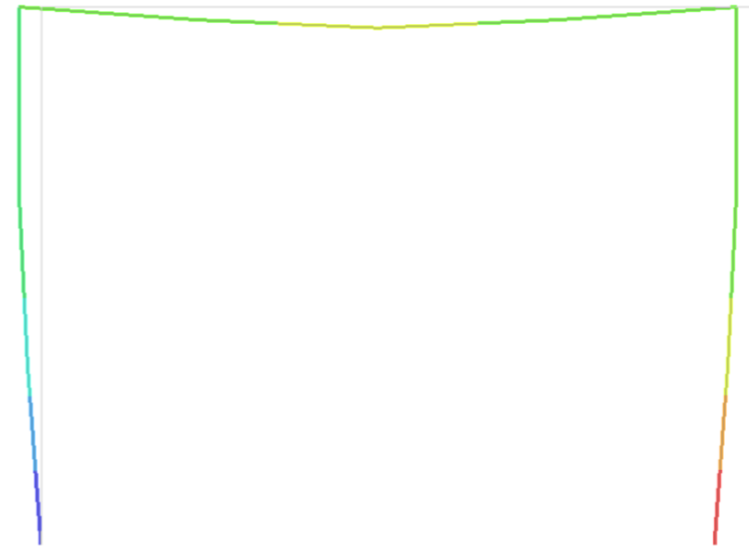


Tutorial

## Linear Analysis of a Simple Lattice Frame



## 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	8
2.4	Loads	9
2.5	Mesh	11
<b>3</b>	<b>Structural linear static analysis</b>	<b>12</b>
3.1	Commands	12
3.2	Results	13
	<b>Appendix A Additional Information</b>	<b>15</b>

# 1 Description

We analyze a simple lattice frame that is supported and loaded as illustrated in Figure 1. We chose to model the frame with ten beam elements in a two-dimensional analysis. We perform a structural linear static analysis on this structure.

