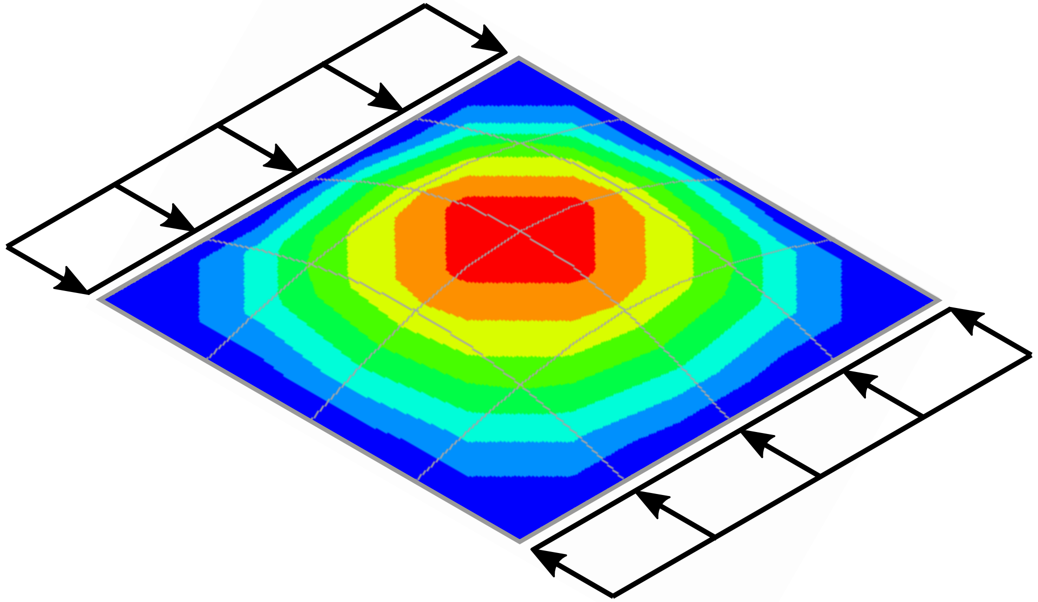


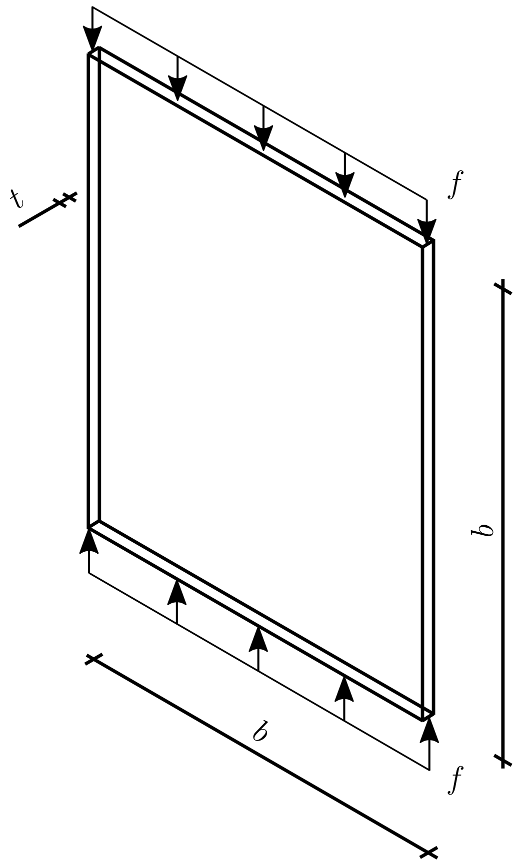
Buckling Analysis of a Thin Plate

DIANA FEA

Outline

1 Description

In this example, the buckling behavior of a thin square plate is analyzed. Figure 1 shows that the plate is subjected to a uniform compression load f.



$b = 2000$ mm

$t = 10$ mm

$E = 2\text{x}10^5$ N/mm^2

$\nu = 0.3$

Figure 1: Thin plate under uniform compression loading

Beyond a certain load value f_{crit} (i.e., the critical load) the plate becomes unstable and buckles. According to Timoshenko and Gere [1], the critical buckling load for this example is:

$$f_{\text{crit}} = \frac{4\pi^2 D}{b^2} \qquad \text{with } D = \frac{Et^3}{12\,(1-\nu^2)},$$

where b is the length of the plate, t the thickness, E the Young's modulus and ν the Poisson's ratio. With reference to Figure 1, for the present example we have $f_{\text{crit}} = 180.76$ N/mm.

[1] Timoshenko and Gere, *Theory of Elastic Stability*, 1963

2 Modeling Approach

As both membrane stresses and bending stresses are of importance, shell elements are used. No use is made of symmetry conditions and thus the entire domain is subdivided in finite elements. A rather coarse mesh of 4×4 elements is used and quadratic square elements will be employed. The plate is supported in Z direction along its edges. In-plane displacements in the mid-points of the edges are supported as shown in Figure 2.

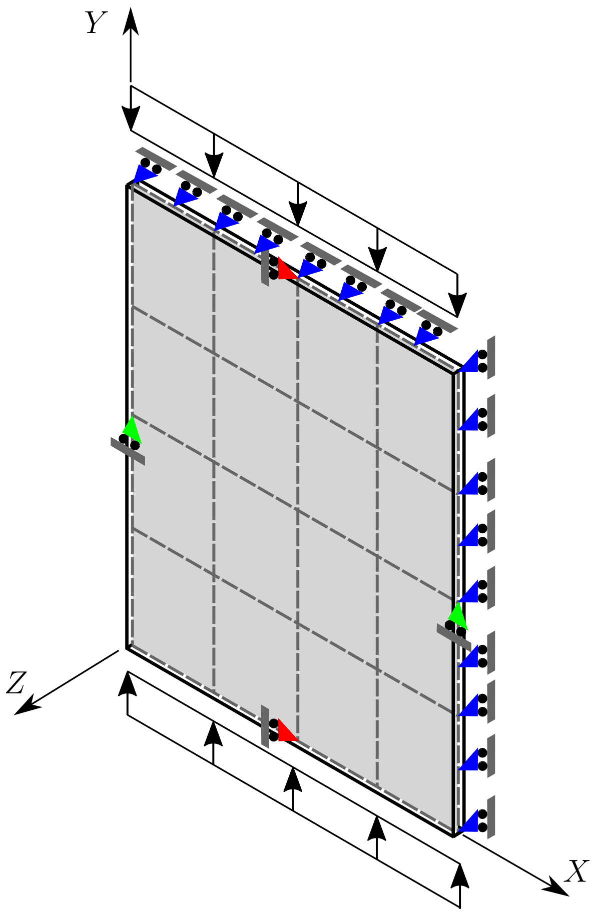


Figure 2: Model used in the finite element analysis (the supports in the X, Y and Z directions are colored in red, green and blue, respectively)

Note that in spite of the fact that the geometry of the structure is two-dimensional, we should perform a three-dimensional structural analysis. That is because the model shows a phenomenon in the third dimension (i.e., the out-of-plane displacement).

3 Finite Element Model

For the modeling session we start a new project for structural analysis [Fig. 3]. The dimensions of the domain for the three-dimensional model are set equal to 10 m. We will use quadratic quadrilateral finite elements.

