SCIA
BY ALLPLAN



\ SCIA ENGINEER

TUTORIAL

STEEL PLATE DESIGN

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 2026 SCIA nv. All rights reserved.

Table of Contents

Table of Contents **3**

Introduction **4**

Steel plate design according to EN1993-1-5 ..... **5**

1. Model the steel structure within SCIA Engineer using finite elements 5

2. Insert loads 7

3. Create linear stability combinations 8

4. Perform the linear stability analysis and check the buckling shape 9

5. Insert this buckling shape as an imperfection for the second order analysis 10

6. Run the nonlinear analysis 12

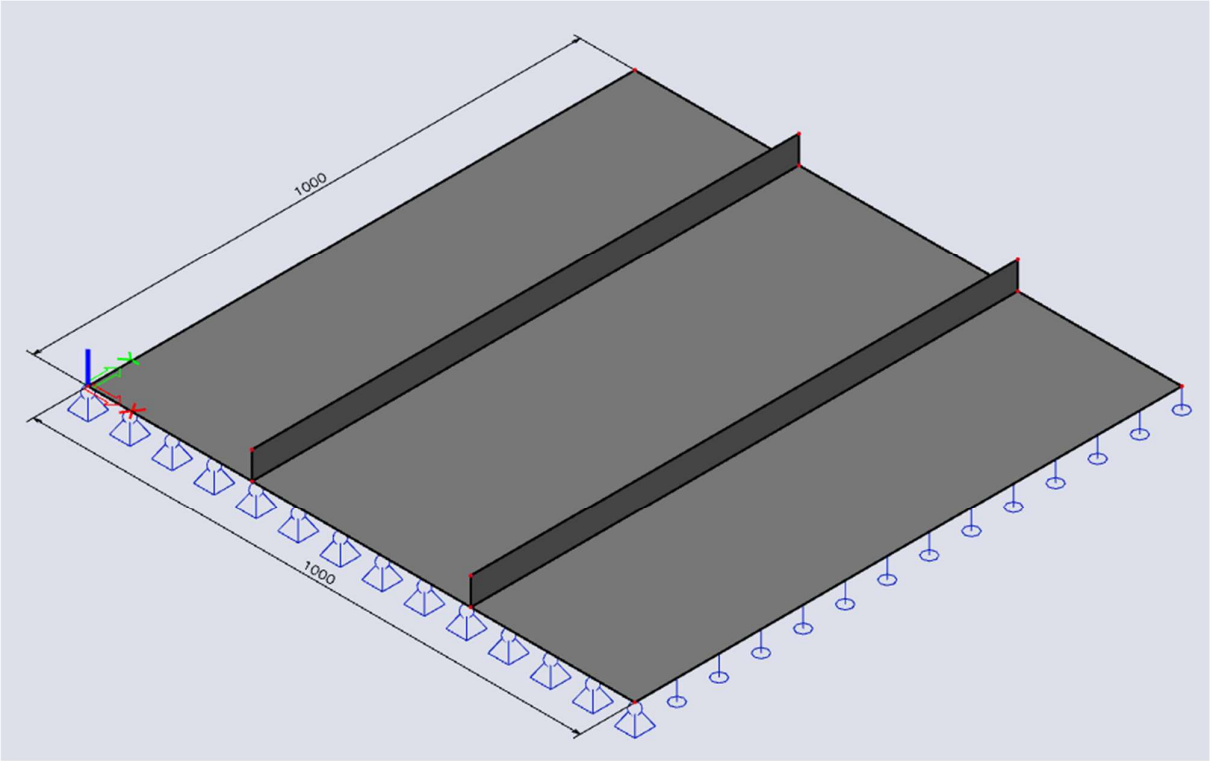
7. Check the plastic stresses and strains 13

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

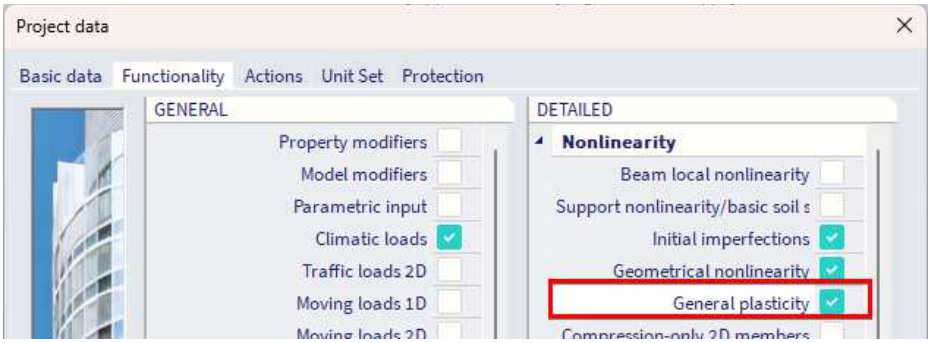
1. Model the steel structure within SCIA Engineer using finite elements

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).