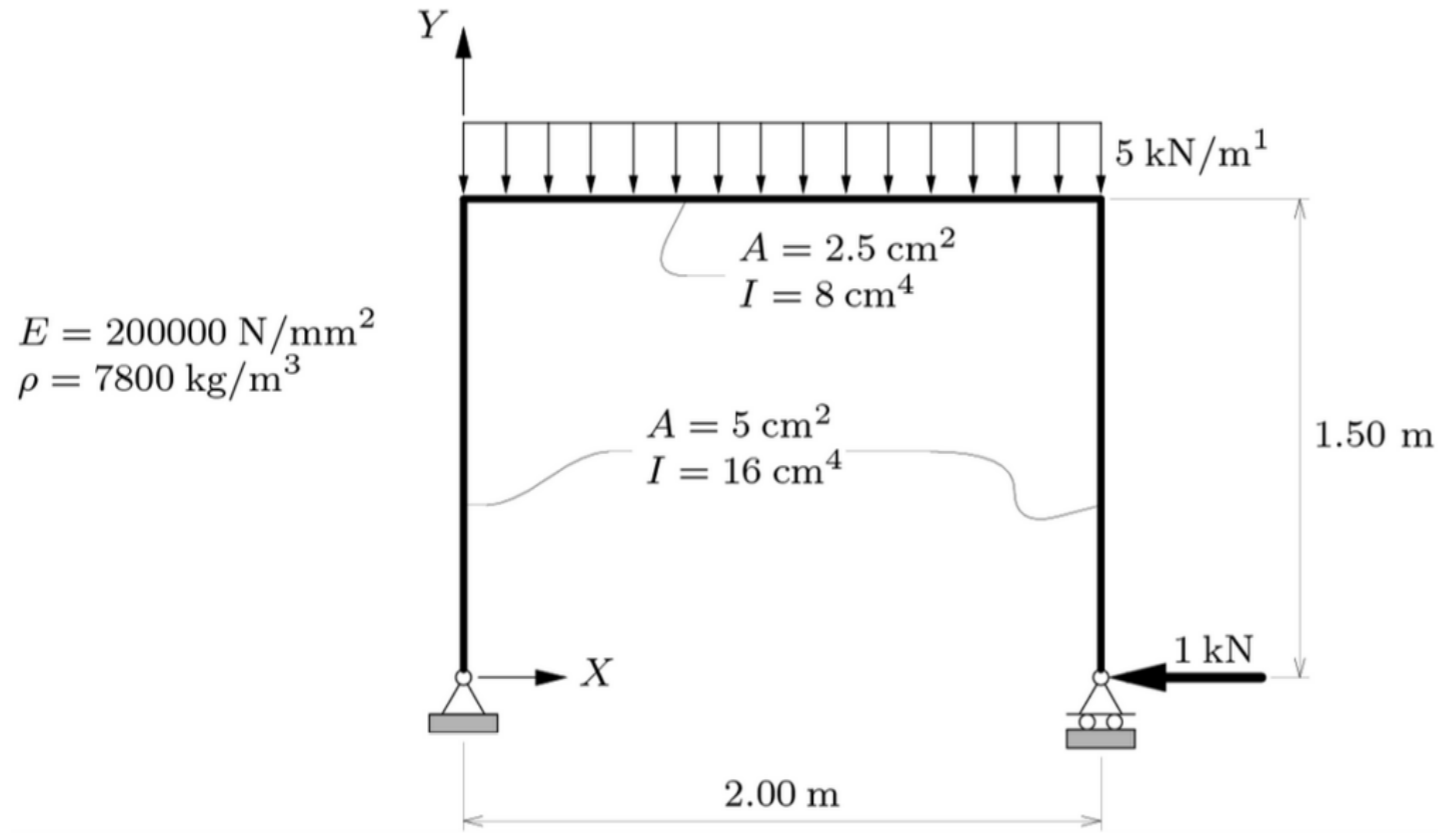


Figure 1: Geometry and characteristic of the frame

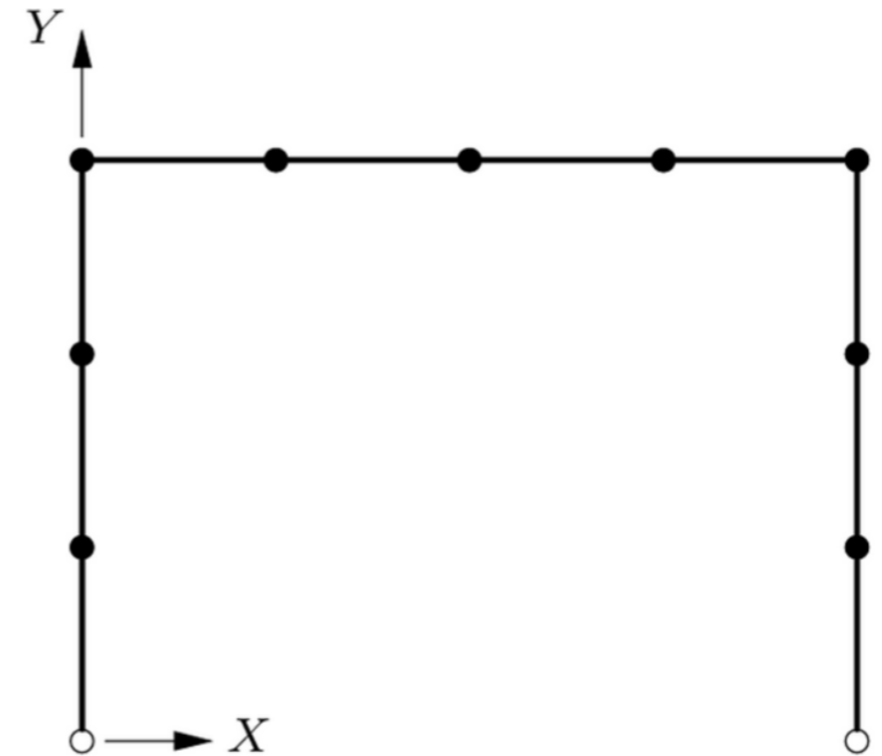



Figure 2: Discretization of the model

## 2 Finite Element Model

We start a new project for a two-dimensional structural analysis. The model size is set to 10 m. The units used are shown in Figure 5.

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

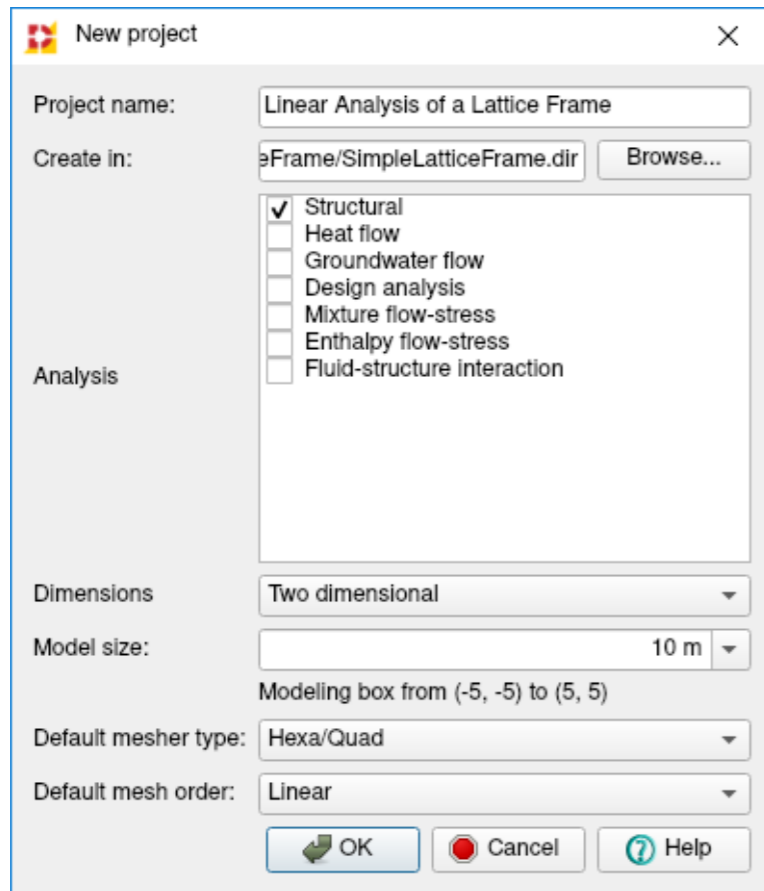


Figure 3: New project dialog

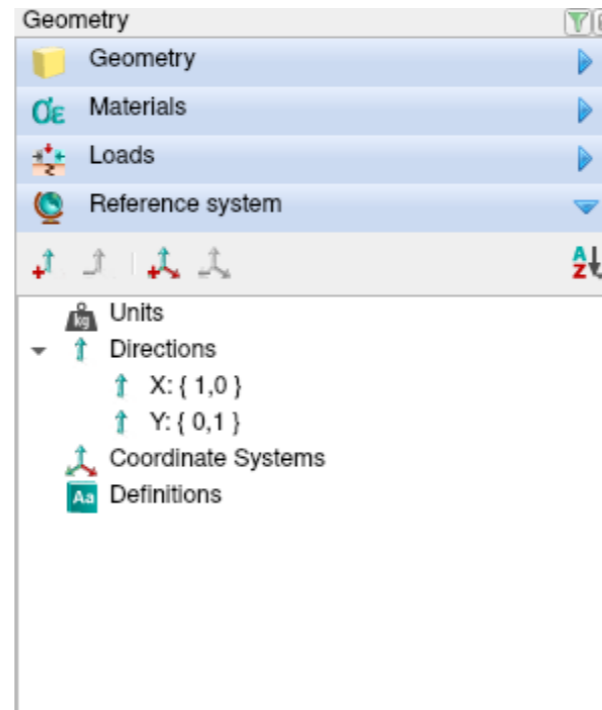


Figure 4: Geometry browser - units


Quantity	Unit	Symbol
Length	meter	m
Mass	kilogram	kg
Force	newton	N
Time	second	s
Temperature	kelvin	K
Angle	radian	rad

[Reset to defaults](#)

Figure 5: Property panel - units

## 2.1 Geometry

We create three lines which form the lattice frame.