DianaIE

Main menu → File → New [Fig. 3]

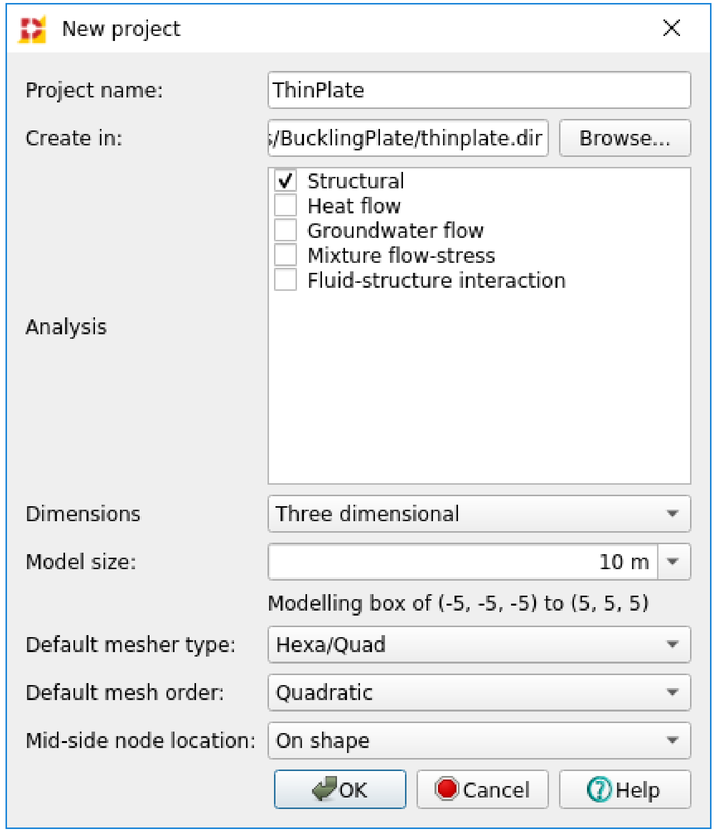


Figure 3: New project dialog

We choose millimeter for length, ton for mass, and newton for force.

DianaIE

Geometry browser → Reference system → Units [Fig. 4]
Property Panel [Fig. 5]

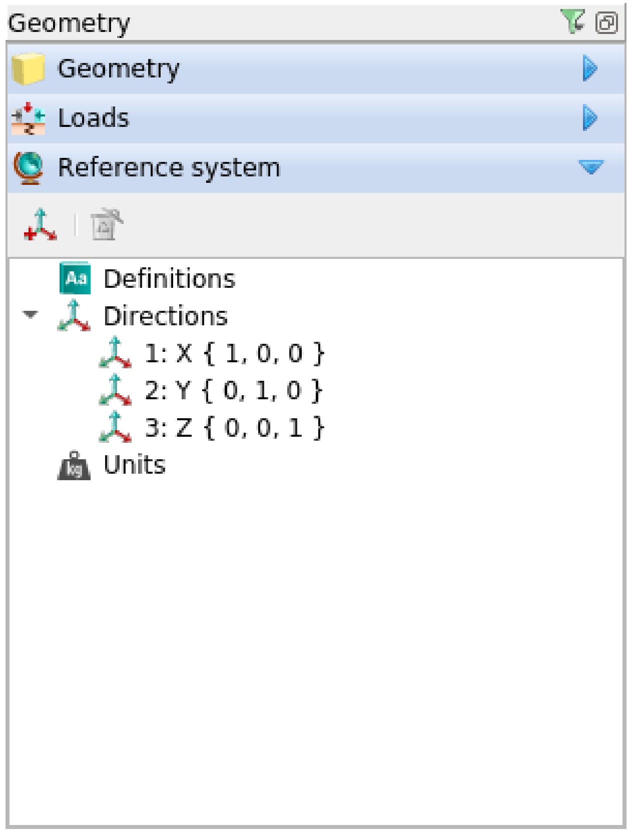


Figure 4: Geometry browser

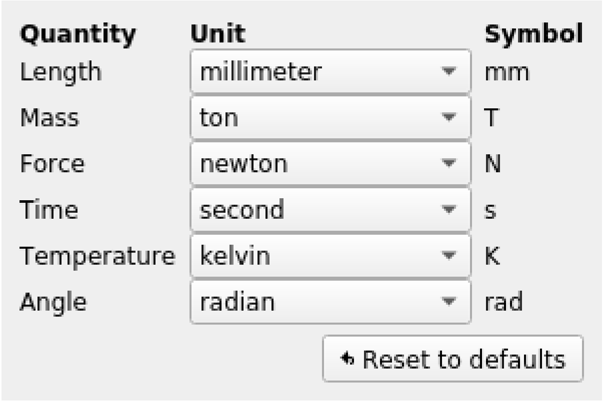


Figure 5: Property panel - units

3.1 Geometry

We create the *PlateZ* sheet. The coordinates used for this geometry are in Table 1.

DianaIE

Main menu → Geometry → Create → Add polygon sheet [Fig. 6]
Main menu → Viewer → Viewpoints → Top view

PlateZ		
0	0	0
1000	0	0
2000	0	0
2000	1000	0
2000	2000	0
1000	2000	0
0	2000	0
0	1000	0

Table 1: Plate coordinates

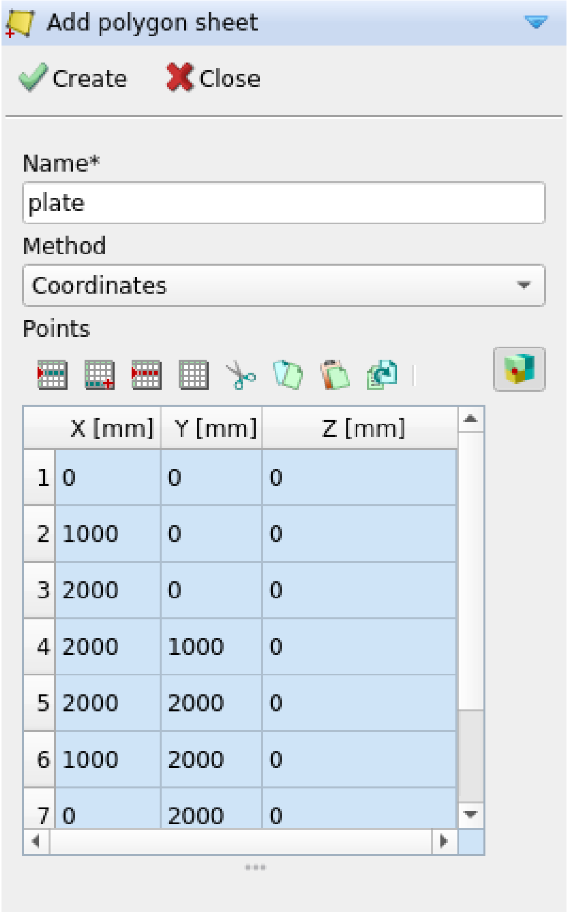


Figure 6: Add *plate* sheet

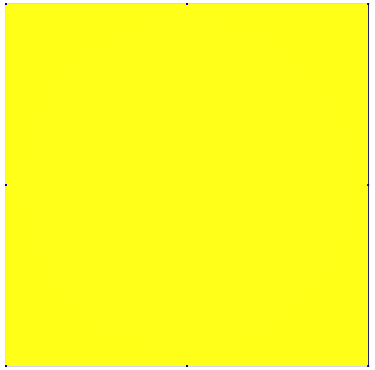


Figure 7: View of the *plate* geometry

3.2 Properties

We assign the element class, the material, and geometrical properties to the *PlateZ*. We use curved shell elements with a thickness of 10 mm [Fig. 1]. The material is assumed isotropic linear elastic and the material properties presented in Section 1 are used.

