



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	8
2.5	Mesh	9
3	Structural Stability Analysis	10
3.1	Commands	10
3.2	Results	11
	Appendix A Additional Information	12

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.

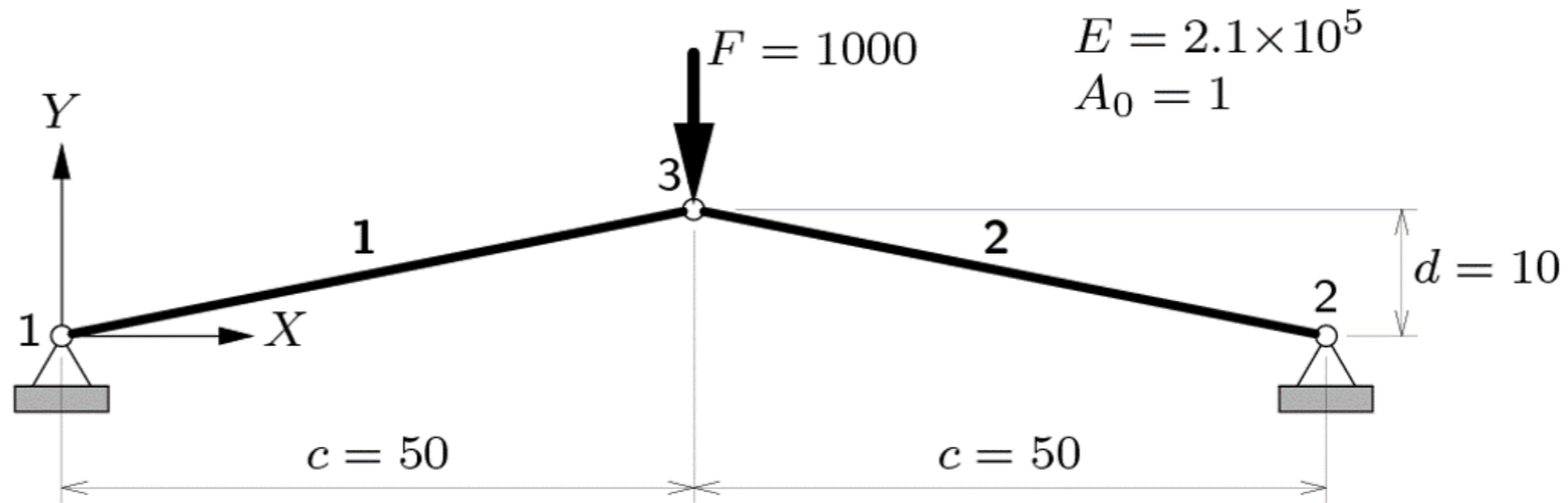


Figure 1: Geometry and input data

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.