**Main menu** → Geometry → Create → Line  [Fig. 6] - [Fig. 8]  
 < Repeat 3x >

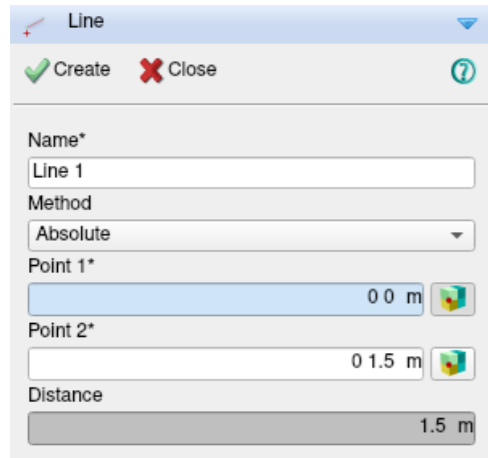


Figure 6: Add Line 1

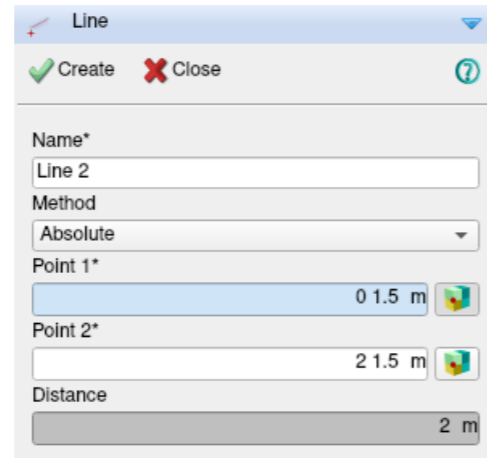


Figure 7: Add Line 2

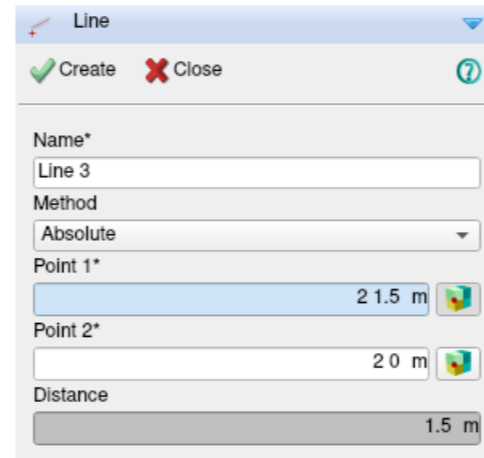


Figure 8: Add Line 3

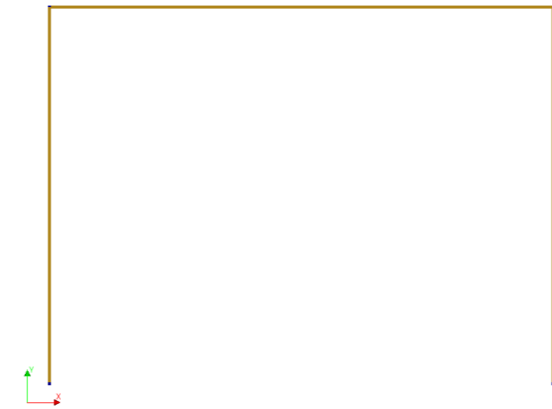


Figure 9: Geometry

## 2.2 Properties

We define the material and geometry properties of the frame. The element class is *Class-I Beams 2D*. Distinction is made between the vertical beams and the horizontal beam. The vertical beams have a larger cross-sectional area and a higher moment of inertia. The material properties are the same: Young's modulus  $E = 2e+11 \text{ N/m}^2$ , Poisson's ratio  $\nu = 0.3$  and mass density  $\rho = 7800 \text{ kg/m}^3$ . First, we assign the properties to the vertical beams.

**Main menu** → Geometry → Assign → Shape properties [Fig. 10]  
 Shape properties → Material → Add material [Fig. 11] → Edit material [Fig. 12]  
 Shape properties → Geometry → Add new geometry [Fig. 13]

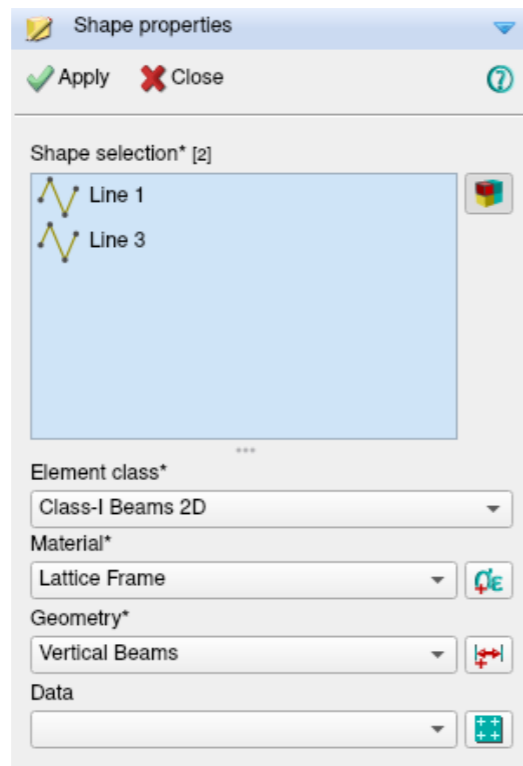


Figure 10: Property assignments vertical beams

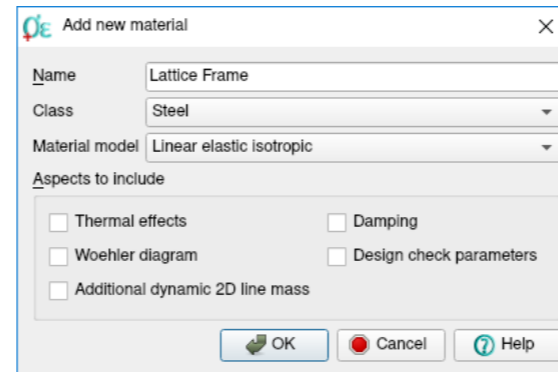


Figure 11: Add new material *Lattice Frame*

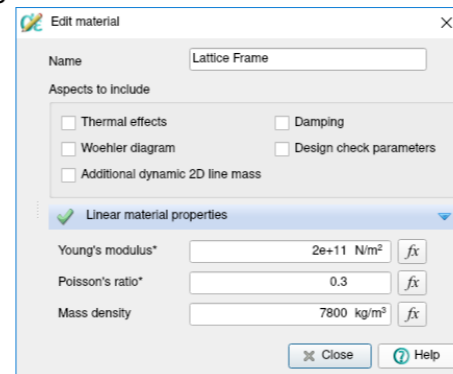


Figure 12: Edit material *Lattice Frame*

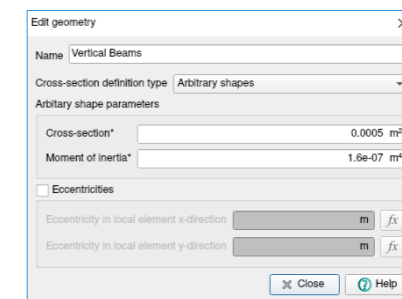




Figure 13: Edit geometry

Now we define the properties of the horizontal beam.

**Main menu** → Geometry → Assign → Shape properties  [Fig. 14]

Shape properties  → Geometry → Add new geometry  [Fig. 15]