DianaIE

Main menu → Geometry → Assign → Shape Properties [Fig. 8]
Shape Properties → Material → Add material [Fig. 9] → Edit material [Fig. 10]

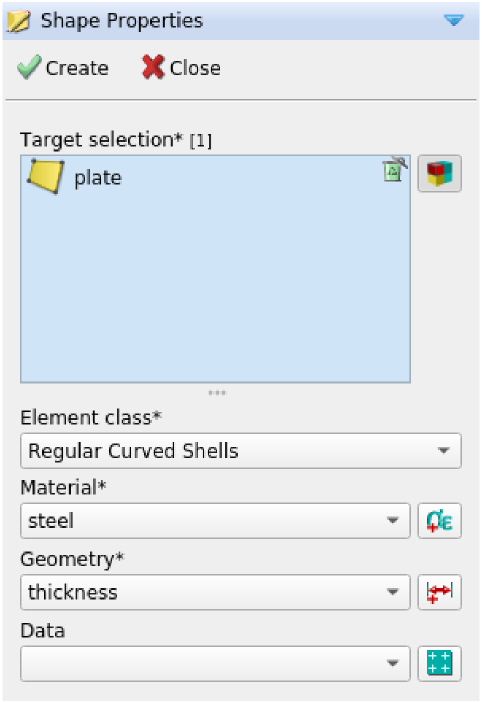


Figure 8: Property assignments to *plate*

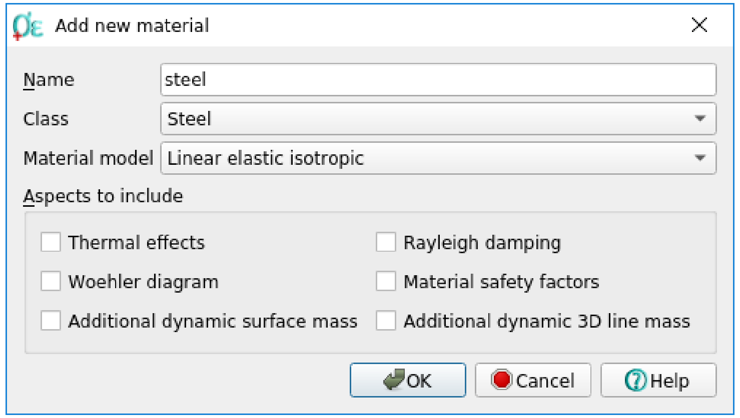


Figure 9: Add new material *steel*

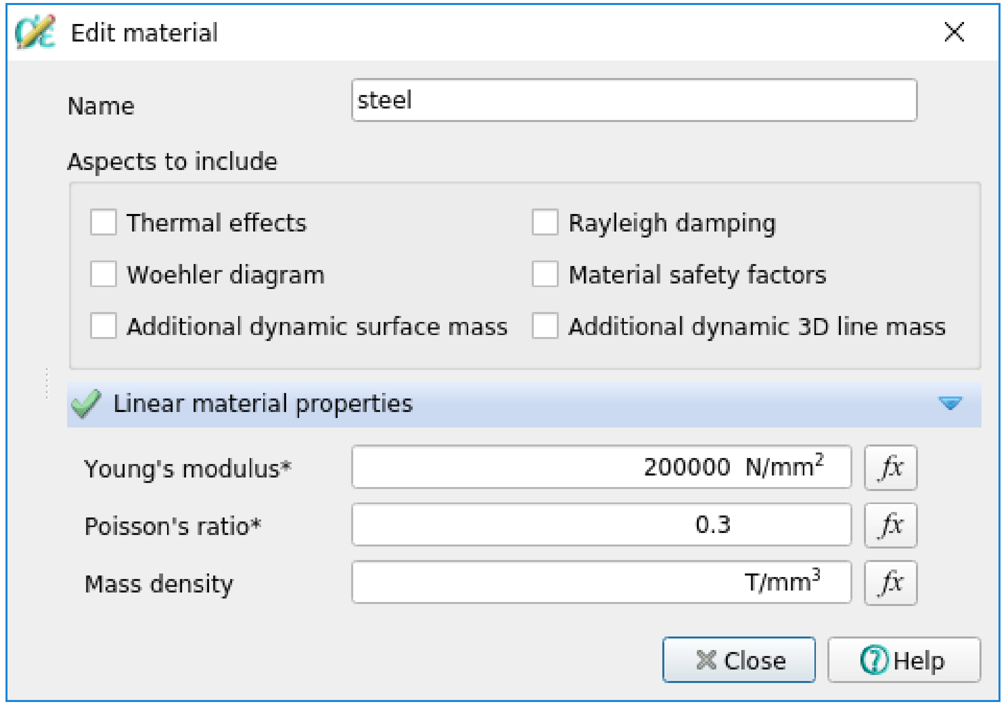


Figure 10: Edit material *steel*

Shape Properties → Geometry → Add new geometry [Fig. 11]

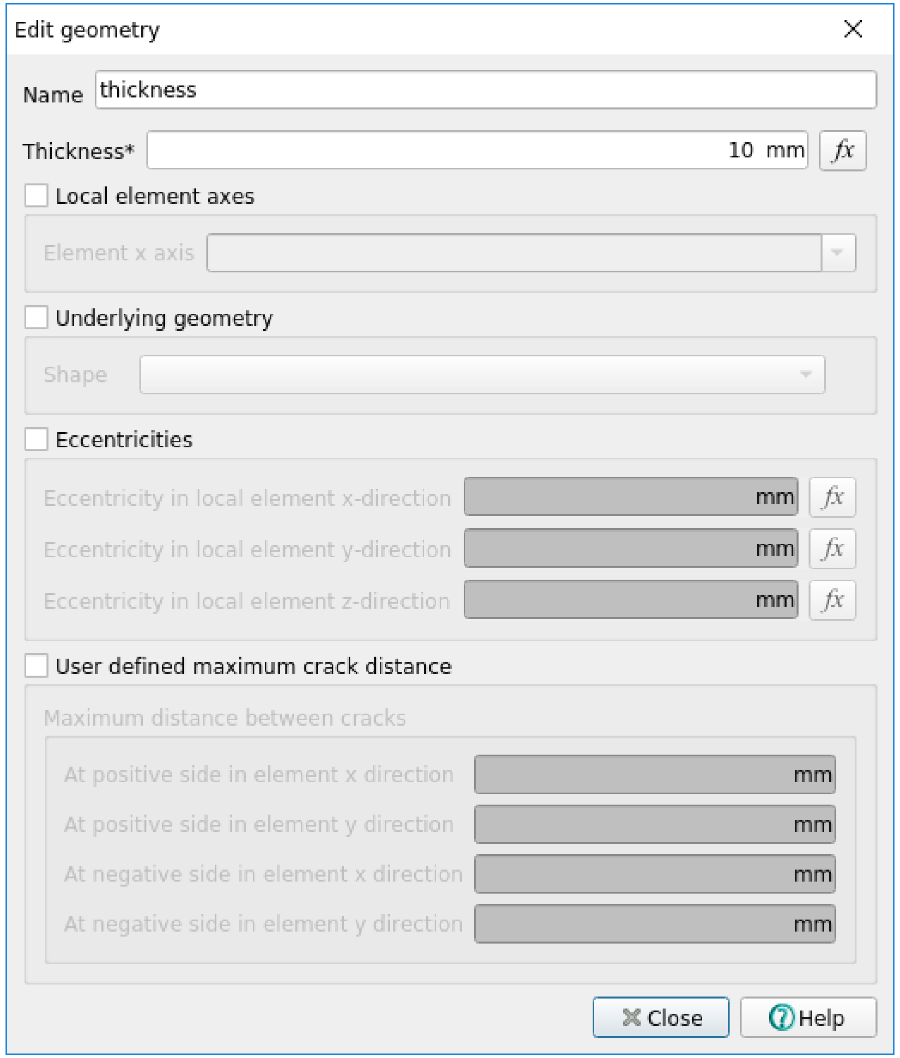


Figure 11: Edit geometry *thickness*

3.3 Boundary Conditions

We now apply the boundary conditions shown in Figure 2. We first constrain the displacement in the X direction at the top and bottom mid-nodes, then the displacement in the Y direction at the left and right mid-nodes and, finally, the displacement of the edges in the Z direction.

