefficients for each load case to perform a ULS design.



3. Create linear stability combinations

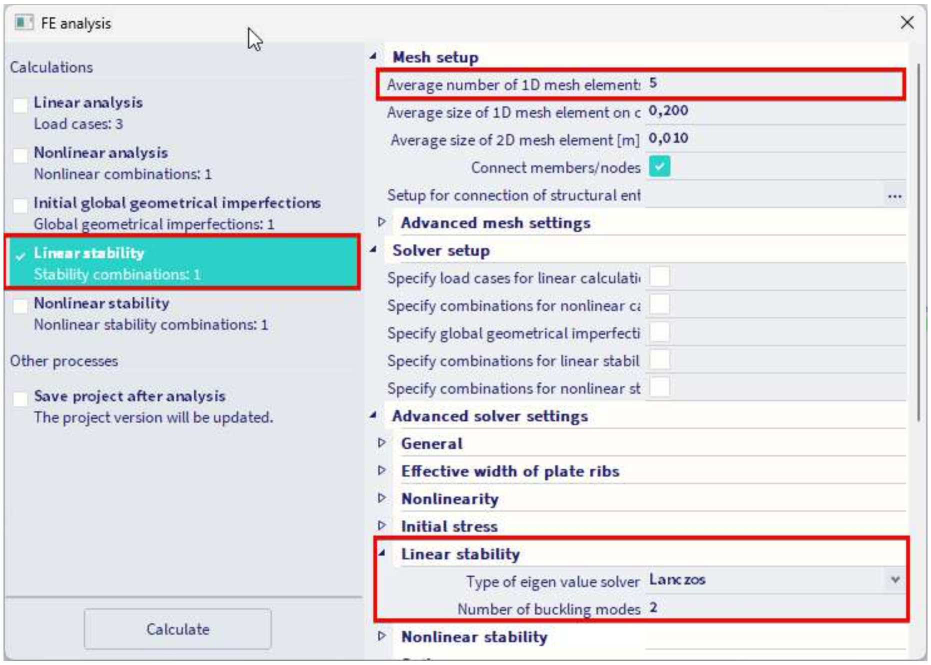
A stability analysis within SCIA Engineer is an elastic buckling shape analysis. These elastic buckling shapes need to be inserted as being imperfections for the second order analysis according to EN1993-1-5.

Note: in this example, a unit value is used for the stability combination. In reality, you need to make stability combinations using ULS coefficients for the loads since this is a buckling analysis.