DIANAIE

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

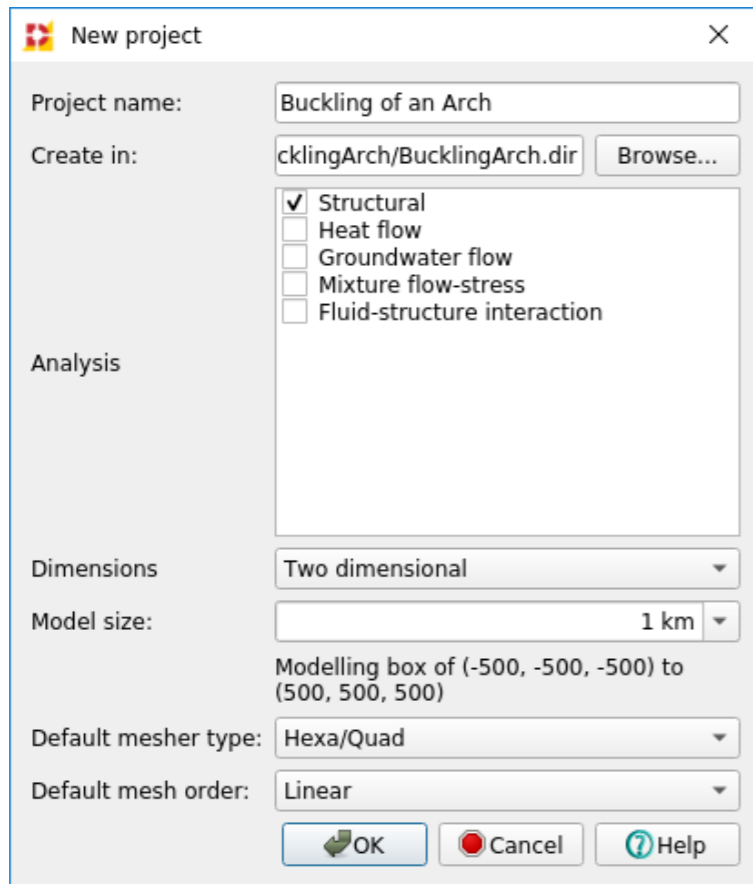


Figure 2: New project dialog

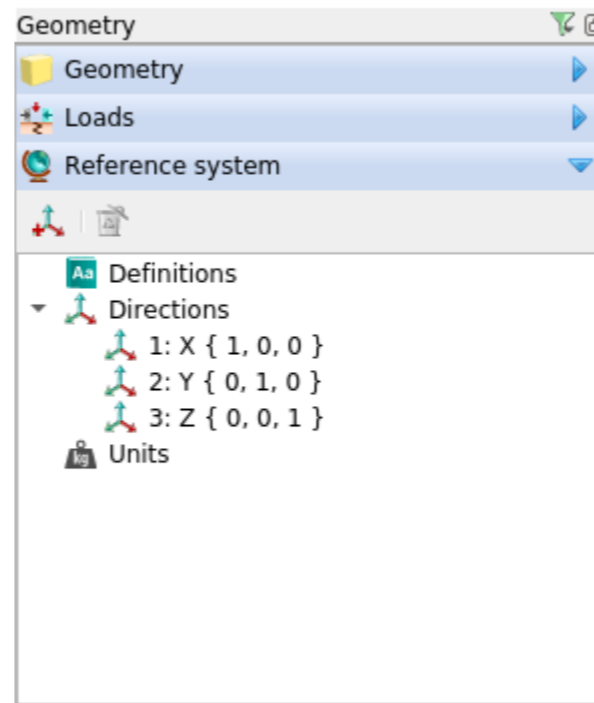


Figure 3: Geometry browser - units


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 → Add line  [Fig. 5]