DianaIE

Main menu → Geometry → Assign → Add supports [Fig. 12] – [Fig. 15]
<*Repeat 3×*>

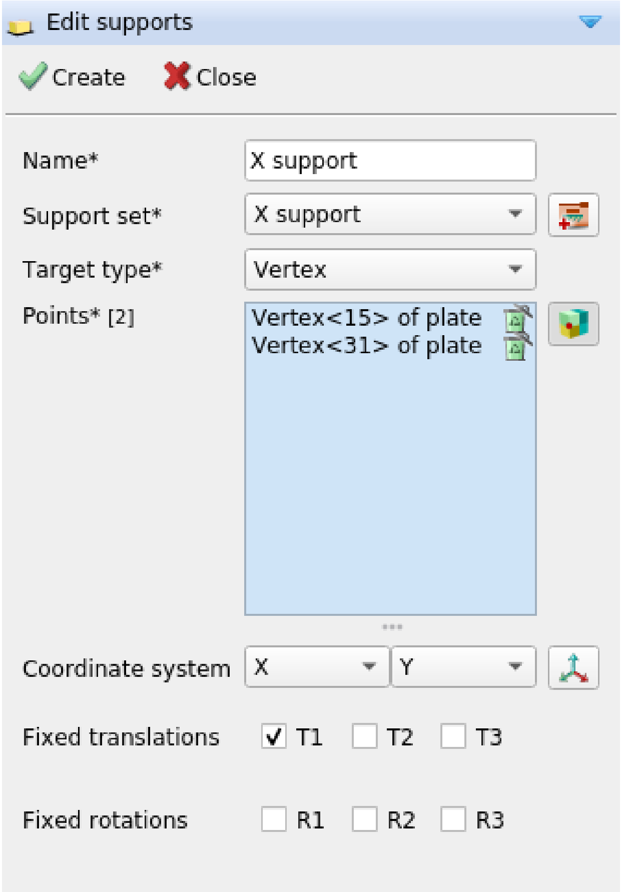


Figure 12: Constraints in the X direction

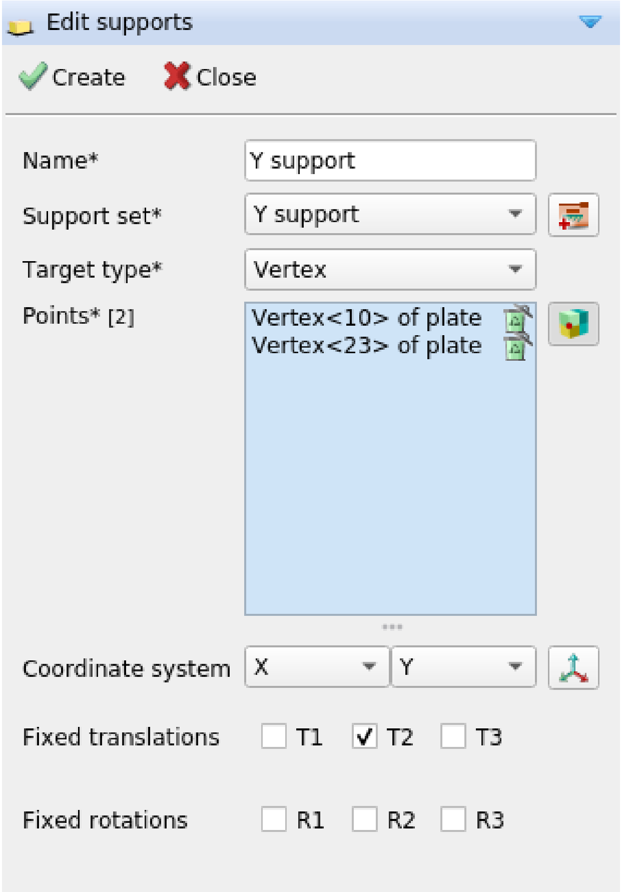


Figure 13: Constraints in the Y direction

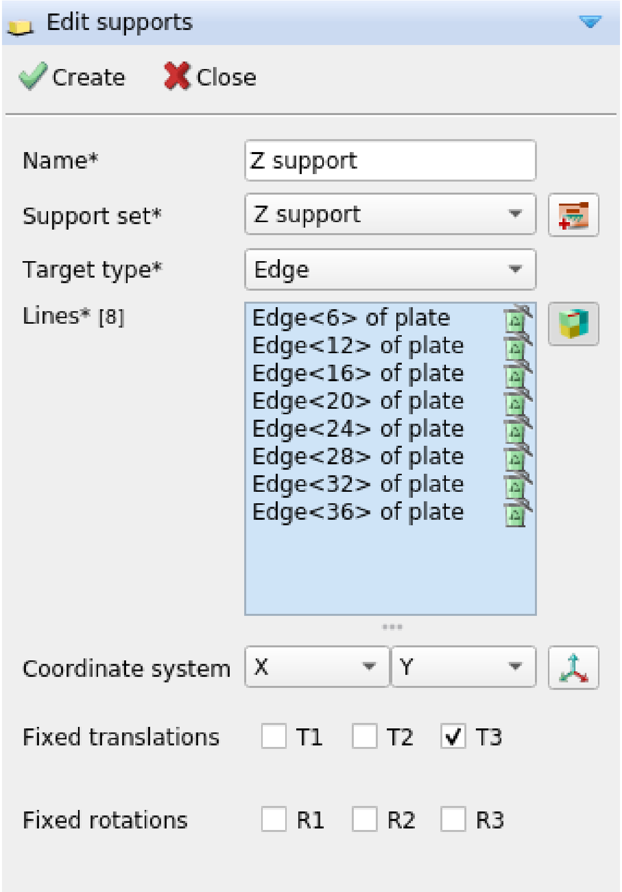


Figure 14: Constraints in the Z direction

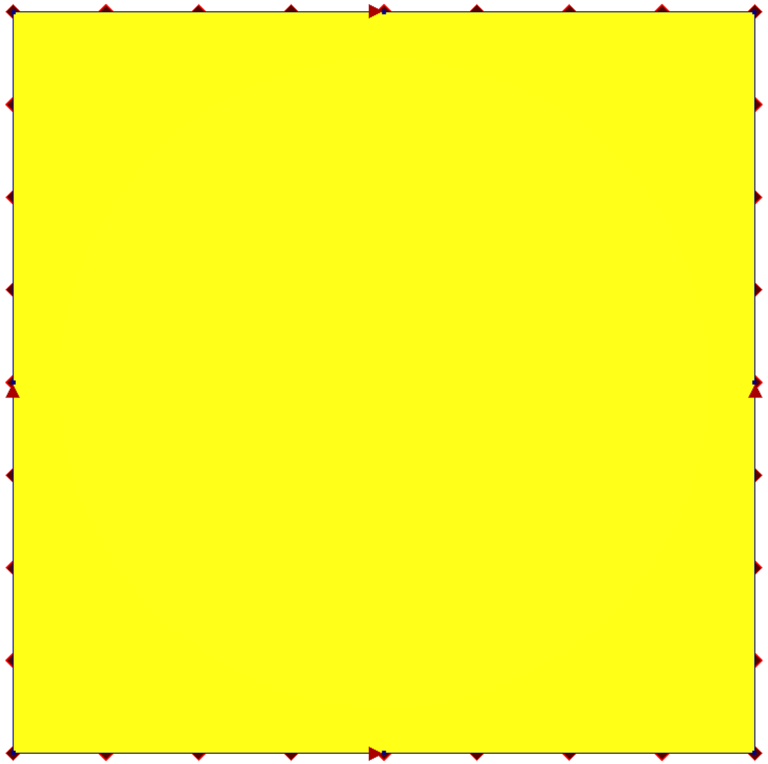


Figure 15: Geometry of the model with constraints

3.4 Loading

We apply a vertical pressure to the *PlateZ* by assigning a vertical distributed load at the top edge (-100 N/mm) and bottom edges (100 N/mm).