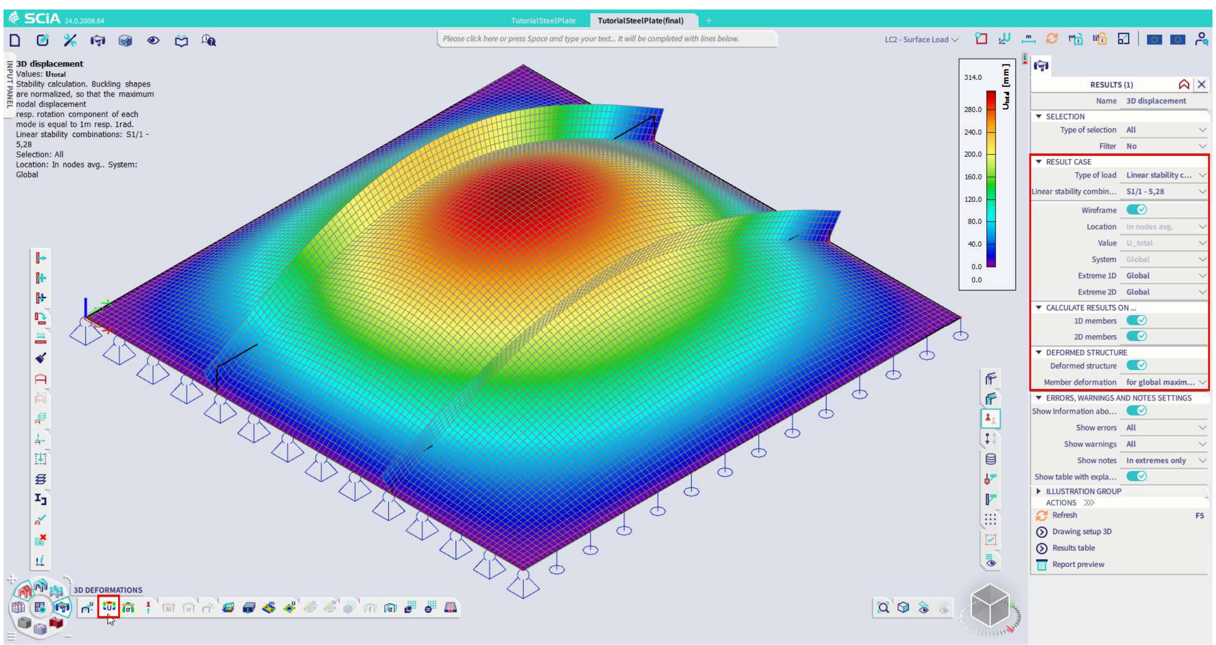


4. Perform the linear stability analysis and check the buckling shape

Go to the process toolbar and press the Calculate button in the middle of the wheel. Set the average number of mesh elements on 1D members to a value of 5 or higher and define the number of buckling modes that you want to calculate.



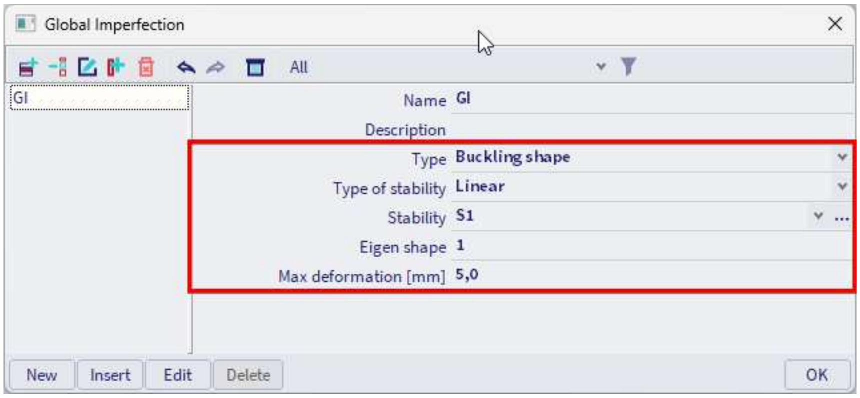
Go to the Results workstation in the process toolbar and choose 3D deformations. Now you can visualize the buckling shape per calculated mode of the stability combination.



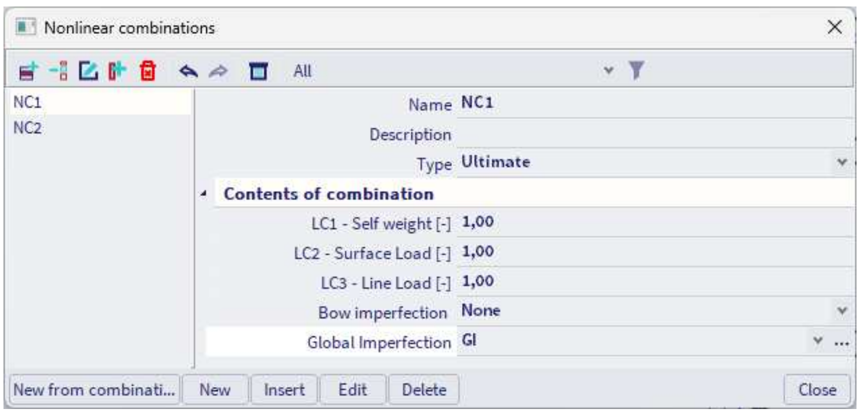
Note: the deformations from a stability combination are normalized values. So these aren't actual displacements but can be used to show the relative deformation between the different elements in your model.