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

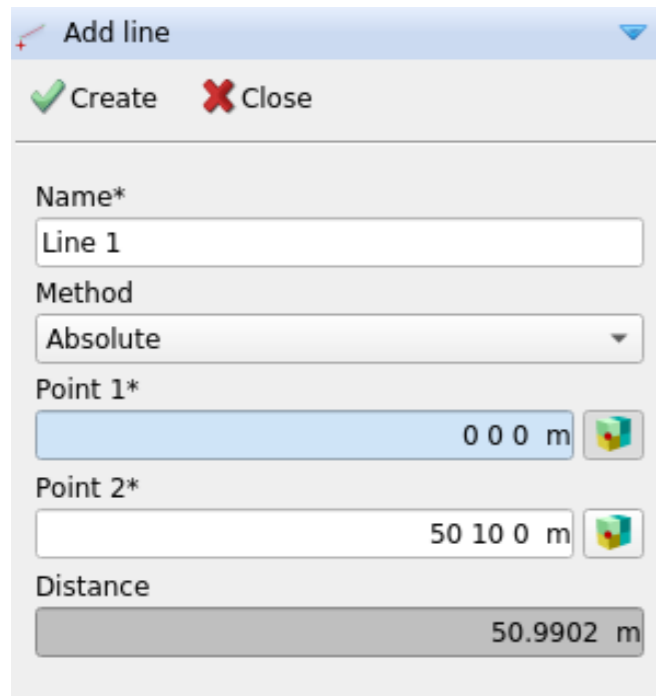


Figure 5: Add line 1

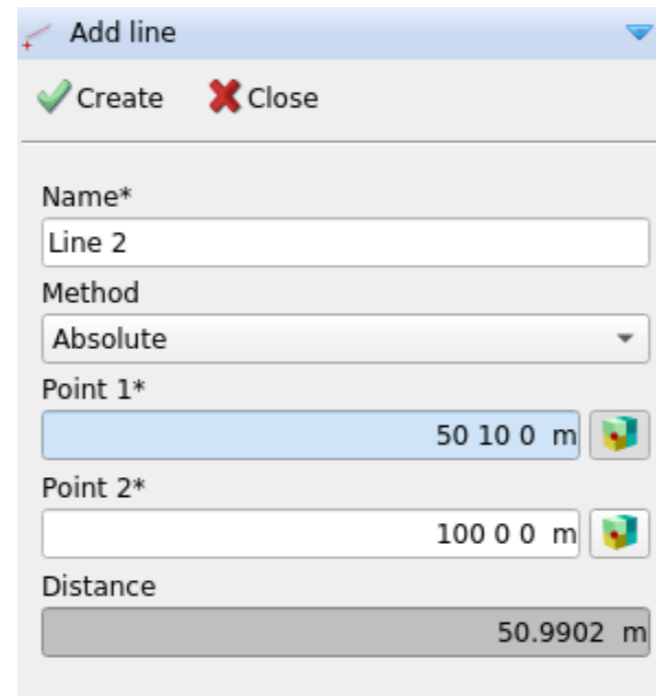


Figure 6: Add line 2



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.

DIANAIE

Main menu → Geometry → Assign → Add supports 📁 [Fig. 8]

