



## Linear Static Analysis of a Cantilever Beam

## Outline

<b>1</b>	<b>Description</b>	<b>3</b>
<b>2</b>	<b>Finite Element Model</b>	<b>4</b>
2.1	Geometry . . . . .	5
2.2	Properties . . . . .	6
2.3	Boundary Conditions . . . . .	7
2.4	Loads . . . . .	8
2.5	Mesh . . . . .	9
<b>3</b>	<b>Linear Static Analysis</b>	<b>10</b>
3.1	Commands . . . . .	10
3.2	Results . . . . .	11
3.2.1	Displacements . . . . .	11
3.2.2	Reaction Forces . . . . .	12
3.2.3	Cross-section Moments . . . . .	13
3.2.4	Cauchy Total Stresses . . . . .	14
	<b>Appendix A Additional Information</b>	<b>15</b>

# 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:

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

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

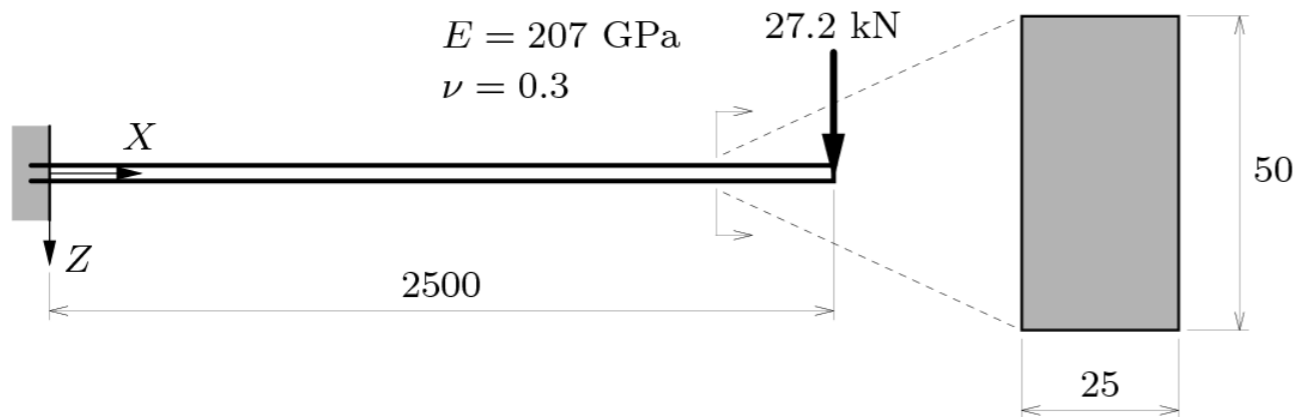


Figure 1: Model of cantilever beam [mm]

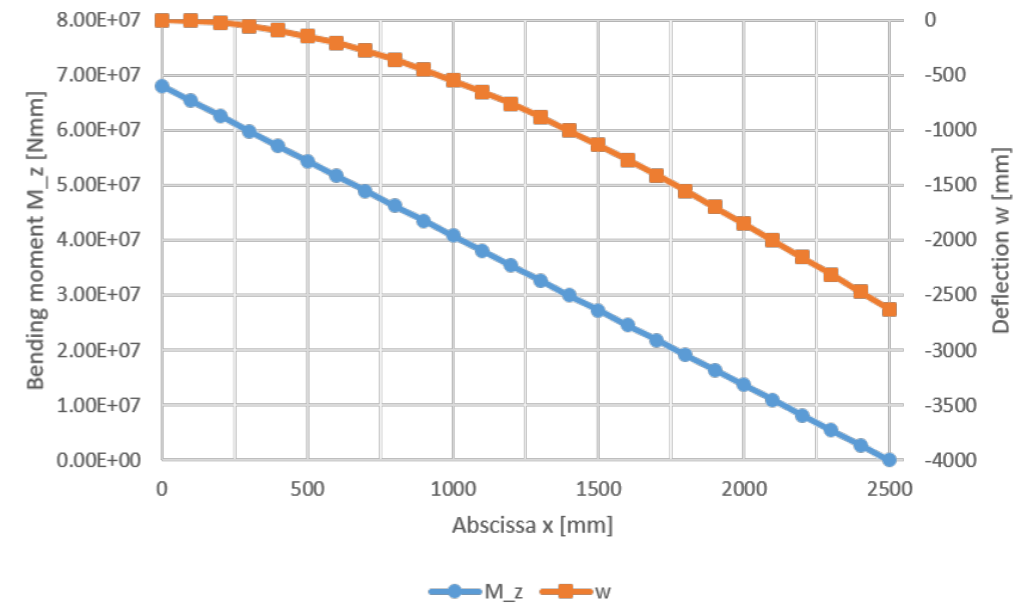


Figure 2: Distribution of bending moment and deflection along the beam

## 2 Finite Element Model

For the model we use 2D beam elements. We choose millimeter for the quantity length, ton for mass and newton for force.

DIANAIE

**Main menu** → File → New  [Fig. 3]  
**Geometry browser** → Reference system → Units [Fig. 4]  
**Property Panel** [Fig. 5]