DianaIE

Main menu → Geometry → Assign → Add loads [Fig. 16] – [Fig. 18]

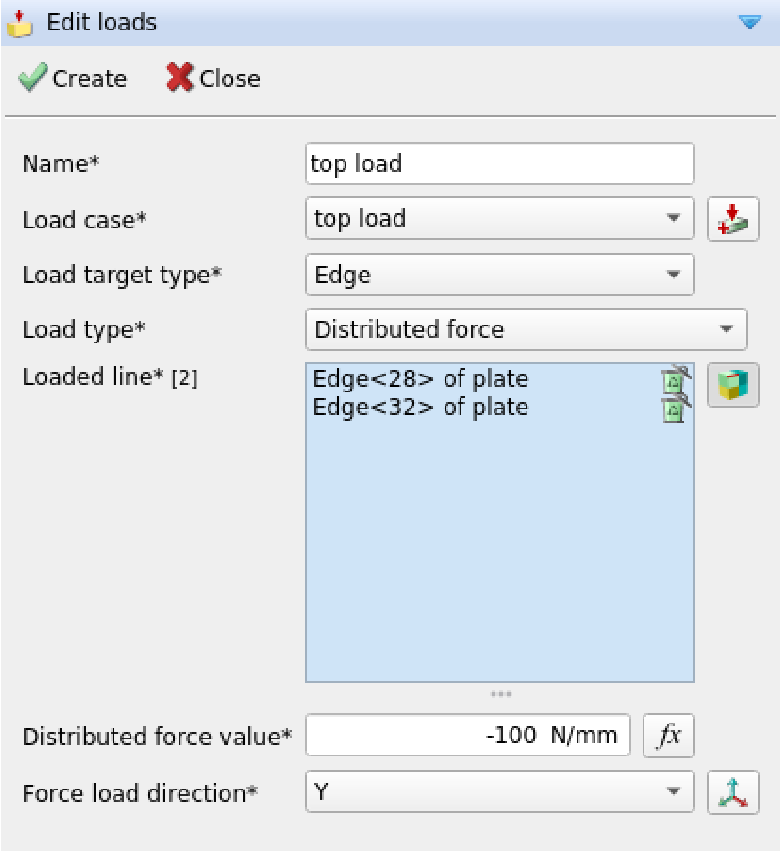


Figure 16: Vertical load at top edge

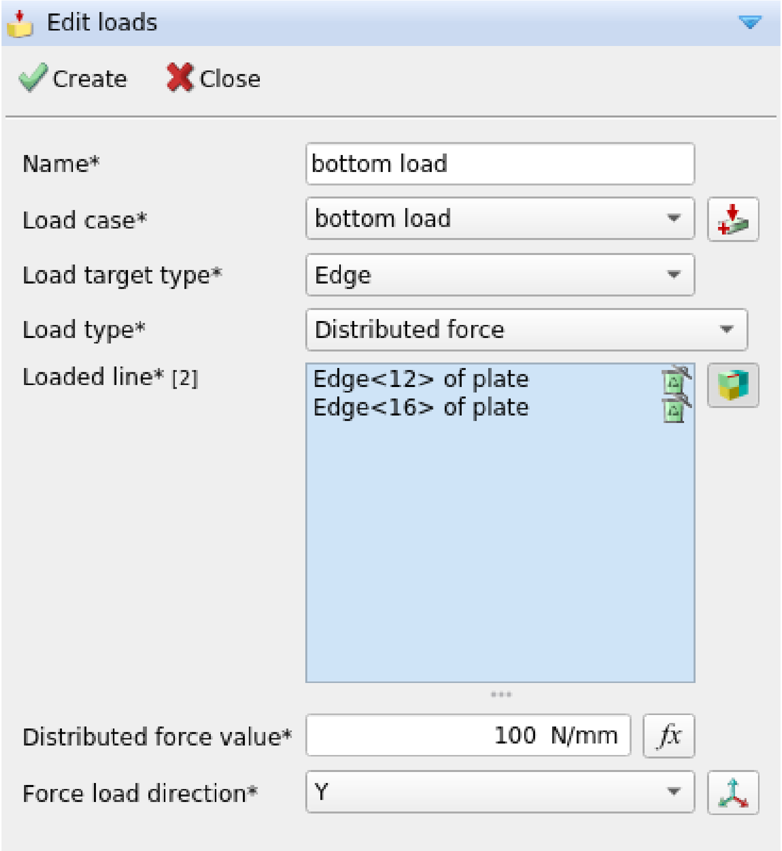


Figure 17: Vertical load at bottom edge

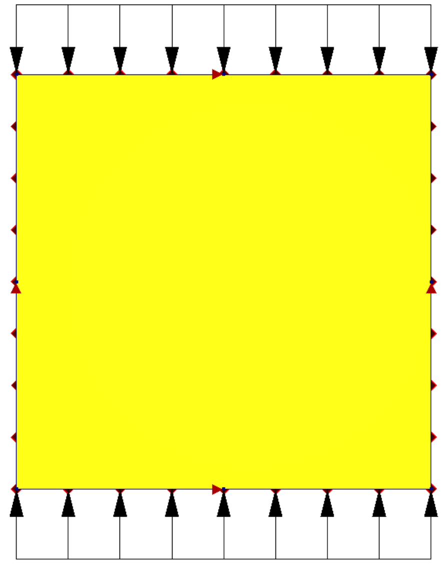


Figure 18: Top view of the model with the vertical loads

Then, we need to create a load combination that considers the two vertical loads. This is needed during the set-up of the buckling analysis as it will be considered as the buckling load.

