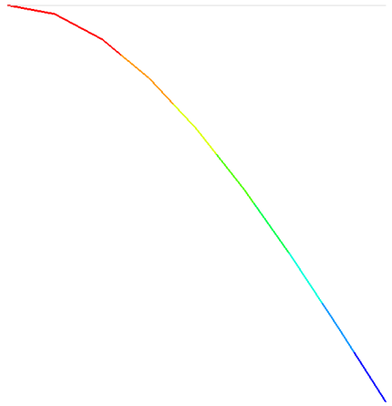

Linear Static Analysis of a Cantilever Beam

DIANA FEA

Outline

1 Description

In this example we perform a linear analysis on a cantilever beam subjected to a static load [Fig. 1]. The Euler-Bernoulli beam theory determines that at a distance x along the beam the bending moment M_z and deflection w are:

$$M_z(x) = P(L - x)$$

$$w(x) = \frac{2Px^2}{Ebh^3}(x - 3L)$$

The bending moment and deflection distributions along the beam are presented in Figure 2. The largest stress values arise at the support and are equal to:

$$\sigma_{max}^{+h/2} = \frac{6PL}{bh^2} = \frac{6 \times 27200 \times 2500}{25 \times 50 \times 50} = 6528 \text{ MPa}$$

We will construct a model for the cantilever beam and run a linear analysis. Afterwards we will compare the computed results with the theoretical values in terms of bending moment, displacement field and maximum stress values.

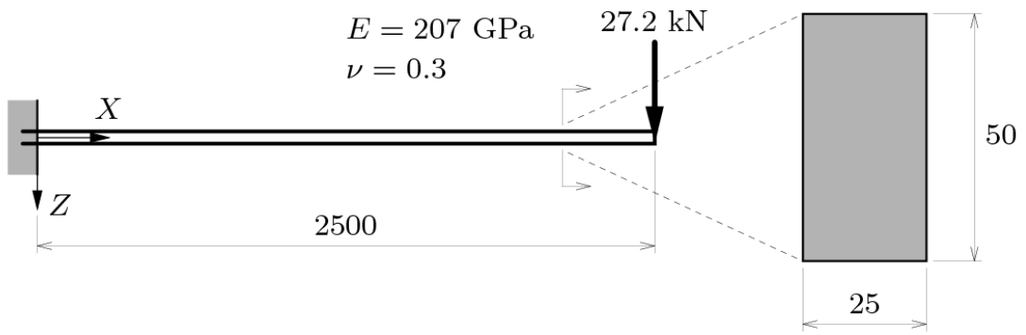

Figure 1: Model of cantilever beam [mm]

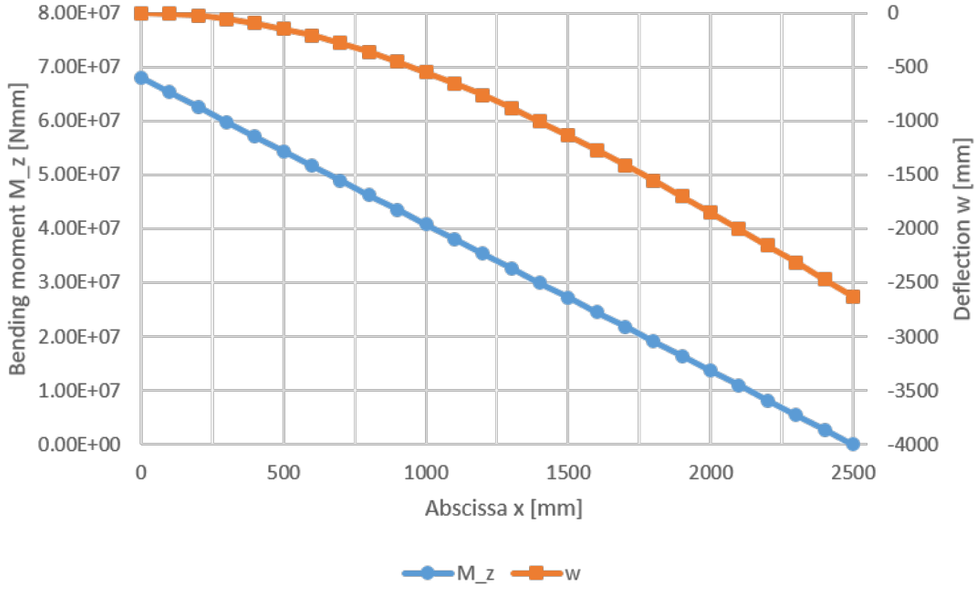

Figure 2: Distribution of bending moment and deflection along the beam

2 Finite Element Model

For the model we use 2D beam elements.

DianaIE

Main menu → File → New [Fig. 3]