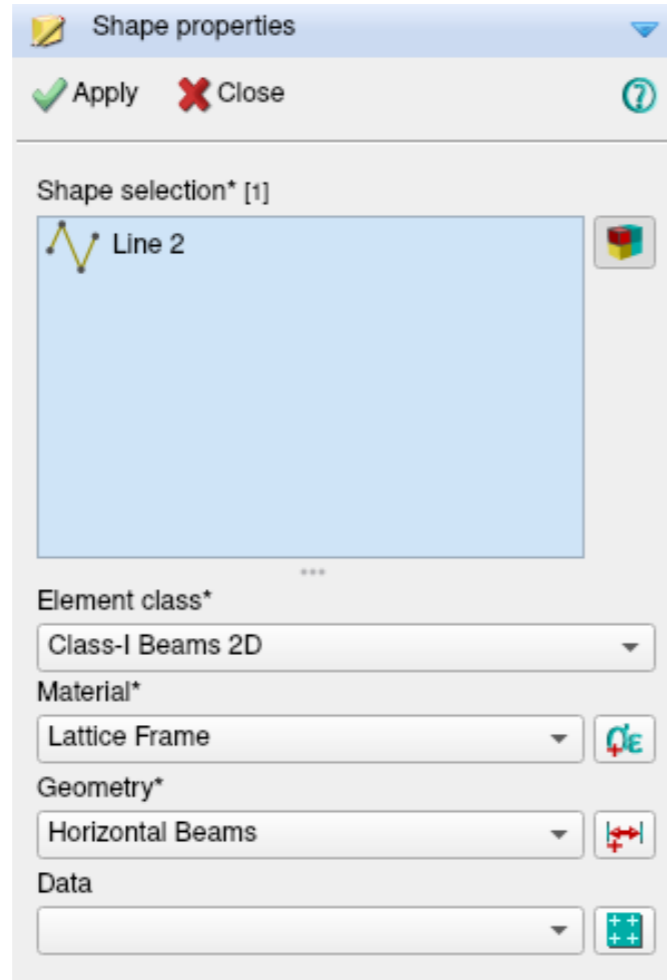


Figure 14: Property assignments horizontal beam

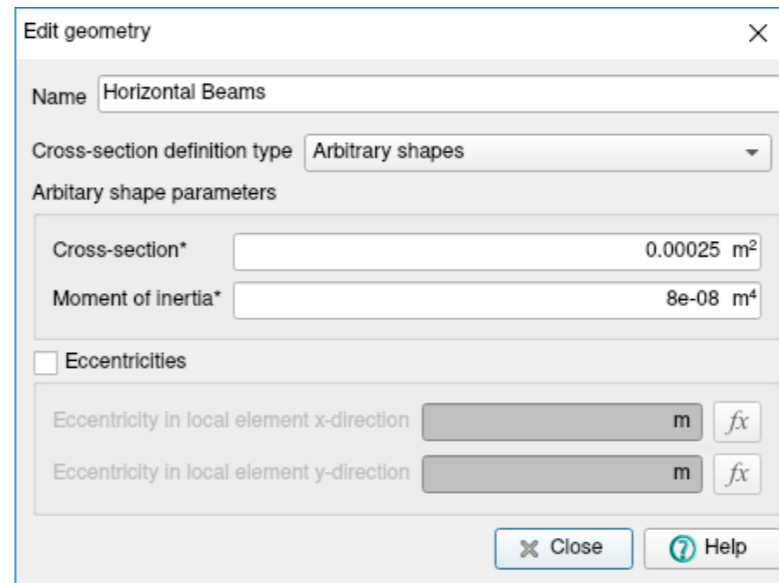


Figure 15: Edit geometry

## 2.3 Boundary Conditions

We create the supports of the frame. The bottom left corner is constrained in both  $X$  and  $Y$  directions and the bottom right corner in  $Y$  direction [Fig. 18].

**Main menu** → Geometry → Assign → Supports [Fig. 16]

**Main menu** → Geometry → Assign → Supports [Fig. 17]

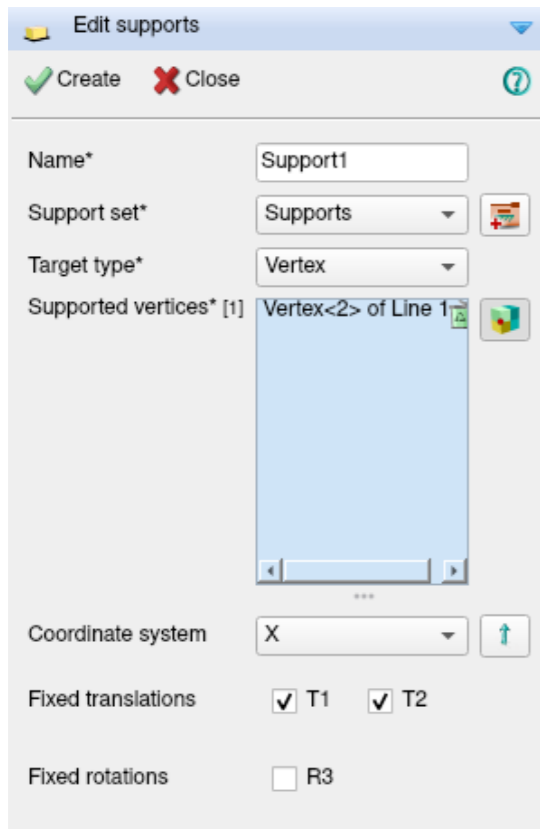


Figure 16: Attach left support

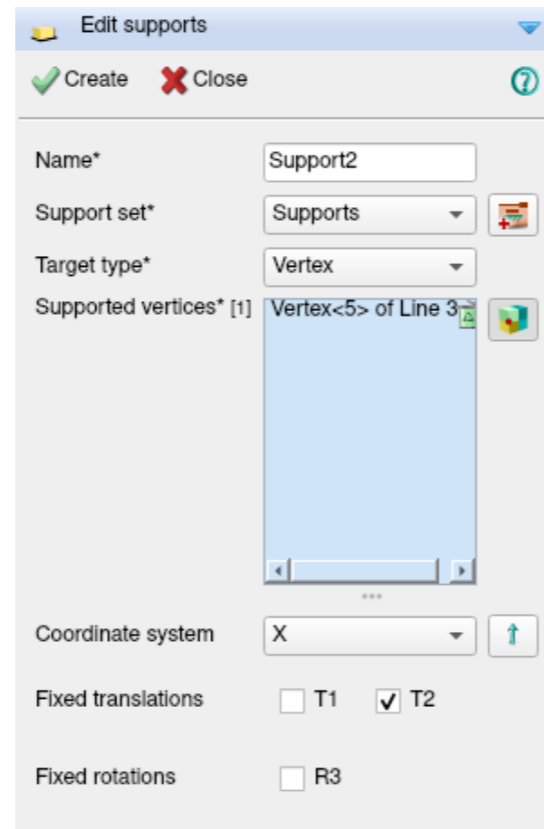


Figure 17: Attach right support

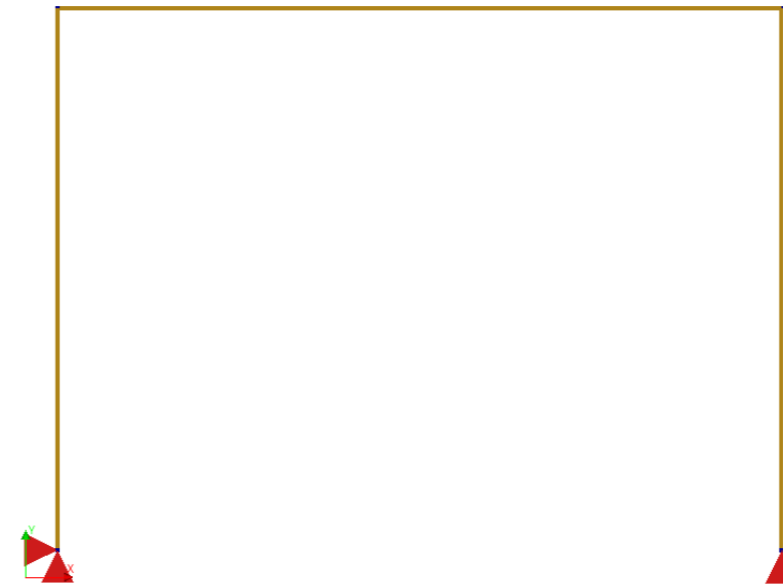



Figure 18: Supports



## 2.4 Loads

The loading consists of three components:

- at the bottom right corner a point load of 1000 N in the negative  $X$  direction
- on the horizontal beam a line load of 5000 N/m in the negative  $Y$  direction
- self-weight

**Main menu** → Geometry → Assign → Loads  [Fig. 19] [Fig. 20]

**Main menu** → Geometry → Assign → Global loads  [Fig. 21]

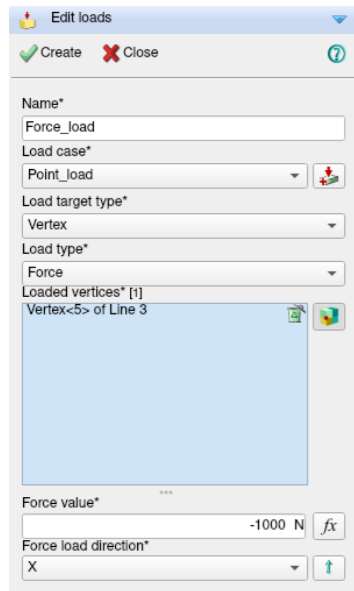


Figure 19: Attaching a point load

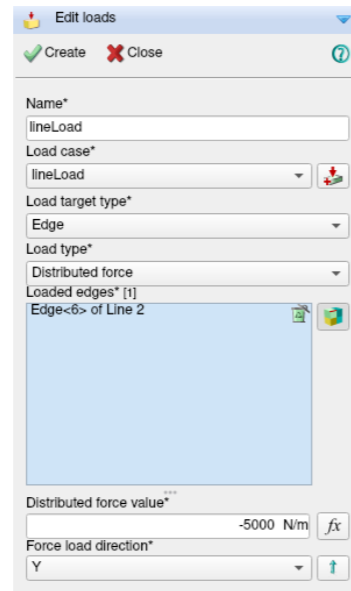


Figure 20: Attaching a line load



Figure 21: Define self-weight

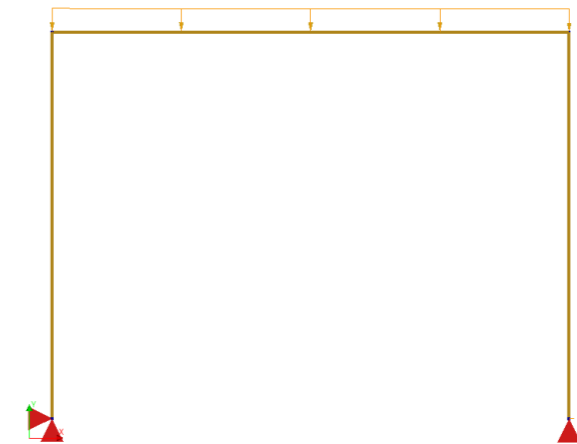




Figure 22: Overview of the applied loads

We define a load combination table. This enables us to simultaneously take into account various load cases. In this case we create a *Geometry load combination 4* to combine the point load and the line load.

**Main menu** → Geometry → Loads → Open geometry load combinations table  [Fig. 23] → Add geometry load combination  [Fig. 24]