5. Insert this buckling shape as an imperfection for the second order analysis

Now you can create a Global imperfection (Main menu > Libraries > Load cases, combinations > Global Imperfections) based on the desired buckling shape.



Then we can apply this imperfection to the nonlinear combination. You can create a second nonlinear combination NC2 without imperfection to compare with the one with imperfection.



Again, in reality here should be used ULS coefficients for the loads.

As stated in step 4, the buckling shape is shown for a normalized value. We have to define an amplitude if we want to use this shape as an imperfection. You can chose an imperfection based on the tables from EN1993-1-5 shown on the next page. We will use an imperfection value of 1000 mm / 200 = 5 mm.

Table C.2: Equivalent geometric imperfections

Type of imperfection	Component	Shape	Magnitude
global	member with length ℓ	bow	see EN 1993-1-1, Table 5.1
global	longitudinal stiffener with length a	bow	min (a/400, b/400)
local	panel or subpanel with short span a or b	buckling shape	min (a/200, b/200)
local	stiffener or flange subject to twist	bow twist	1 / 50

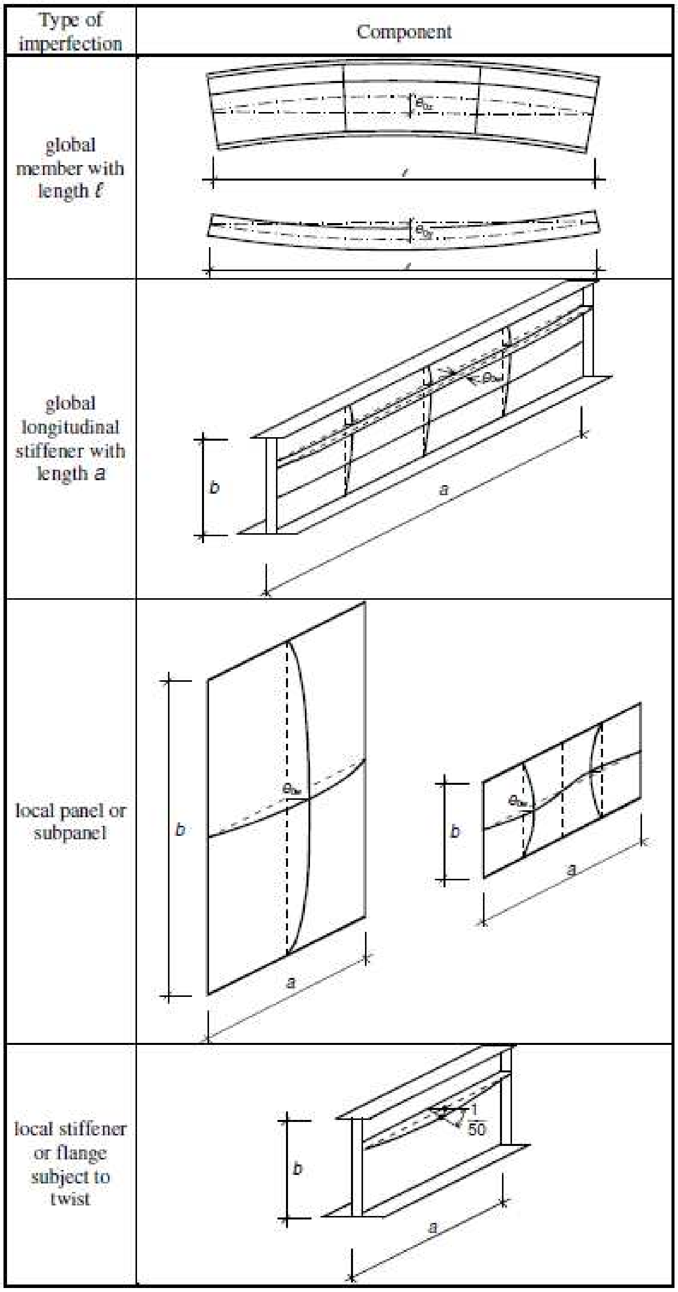
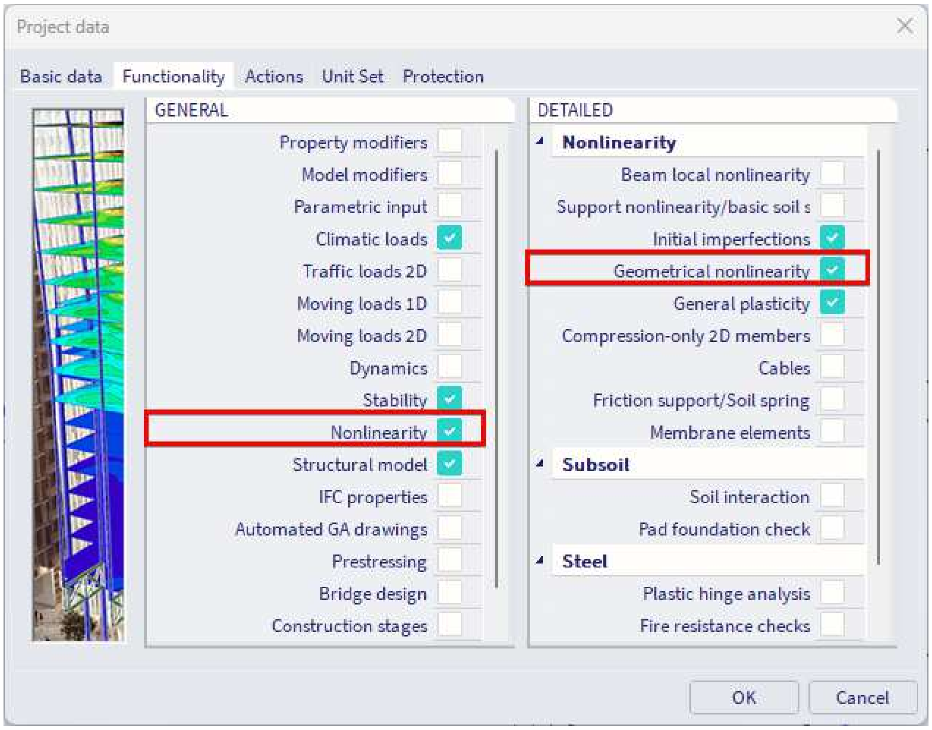