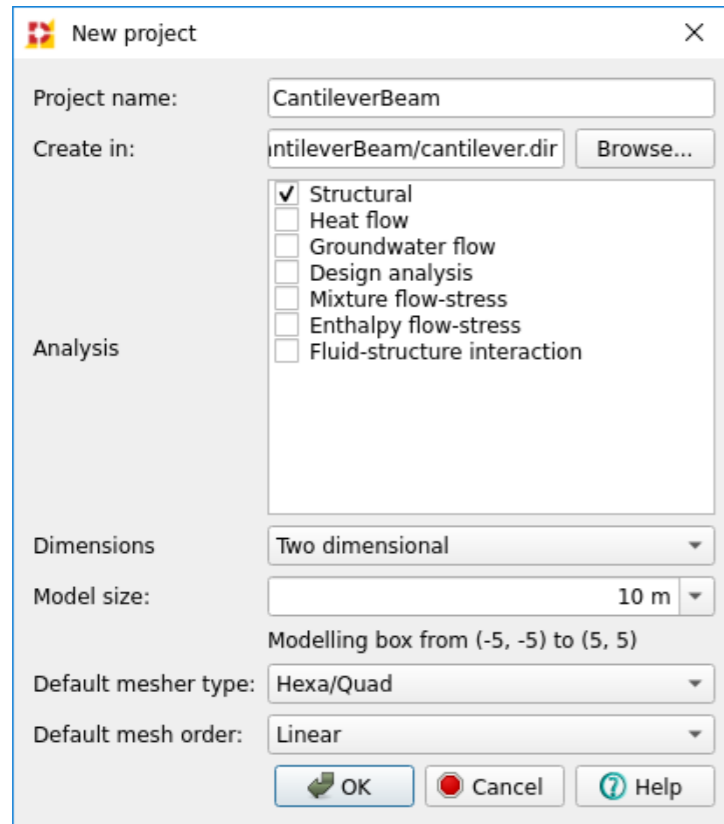


Figure 3: New project dialog

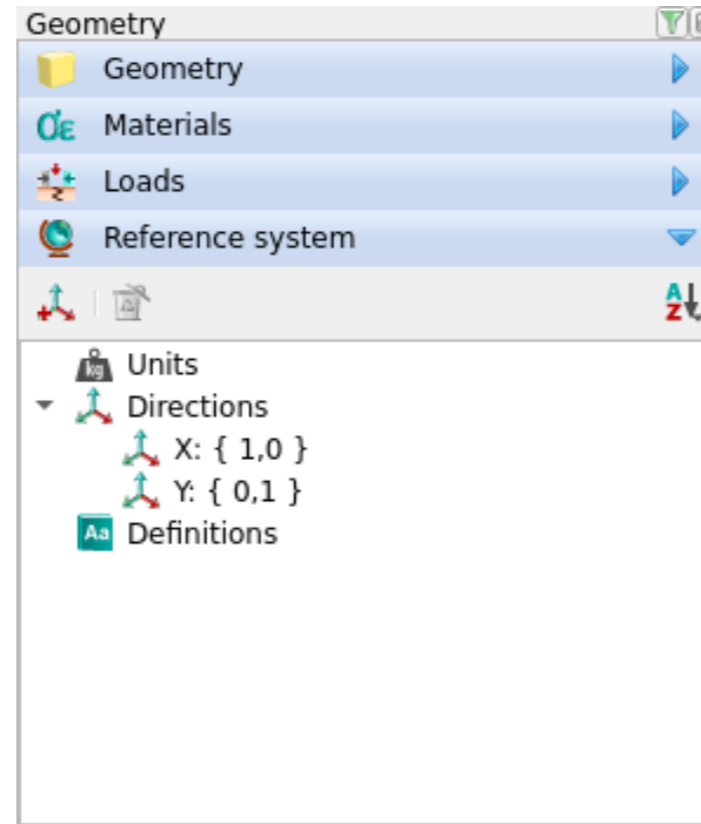


Figure 4: Geometry browser

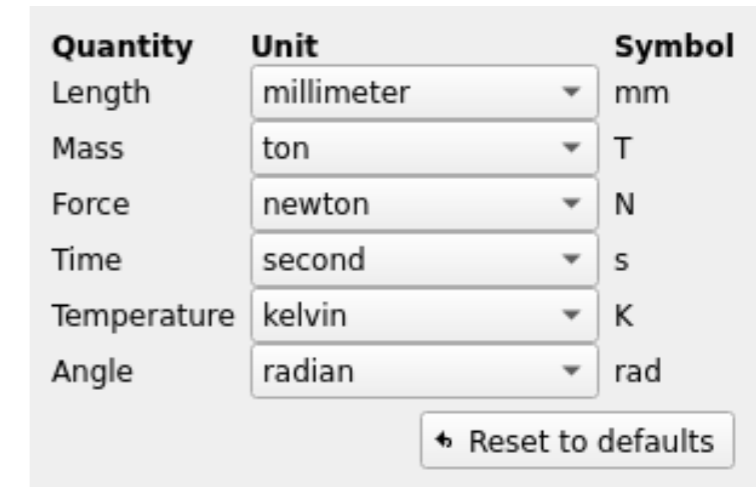


Figure 5: Property panel - units

## 2.1 Geometry

One line is required to build the model.

DIANAIE

**Main menu** → Geometry → Create → Line  [Fig. 6]

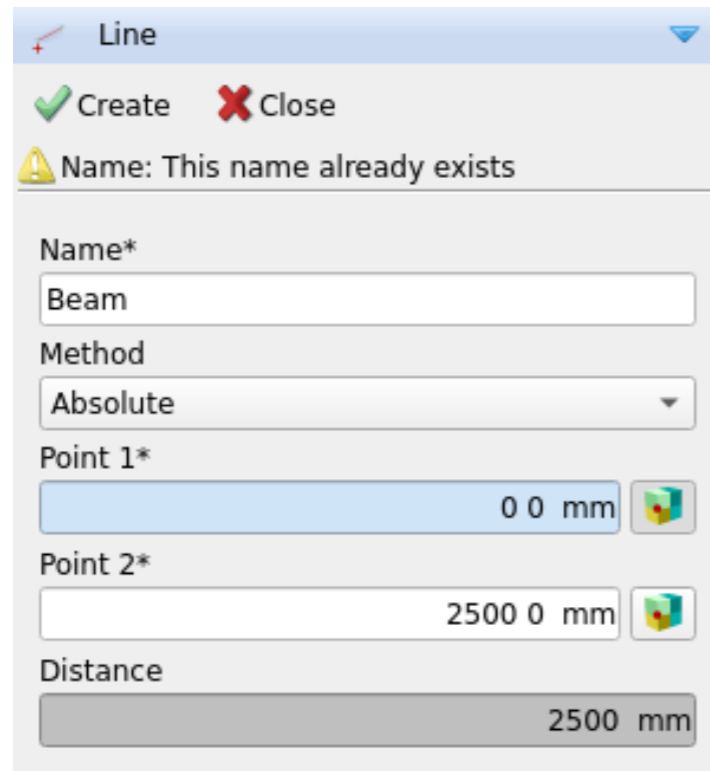


Figure 6: Add line









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]