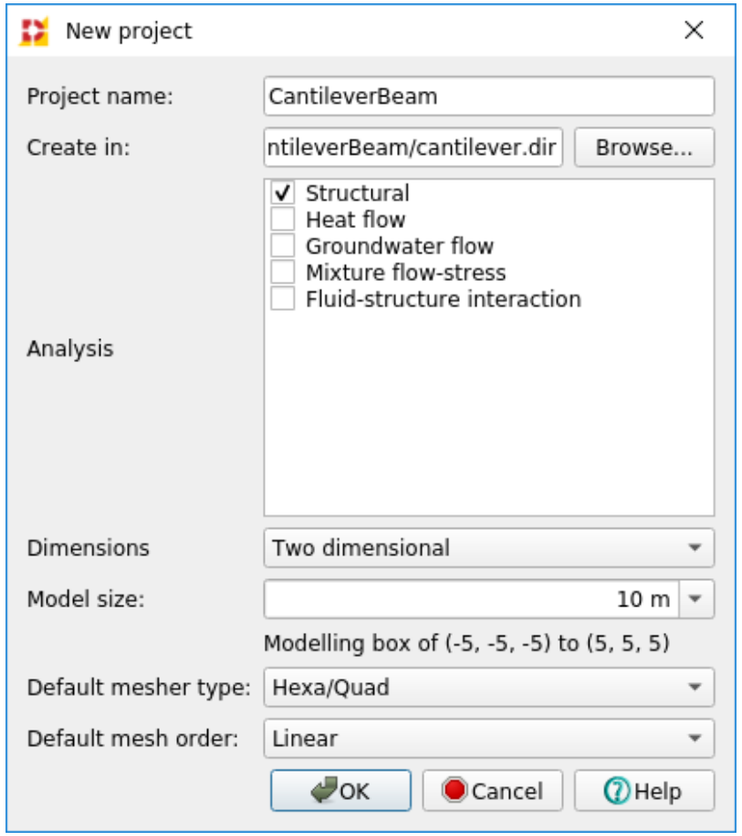

Figure 3: New project Dialog

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

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

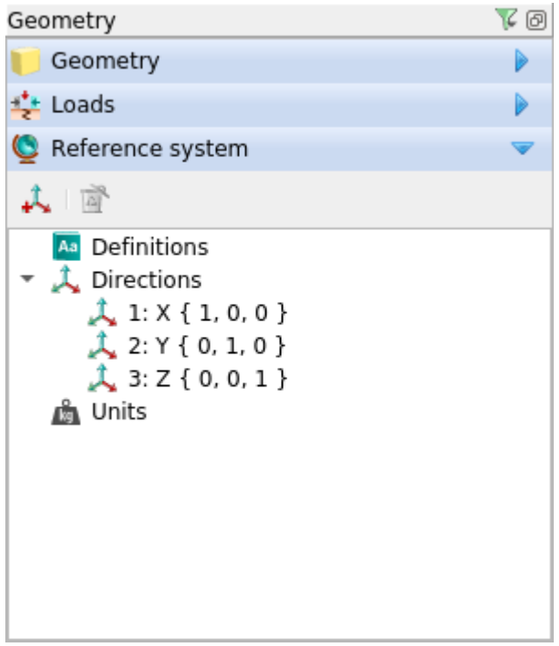

Figure 4: Geometry browser

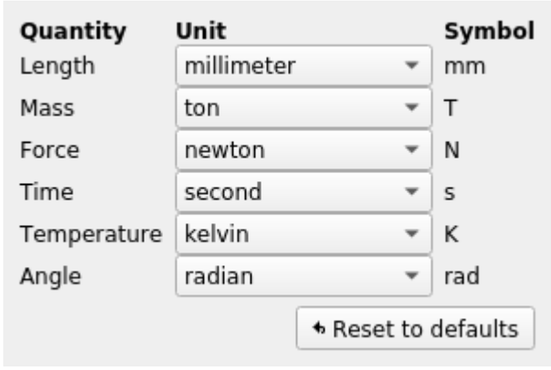

Figure 5: Property Panel - Units

2.1 Geometry

One line is required to build the model.

DianaIE

Main menu → Geometry → Create → Add line [Fig. 6]

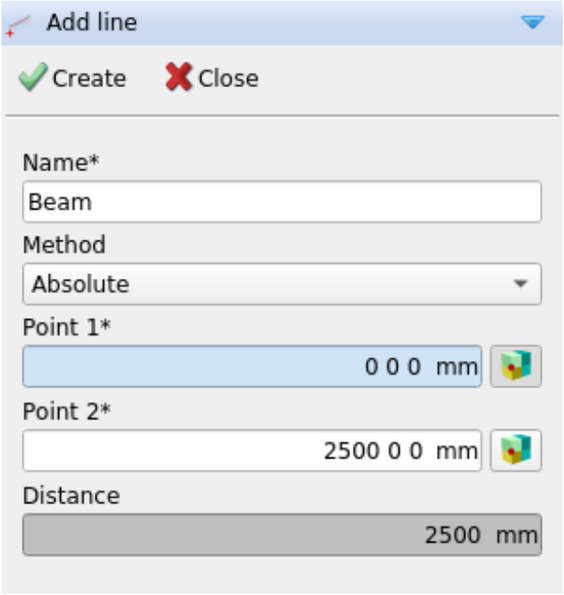

Figure 6: Geometry - add Line

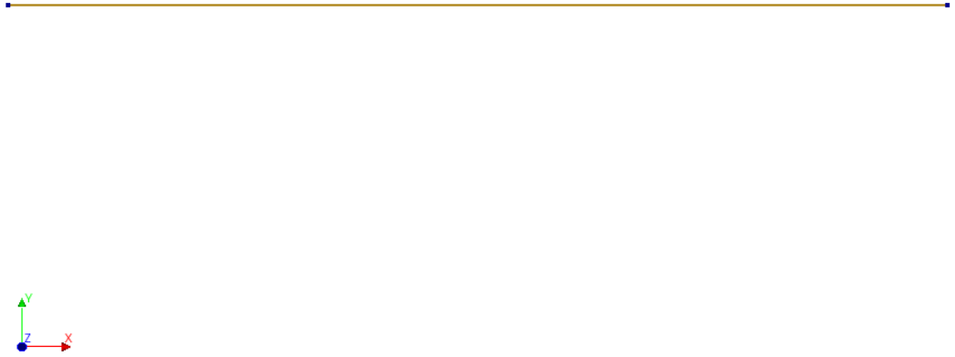

Figure 7: View of model

2.2 Properties

We assign the element class and the material and geometrical properties to the line. Two-noded beam elements are applied and the material is linear elastic. The geometry of the cross-section is specified in Figure 1.