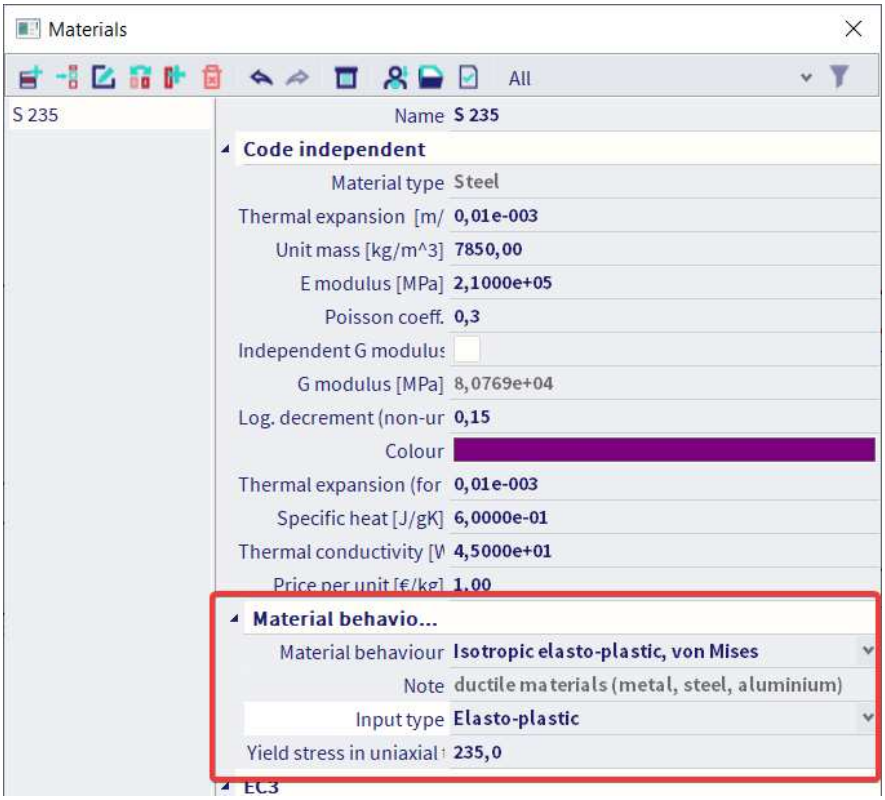
More info about the module 'General Plasticity' can be found in our online help:

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).



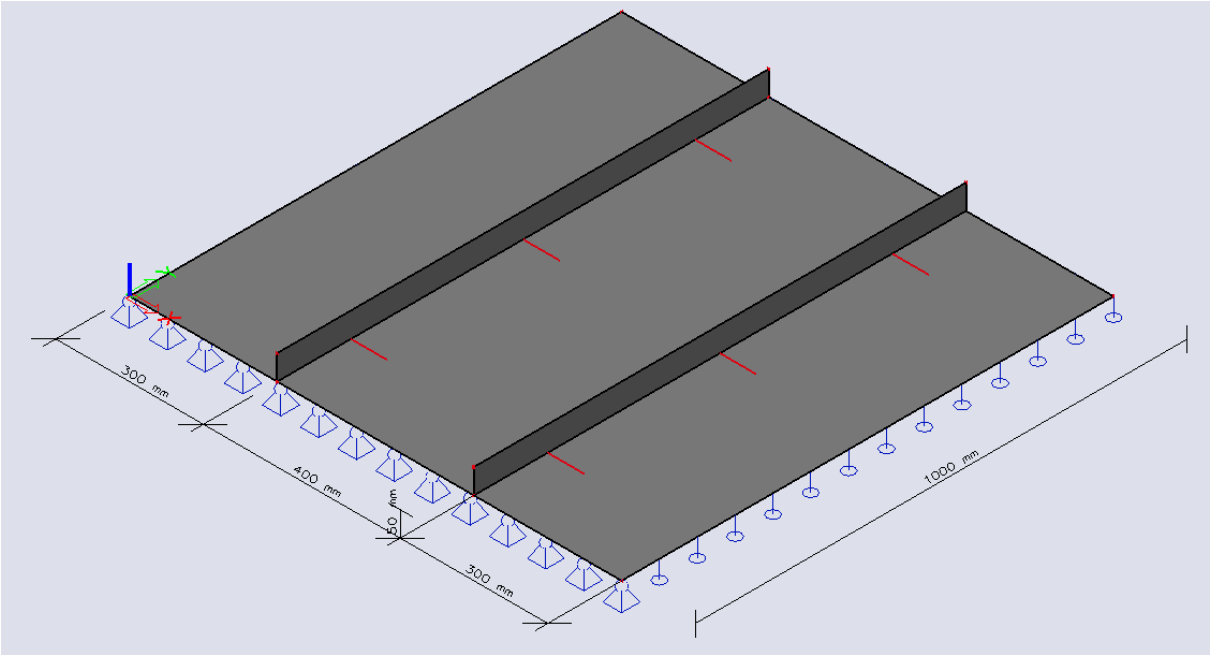
Now you can activate the plasticity behaviour in the material properties. Go to 'Libraries' in the main menu and choose 'Materials from the dropdown menu:



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

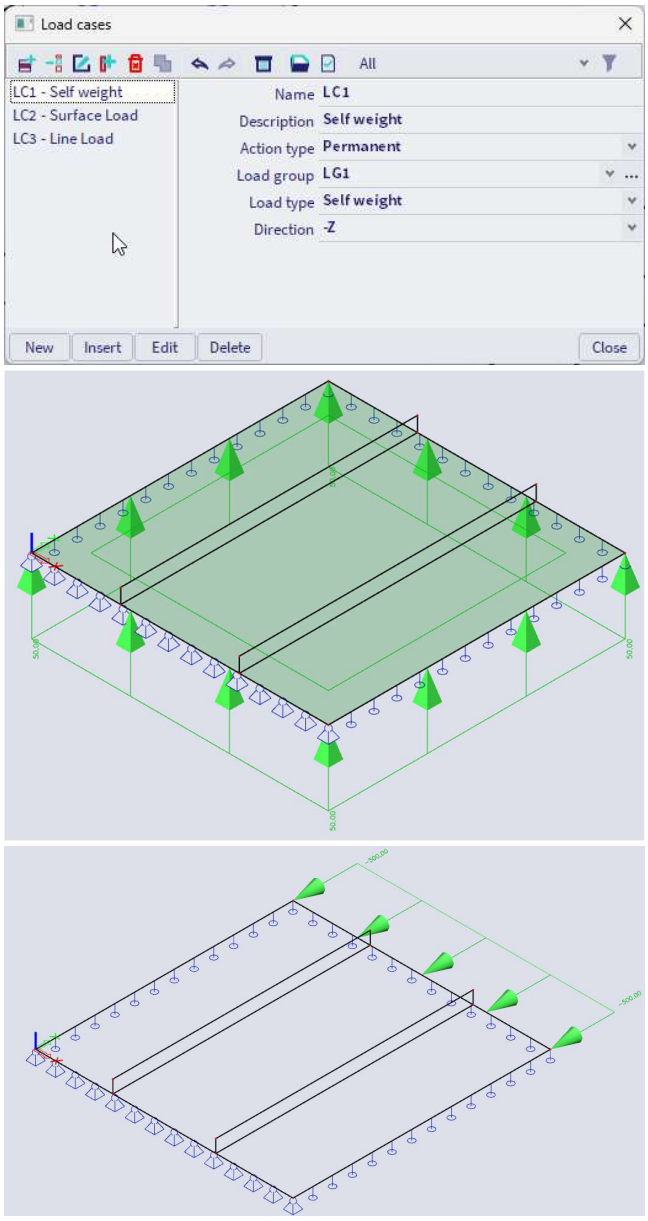
On one face, parallel to the loaded direction, at 1/3 and 2/3 of the width you can model two stiffeners of 0.05 m height in the same material quality and with the same thickness of 10 mm.

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. Set the X and Y directions for one edge to Fixed to avoid that the model becomes unstable.

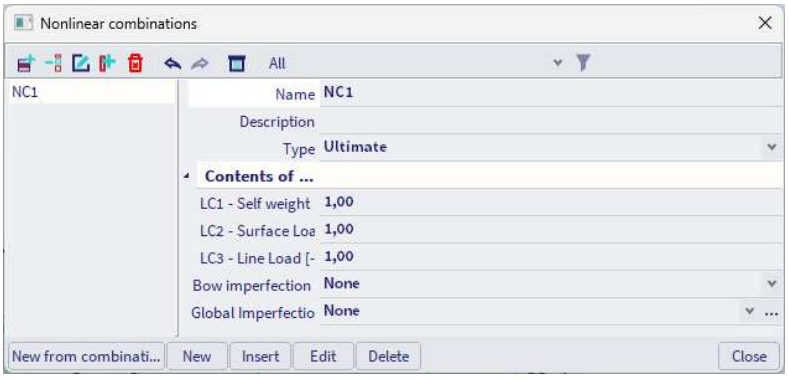


2. Insert loads

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 containing these three load cases.



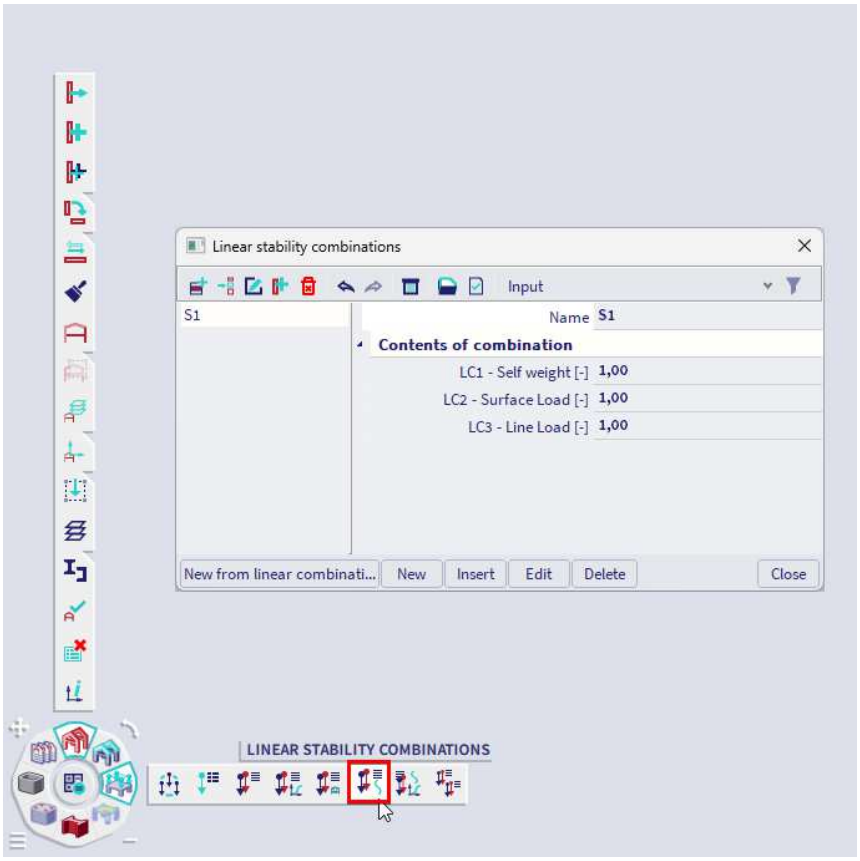
Put all load cases also in a nonlinear combination (to take into account plasticity of the material). For a real structure you will define specific coefficients for each load case to perform a ULS design.