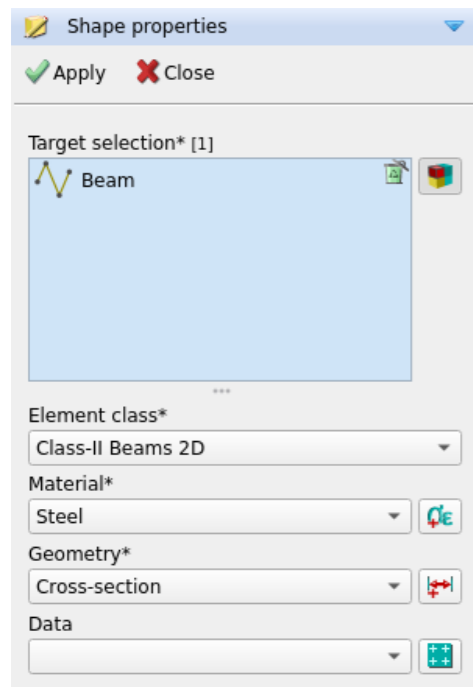


Figure 8: Property assignments

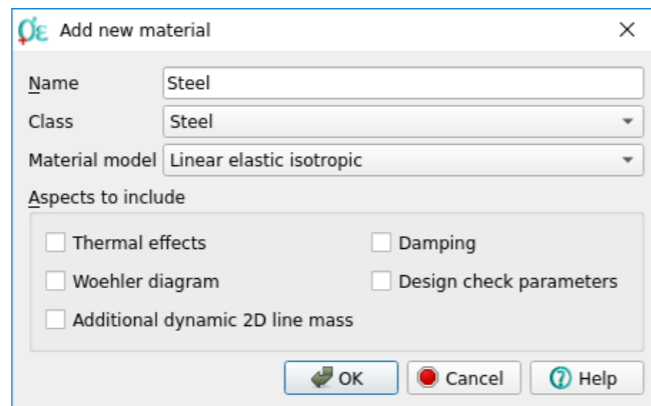


Figure 9: Add new material

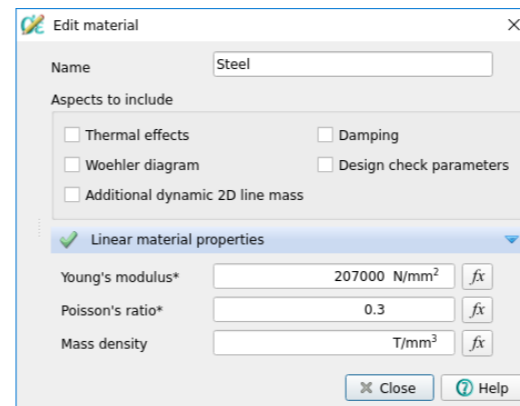


Figure 10: Edit material

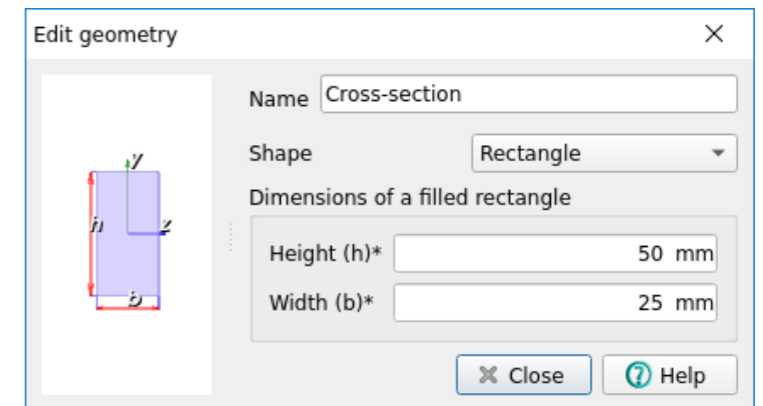



Figure 11: Element geometry properties

## 2.3 Boundary Conditions

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

DIANAIE

**Main menu** → Geometry → Assign → Supports  [Fig. 12] [Fig. 13]

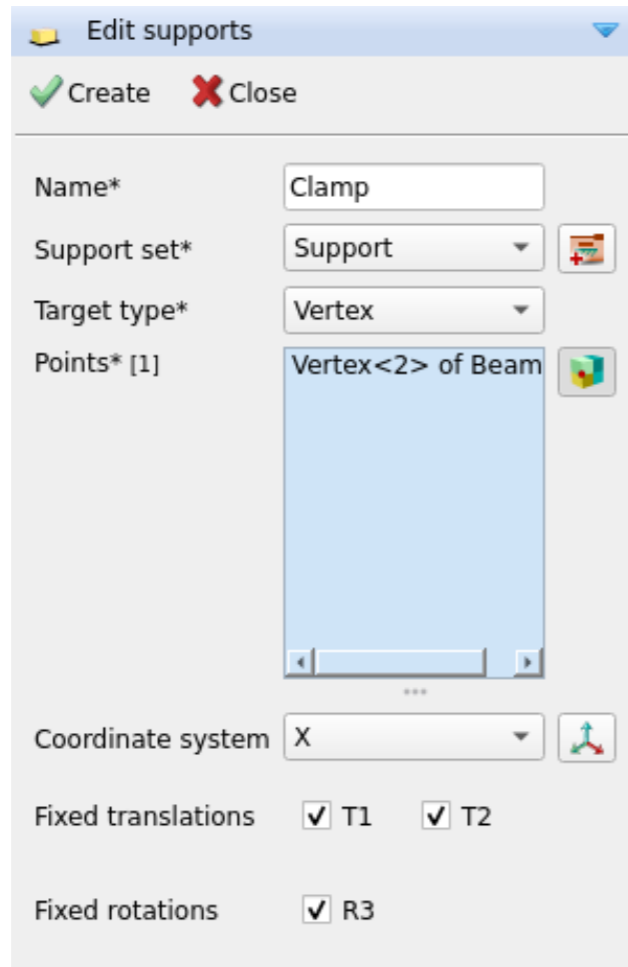