DianaIE

Main menu → Geometry → Loads → Open geometry load combinations table [Fig. 19]

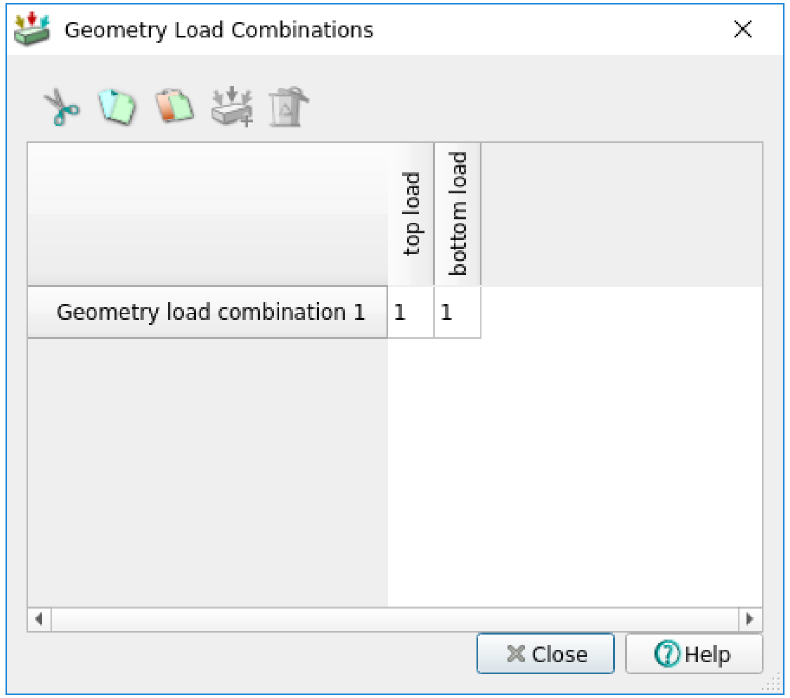


Figure 19: Load combination table

3.5 Mesh

For the mesh properties we specify an element size of 500 mm. Then, we generate the mesh.

DianaIE

Main menu → Geometry → Assign → Mesh properties [Fig. 20]
Main menu → Geometry → Generate mesh [Fig. 21]

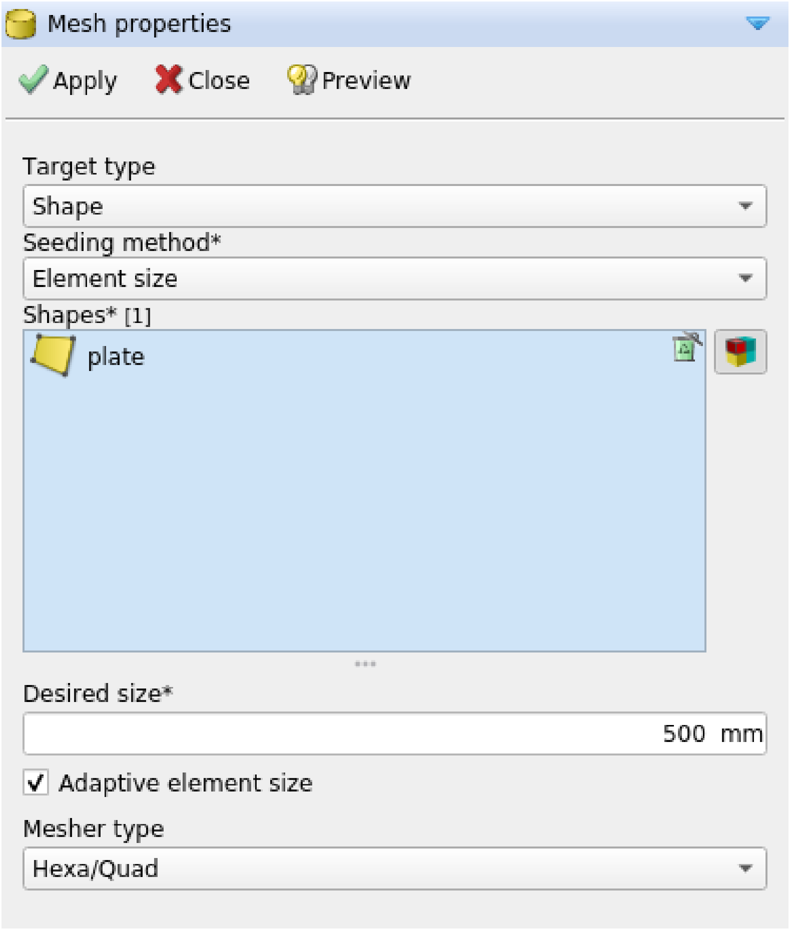


Figure 20: Mesh properties

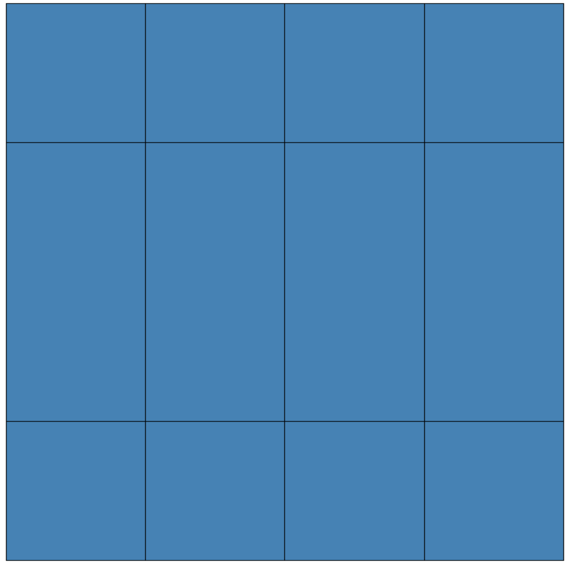


Figure 21: Finite element mesh

4 Structural Linear Static Analysis

4.1 Commands

We first perform a linear elastic analysis for a quick assessment of the model.

DianaIE

Main menu → Analysis → Add analysis
Analysis browser → Analysis1 → Rename → Linear [Fig. 22]
Main menu → Analysis → Run all analyses