DianaIE

Main menu → Geometry → Assign → Shape Properties [Fig. 8]
Shape Properties → Material → Add material [Fig. 9] → Edit material [Fig. 10]
Shape Properties → Geometry → Add new geometry [Fig. 11]

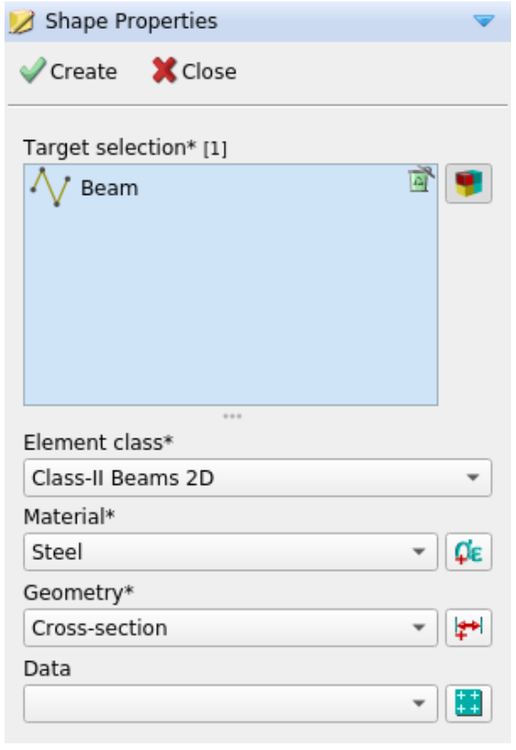

Figure 8: Property assignments

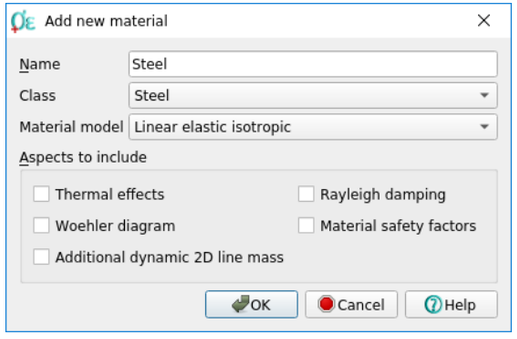

Figure 9: Add new material

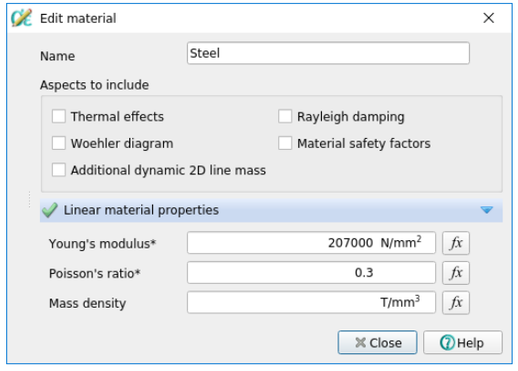

Figure 10: Material properties

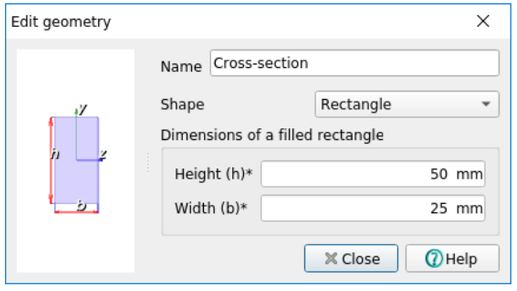

Figure 11: Geometrical properties

2.3 Boundary Conditions

We clamp the beam by fixing the translations and rotations at the left end of the horizontal line. See Figure 12 and Figure 13.

DianaIE

Main menu → Geometry → Assign → Add supports [Fig. 12] [Fig. 13]

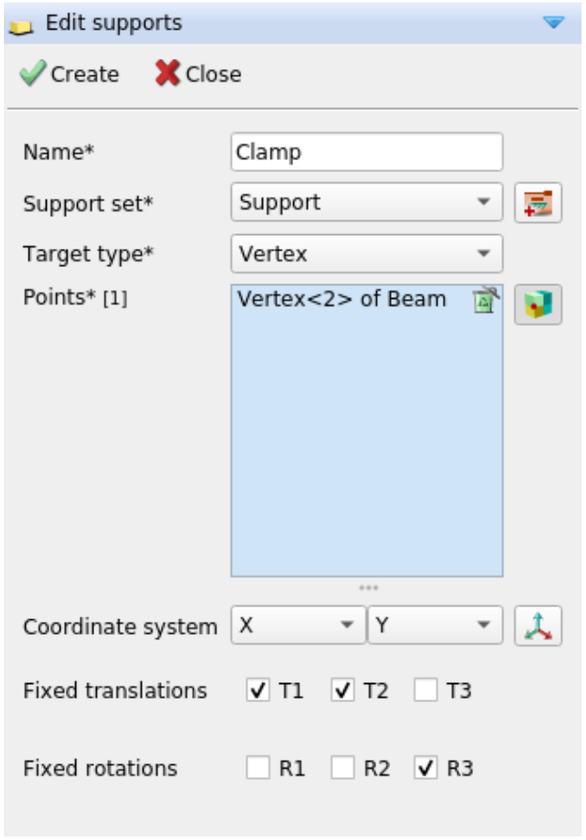

Figure 12: Clamping the left end

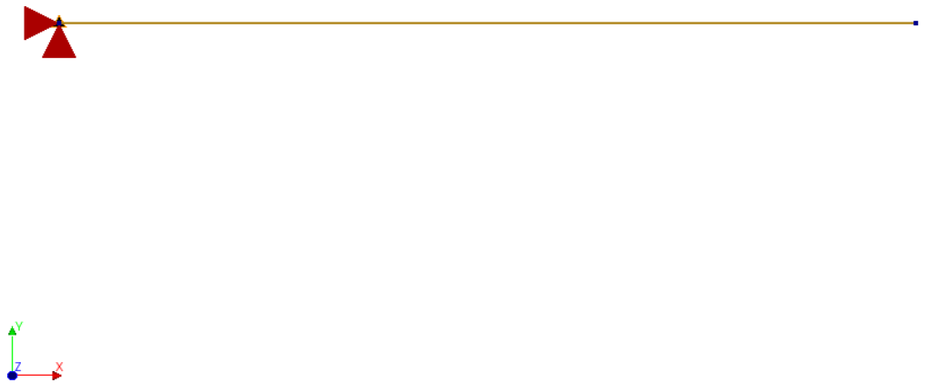

Figure 13: View of model - Support

2.4 Loads

A vertical nodal load pointing in the negative Y direction is applied at the right end of the beam. See Figure 14 and Figure 15.