3. Create linear stability combinations

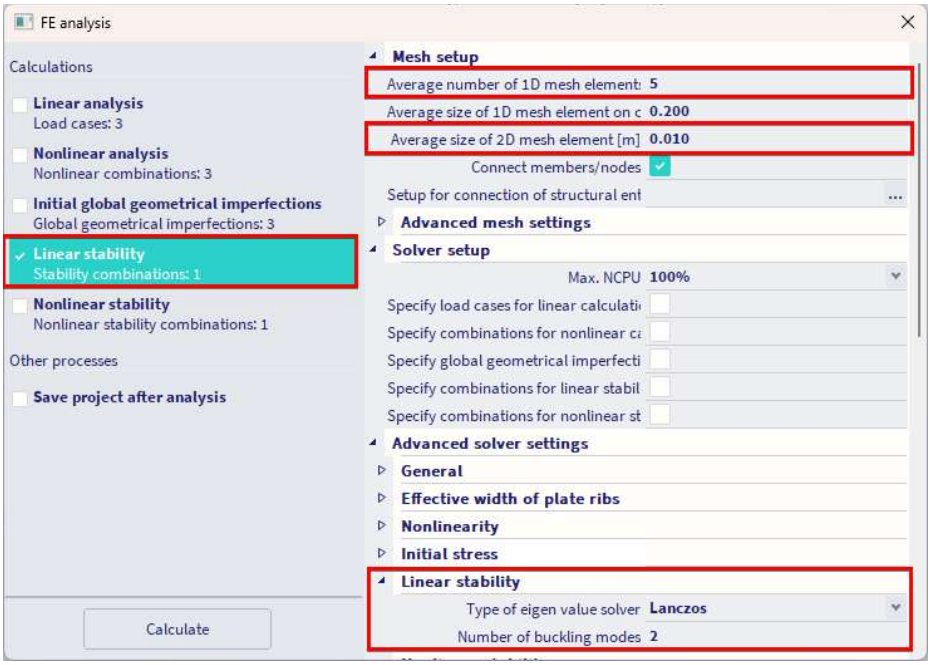
A stability analysis within SCIA Engineer is an elastic buckling shape analysis. These elastic buckling shapes need to be inserted as being imperfections for the second order analysis according to EN1993-1-5.

Note: in this example, load coefficients of 1.0 are 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.

