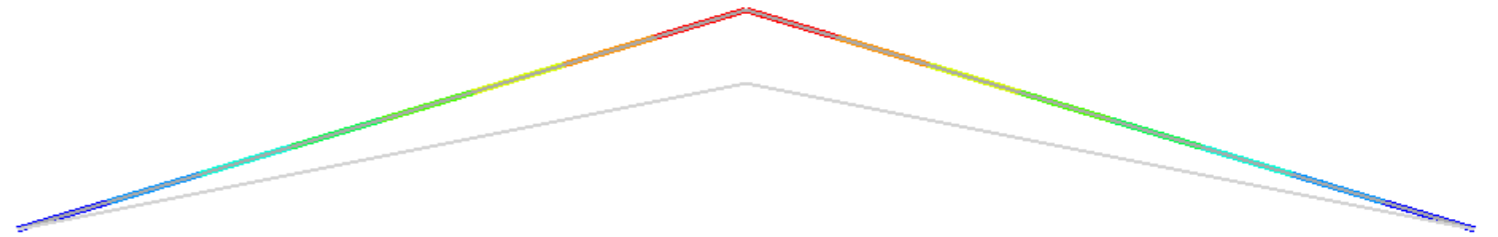


Tutorial

Buckling of an Arch



Outline

1	Description	3
2	Finite Element Model	4
2.1	Geometry	5
2.2	Boundary Conditions	6
2.3	Properties	7
2.4	Loads	9
2.5	Mesh	10
3	Structural Stability Analysis	11
3.1	Commands	11
3.2	Results	12
	Appendix A Additional Information	13

1 Description

We analyze an arch structure as an example of an Euler buckling analysis. Figure 1 shows the geometry and input data of the model. The used units are newton [N] and meter [m]. The vertical displacement u_y of node 3 is the only actual degree of freedom, since the displacement of nodes 1 and 2 are restrained. Due to symmetry the horizontal displacement u_x of node 3 is zero. Application of a load F leads to a vertical displacement of node 3.

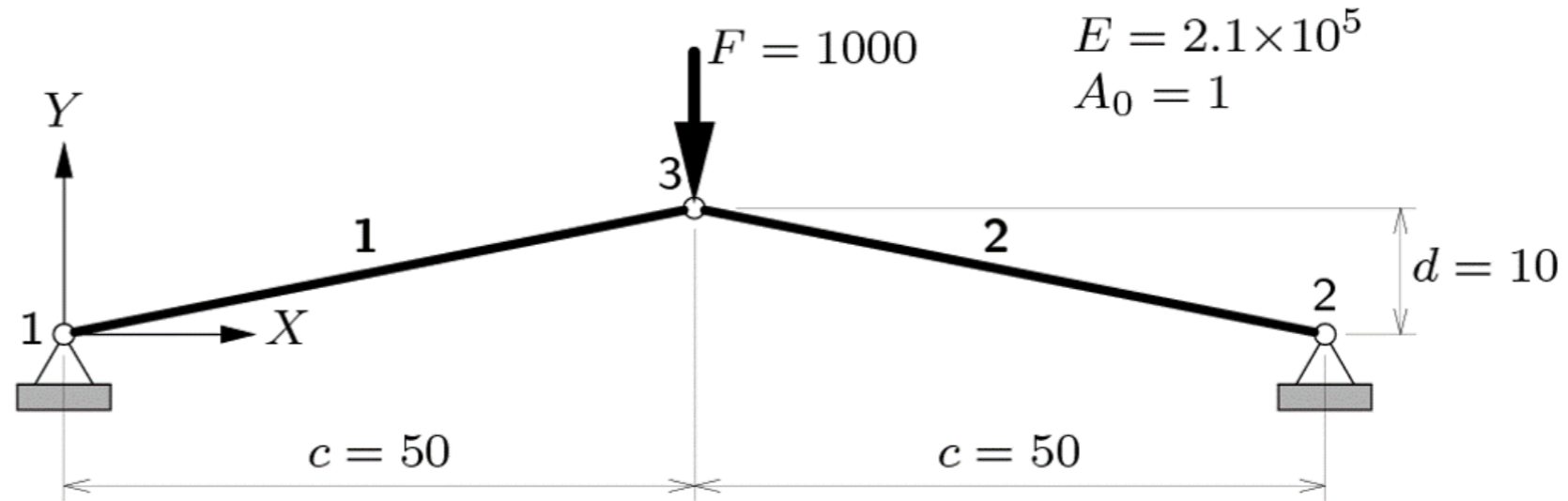



Figure 1: Geometry and input data

2 Finite Element Model

We start a new project for a two-dimensional structural analysis. The *Model Size* is set to 1 km. The used units are depicted in Figure 4.