DianaIE

Main menu → Geometry → Assign → Add loads [Fig. 14] [Fig. 15]

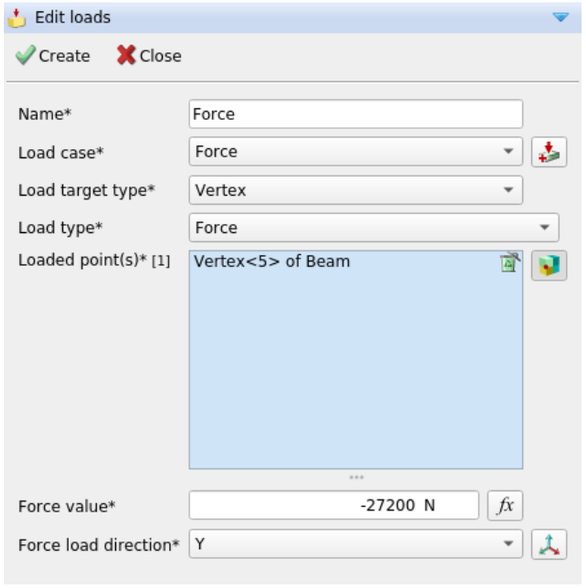

Figure 14: Attaching a nodal load to the right end

Figure 15: View of model - Support and load

2.5 Mesh

The mesh is defined by dividing the line into eight parts. We can observe the cross-section of the beam.

DianaIE

Main menu → Geometry → Assign → Mesh properties [Fig. 16]
Main menu → Geometry → Generate mesh
Main menu → Mesh → Mesh → 3D beams and shells [Fig. 17]

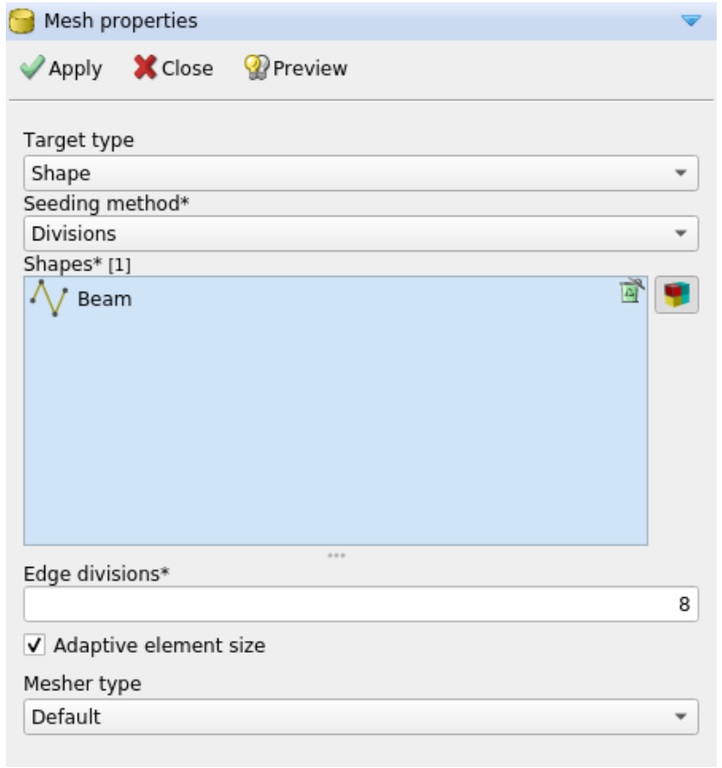

Figure 16: Divide the line geometry

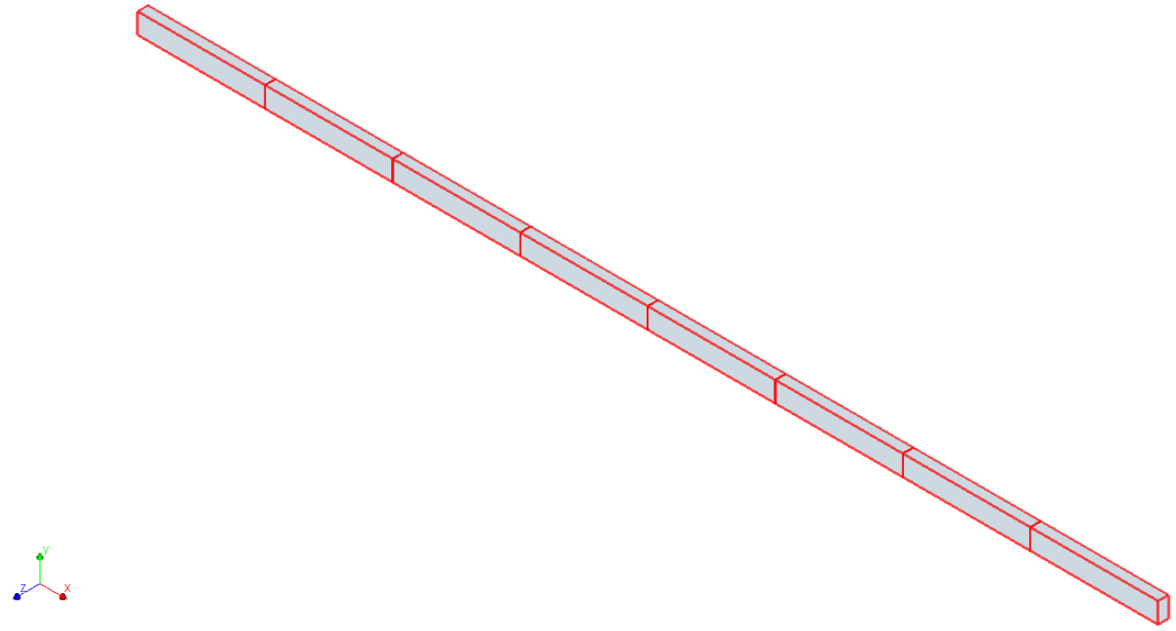

Figure 17: Mesh

3 Linear Static Analysis

3.1 Commands

We perform a structural linear static analysis.