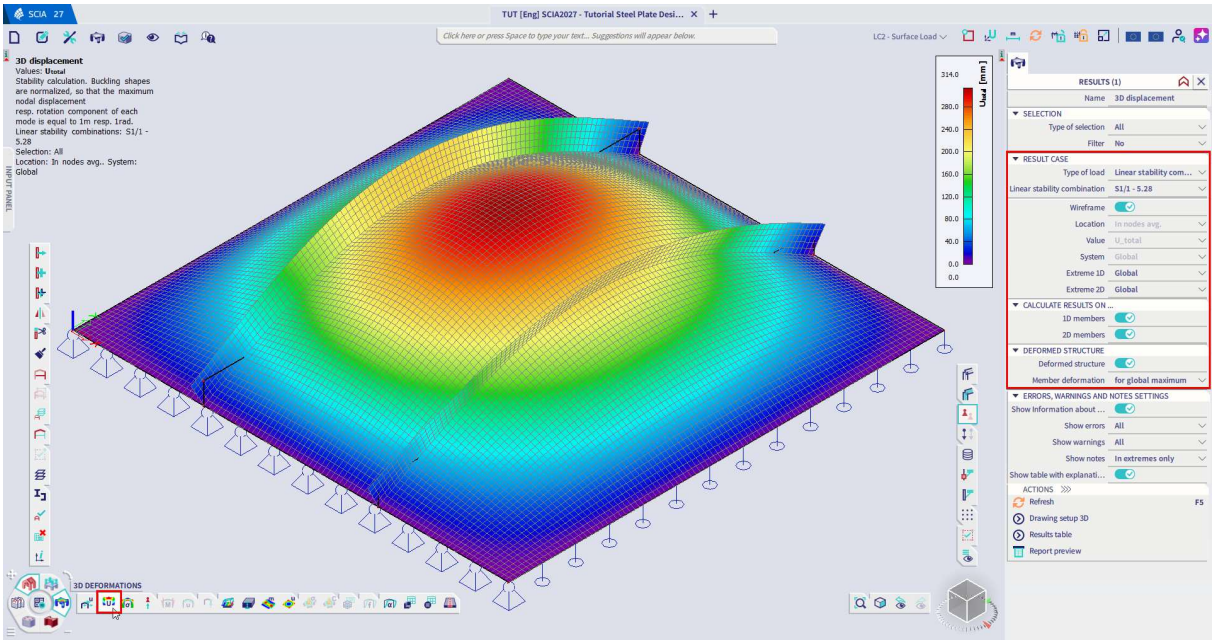


4. Perform the linear stability analysis and check the buckling shape

Go to the process toolbar and press the Calculate button in the middle of the process wheel. Here the 1D mesh setting (a value of 5 or higher is recommended in case 1D members are present in the model) doesn't matter much for a plate model. What matters is the 2D element size: enough elements per buckling half-wave, and several elements over the 50 mm stiffener height (here 0.010 m would be a good mesh to start with) 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.

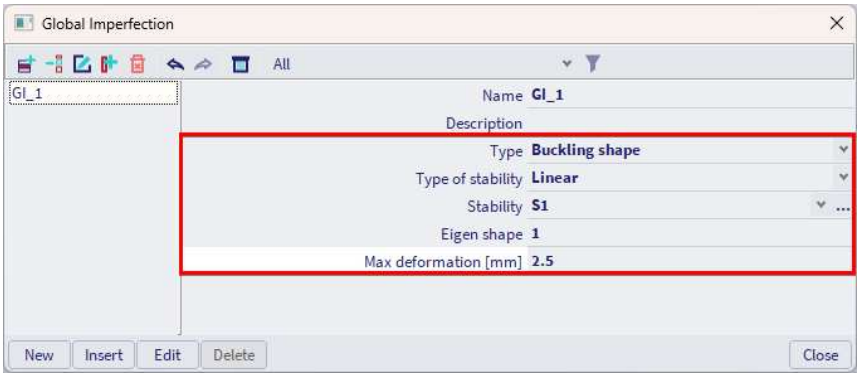


Note: the deformations from a stability combination are normalized values. So these aren't actual displacements but can be used to show the relative deformation between the different elements in your model.

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.

As stated in step 4, the buckling shape is shown for a normalized value. You have to define an amplitude if you want to use this shape as an imperfection. You can chose an imperfection based on EN1993-1-5, Annex C, Table C.2 (see next page). For the global mode you can use an imperfection value of min(a,b)/400 or 1000 mm / 400 = 2.5 mm.



Then you can apply this imperfection to the nonlinear combination NC1. You can create a second nonlinear combination NC2 without imperfection to compare with NC1 (with imperfection).



In practice, use ULS load coefficients.

Since SCIA Engineer 2027 it's also possible to combine imperfections (a leading imperfection and accompanying imperfections) via the Global imperfection type Superposed eigenschapes or inclination. First create two global imperfection based on the second (accompanying) buckle mode and imperfection value 400 mm / 200 = 2.0 mm and -2.0 mm. Then combine GI_1 with GI_2 and GI_1 with GI_3.



You can assign this imperfection to nonlinear combination NC3.
For a real model you can check both signs of the buckling shape.