Main menu → File → New  [Fig. 2]
Geometry browser → Reference system → Units [Fig. 3]
Property Panel [Fig. 4]

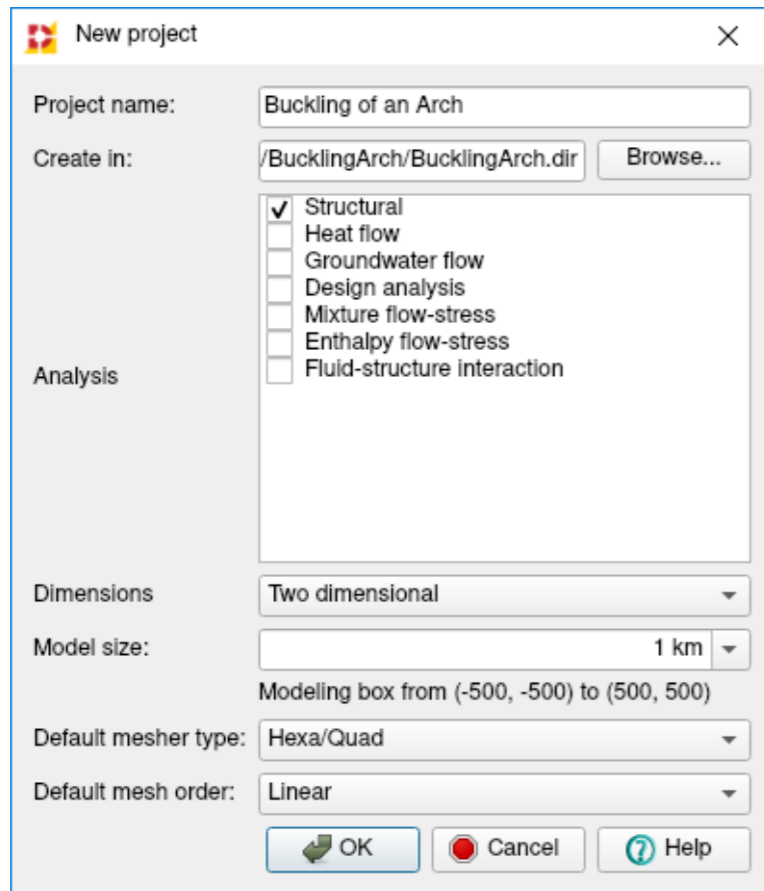


Figure 2: New project dialog

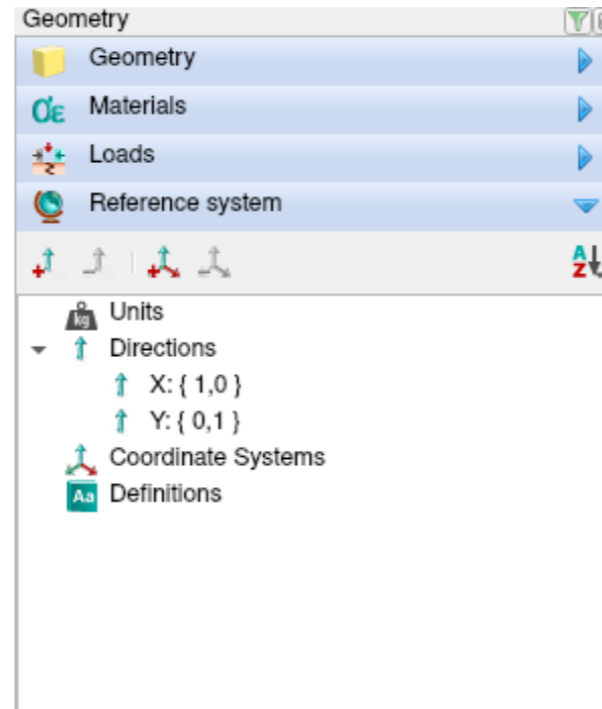


Figure 3: Geometry browser - units


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 4: Property panel - Units

2.1 Geometry

We create two lines representing the arch.

Main menu → Geometry → Create → Line  [Fig. 5]