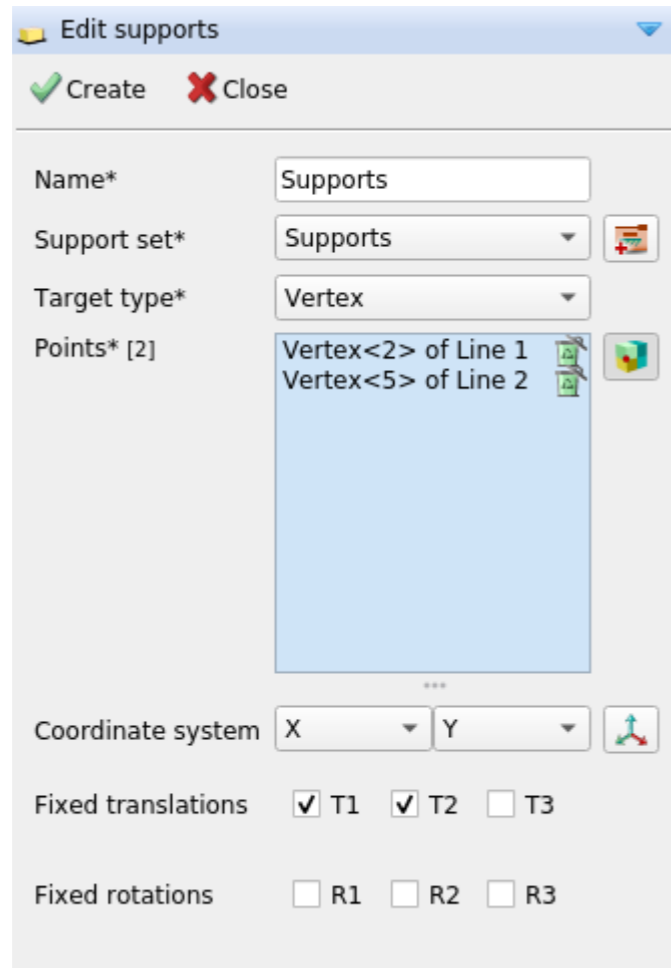


Figure 8: Application of the supports

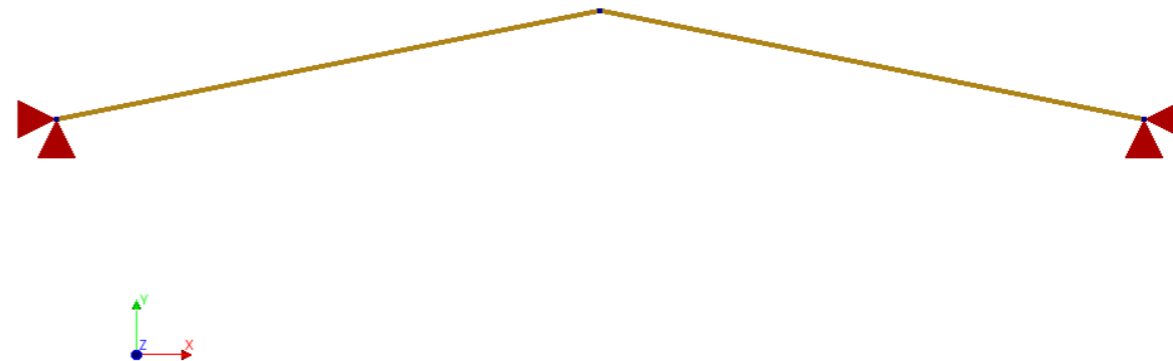








Figure 9: View supports

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 $\nu = 0$. For the geometry a cross-section of 1 m^2 is chosen.

DIANAIE

Main menu → Geometry → Assign → Shape Properties  [Fig. 10] [Fig. 14]
Shape Properties  → Material → Add material  [Fig. 11] → Edit material  [Fig. 12]
Shape Properties  → Geometry → Add new geometry  [Fig. 13]

