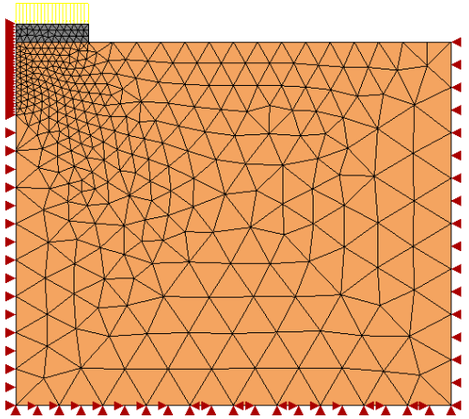
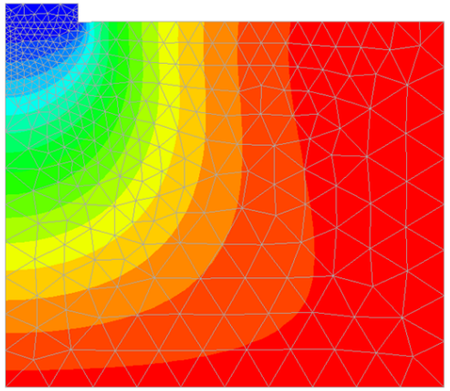
Tutorial

Settlement of a Strip Footing – Linear Static Analysis





DIANA

DIANA

Outline

1 Description **3**

2 Finite Element Model **5**
2.1 Geometry . . . 8
2.2 Properties . . . 12
2.3 Boundary Conditions . . . 14
2.4 Loads . . . 17
2.5 Mesh . . . 18

3 Structural Linear Static Analysis **20**
3.1 Commands . . . 20
3.2 Results . . . 22
3.3 Comparison with Analytical Solution . . . 25

Appendix A Additional Information **27**

1 Description

The main objective of this tutorial is to perform a settlement analysis (long-term) of a strip footing using Linear Elasticity. This is a benchmark example that will be compared with an analytical solution for settlement of an infinitely long rigid footing on smooth finite homogeneous isotropic ground, according to Giroud (1972)[1].

The geometry of the model consists on a rigid concrete strip footing on an elastic soil layer of 5 m height. Due to symmetry, half of the geometry is simulated. Plane strain finite elements are used in this model.

The work flow used in this tutorial is the following:

- define geometry
- define material properties
- define boundary conditions and loads
- generate the mesh
- setup a linear elastic analysis
- perform the analysis
- interpret the results and compare with an analytical solution.

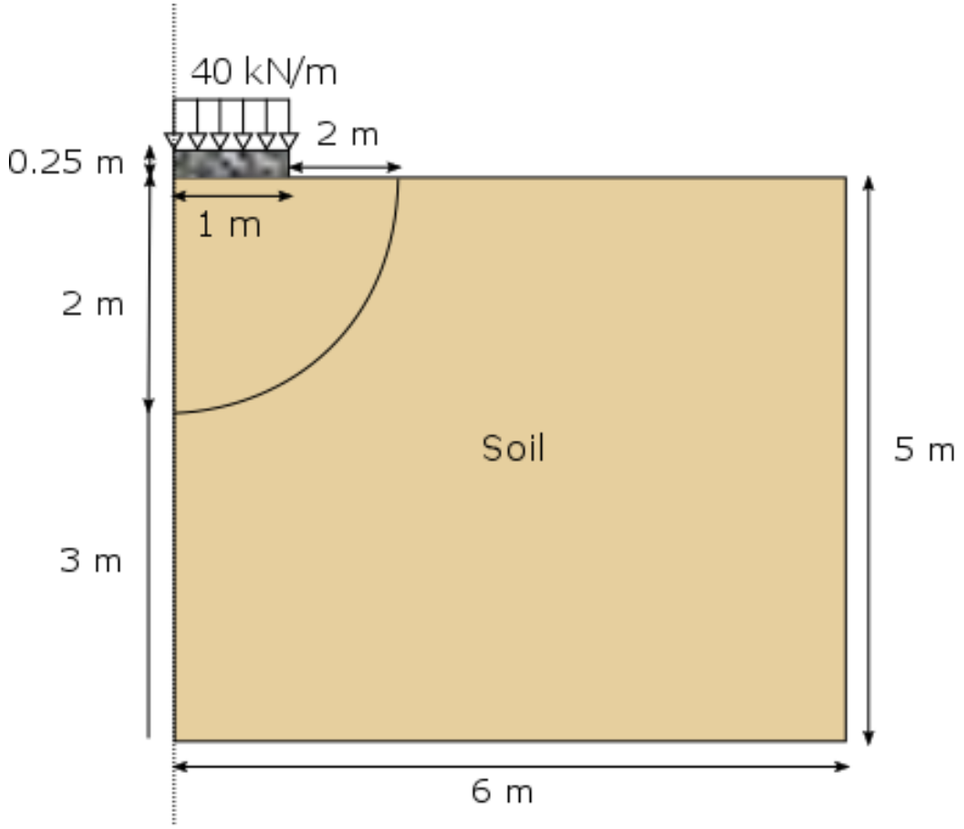


Figure 1: Geometry of the example

[1] Giroud, *Settlement of rectangular foundation on soil layer*, 1972

The material models used are linear elastic isotropic with the properties described in Table 2.

Figure 2: Material properties

Soil		
Young's modulus E	1.0e+04	kN/m^2
Poisson's ratio ν	0.33	
Mass density (dry) ρ	1.63165	T/m^3
Porosity (dry) ϕ	0.25	
Concrete		
Young's modulus E	3.5e+07	kN/m^2
Poisson's ratio ν	0.15	
Mass density ρ	2.44748	T/m^3

2 Finite Element Model

For the modeling session we start a new project. We use plane strain elements.