DianaIE

Main menu → Analysis → Add analysis
Analysis browser → Analysis1 → Rename → LinSta [Fig. 18]
Analysis browser → LinSta → Add command → Structural linear static [Fig. 19] [Fig. 20]
Main menu → Analysis → Run selected analysis

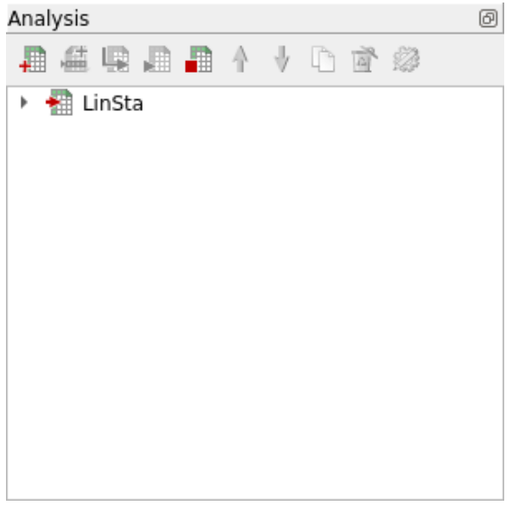

Figure 18: Analysis window

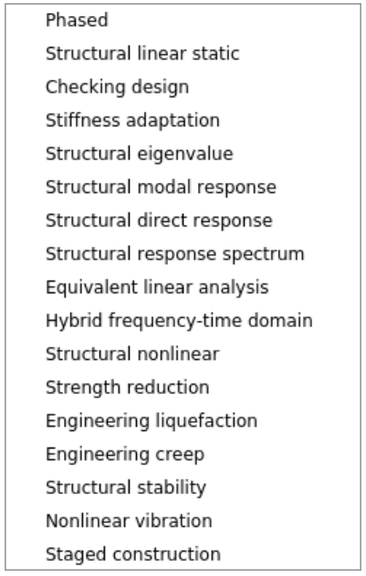

Figure 19: Add command

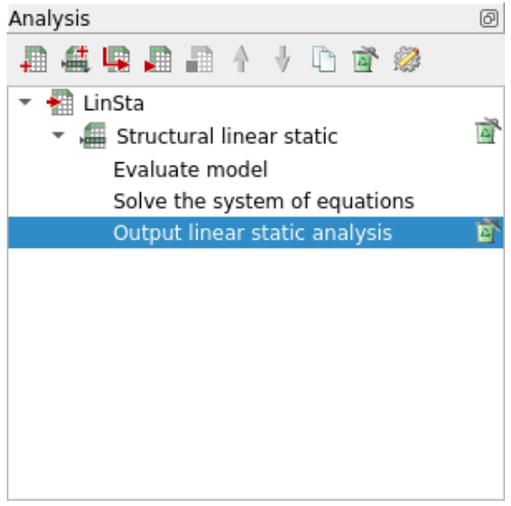

Figure 20: Analysis tree

3.2 Results

3.2.1 Displacements

The displacements can be graphically represented to check whether the structure behaves as expected, i.e. deflecting in the negative Y direction [Fig. 22].

DianaIE

Results browser → LinSta → Output linear static analysis → Nodal results → Displacements → DtY → Show contours [Fig. 21]