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

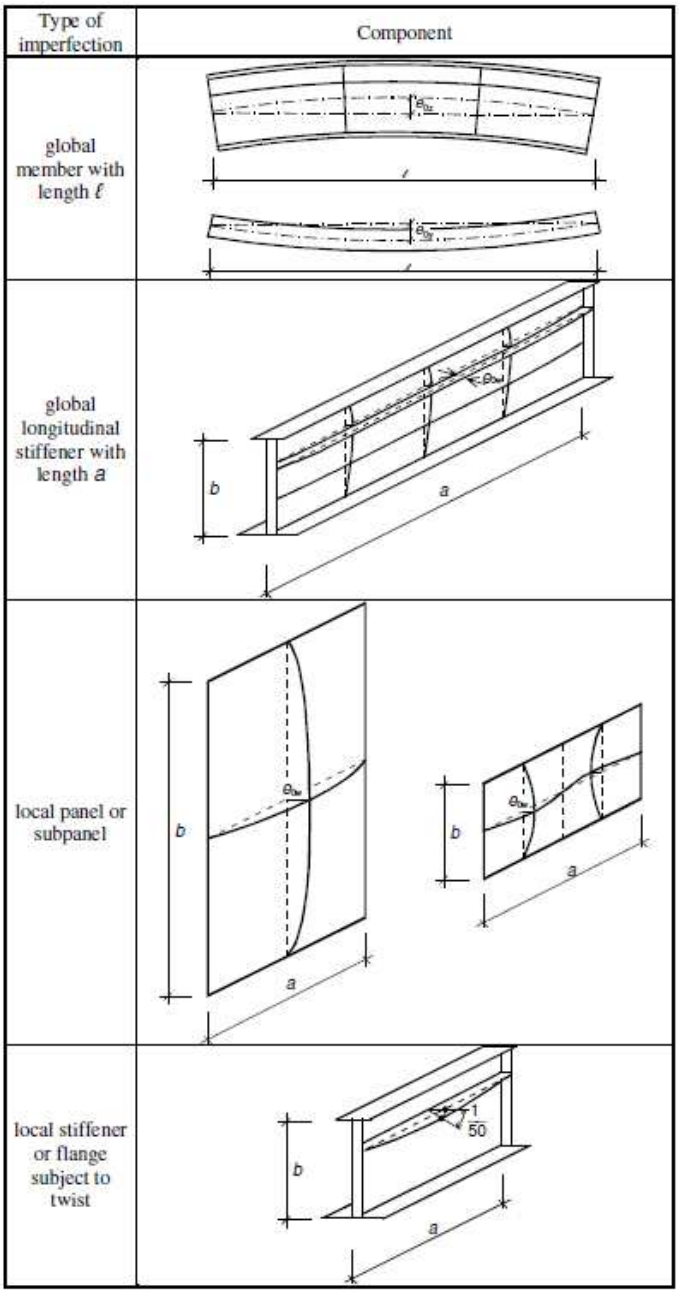
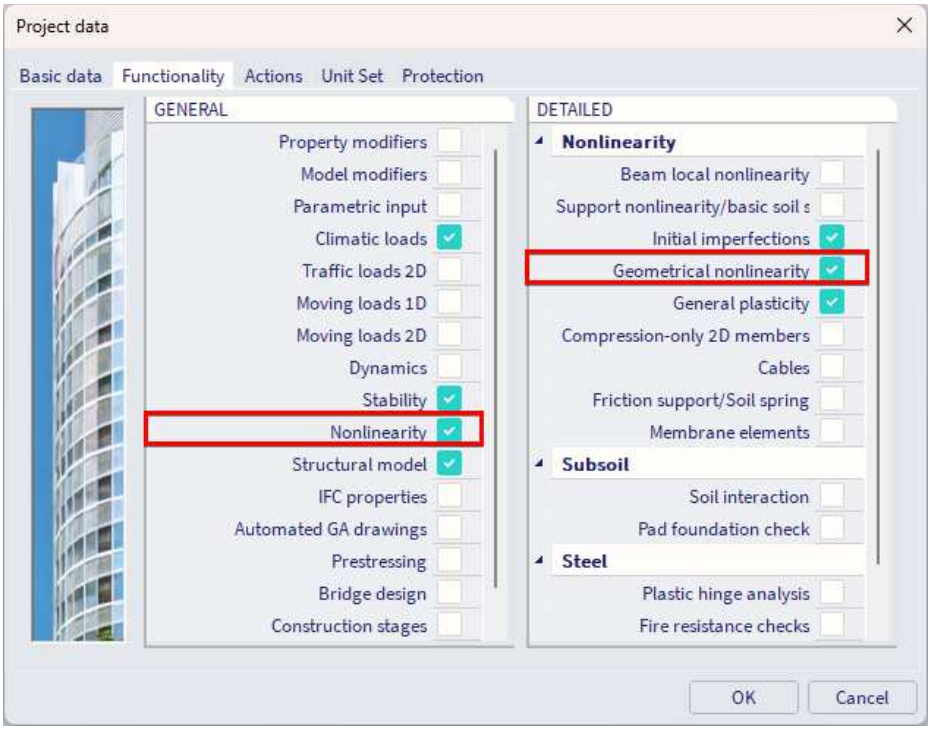
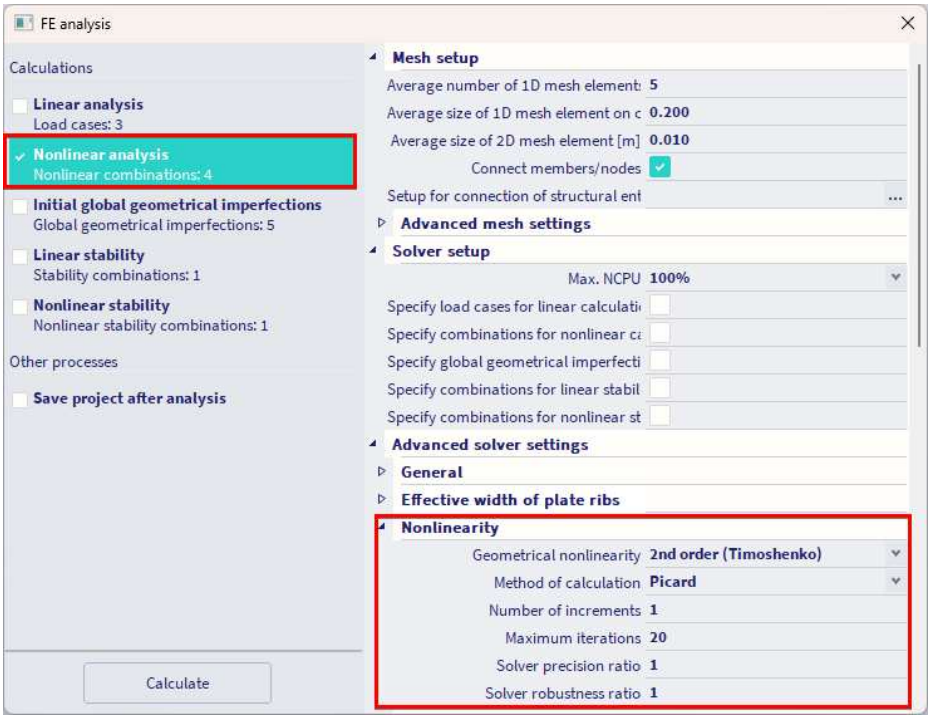


