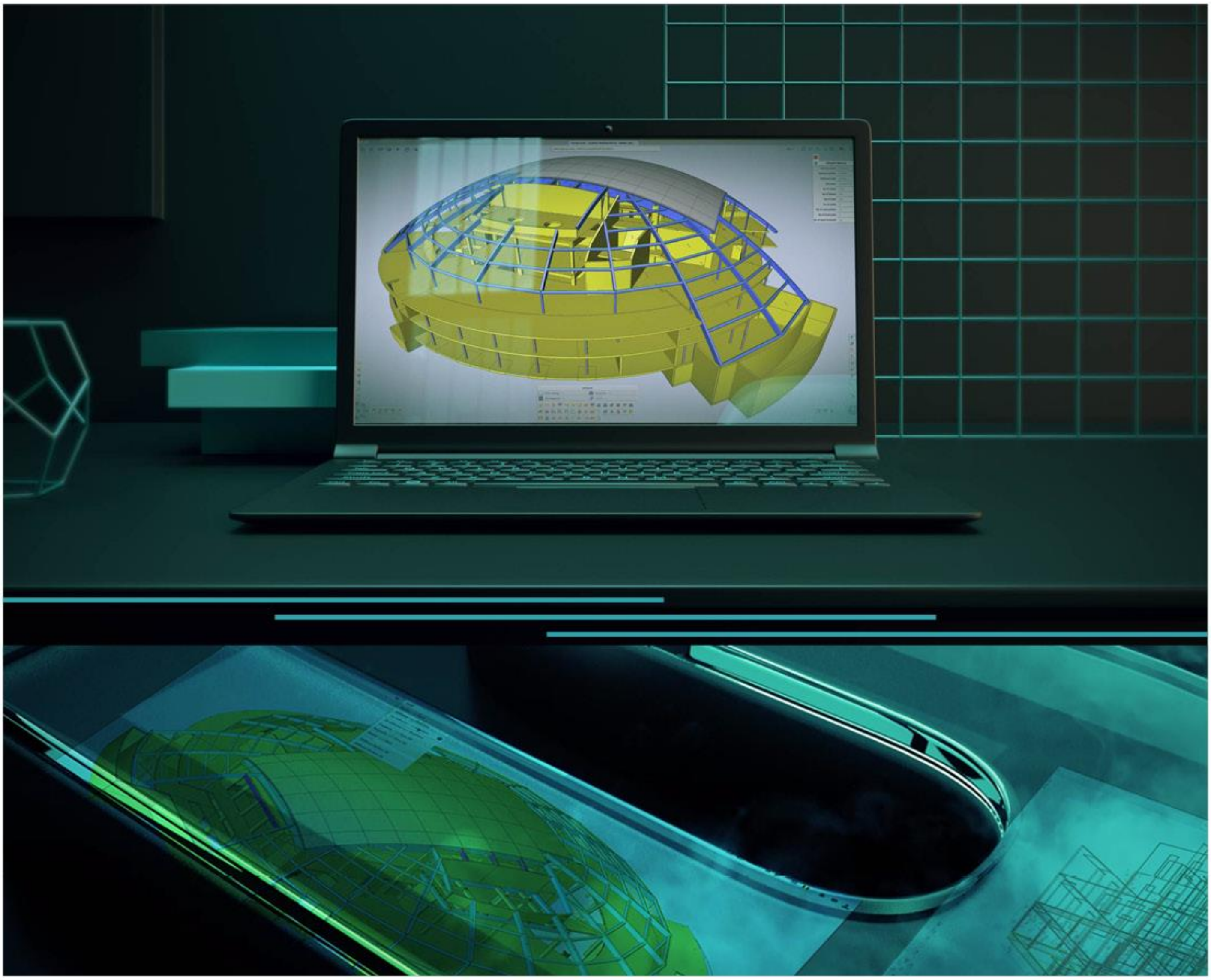


TUTORIAL

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

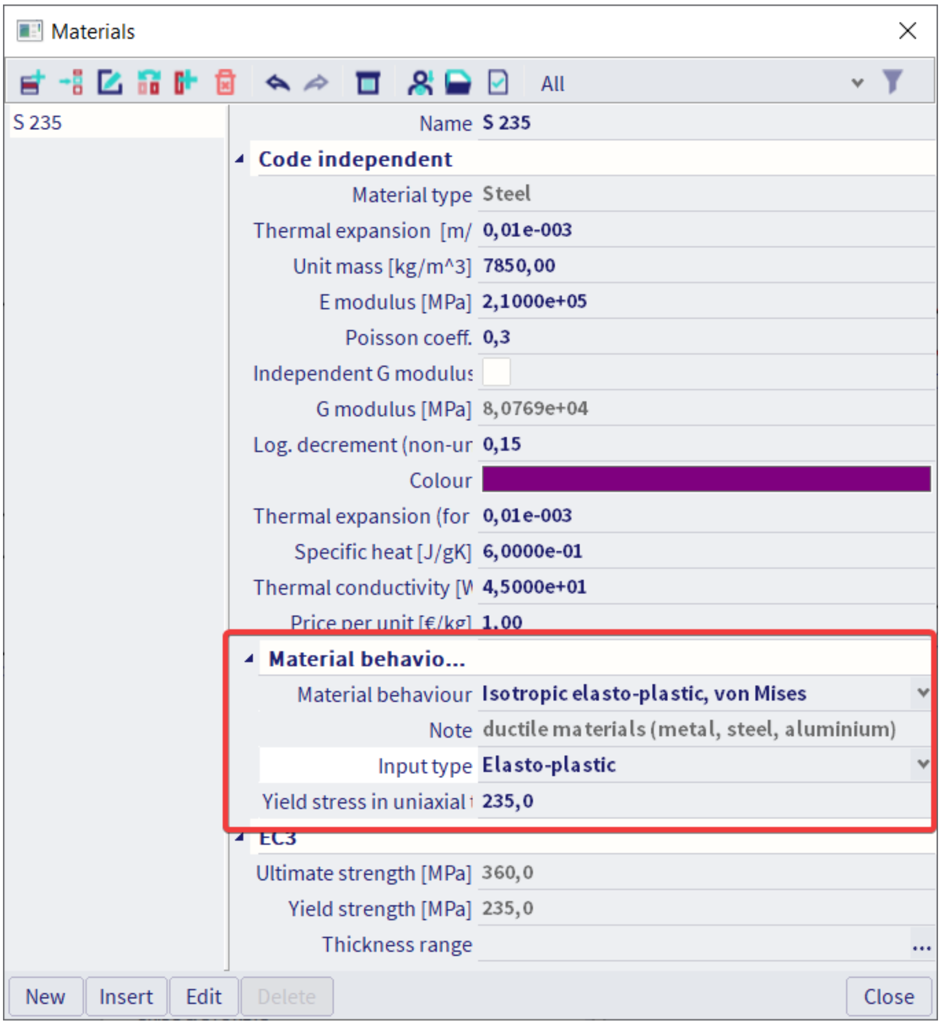
1) Model the steel structure within SCIA Engineer using finite elements.

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' branch 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/#kik/functionalities/nonlinearities/general_plasticity/general_plasticity.htm