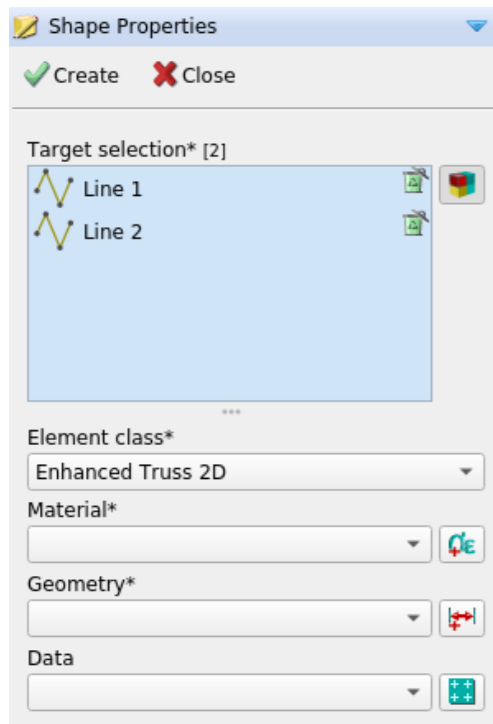


Figure 10: Property assignment - lines

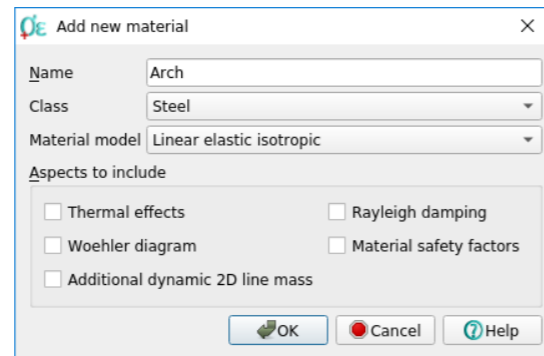


Figure 11: Add new material *Arch*

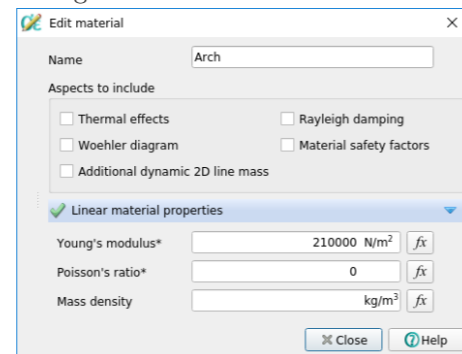


Figure 12: Edit material *Arch*

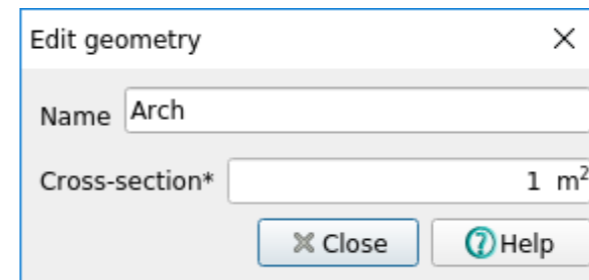


Figure 13: Edit line geometry

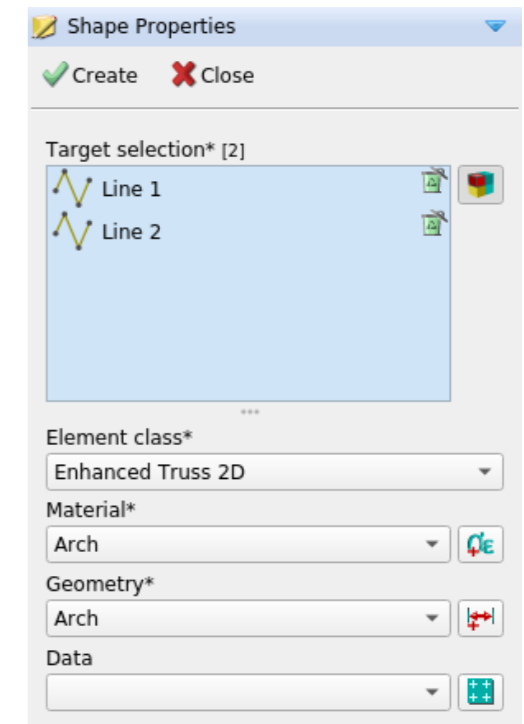


Figure 14: Property assignment -: lines

2.4 Loads