Figure C.1: Modelling of equivalent geometric imperfections

6. Run the nonlinear analysis

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

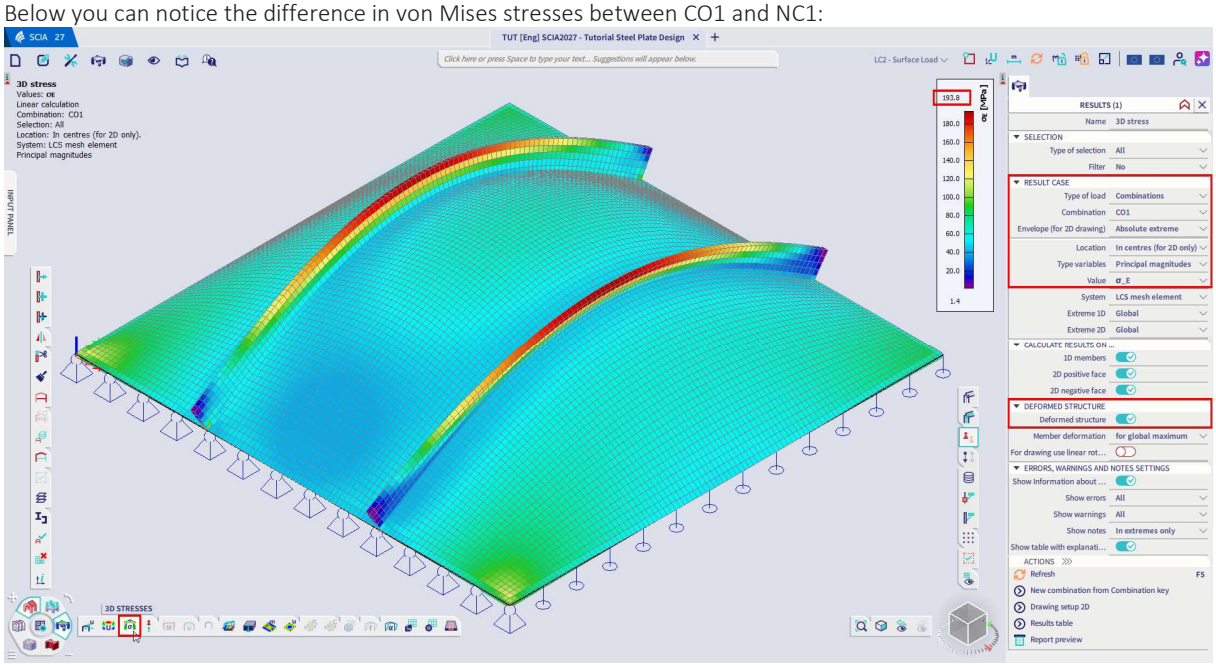


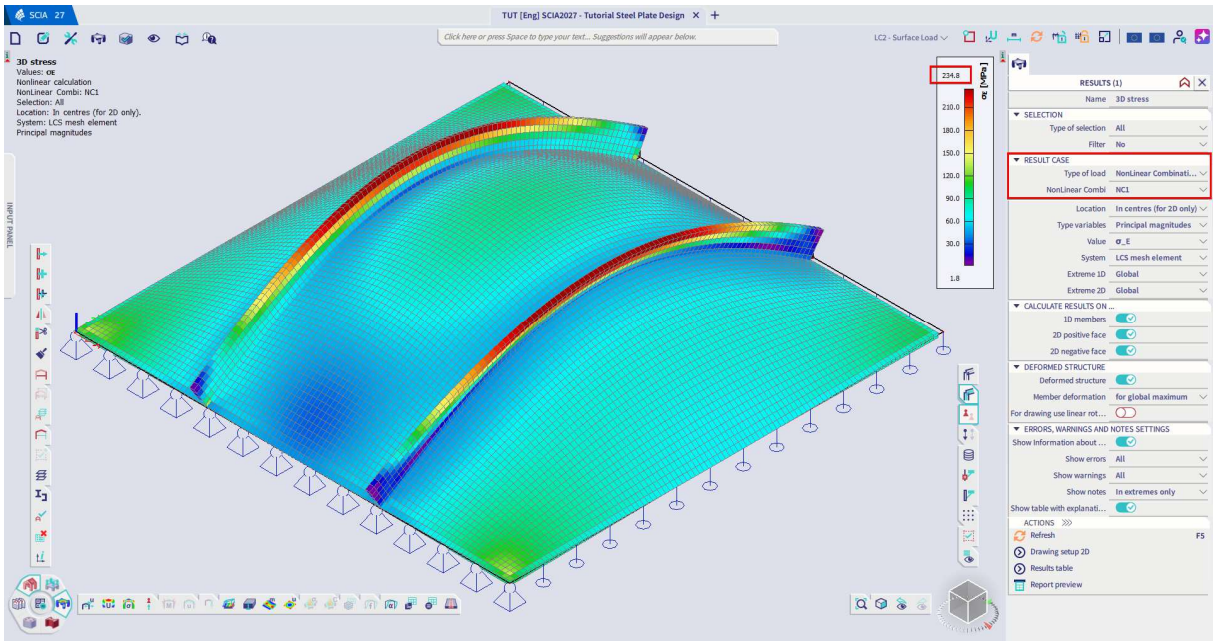


7. Check the plastic stresses and strains

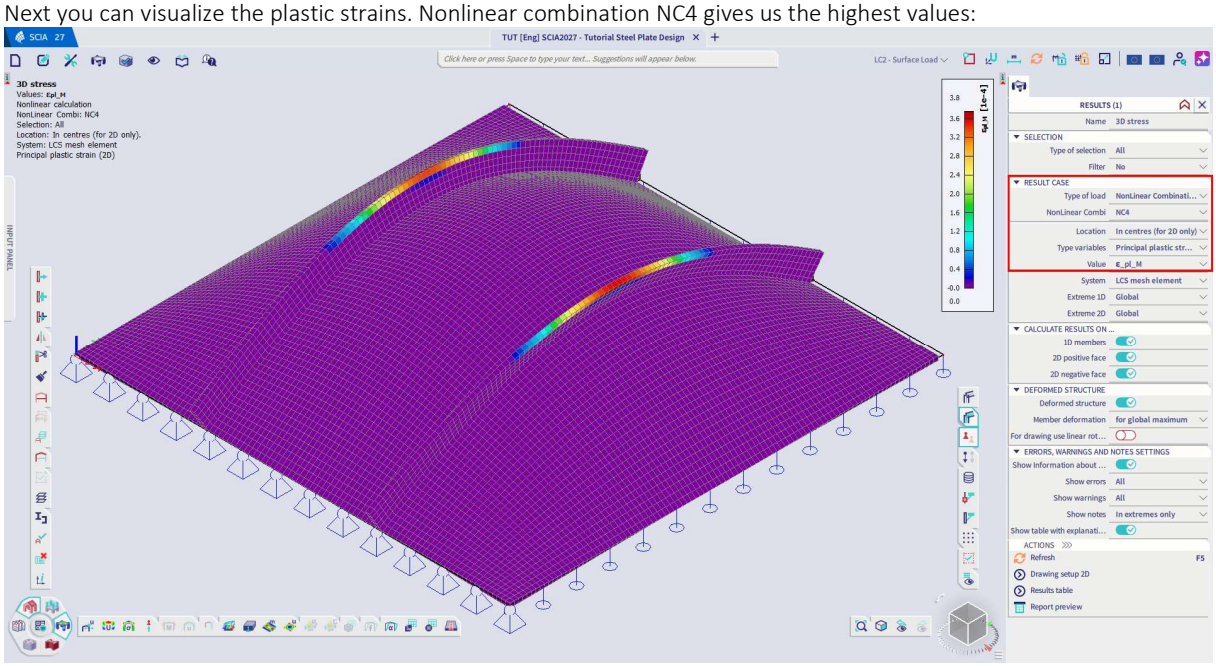
Notice the difference between the linear results (combination CO1) and the nonlinear results taking into account the plastic material behaviour and imperfections (nonlinear combinations NC1, NC3 and NC4). For the three nonlinear combinations you can notice that the von Mises stresses are maximal at 235 MPa, which makes sense since plasticity of the material limits the stress to the yield strength in case that the stresses can be redistributed over the plate.

Below you can notice the difference in von Mises stresses between CO1 and NC1:





Next you can visualize the plastic strains. Nonlinear combination NC4 gives us the highest values:



The plastic strains in this last image can be used to check if the strains are below the given limits in the design code of your country (EN 1993-1-5, Annex C C.8 recommends 5% as principal strain). Note that the default unit value is [1e-4] meaning a value of 3.8 is a plastic strain of 0.038%. The results can of course also be shown in table results via the Report preview or Results table.

3D stress

Nonlinear calculation
NonLinear Combi: NC4
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	ε_{pl_1+} [1e-4] ε_{pl_1-} [1e-4]	ε_{pl_2+} [1e-4] ε_{pl_2-} [1e-4]	$\alpha+$ [deg] $\alpha-$ [deg]	ε_{pl_M+} [1e-4] ε_{pl_M-} [1e-4]
S3	Element: 10954	0.700 0.535 0.045	NC4	**3.6** 3.2	**-1.7** -1.6	0.11 -0.11	**3.8** 3.4
S2	Element: 10454	0.300 0.535 0.045	NC4	3.2 **3.6**	-1.6 **-1.7**	-0.11 0.11	3.4 **3.8**
S1	Element: 1	0.005 0.005 0.000	NC4	**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.