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

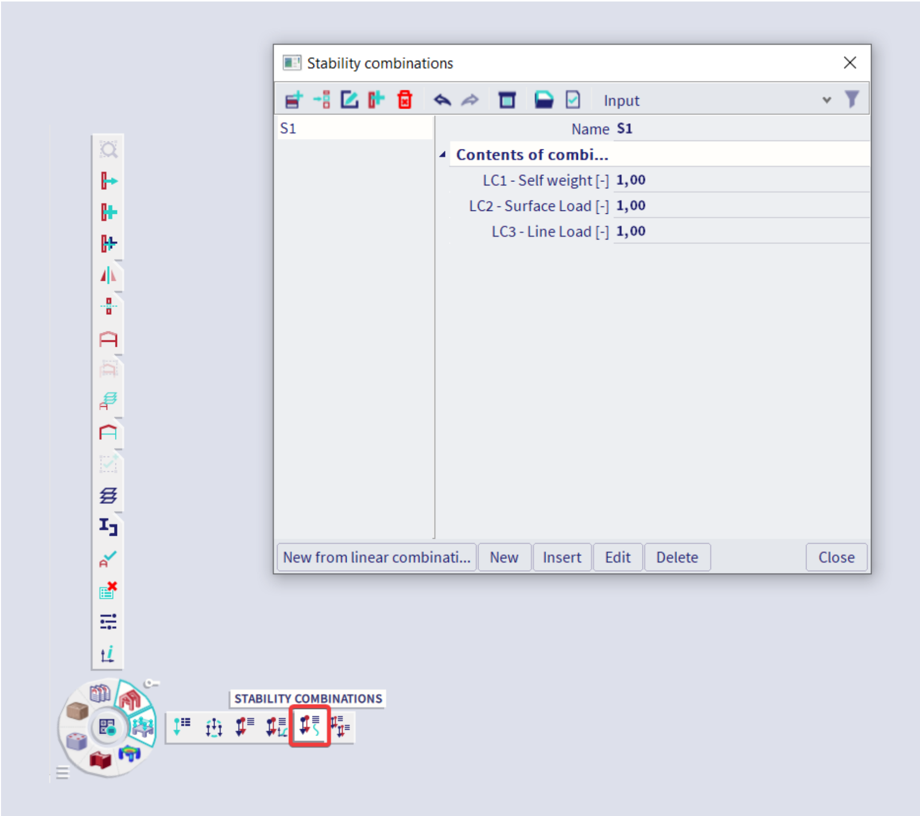


2) Insert the loads.
In the demo file already some loads and combinations are predefined for you.

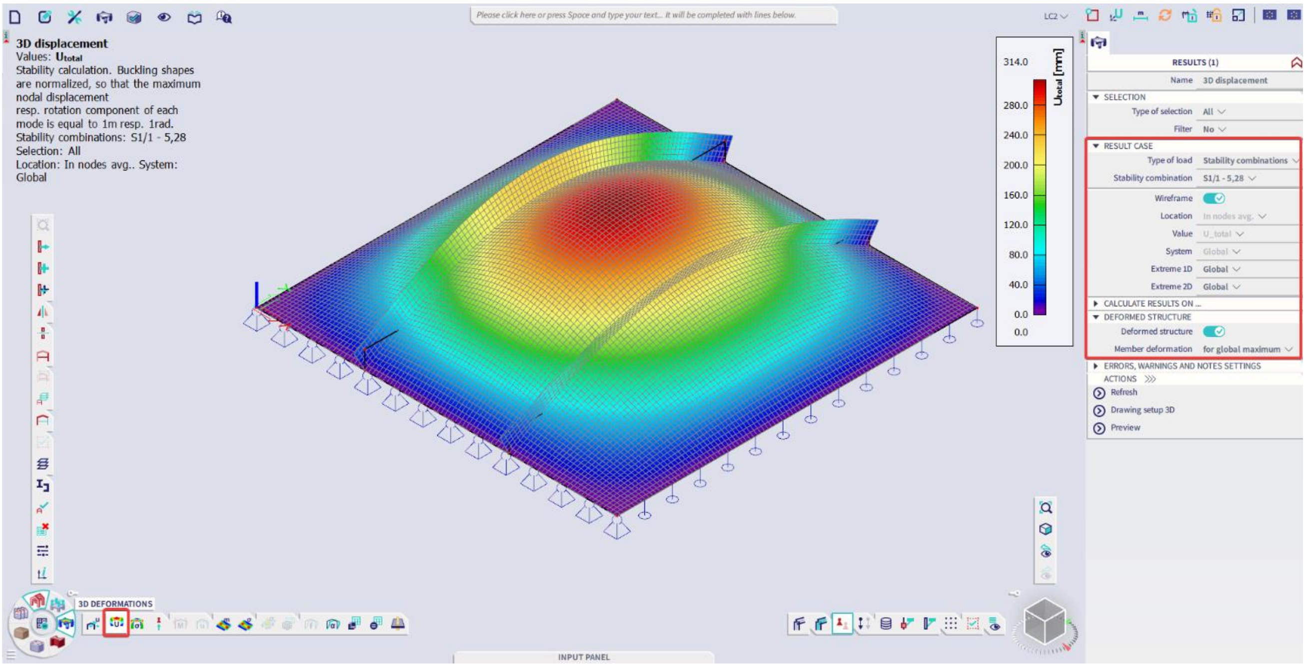
3) Create stability combinations.