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

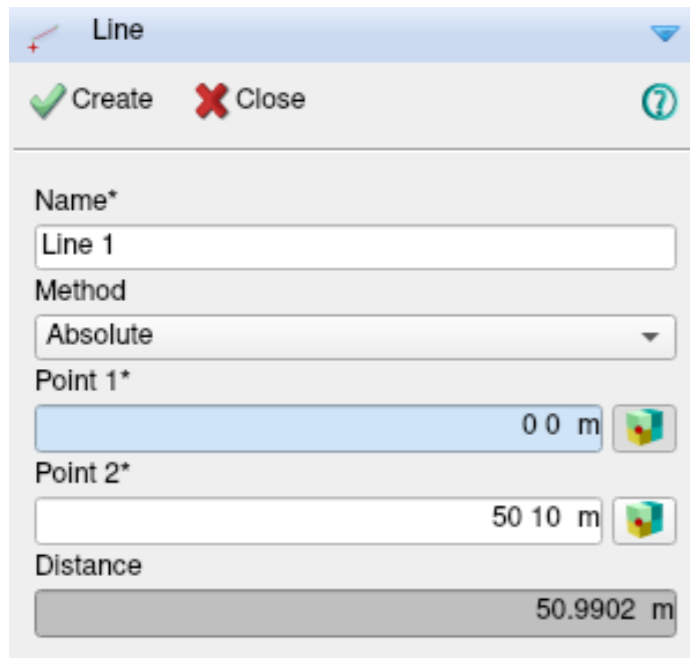


Figure 5: Add line 1

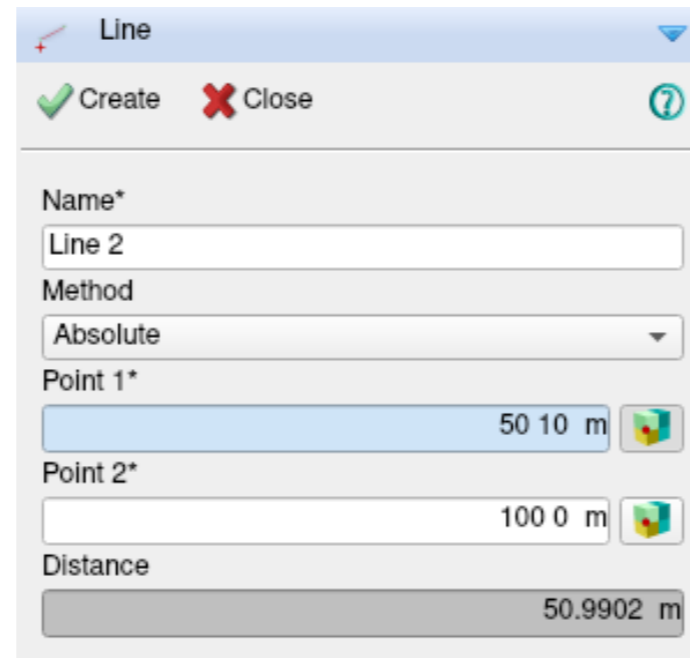


Figure 6: Add line 2



Figure 7: Geometry view

2.2 Boundary Conditions

We create supports at the left and right end of the structure. We constrain the translation in X and Y direction.

Main menu → Geometry → Assign → Supports  [Fig. 8]