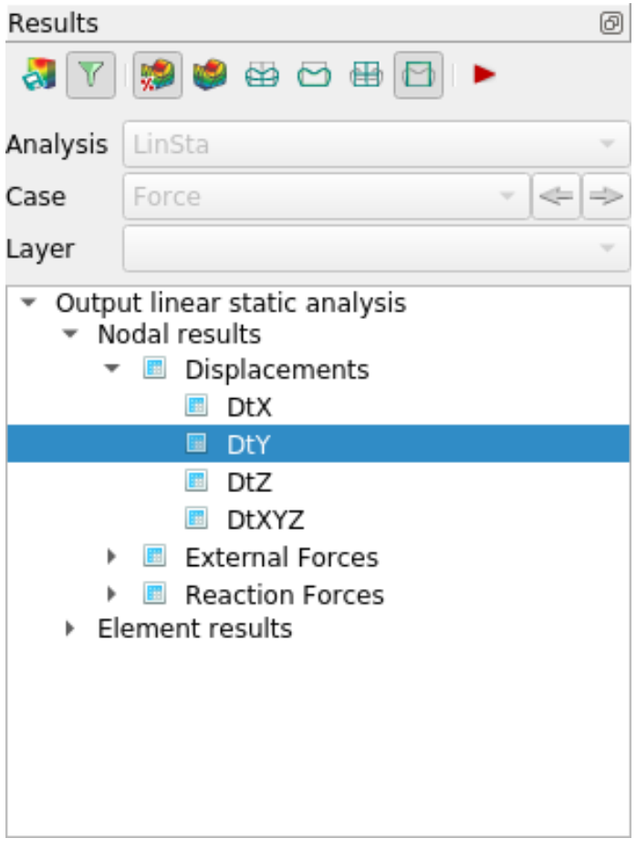

Figure 21: Results browser

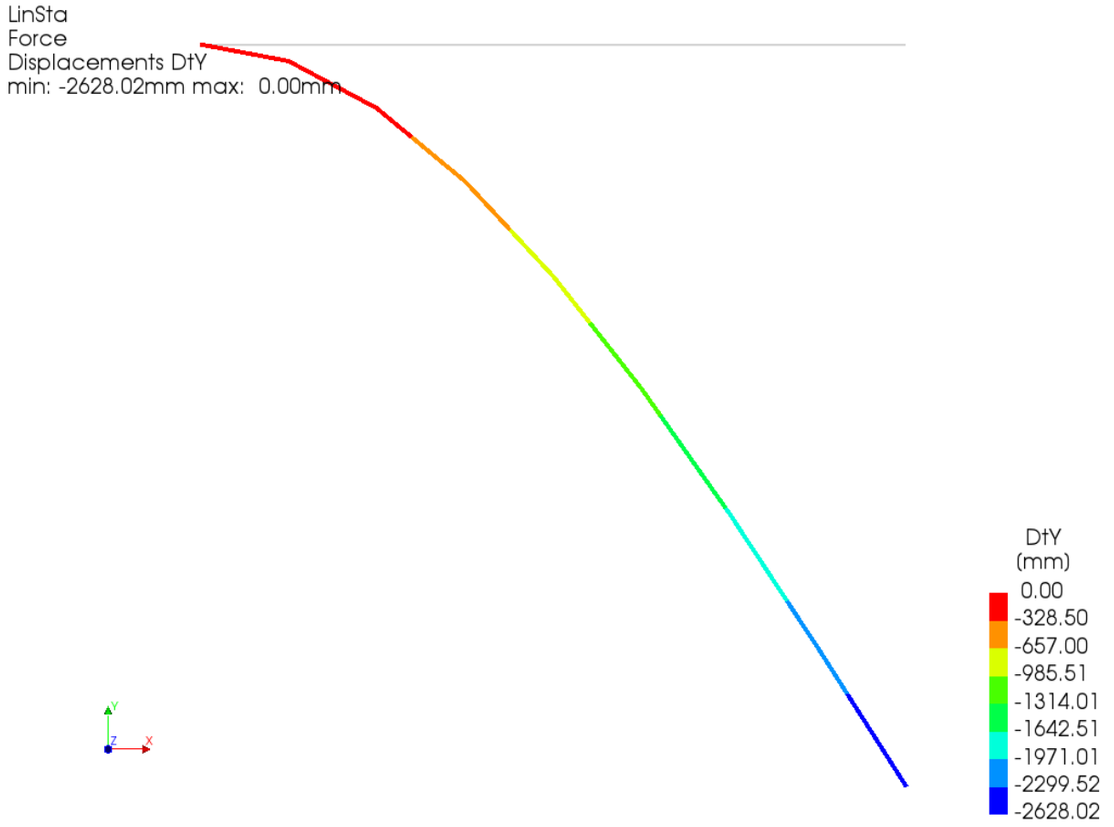

Figure 22: Absolute deflection

Comparing the finite element result [Fig. 22] with the analytical result [Fig. 2] shows that the finite element model returns the same deflection as the theory.

3.2.2 Reaction Forces

The reaction forces can also be graphically represented to check whether force equilibrium is met [Fig. 24].

DianaIE

Results browser → LinSta → Output linear static analysis → Nodal results → Reaction Forces → FBY → Show vectors [Fig. 23]

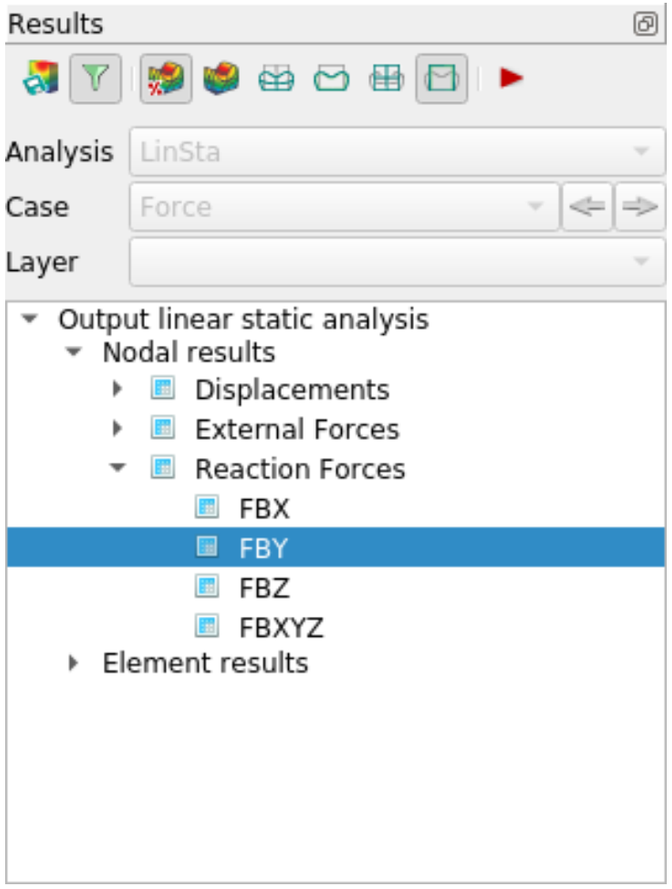

Figure 23: Results browser

Figure 24: Reaction force

According to Figure 24 vertical force equilibrium is fulfilled.

3.2.3 Cross-section Moments

Representing the bending moment distribution by a diagram can be done in DianaIE [Fig. 25] [Fig. 26].