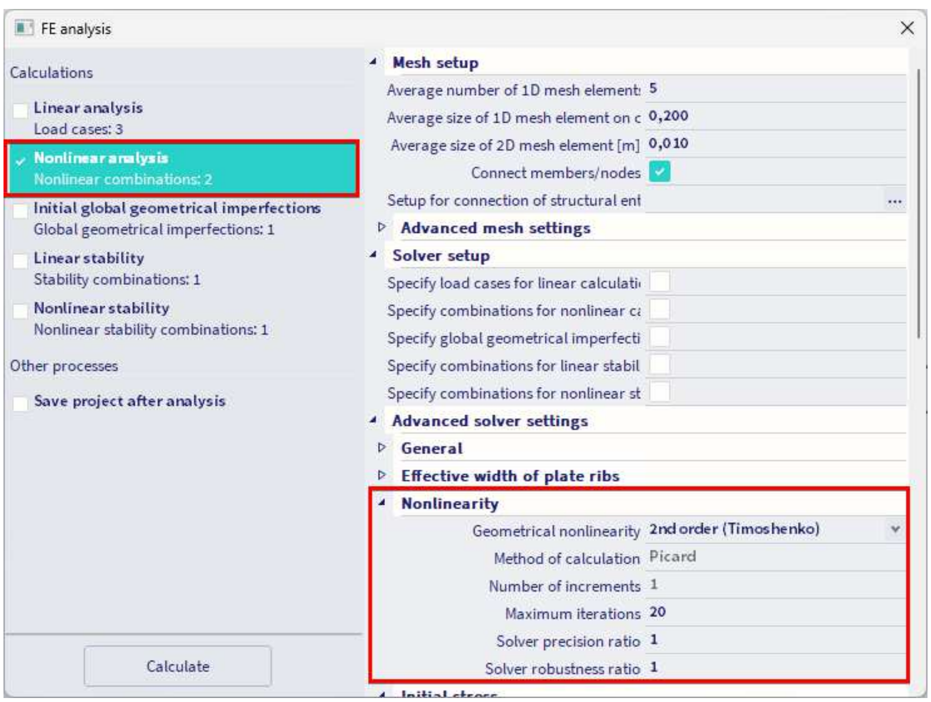
Figure C.1: Modelling of equivalent geometric imperfections