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.



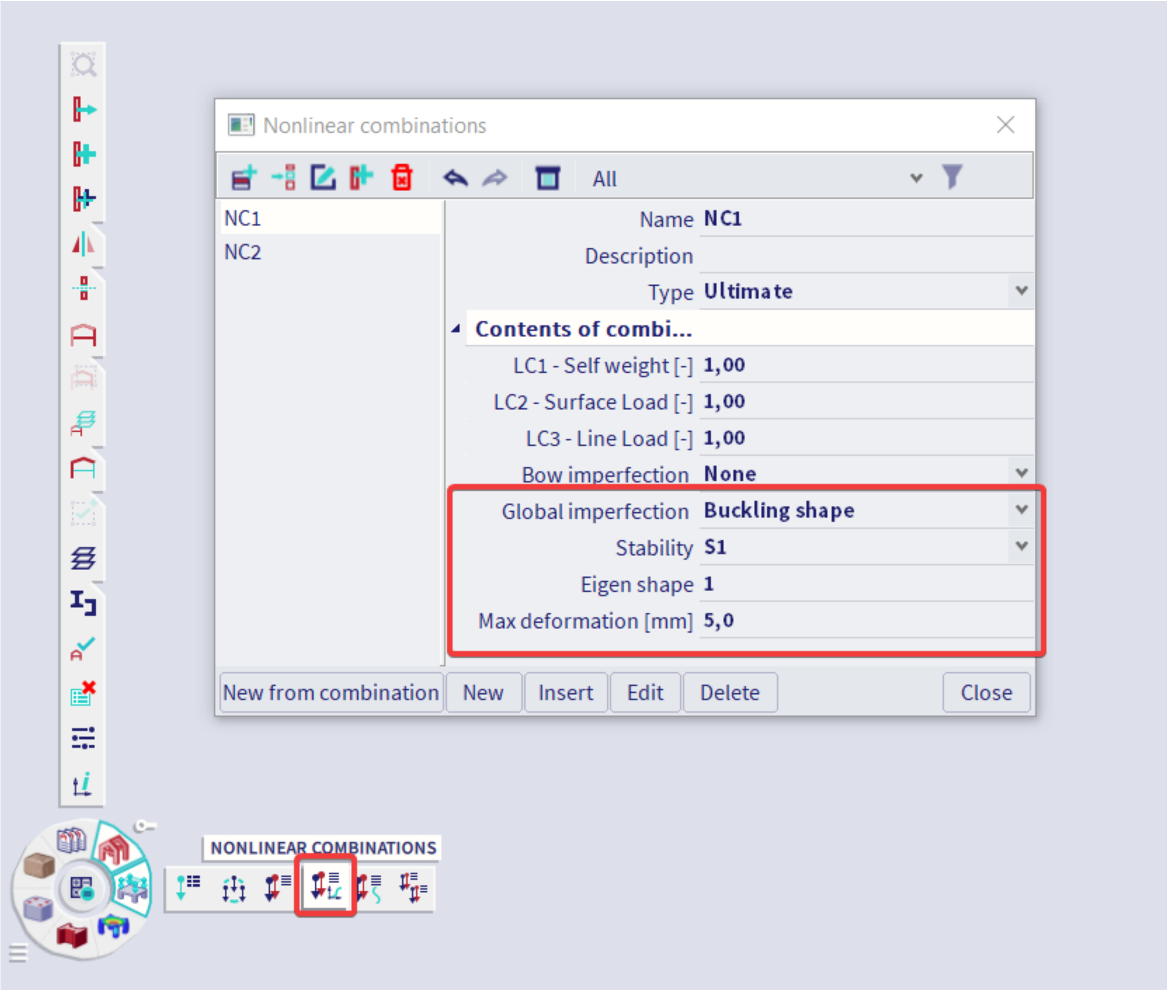
4) Perform the stability analysis and check the buckling shape.



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

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.