Figure 12: Clamp the left end

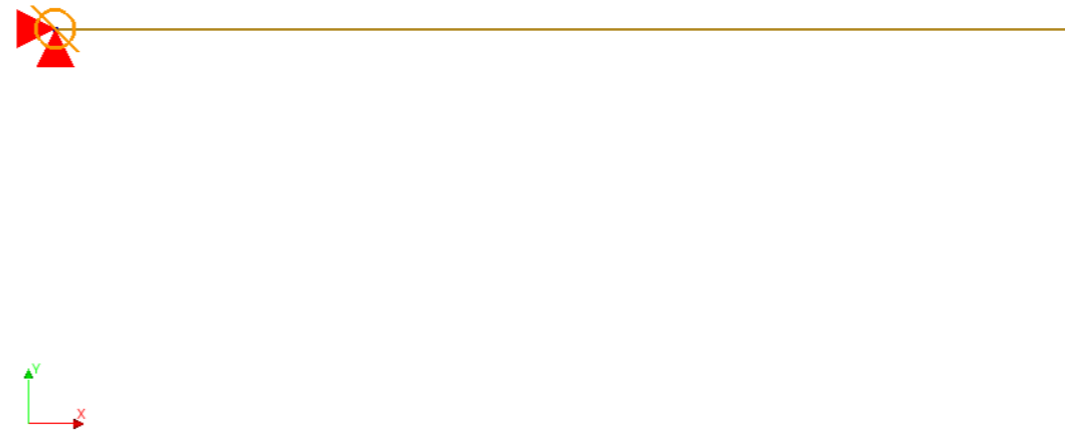



Figure 13: View of model - supports

## 2.4 Loads

A vertical nodal load pointing in the negative  $Y$  direction is applied at the right end of the beam.

DIANAIE

**Main menu** → Geometry → Assign → Loads  [Fig. 14] [Fig. 15]

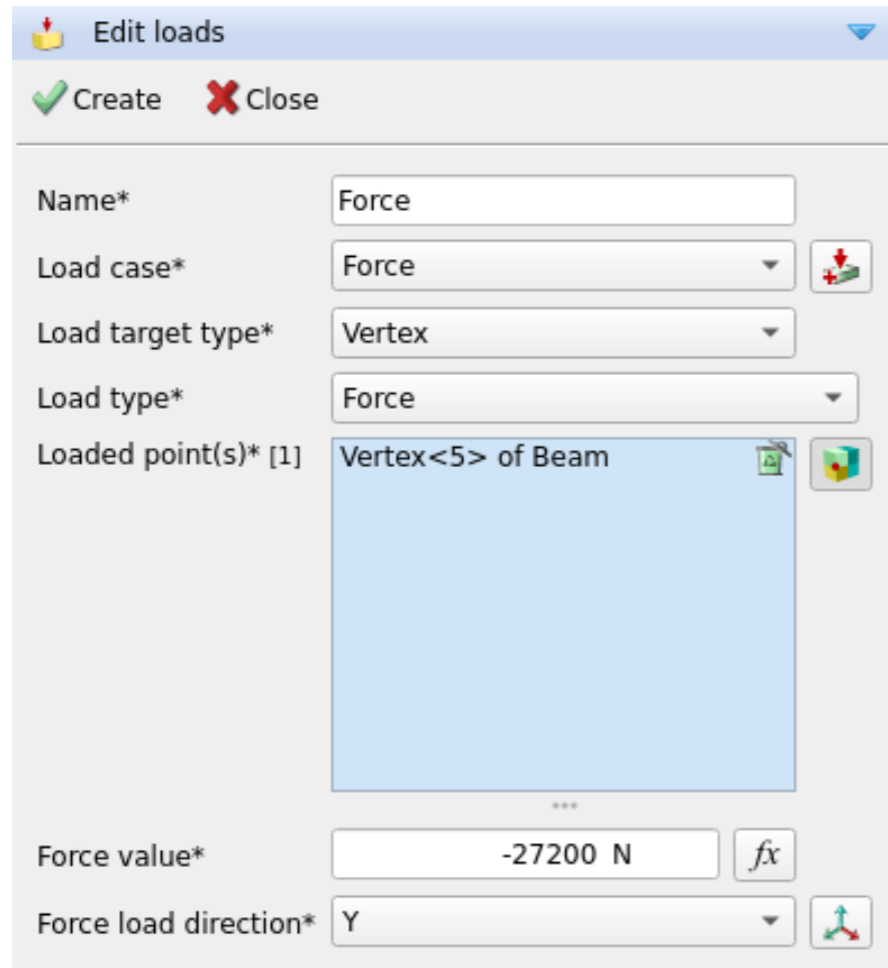


Figure 14: Attach nodal load to the right end



Figure 15: View of model - supports and load




## 2.5 Mesh

The mesh is defined by dividing the line into eight parts.