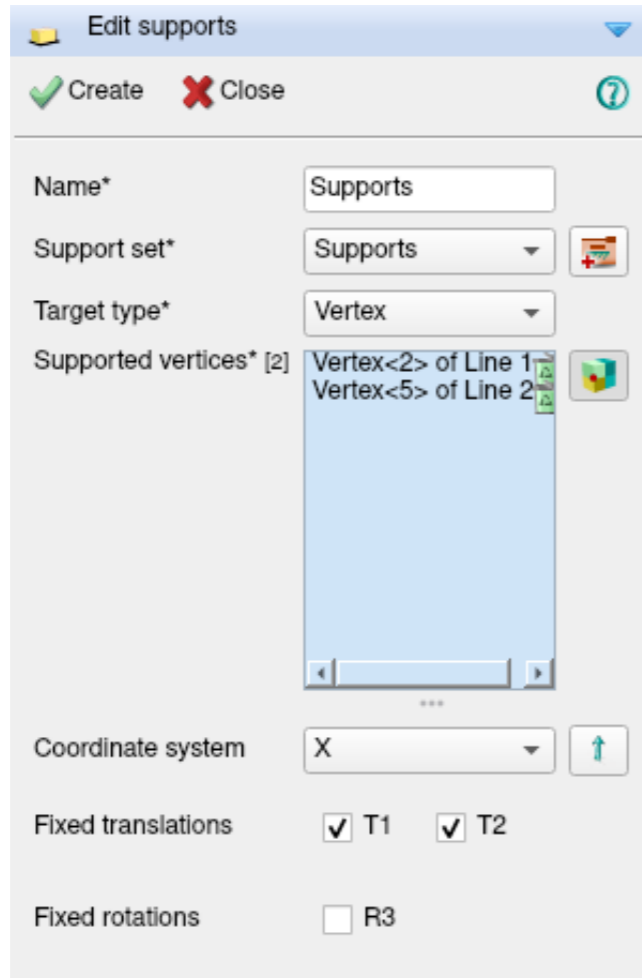


Figure 8: Application of the supports

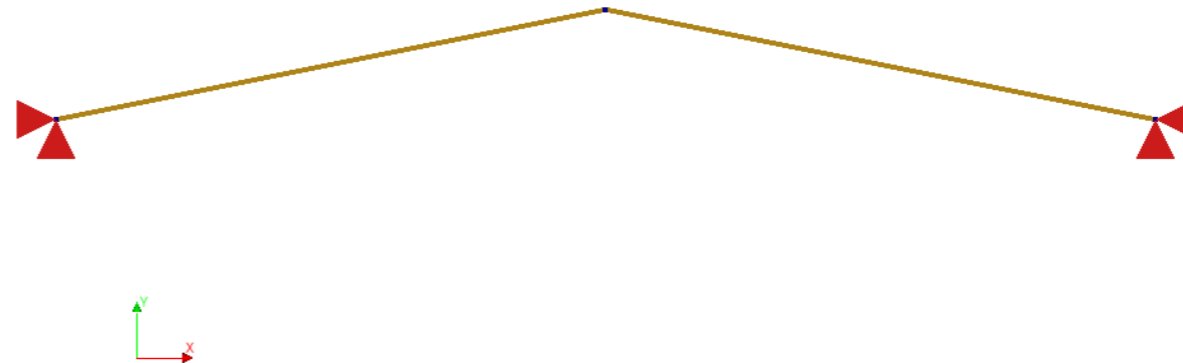


Figure 9: View supports

2.3 Properties

We give the lines material and geometrical properties. The element class is *Enhanced Truss 2D*. The following material parameters are chosen: Young's modulus $E = 2.1E+5 \text{ N/m}^2$ and Poisson's ratio $\nu = 0$.