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

DIANAIE

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

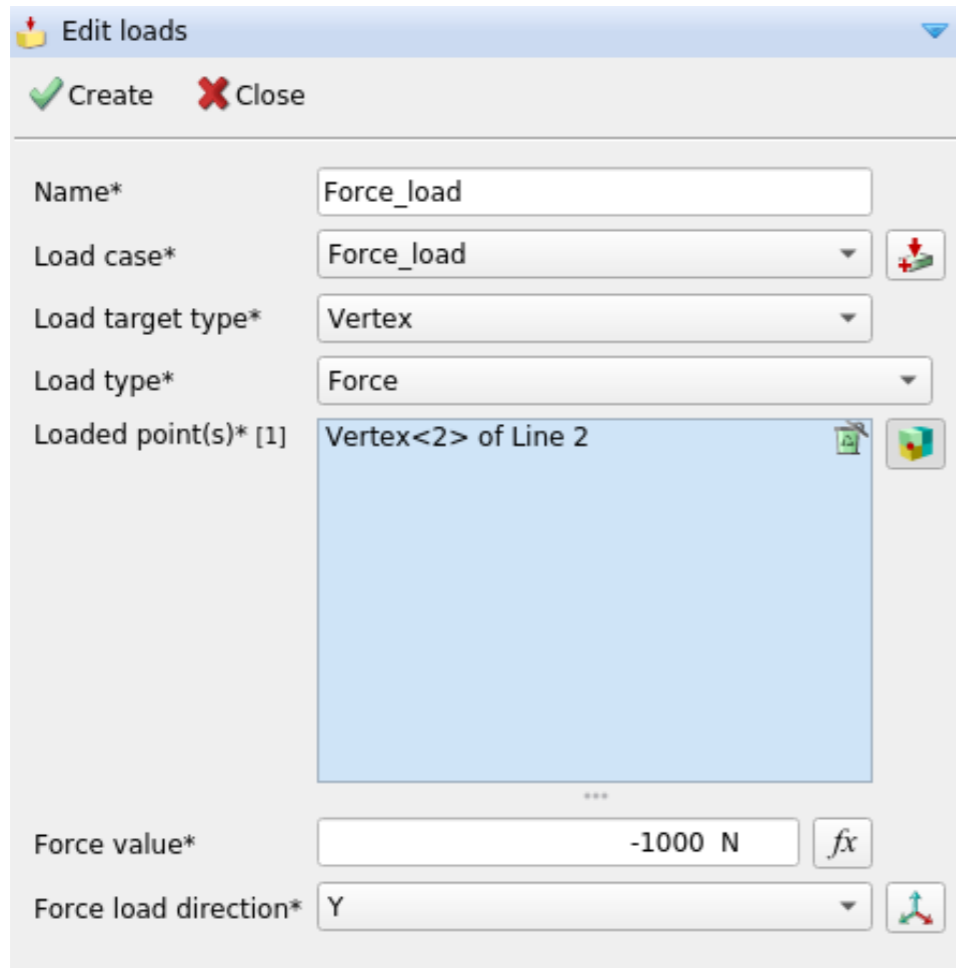


Figure 15: Attach point load

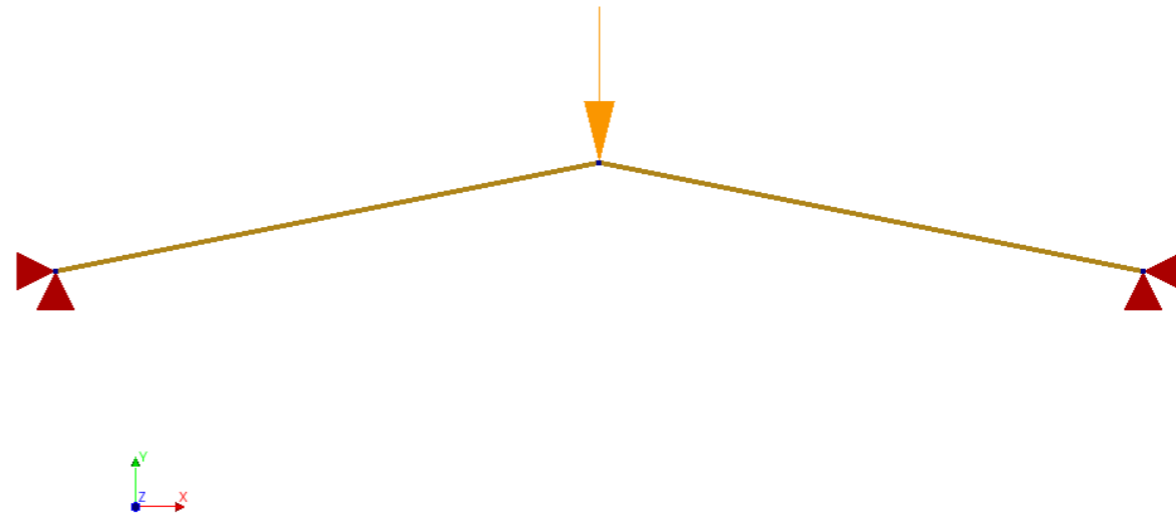


Figure 16: View of the applied loads

2.5 Mesh

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

DIANAIE

Main menu → Geometry → Assign → Mesh properties  [Fig. 17]