DIANAIE

**Main menu** → Geometry → Mesh → Mesh properties  [Fig. 16]

**Main menu** → Geometry → Mesh → Generate mesh 

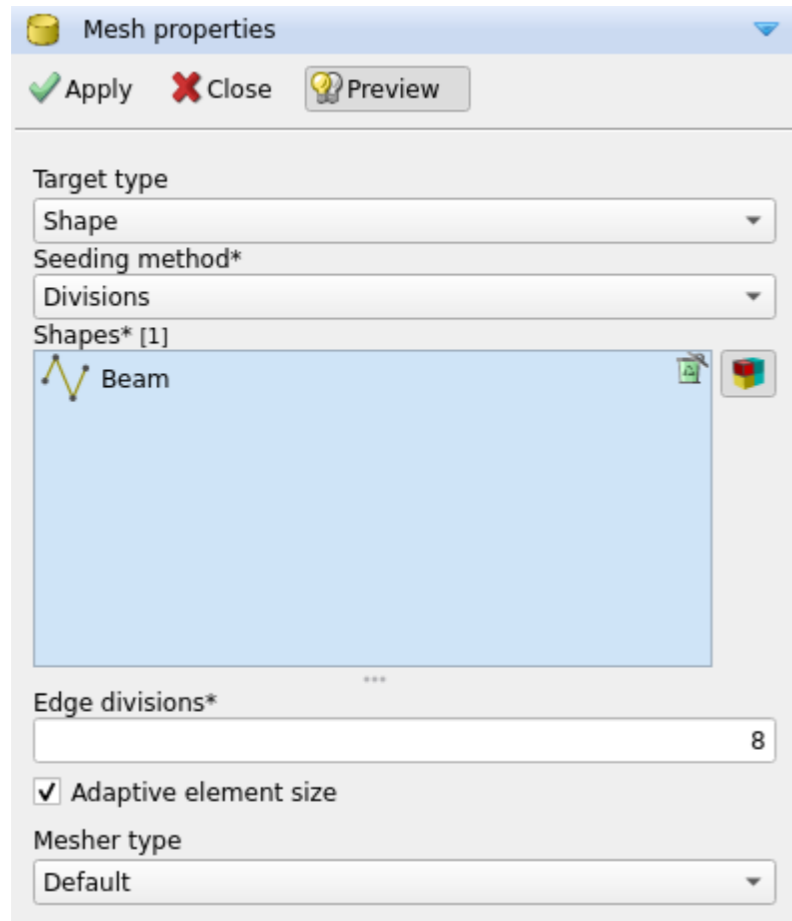


Figure 16: Mesh properties

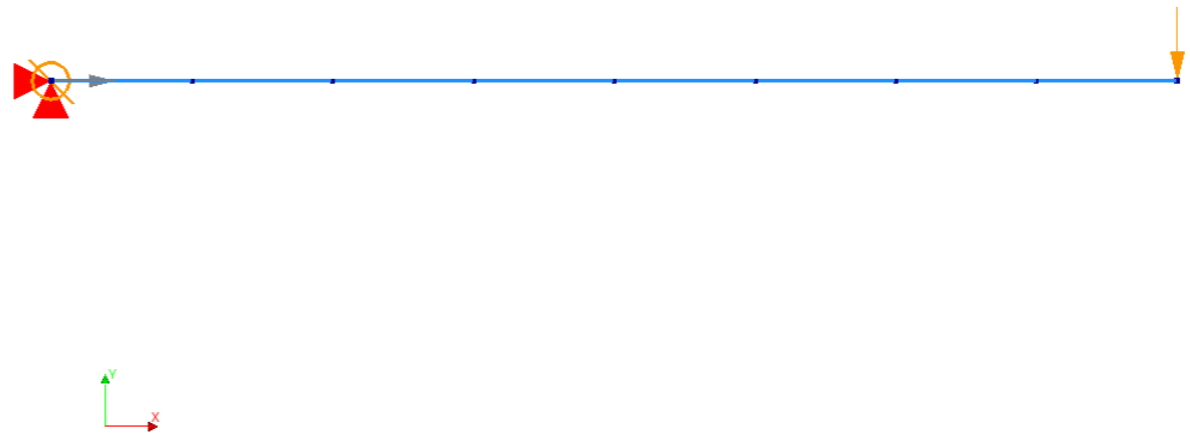





Figure 17: Finite element mesh


# 3 Linear Static Analysis


## 3.1 Commands

We perform a structural linear static analysis.