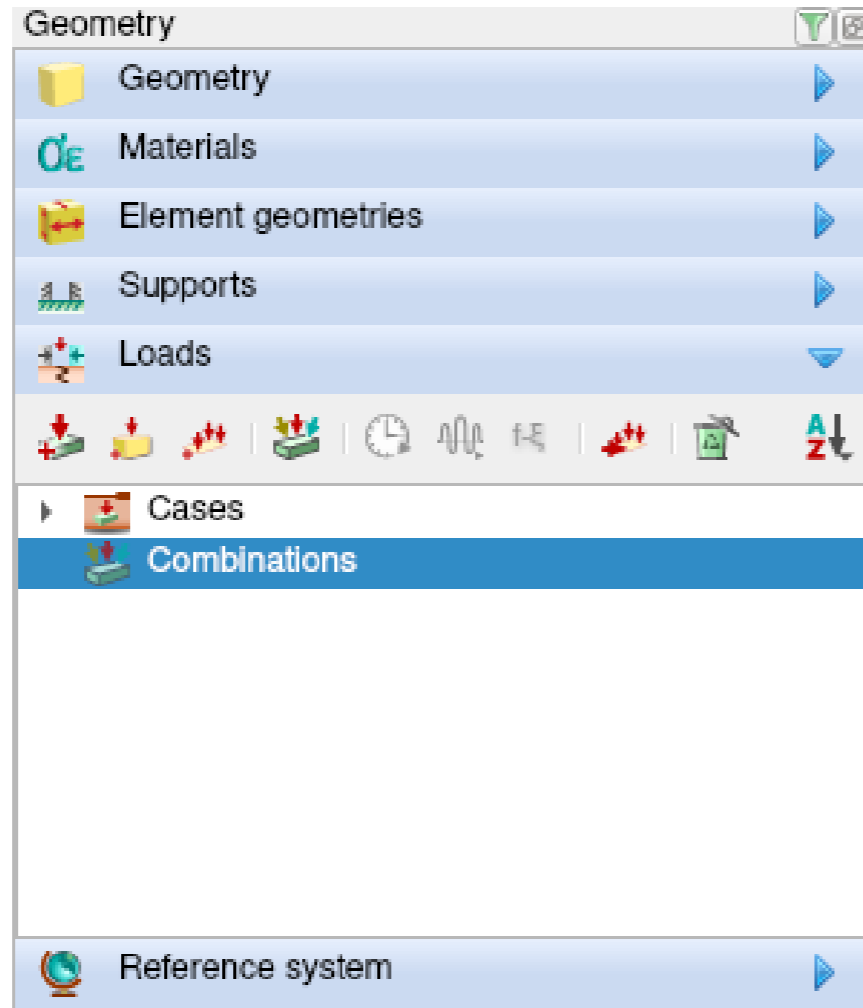


Figure 23: Geometry browser - load combinations

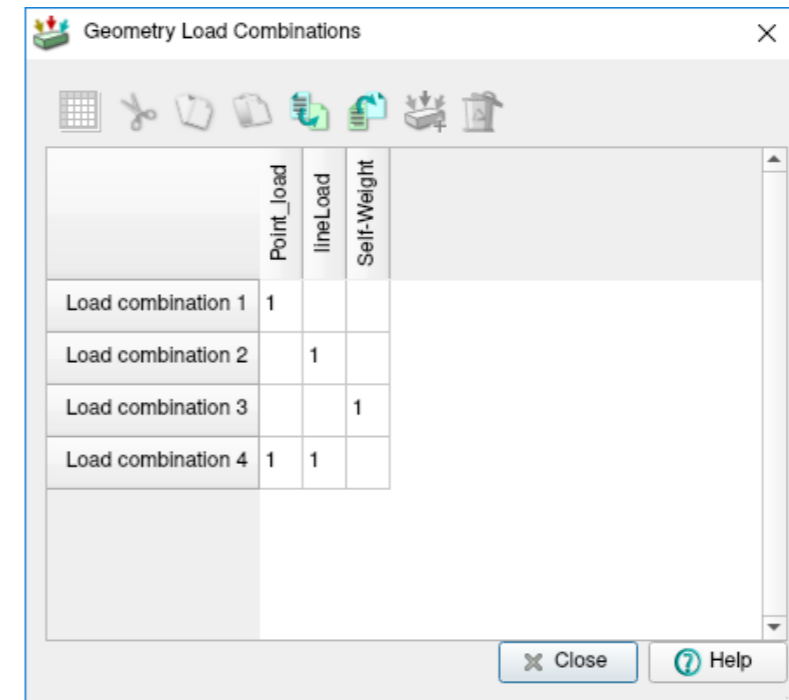


Figure 24: Load combinations

## 2.5 Mesh

We create a mesh by using the seeding method of *divisions*. We use 3 divisions per edge for the vertical beams and 4 divisions for the horizontal beam.

**Main menu** → Geometry → Mesh → Mesh properties  [Fig. 25] [Fig. 26]

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

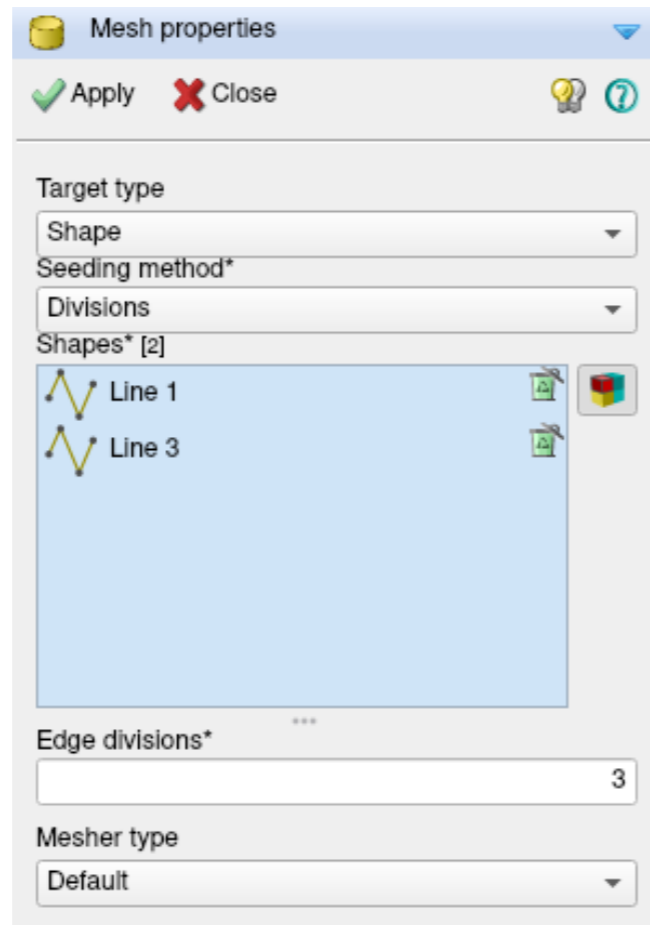


Figure 25: Mesh properties for vertical beams

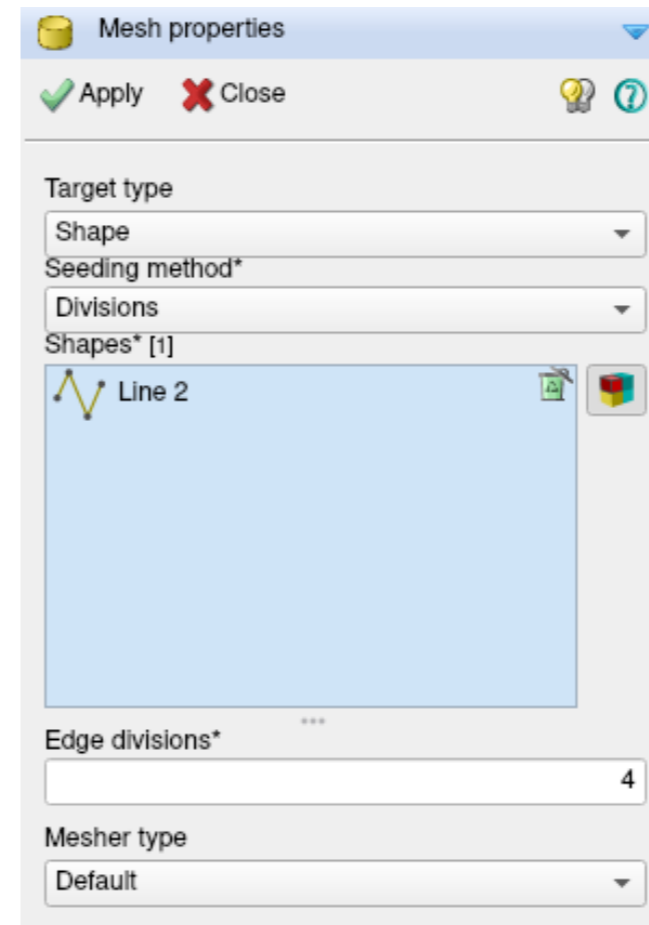




Figure 26: Mesh properties for horizontal beam


### 3 Structural linear static analysis

#### 3.1 Commands

We set up the commands for a structural linear static analysis.