Main menu → Geometry → Generate mesh 

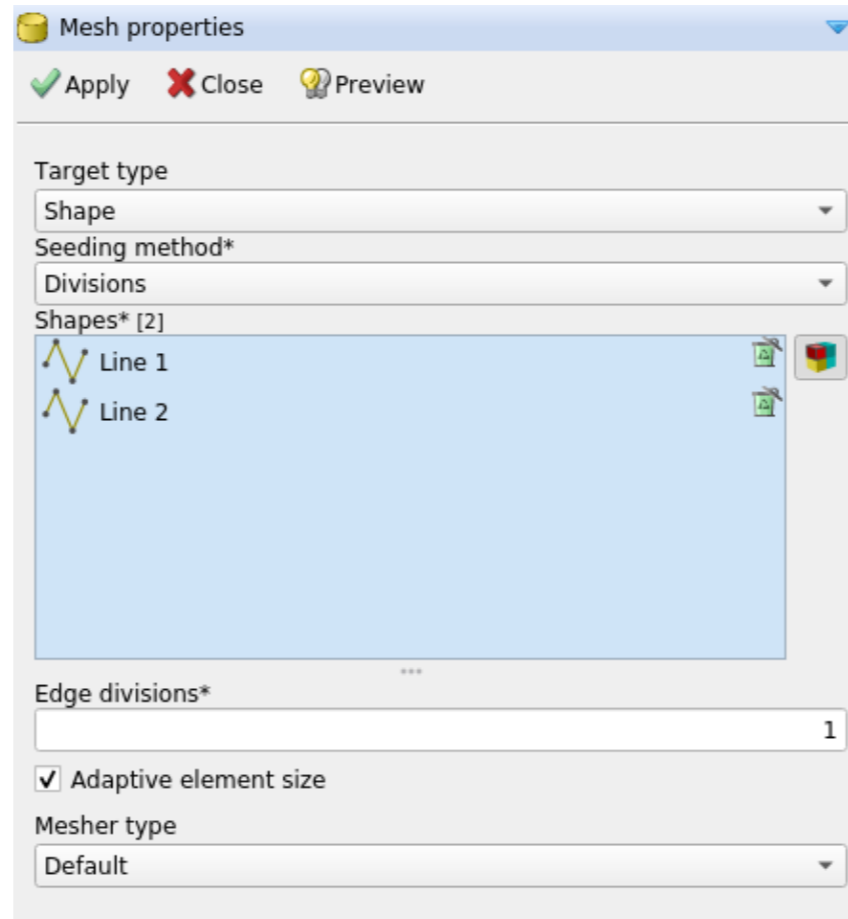



Figure 17: Mesh properties


3 Structural Stability Analysis


3.1 Commands


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

DIANAIE

Main menu → Analysis → Add analysis  [Fig. 18]

Analysis browser → Analysis1  → Add command → Structural stability

Analysis browser → Analysis1 → Structural stability → Eigenvalue analysis → Define stability analysis → Edit properties  [Fig. 19] [Fig. 20]