In case of combining imperfections (a leading imperfection and accompanying imperfections), you'll need to modify the geometry of the model or substitute the imperfections by fictitious forces on the member. SCIA Engineer allows the use of one unique stability mode as imperfection. For each non-linear combination a different stability shape can be set as imperfection. In case more granularity is requested, for example to combine imperfections (a leading imperfection and accompanying imperfections), then as an alternative approach to using imperfections a set of fictitious forces can be used.

6. Run the nonlinear analysis

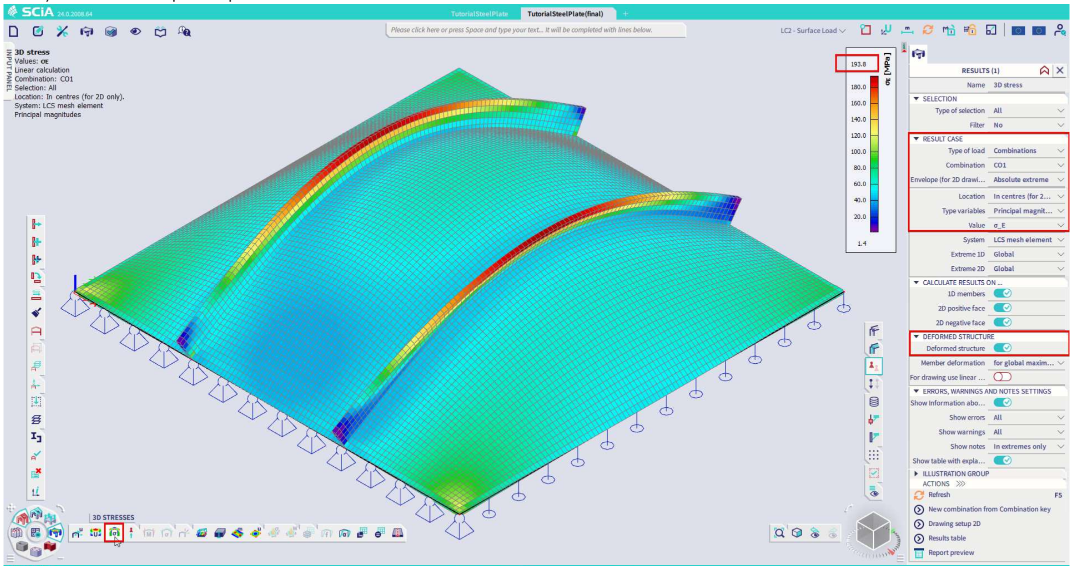
Make sure that you activate the second order analysis with the functionality 'Geometrical nonlinearity' in the 'Project settings' enabled.