**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 

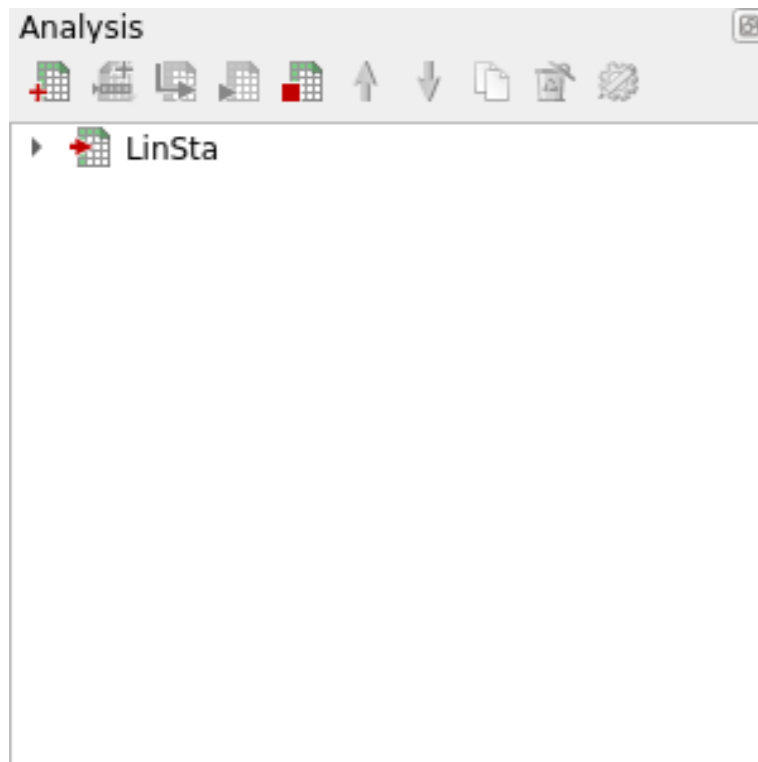


Figure 18: Analysis browser

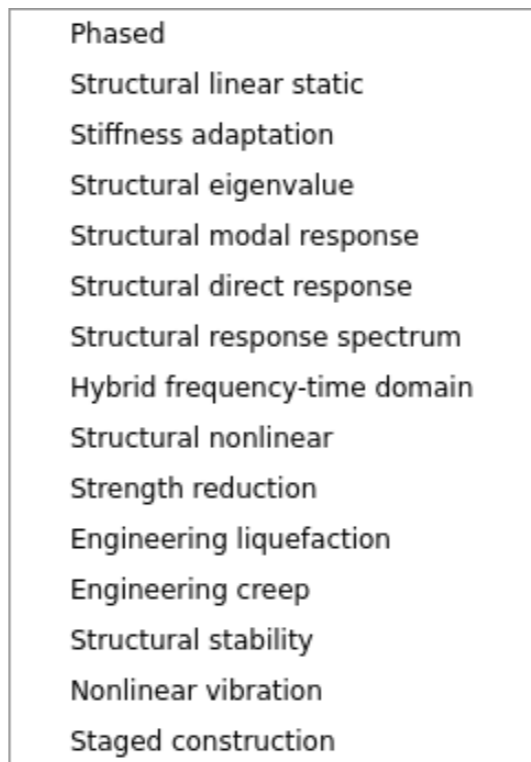


Figure 19: Command menu

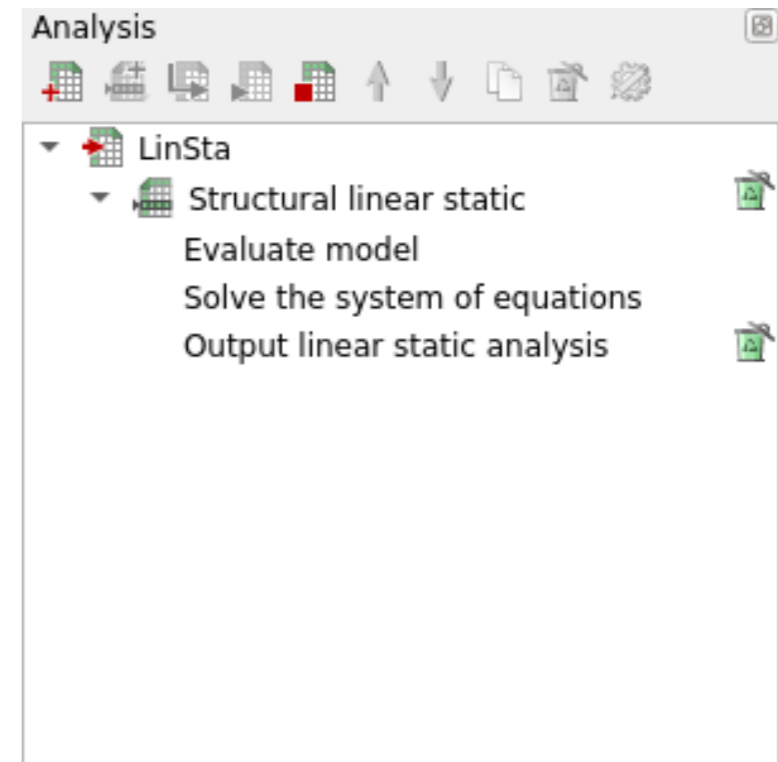


Figure 20: Analysis browser

## 3.2 Results


### 3.2.1 Displacements

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

DIANAIE

**Main menu** → Results → Absolute deformed results 

**Main menu** → Results → Deformed mesh feature edges 

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

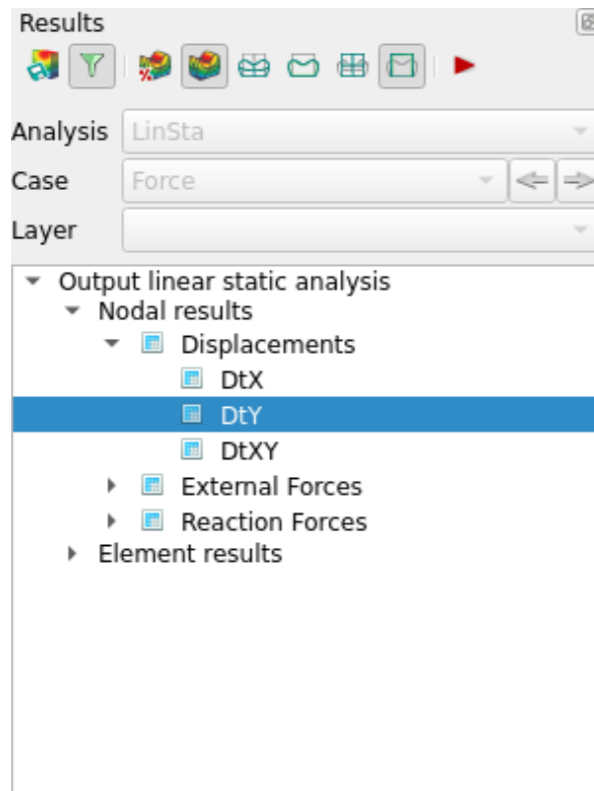


Figure 21: Results browser

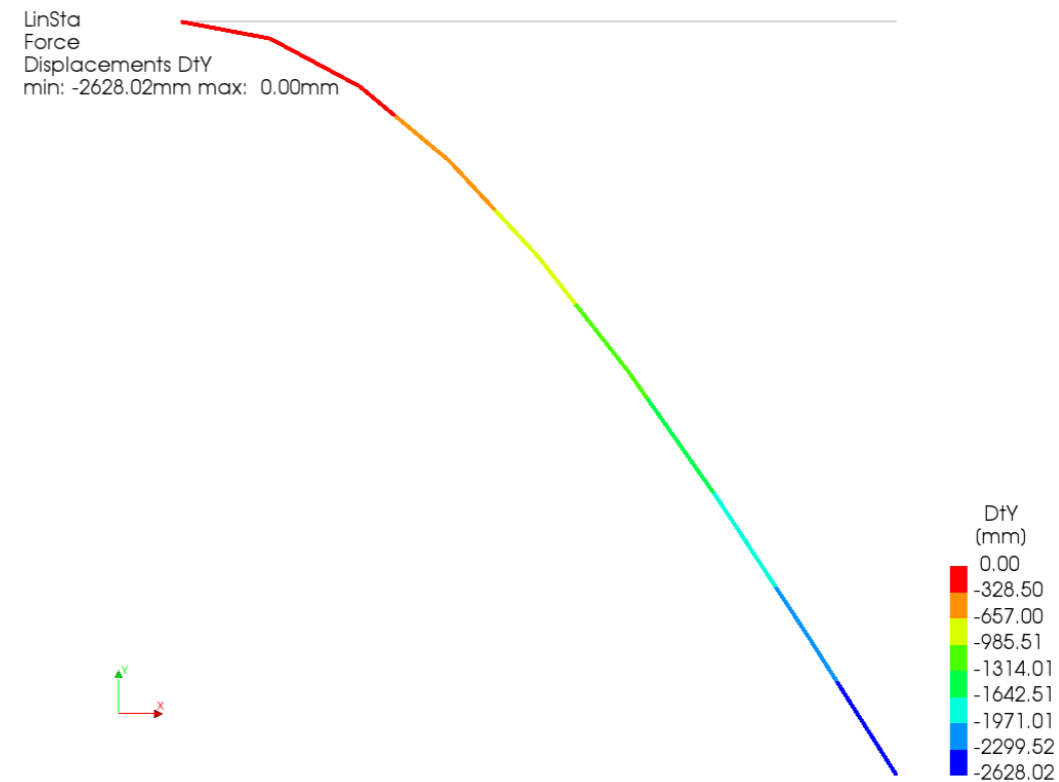



Figure 22: Absolute deflection

### 3.2.2 Reaction Forces

The reaction forces can also be represented to check whether force equilibrium is met. According to Figure 24 vertical force equilibrium is fulfilled.