Main menu → Analysis → Run selected analysis 

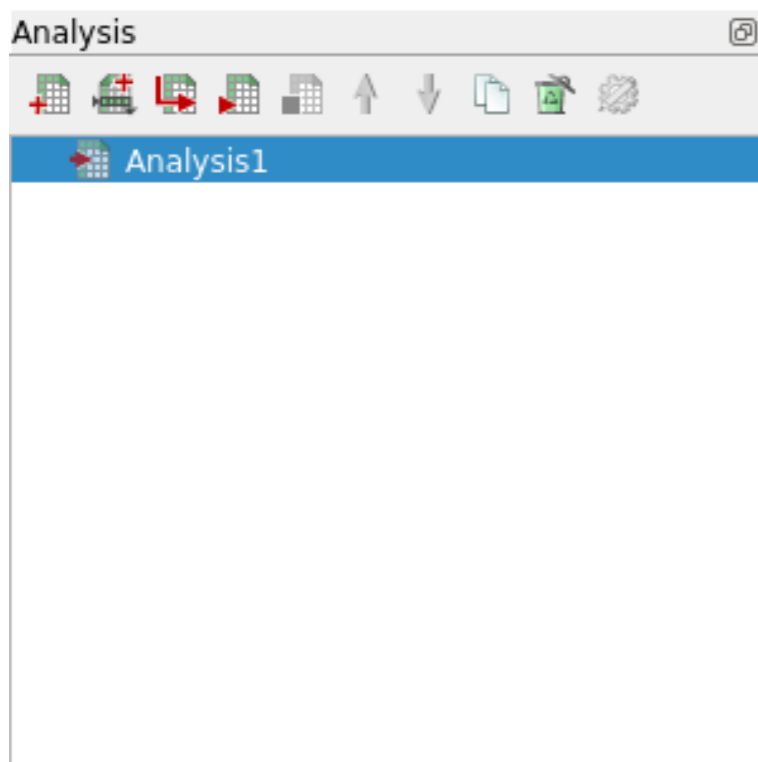


Figure 18: Analysis browser

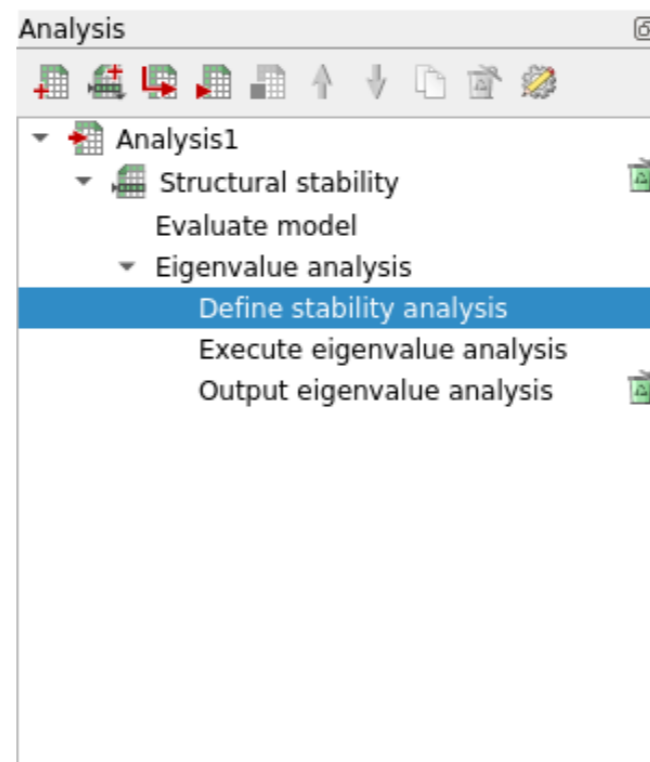


Figure 19: Analysis browser

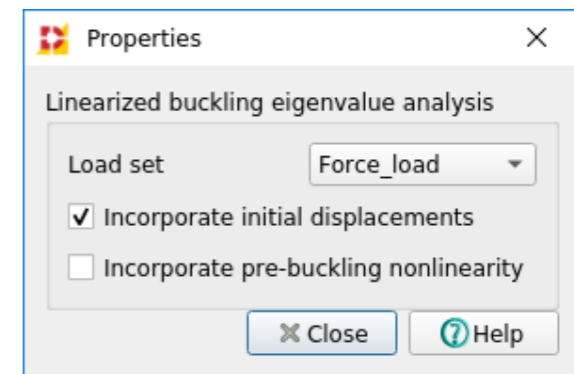


Figure 20: Properties stability analysis

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 21.

DIANAIE

Results browser → Analysis1 → Output eigenvalue analysis → Nodal results → Displacements → TDtXYZ [Fig. 22]

```
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||:
0.39524E-15( 1)

/DIANA/DC/END 18:45:30 0.22-CPU 0.40-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 21: Obtained buckling value

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

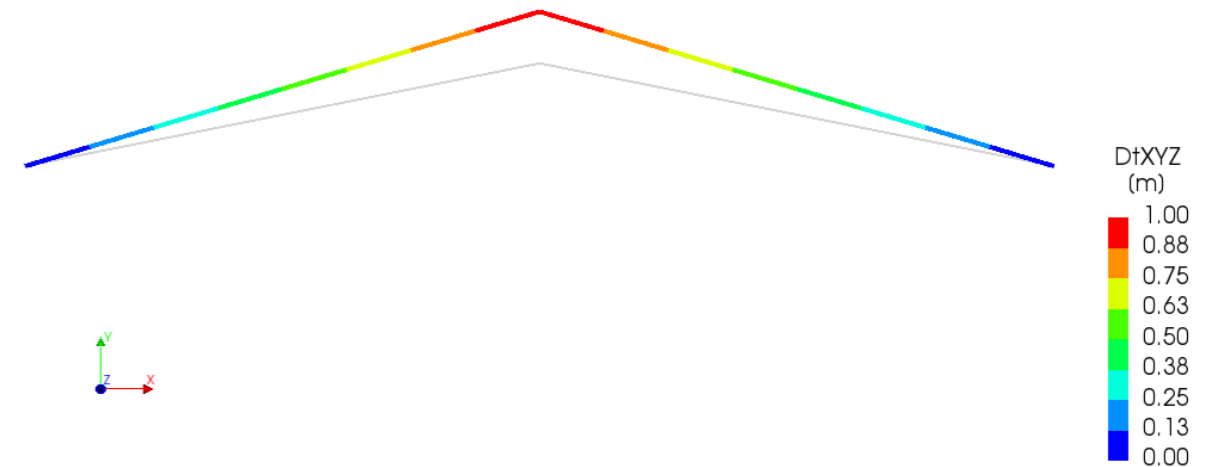


Figure 22: Buckling mode

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.



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