DIANAIE

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

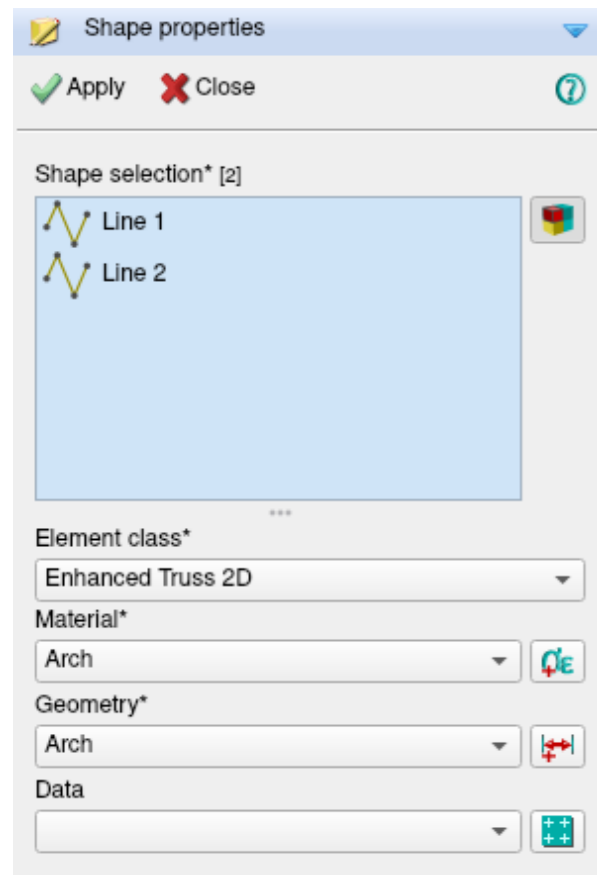


Figure 10: Property assignment -: lines

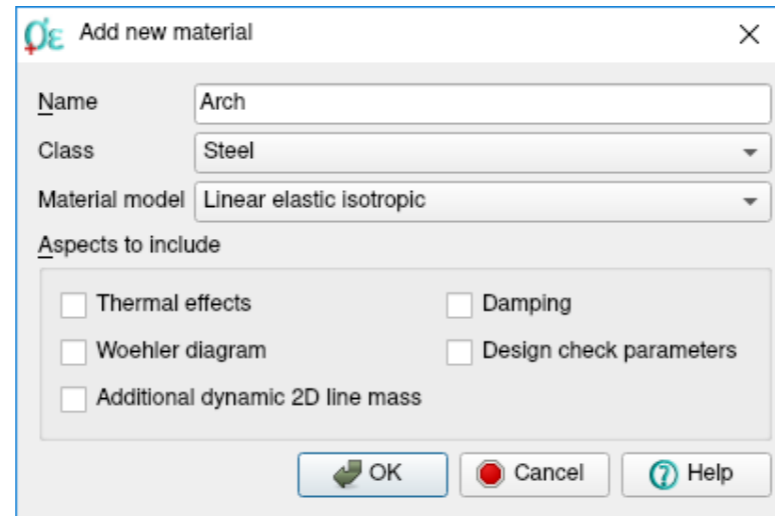


Figure 11: Add new material *Arch*

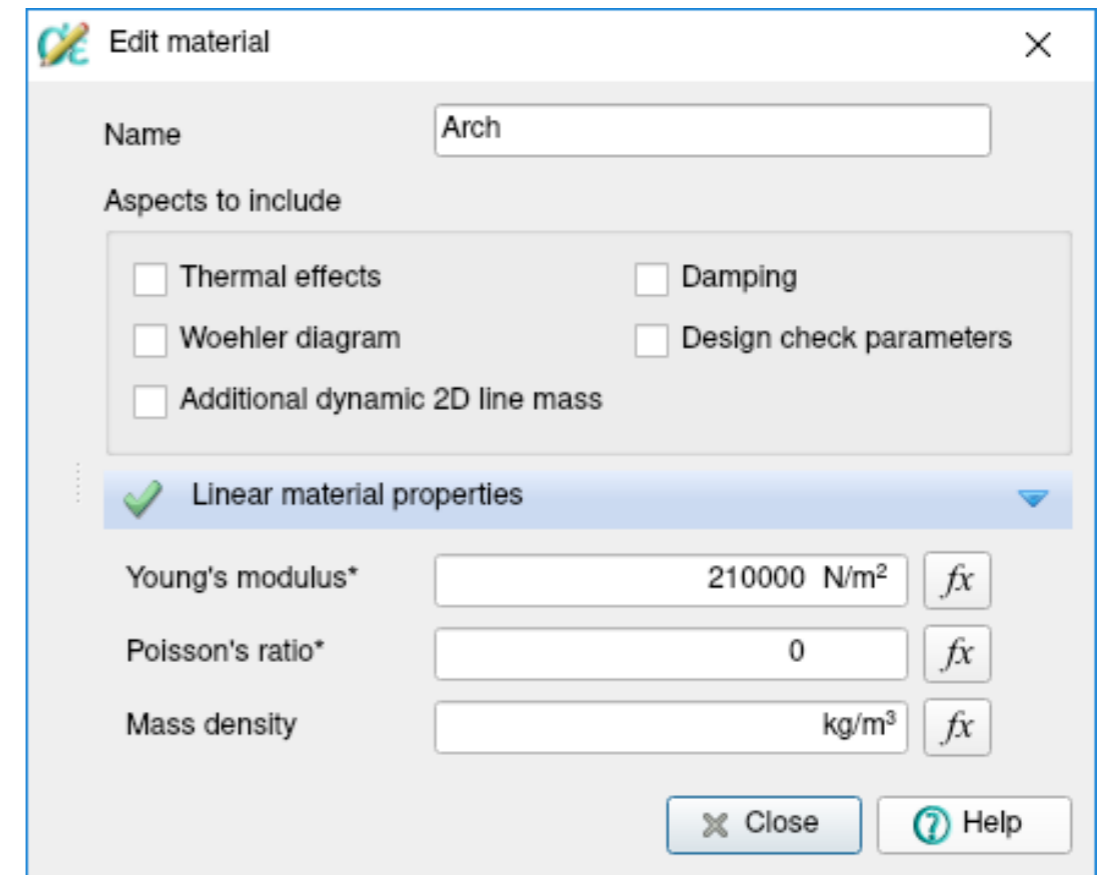


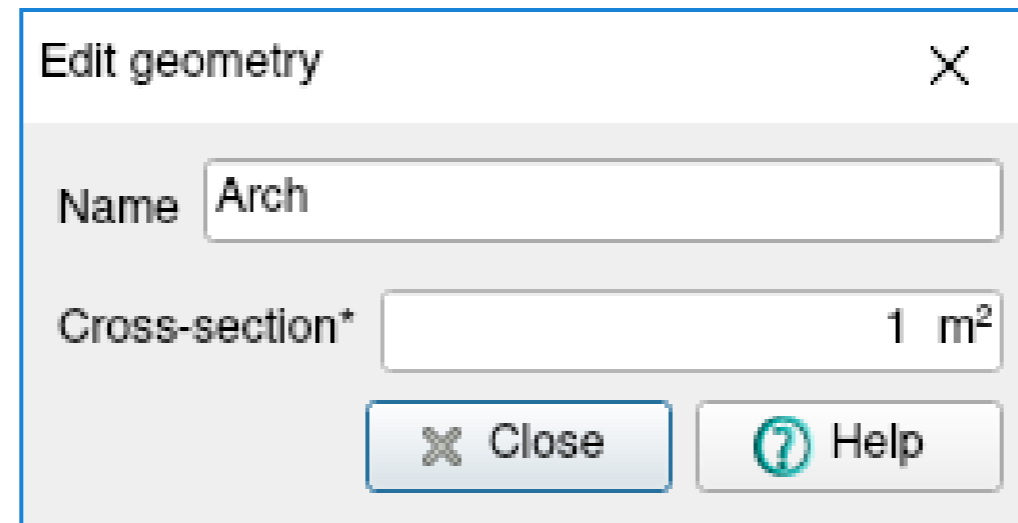


Figure 12: Edit material *Arch*

For the geometry we define a cross-section of 1 m².

Shape properties  → Geometry → Add new geometry  [Fig. 13]



Dialog box titled "Edit geometry" with a close button (X) in the top right corner. The dialog contains two input fields: "Name" with the value "Arch" and "Cross-section*" with the value "1 m²". At the bottom, there are two buttons: "Close" and "Help".