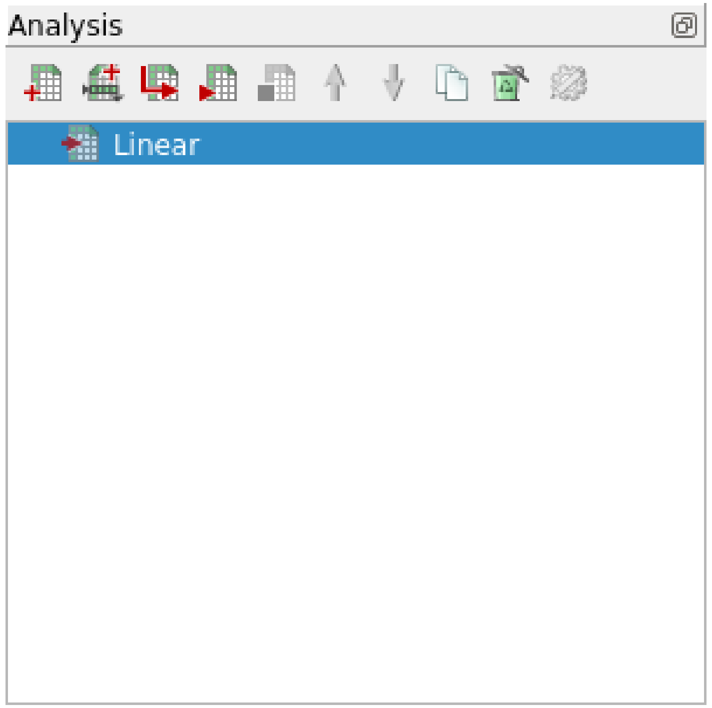


Figure 22: Analysis browser

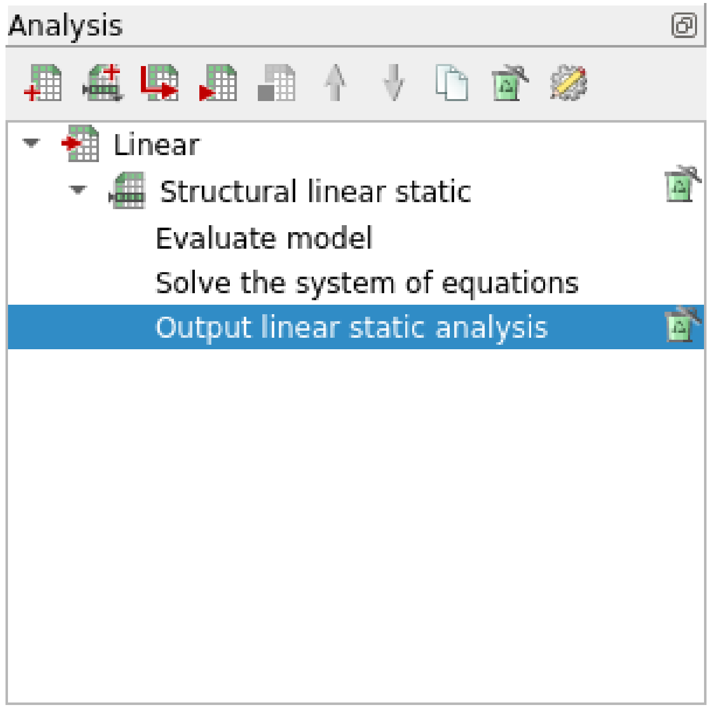


Figure 23: Analysis browser

4.2 Results

Here we compare the vertical displacement of the top of the *PlateZ* obtained numerically with DianaIE and analytically from the equation:

$$u_y = \frac{fb}{2Et} = 0.05 \text{ mm.}$$

DIANAIE

Results browser → Linear → Output eigenvalue analysis → Nodal results → Displacements → DtY [Fig. 24] [Fig. 25]

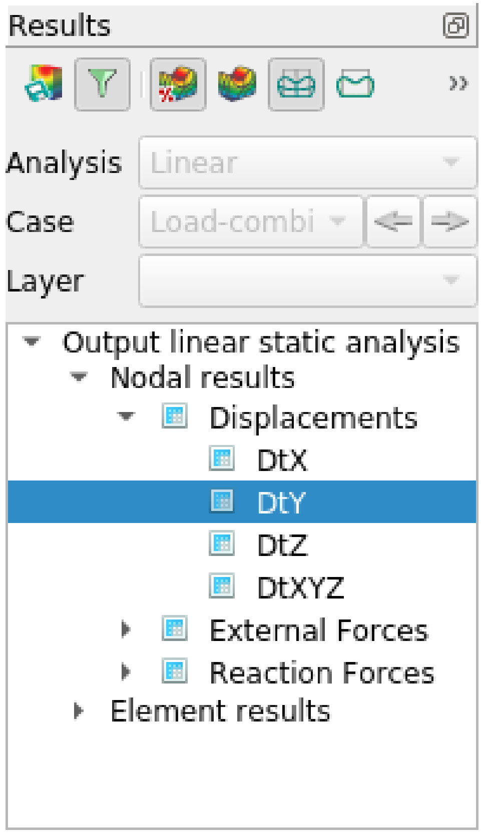


Figure 24: Results browser - DtY

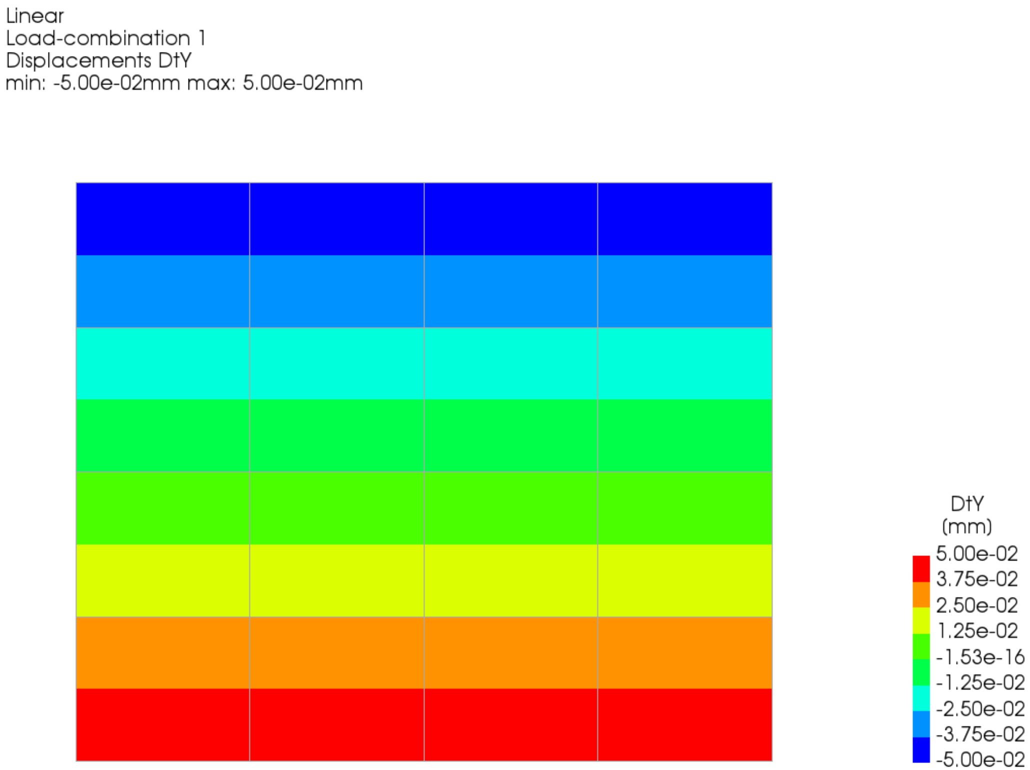


Figure 25: Displacement DtY

Figure 25 shows that the numerical result obtained with DianaIE is the same as the analytical solution.

5 Structural Stability Analysis

5.1 Commands

Now, we perform a structural stability analysis to derive the buckling load for the *PlateZ*.

DianaIE

Analysis browser → Linear 🖱 → Add command → Structural stability [Fig. 26]
Analysis browser → Linear → Structural stability → Eigenvalue analysis → Define stability analysis → Edit properties [Fig. 27]