DIANAIE

**Main menu** → Results → Deformed mesh feature edges 

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

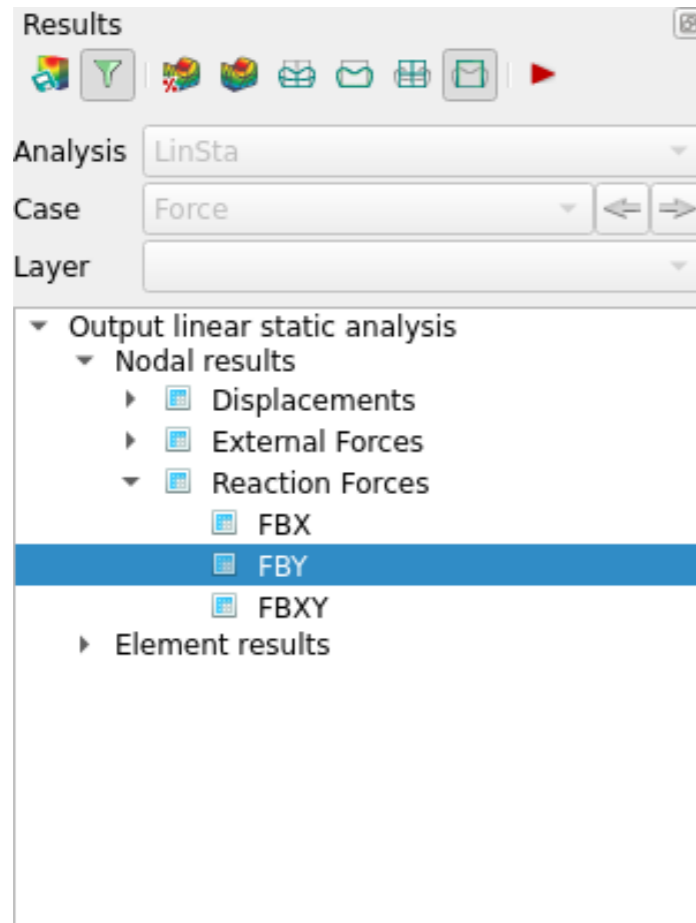


Figure 23: Results browser

LinSta  
Force  
Reaction Forces FBY  
min: 2.72e+04N max: 2.72e+04N

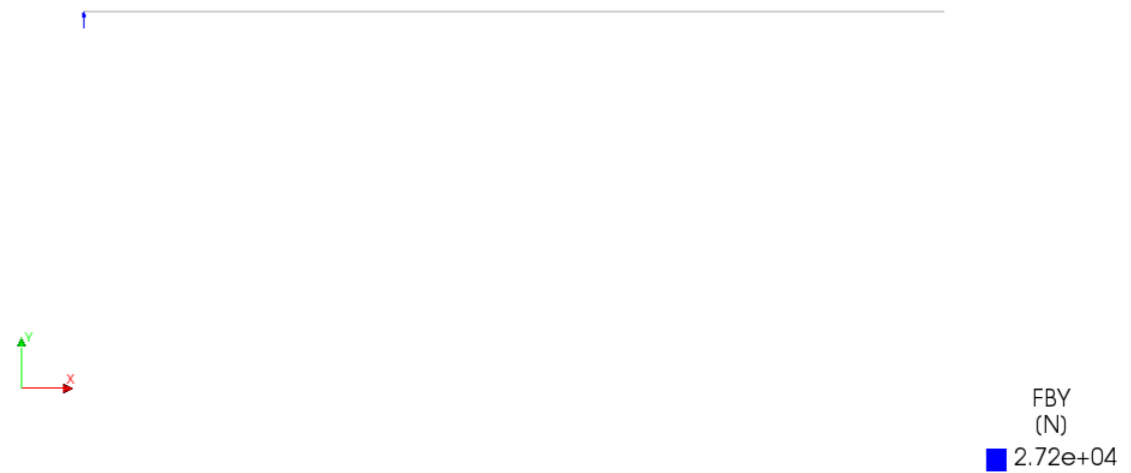




Figure 24: Reaction force

### 3.2.3 Cross-section Moments

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

DIANAIE

**Main menu** → Results → Deformed mesh feature edges 

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

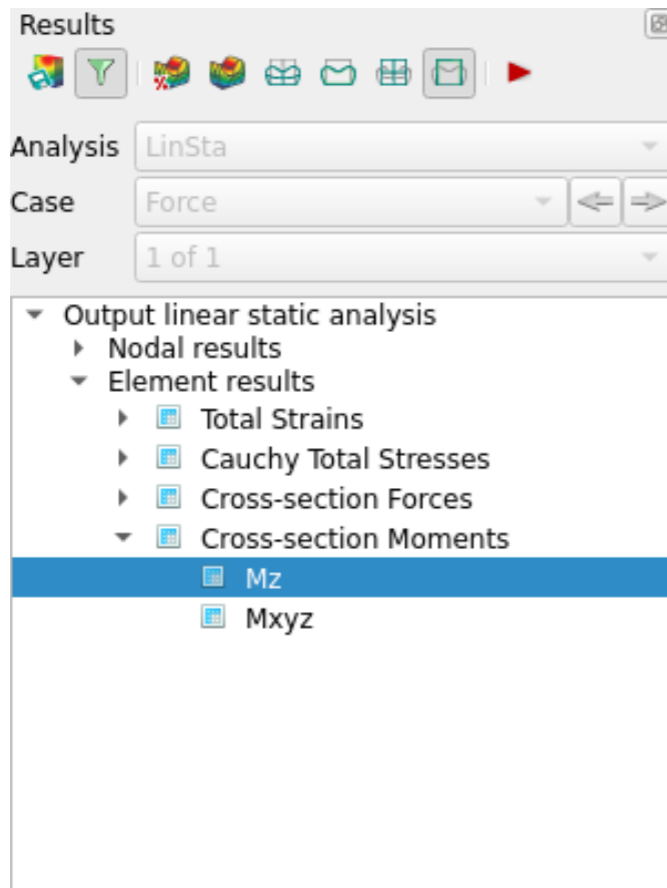


Figure 25: Results browser

LinSta  
Force  
Cross-section Moments Mz  
min: -1.96e-06Nmm max: 6.80e+07Nmm

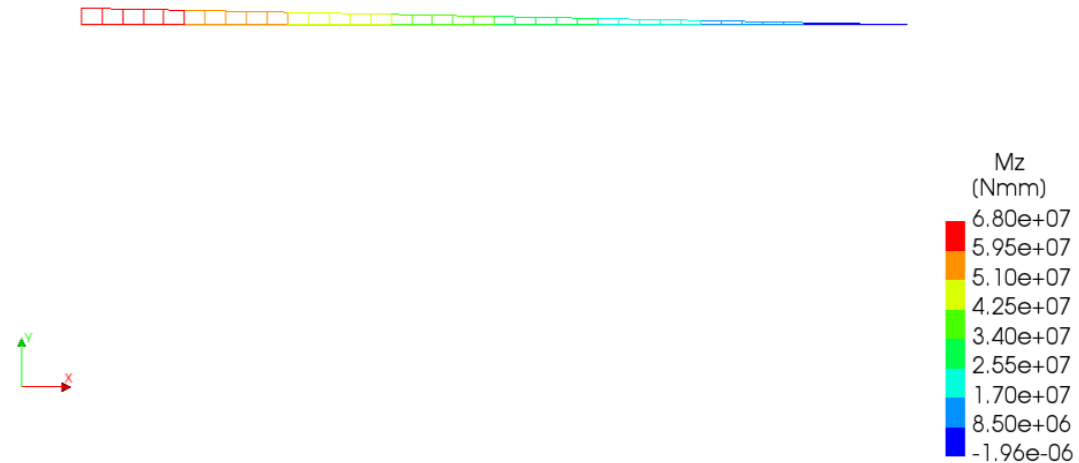


Figure 26: Bending moment distribution