Figure 13: Edit line geometry

2.4 Loads

The loading consists of a point load of 1000 N in the negative Y direction.

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

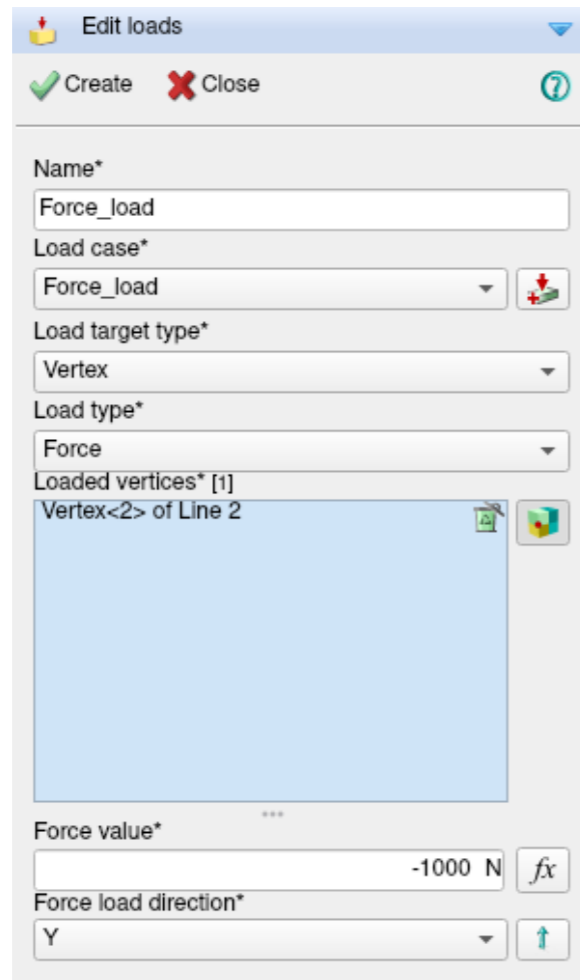


Figure 14: Attach point load

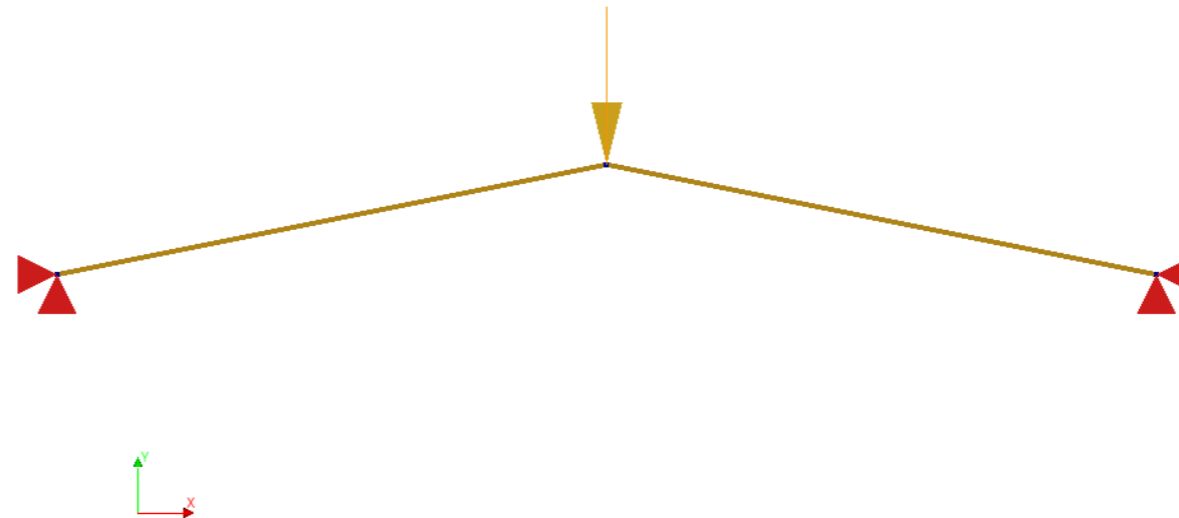



Figure 15: View of the applied loads

2.5 Mesh

We create a mesh by using *Divisions* and set the corresponding value equal to 1.