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

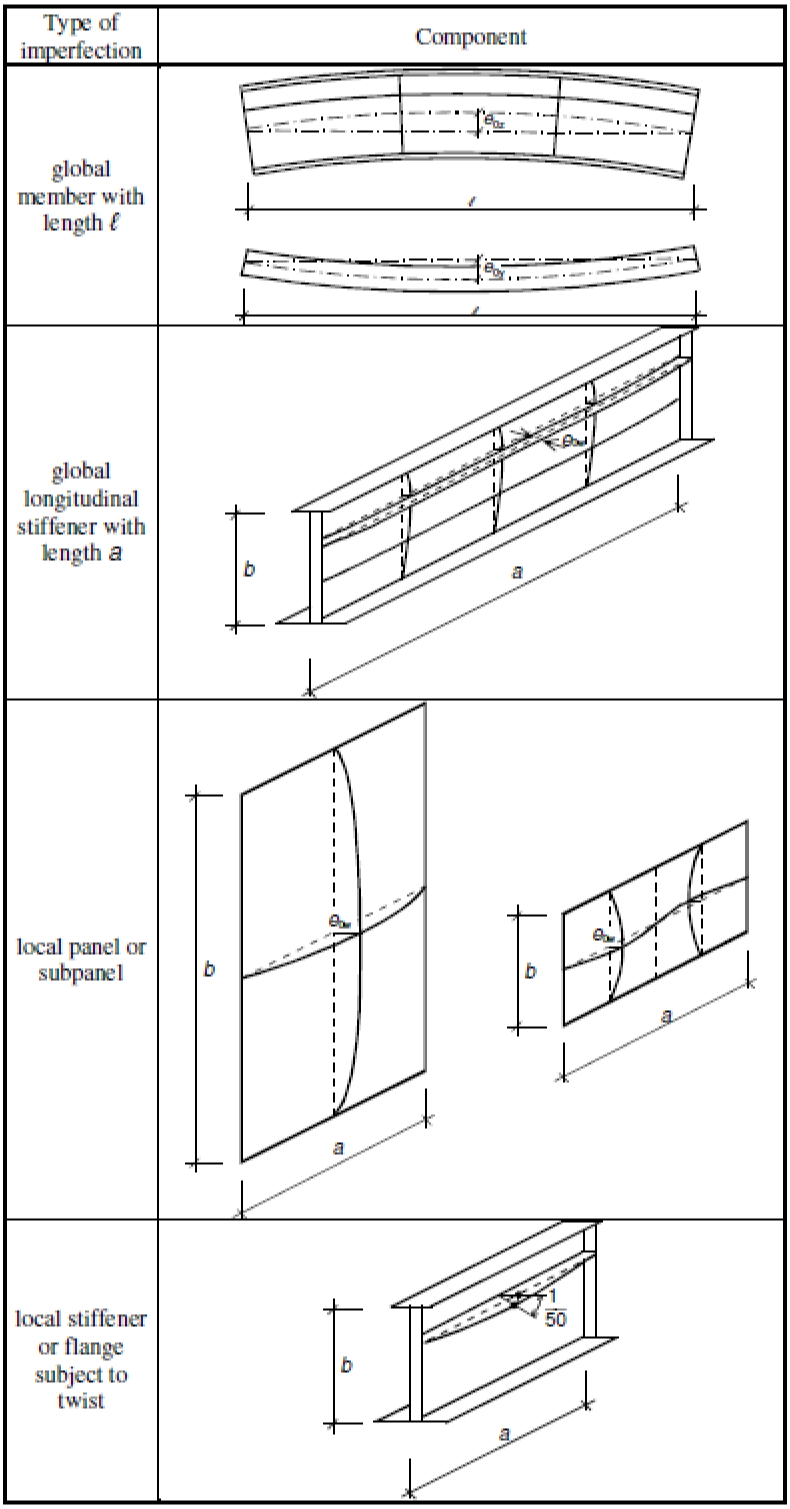


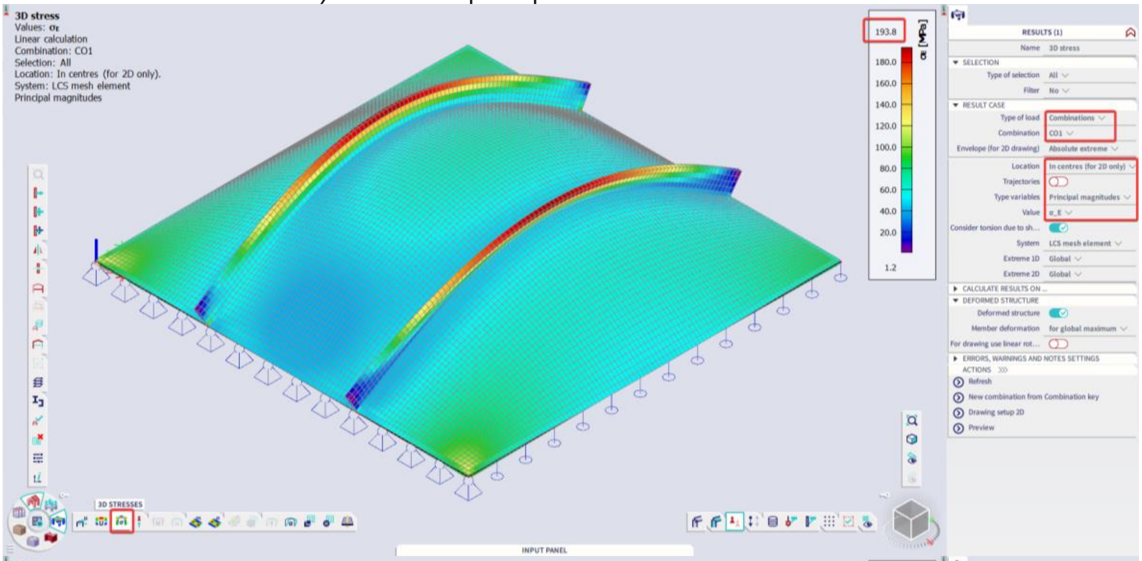
Figure C.1: Modelling of equivalent geometric imperfections