**Main menu** → Analysis → Add analysis  [Fig. 27]

**Analysis browser** → Analysis1  → Add command → Structural linear static [Fig. 28]

**Main menu** → Analysis → Run selected analysis 

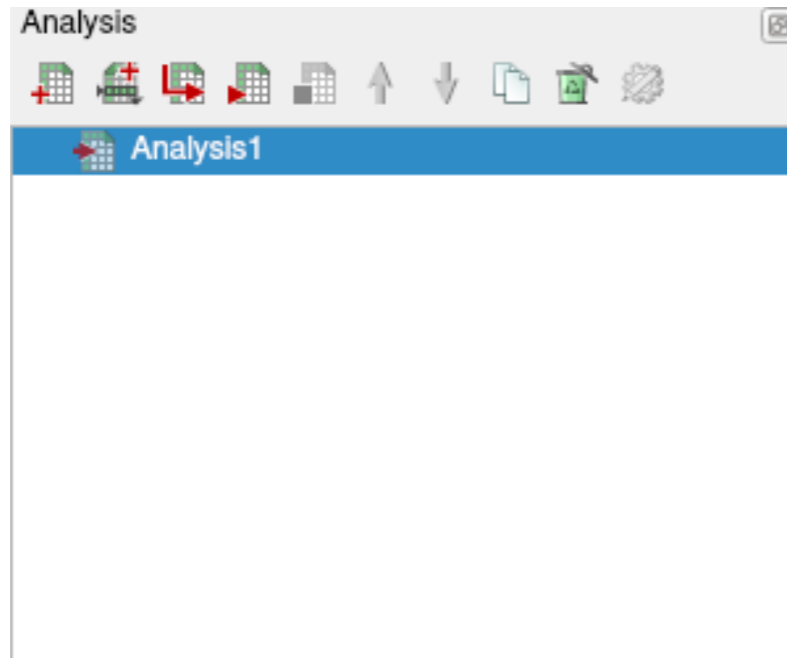


Figure 27: Analysis browser

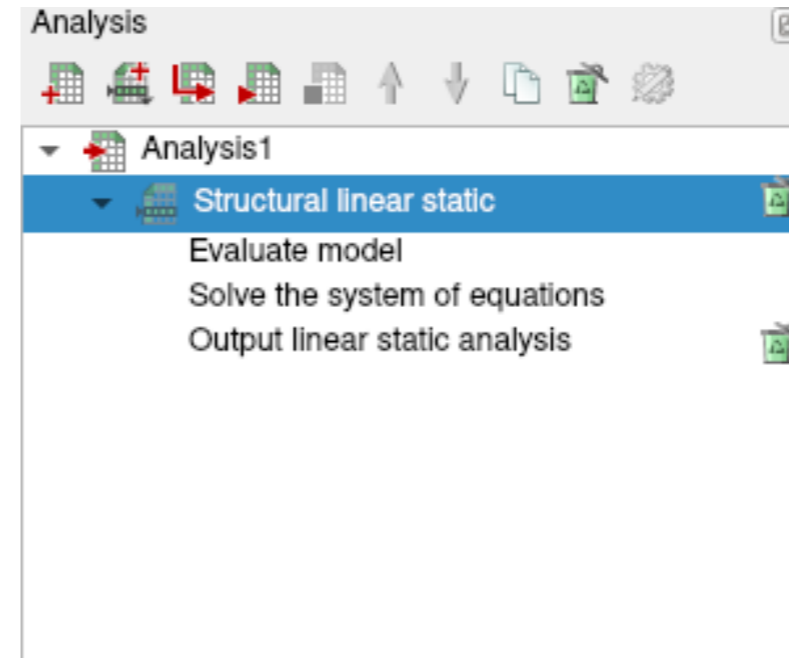



Figure 28: Analysis browser

### 3.2 Results

We present the deformed model and bending moments due to the external loads (without self-weight).

**Results browser** → Geometry load combination 4 → Output linear static analysis → Nodal results → Displacements → TDtXY [Fig. 29]

**Results browser** → Geometry load combination 4 → Output linear static analysis → Element results → Cross-section Moments → Mz [Fig. 30]

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

Analysis1  
Load combination 4  
Displacements DtXY  
min: 0.00e+00m max: 3.91e-02m

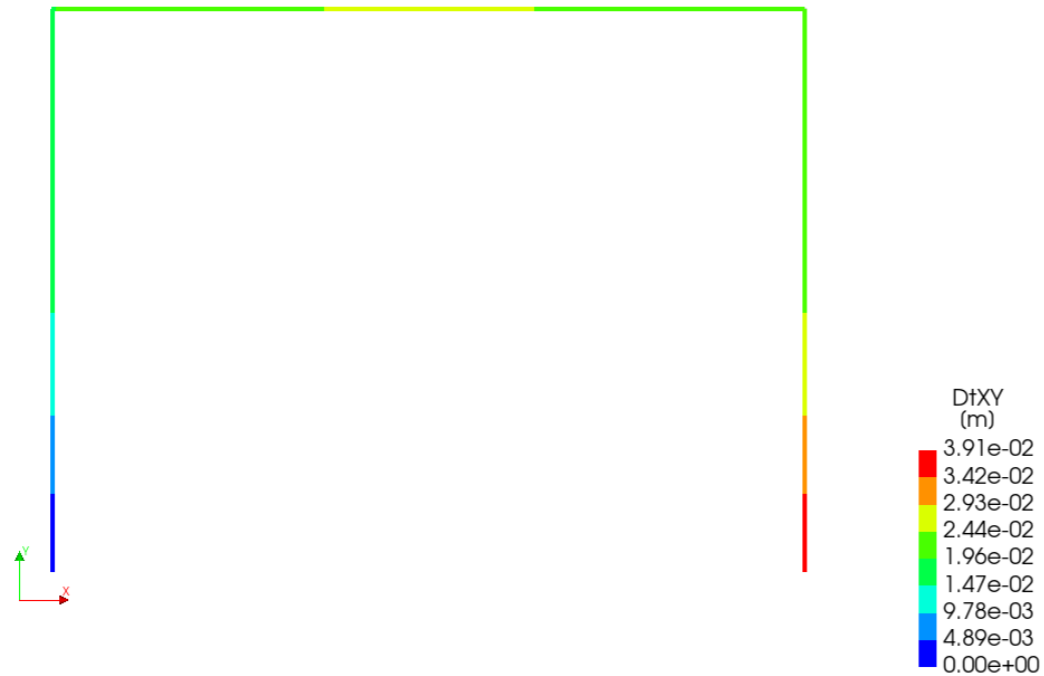


Figure 29: Total displacement DtXY

Analysis1  
Load combination 4  
Cross-section Moments Mz  
min: -1000.00Nm max: 1500.00Nm

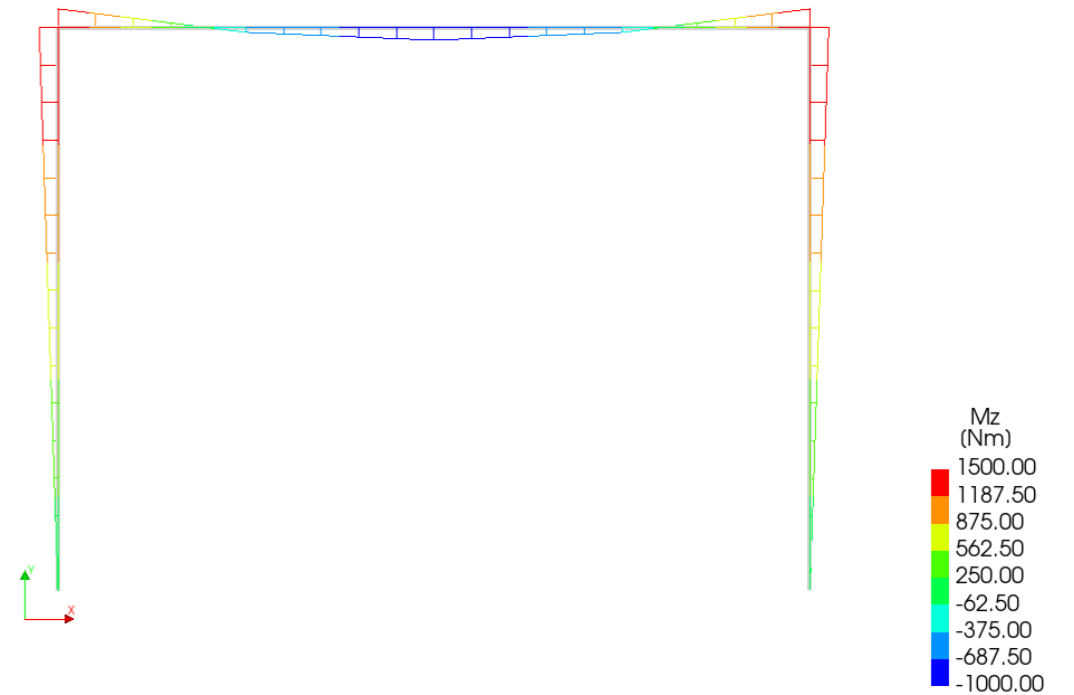



Figure 30: Cross-section moment Mz

We present the deformed model and bending moments only due to the self-weight of the structure.