DianaIE

Results browser → LinSta → Output linear static analysis → Nodal results → Cross-section Moments → Mz → Show line diagram [Fig. 25]

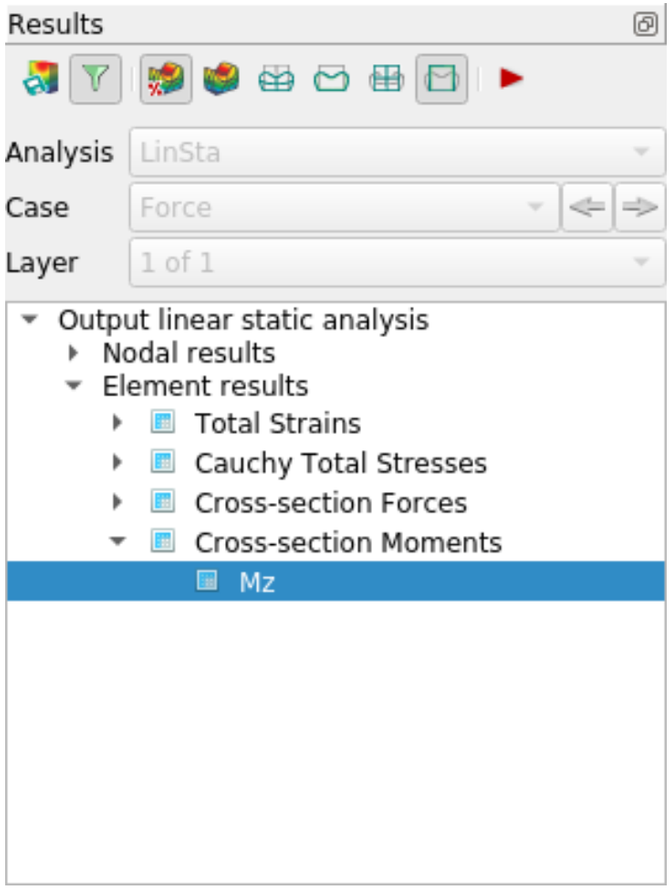

Figure 25: Results browser

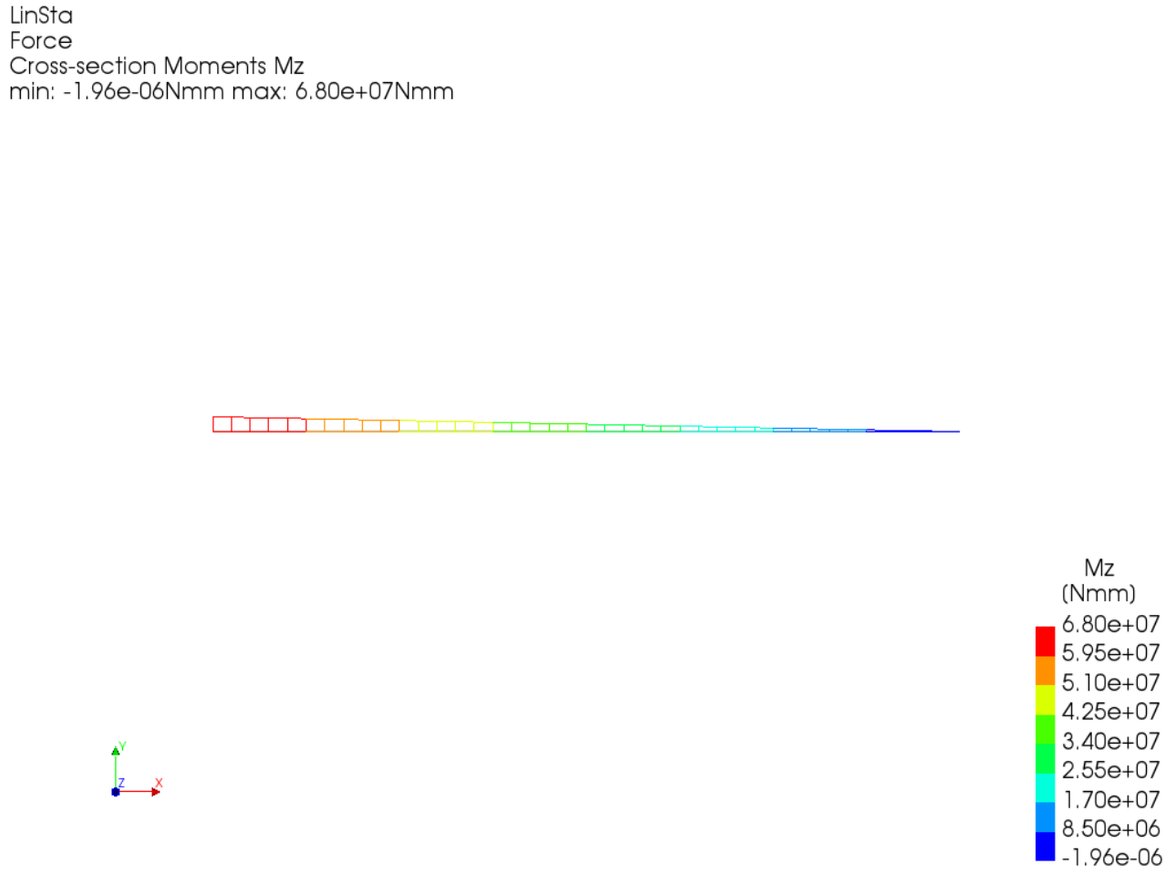

Figure 26: Bending moment distribution

As predicted by the theory, the moment distribution is linear with a maximum value of 68 kNm at the clamp and a null value at the free end [Fig. 2] as determined by the Euler-Bernoulli theory.

3.2.4 Cauchy Total Stresses

Lastly we assess the Cauchy stresses.