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

Main menu → Geometry → Mesh → Generate mesh 

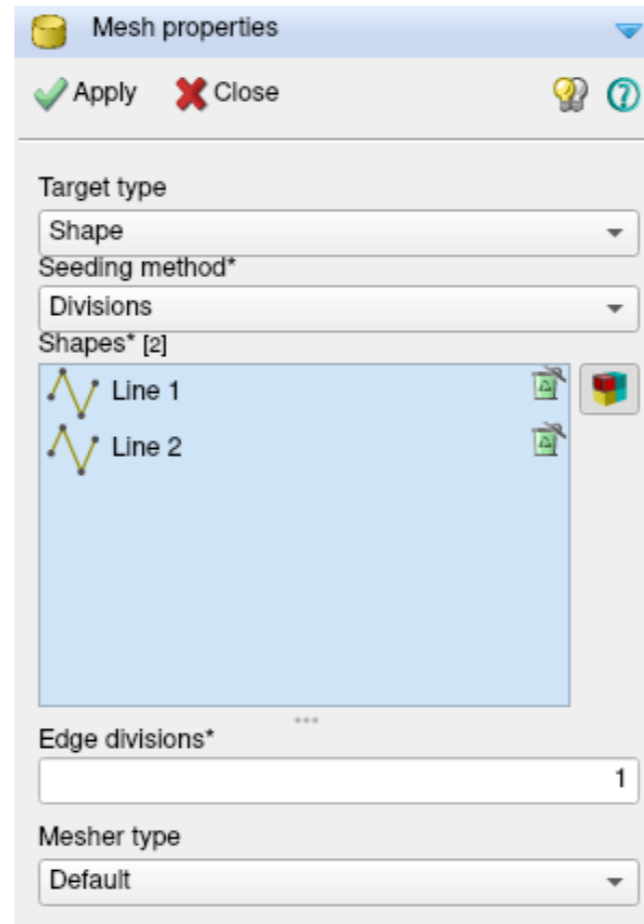




Figure 16: Mesh properties


3 Structural Stability Analysis


3.1 Commands

We set up the commands for a *Structural stability* analysis.

Main menu → Analysis → Add analysis  [Fig. 17]

Analysis browser → Analysis1  → Add command → Structural stability

Analysis browser → Analysis1 → Structural stability → Eigenvalue analysis → Define stability analysis → Edit properties  [Fig. 18] [Fig. 19]