### 3.2.4 Cauchy Total Stresses

Lastly we assess the Cauchy 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.

DIANAIE

**Main menu** → Results → Normalized deformed results 

**Main menu** → Results → Deformed mesh feature edges 

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

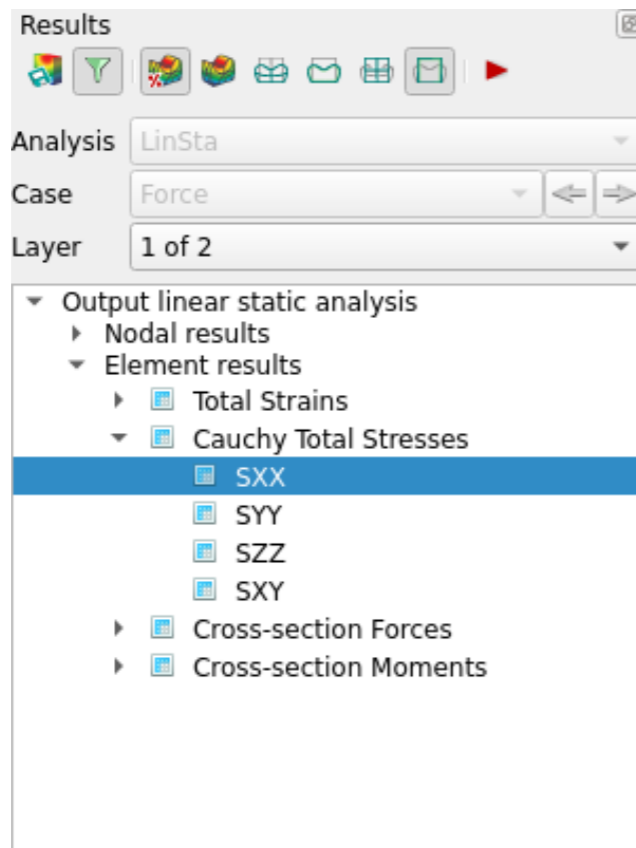


Figure 27: Results browser

LinSta  
Force  
Cauchy Total Stresses SXX layer 1  
min: -6528.00N/mm<sup>2</sup> max: 0.00N/mm<sup>2</sup>

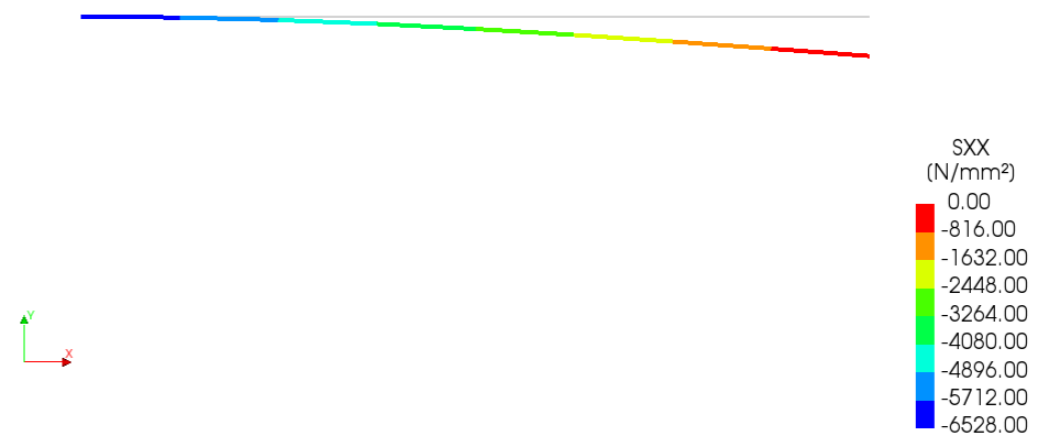


Figure 28: Normal stresses

## Appendix A Additional Information

Folder: Tutorials/CantileverBeam

Number of elements  $\approx 8$

### 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.



[WWW.DIANAFEA.COM](http://WWW.DIANAFEA.COM)

© DIANA FEA BV

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 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