DianaIE

Results browser → LinSta → Output linear static analysis → Element results → Cauchy Total Stresses → SXX → Show contours [Fig. 27]

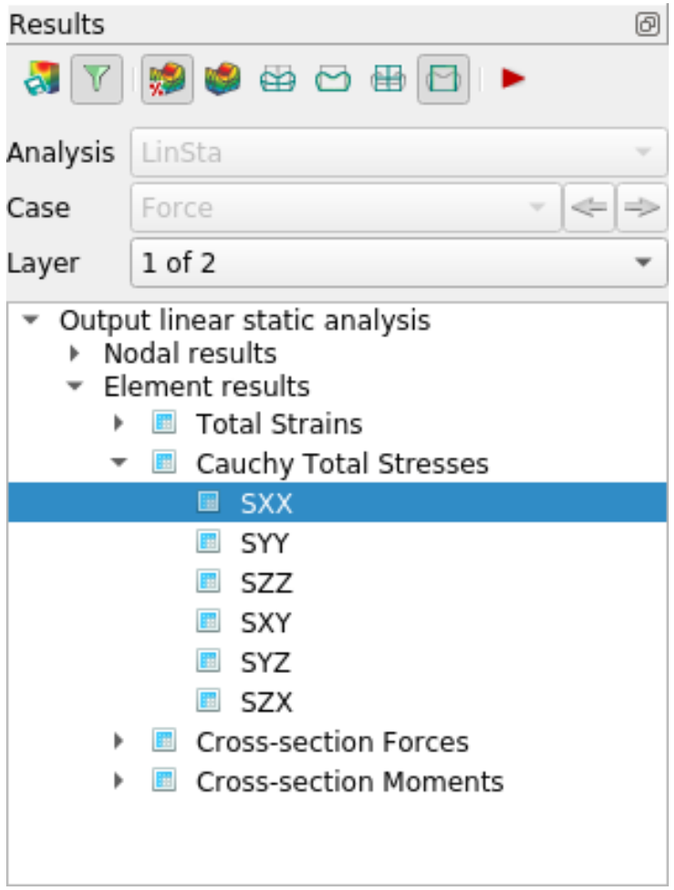

Figure 27: Results browser

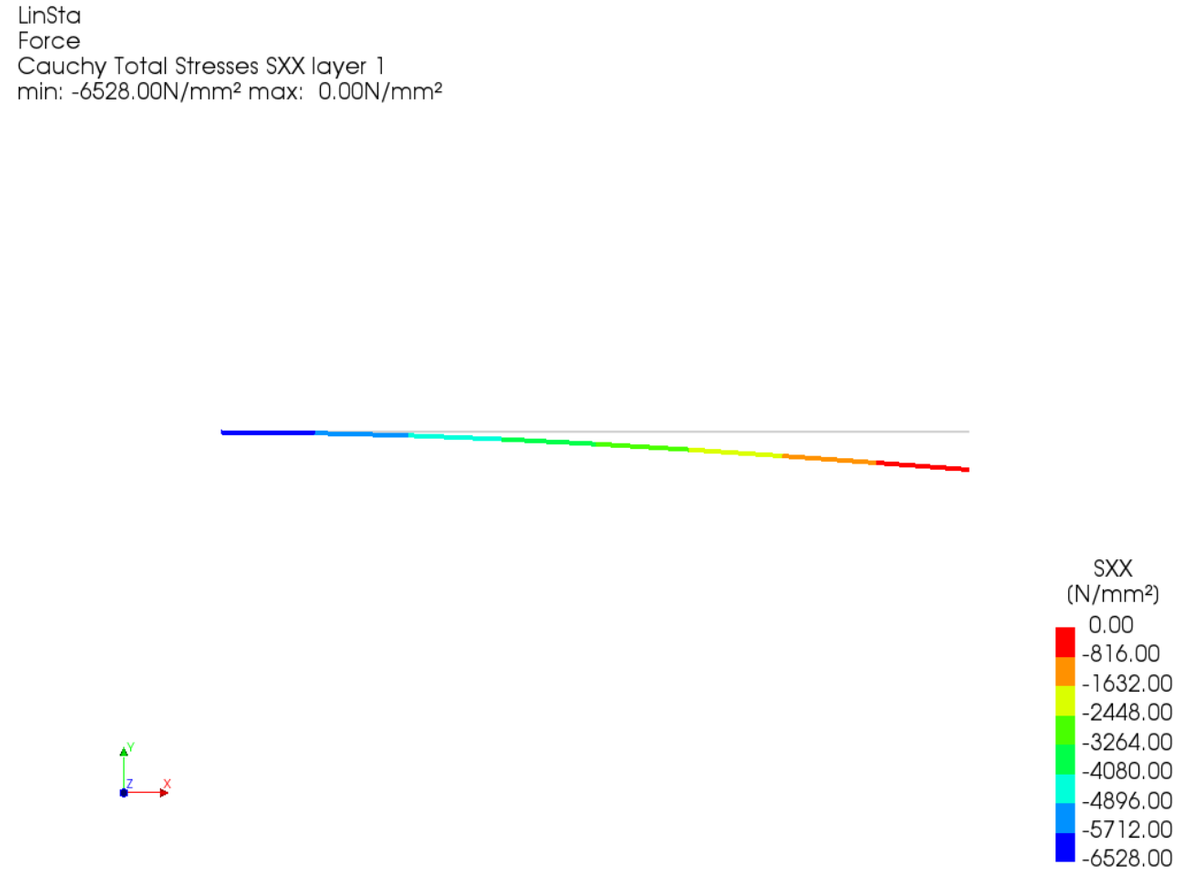

Figure 28: Normal stresses

Note that the legend in Figure 28 displays the negative stress values, i.e. the stresses at the bottom of the beam, while the theory calculates the stress in the top left end cross-section. Regardless, in an absolute sense, the finite element computation is identical to the theory.

Appendix A Additional Information

Folder: Tutorials/CantileverBeam

Number of elements ≈ 8

Keywords:
ANALYS: linear static.
CONSTR: suppor.
ELEMEN: beam class2 l7ben rectan.
LOAD: force node.
MATERI: elasti isotro.
OPTION: direct units.
POST: binary ndiana.
PRE: dianai.
RESULT: cauchy displa extern force green moment reacti strain stress total.

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

DIANA FEA