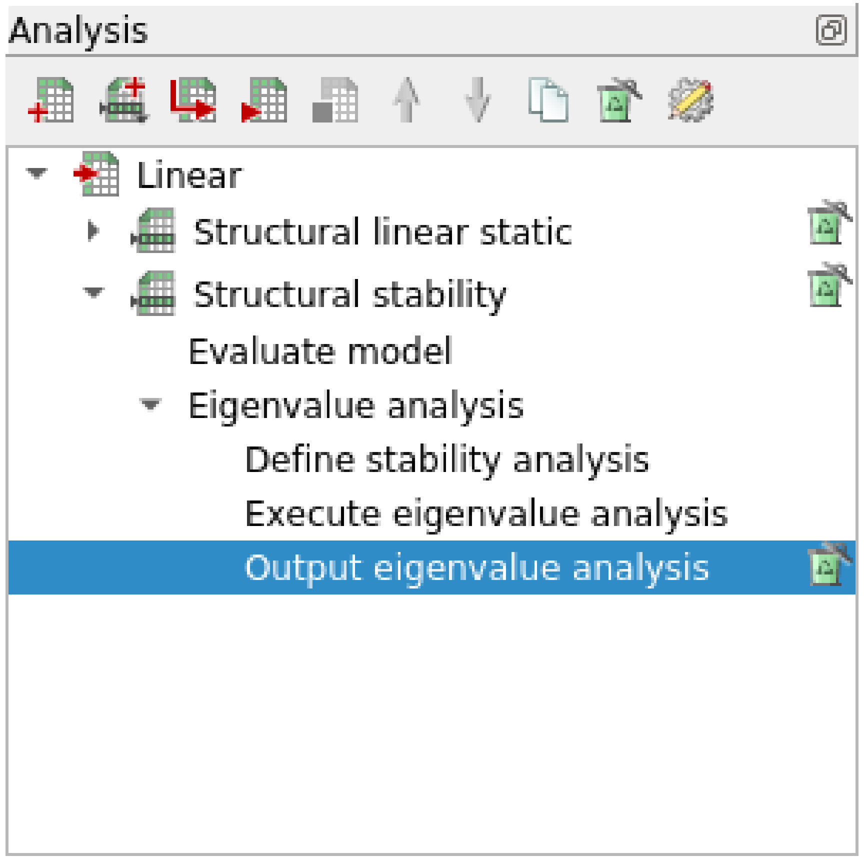


Figure 26: Analysis browser

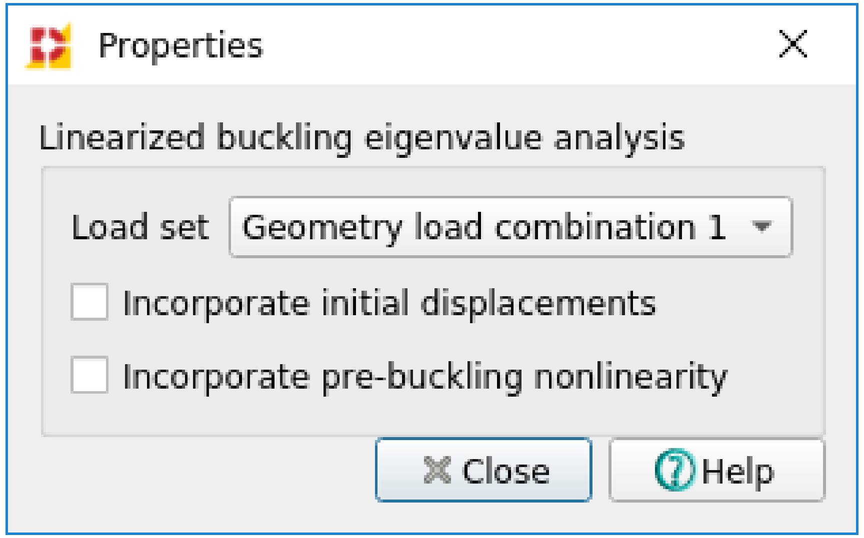


Figure 27: Buckling load

5.2 Results

We create a contour plot of the total displacement of the buckled *PlateZ*.

DianaIE

Results browser → Case → Mode 1, Buckling value 1.7989
Results browser → Linear → Output eigenvalue analysis → Nodal results → Displacements → DtXYZ [Fig. 28] [Fig. 29]

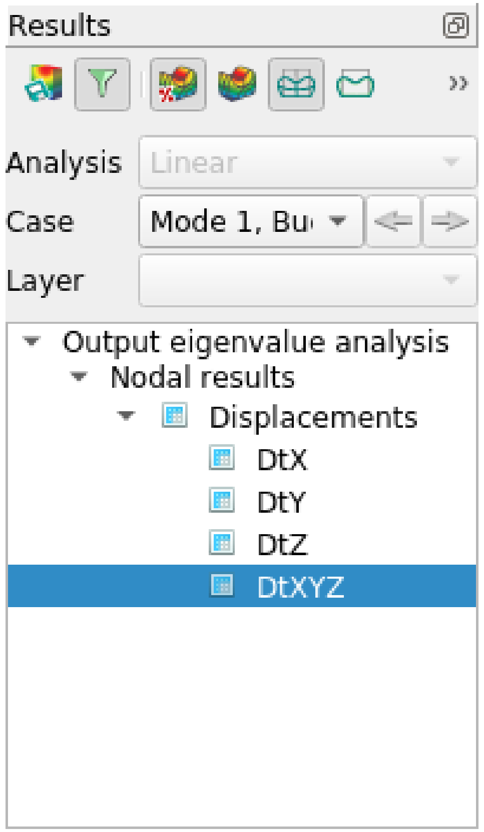


Figure 28: Results browser

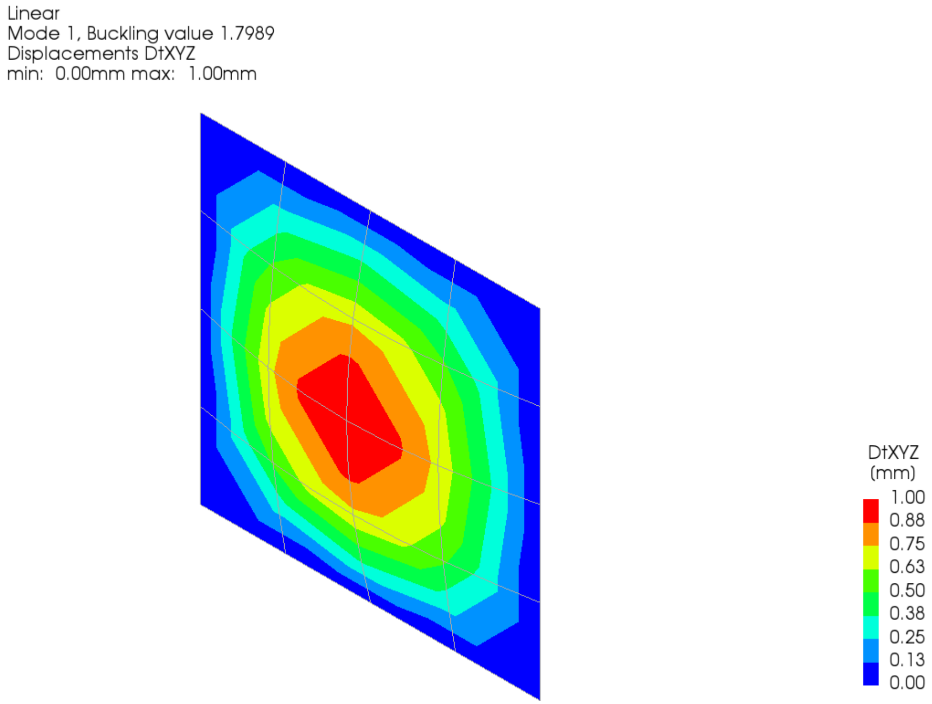


Figure 29: Displacement DtXYZ

Figure 29 shows that the buckling load obtained from DianaIE ($1.7989 \cdot f = 179.89$ N/mm) is in good agreement with the theoretical value of 180.76 N/mm calculated in Section 1.

Appendix A Additional Information

Folder: Tutorials/BucklingPlate

Number of elements ≈ 16

Keywords:
ANALYS: euler linear stabil static.
CONSTR: suppor.
ELEMEN: cq40s curved shell.
LOAD: edge elemen force.
MATERI: elasti isotro.
OPTION: direct units.
POST: binary ndiana.
PRE: dianai.
RESULT: buckli cauchy displa extern force green modes moment reacti strain stress total values.

References:

[1] S. P. Timoshenko and J. M. Gere. *Theory of Elastic Stability*. McGraw-Hill, 2nd edition, 1963.

DIANA FEA BV
Thijsseweg 11
2629JA Delft
The Netherlands
T +31 (0) 88 34262 00

DIANA FEA BV
Vlamoven 34
6826 TN Arnhem
The Netherlands
T +31 (0) 88 34262 00

WWW.DIANAFEA.COM

Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.

DIANA FEA