**Results browser** → Geometry load combination 3 → Output linear static analysis → Nodal results → Displacements → TDtXY [Fig. 31]

**Results browser** → Geometry load combination 3 → Output linear static analysis → Element results → Cross-section Moments → Mz [Fig. 32]

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

Analysis1  
Load combination 3  
Displacements DtXY  
min: 0.00e+00m max: 1.20e-03m

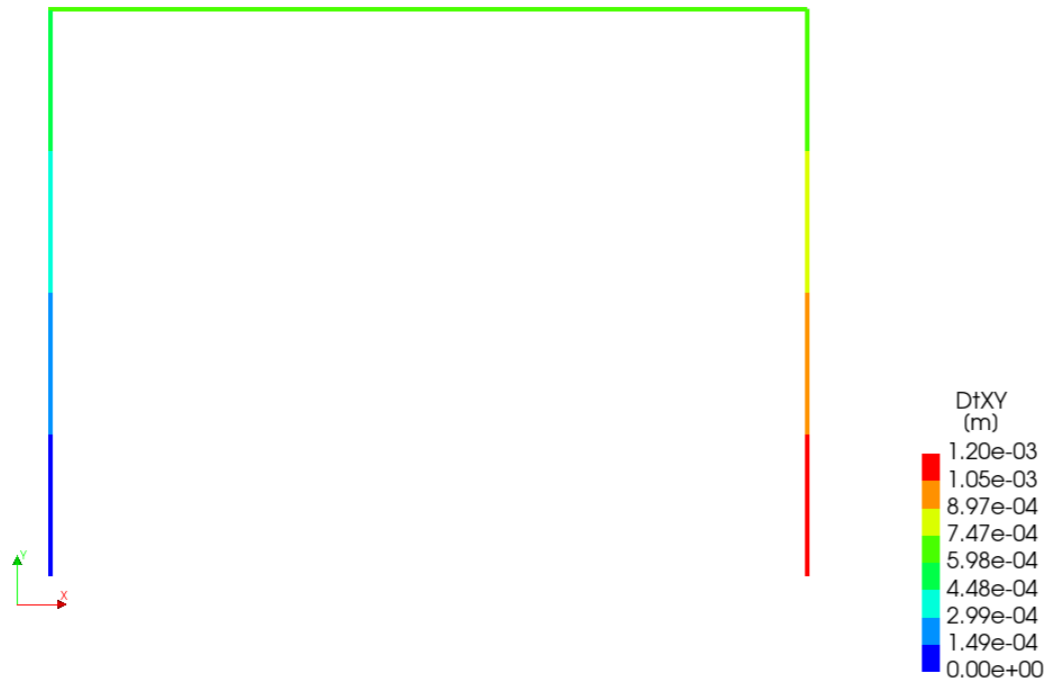


Figure 31: Total displacement DtXY

Analysis1  
Load combination 3  
Cross-section Moments Mz  
min: -9.56Nm max: 0.00Nm

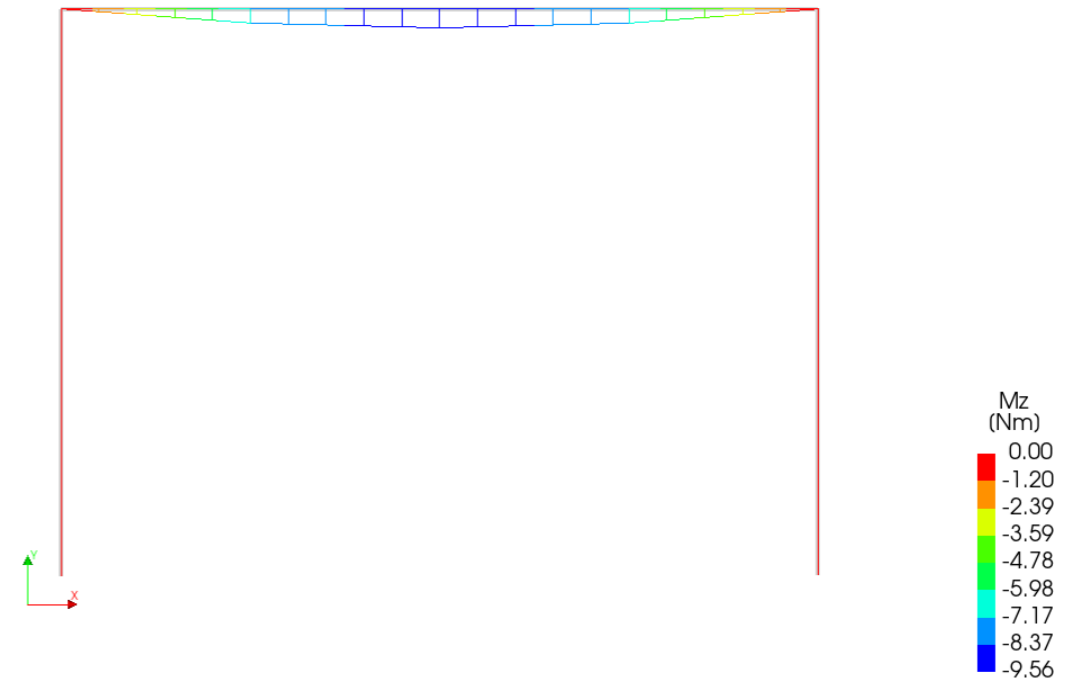


Figure 32: Cross-section moment Mz

## Appendix A Additional Information

Folder: Tutorials/SimpleLatticeFrame

Number of elements  $\approx 10$

Keywords:

ANALYS: linear static.

CONSTR: suppor.

ELEMEN: beam class1 l6ben.

LOAD: elemen force line node weight.

MATERI: elasti isotro.

OPTION: direct.

POST: binary ndiana.

PRE: dianai.

RESULT: cauchy displa extern force green moment reacti strain stress total.

Laan van Waalhaven 462  
2497 GR, The Hague  
The Netherlands

[DIANA.FEA.COM](http://DIANA.FEA.COM)

[INFO@DIANA.FEA.COM](mailto:INFO@DIANA.FEA.COM)

[+31\(0\)88 34 262 00](tel:+31(0)883426200)