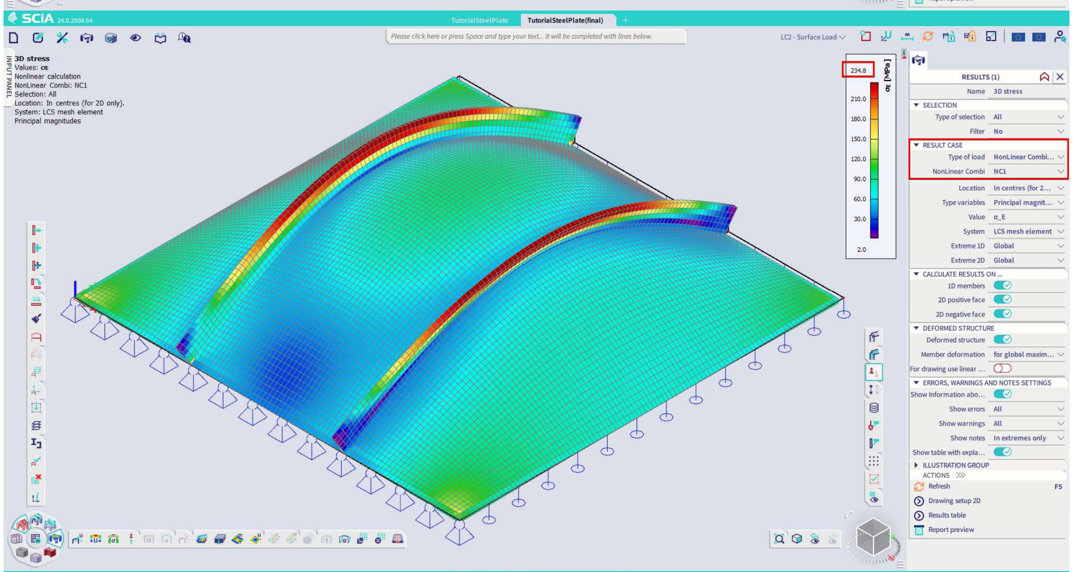


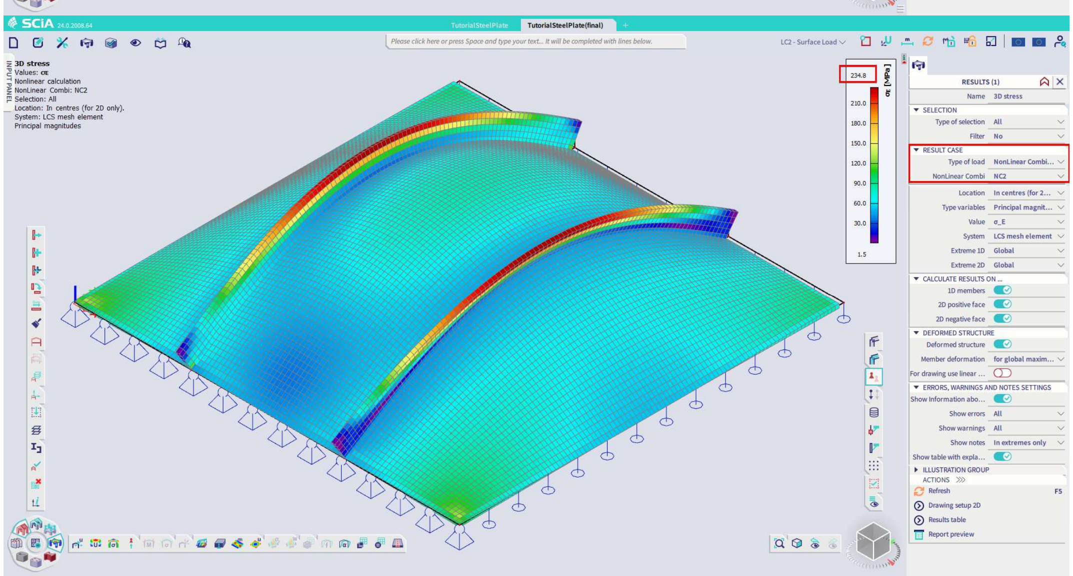


7. Check the plastic stresses and strains

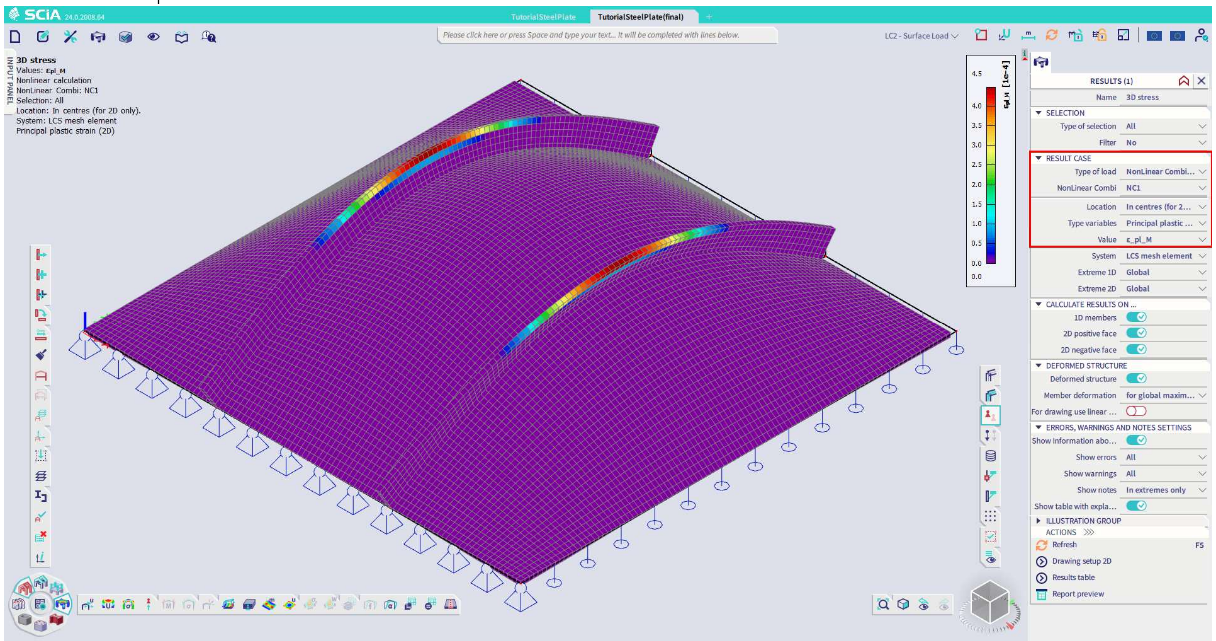
Notice the difference between the linear results (combination CO1) and the nonlinear results taking into account the imperfection(nonlinear combination NC1) as well as the plastic material behavior (nonlinear combinations NC1 and NC2). Below the principal stresses:







And below the plastic strains for NC1:



The plastic strains in this last image can be used to see if the strains are below the given limits in the code of your country. Note that the default unit value is [1e^(-4)] meaning a value of 4,5 is a plastic strain of 0,045%. The results can of course also be shown in table results via the Report preview or Results table.

3D stress

Nonlinear calculation
NonLinear Combi: NC1
Selection: All
Location: In centres (for 2D only). System: LCS mesh element
Principal plastic strain (2D)
Results on 2D member
Extreme 2D: Global

Name	Mesh	Position [m]	Case	εpl_1+ [1e-4] εpl_1- [1e-4]	εpl_2+ [1e-4] εpl_2- [1e-4]	α+ [deg] α- [deg]	εpl_M+ [1e-4] εpl_M- [1e-4]
S3	Element: 10951	0,700 0,505 0,045	NC1	**4,9** 4,6	**-2,4** -2,3	0,03 -0,03	**4,5** **4,5**
S2	Element: 10451	0,300 0,505 0,045	NC1	4,6 **4,9**	-2,3 **-2,4**	-0,03 0,03	4,5 4,5
S1	Element: 1	0,005 0,005 0,000	NC1	**0,0** **0,0**	**0,0** **0,0**	0,00 0,00	**0,0** **0,0**

Note that you'll need to apply a calibration factor α_u according to EN 1993-1-5 Annex C.