Main menu → Analysis → Run selected analysis 

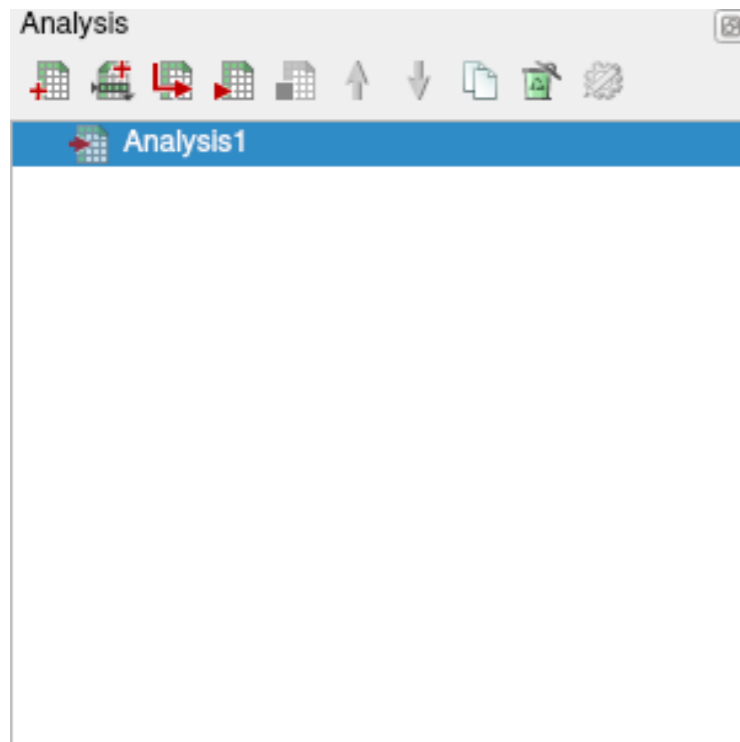


Figure 17: Analysis browser

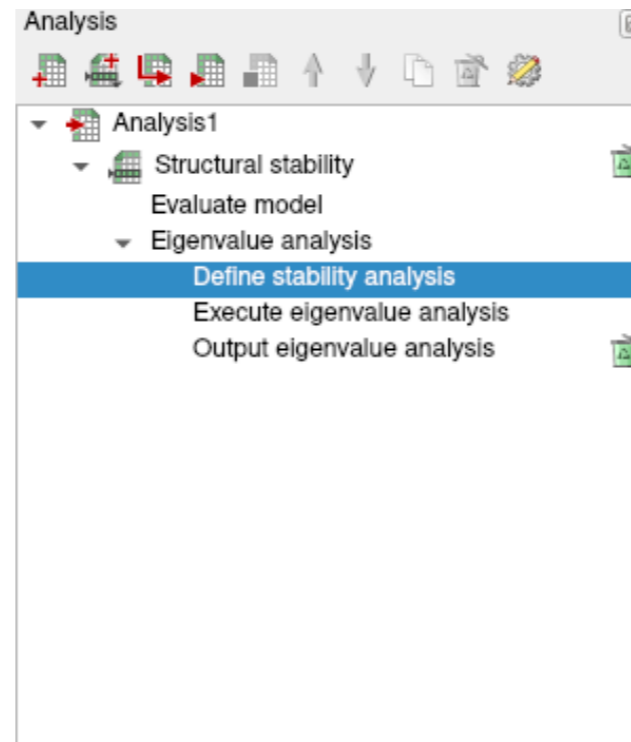


Figure 18: Analysis browser

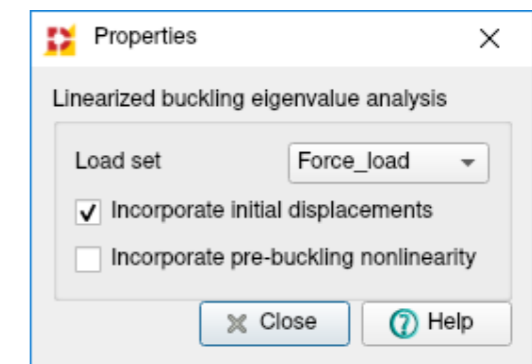


Figure 19: Properties stability analysis

3.2 Results

The *Message window* gives us the information that the analysis is finished. The buckling value gives the multiplication factor for the point load of 1000 N at which the buckling of the arch occurs and is displayed in Figure 20.

Results browser → Analysis1 → Output eigenvalue analysis → Nodal results → Displacements → TDtXY [Fig. 21]

```

2 STORING STRESS STIFFNESS
6 INITIALIZING DISPLACEMENT FIELD FOR D.O.F
2 EVALUTE ELEMENT DATA WITH INITIAL DISPL.
2 STORING EFFECTIVE ELEMENTMATRICES
SOLVING SYSTEM OF EQUATIONS
PARALLEL DIRECT SPARSE SOLVER
1 EIGENVALUES FOUND AFTER 1 ITERATIONS
BUCKLING-VALUES:
0.10560E+01( 1)
RELATIVE ERROR ||R|| / ||Kx||:
0.77695E-15( 1)

/DIANA/DC/END      16:55:37      0.14-CPU      0.19-IO      STOP

Job completed: running analysis 'Analysis1'
Loading results file 'Buckling_of_an_Arch_Analysis1.dnb'
Filos file NOT removed while running regression tests
    
```

Figure 20: Obtained buckling value

Analysis1
 Mode 1, Buckling value 1.0560
 Displacements DtXY
 min: 0.00m max: 1.00m

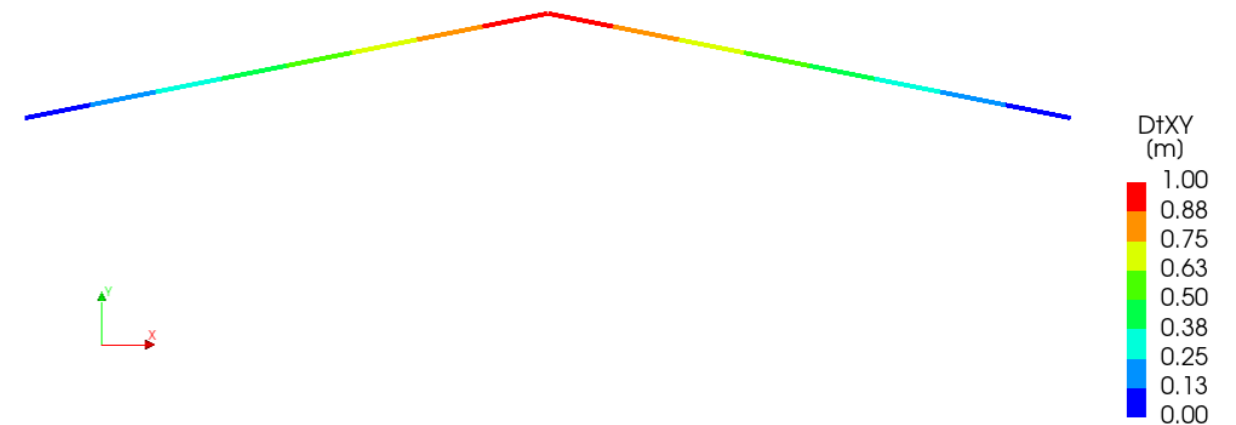


Figure 21: Buckling mode

Appendix A Additional Information

Folder: Tutorials/BucklingArch

Number of elements ≈ 2

Keywords:

ANALYS: euler stabil.

CONSTR: suppor.

ELEMEN: enhanc l4tru truss.

LOAD: force node.

MATERI: elasti isotro.

OPTION: direct.

POST: binary ndiana.

PRE: dianai.

RESULT: buckli displa modes values.

Laan van Waalhaven 462
2497 GR, The Hague
The Netherlands

DIANAFEA.COM

INFO@DIANAFEA.COM

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