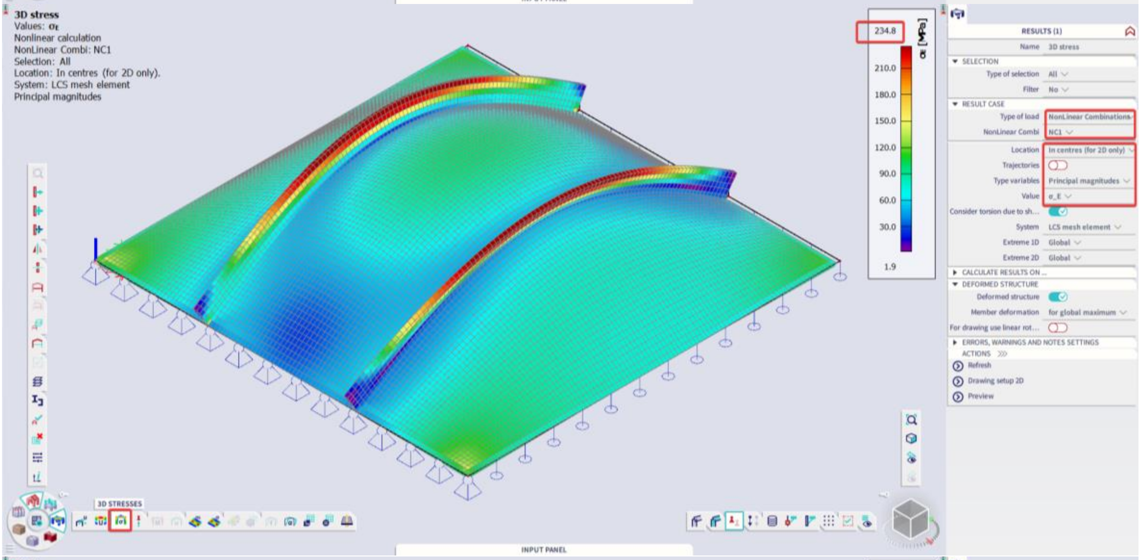
6) Run the nonlinear analysis

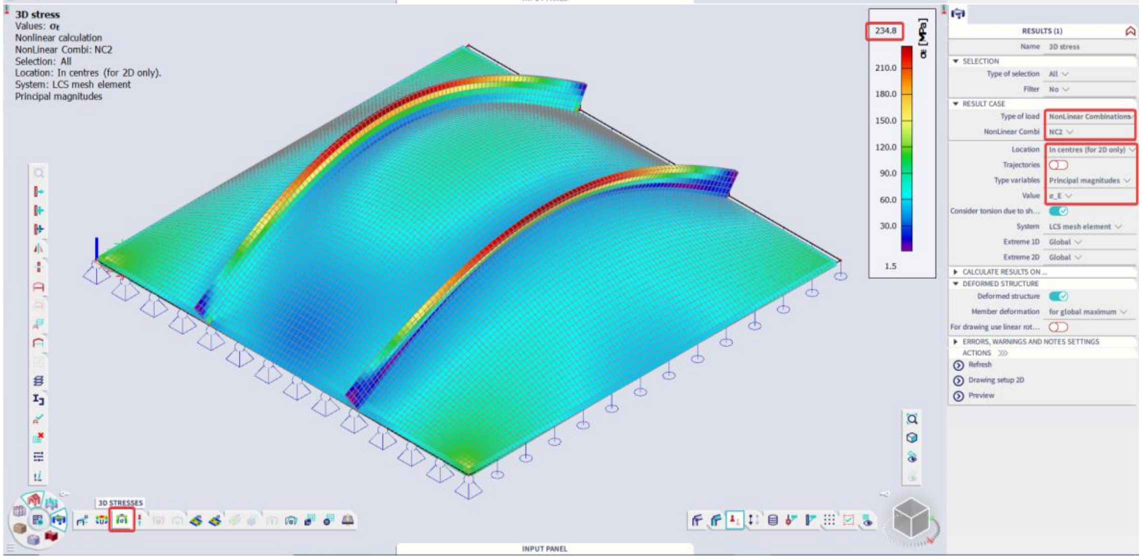
Make sure that you activate the second order analysis with the functionality ‘Geometrical nonlinearity’ in the ‘Project settings’ enabled.

7) Check the plastic stresses and strains.

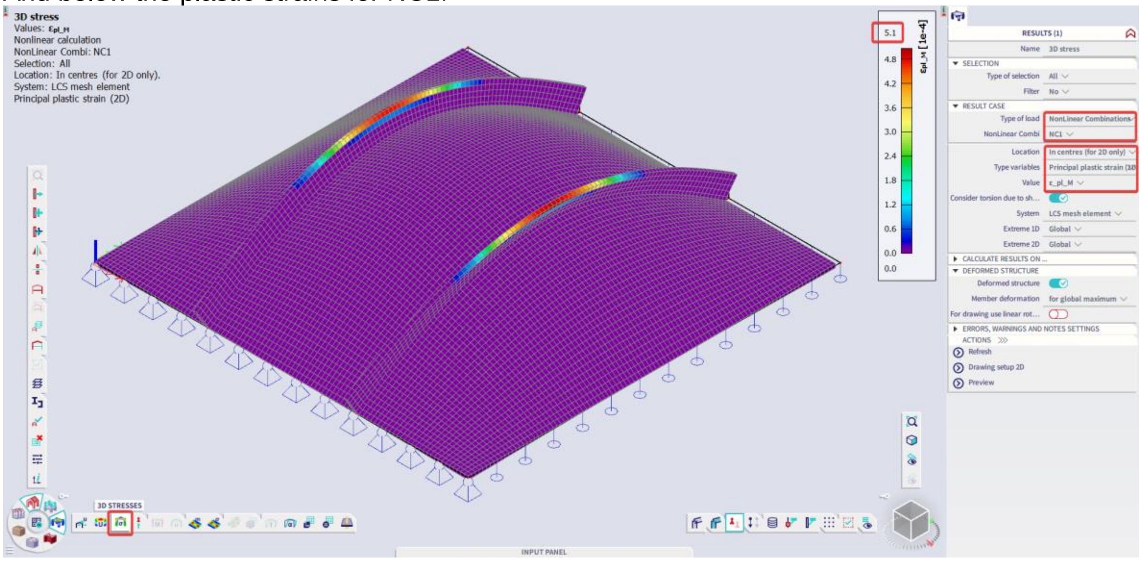
Notice the difference between the linear results (combination CO1) and the nonlinear results taking into account the imperfection(nonlinear combination NC1) as well as the plastic material behavior (nonlinear combination NC1 and NC2). Below the principal stresses:







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 5,1 is a plastic strain of 0,051%. The results can of course also be shown in table results.