DIANAIE

Main menu → File → New [Fig. 3]

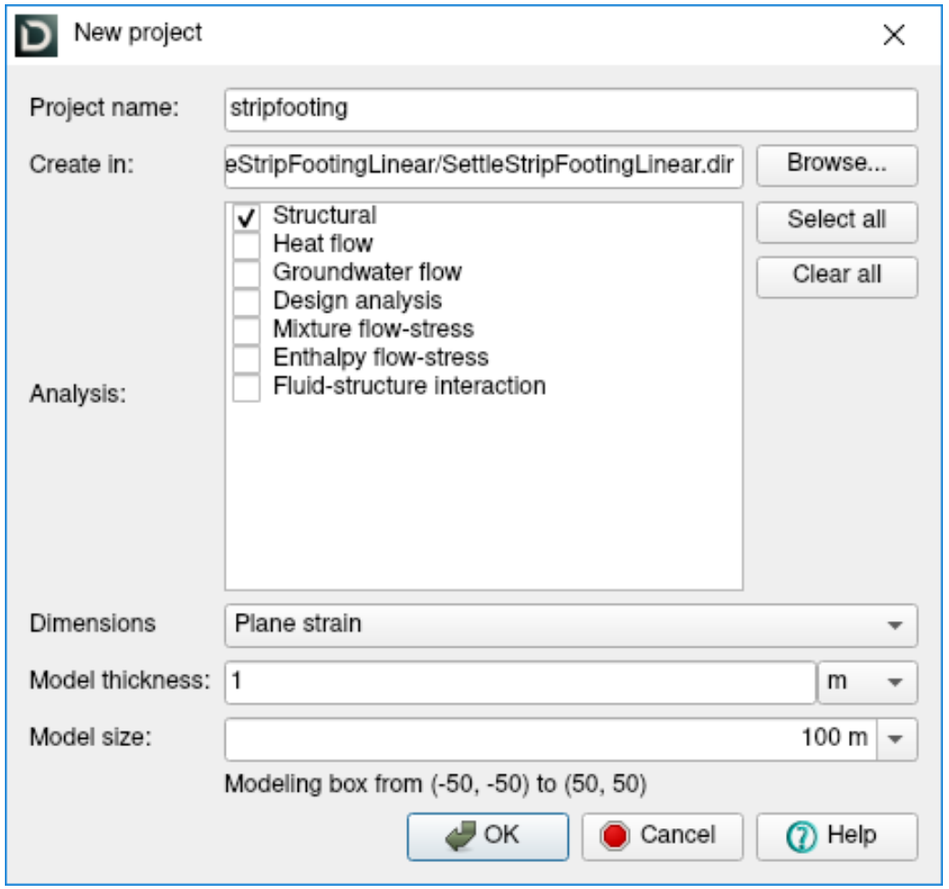


Figure 3: New project dialog

For units we choose kilonewton for force and meter for length.

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

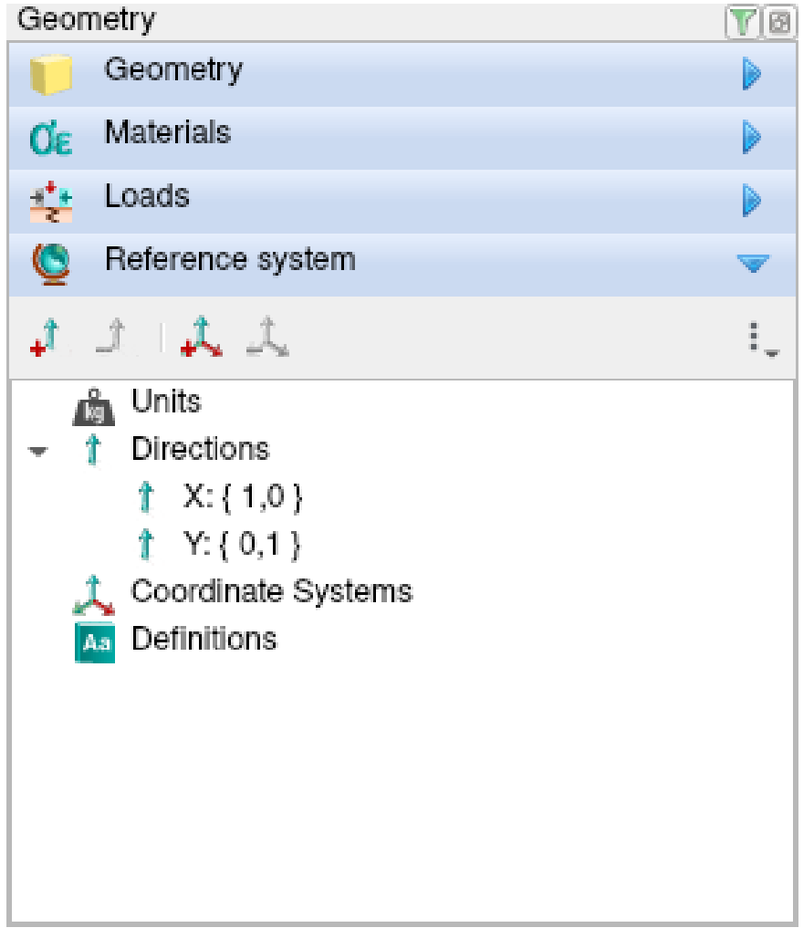


Figure 4: Geometry browser

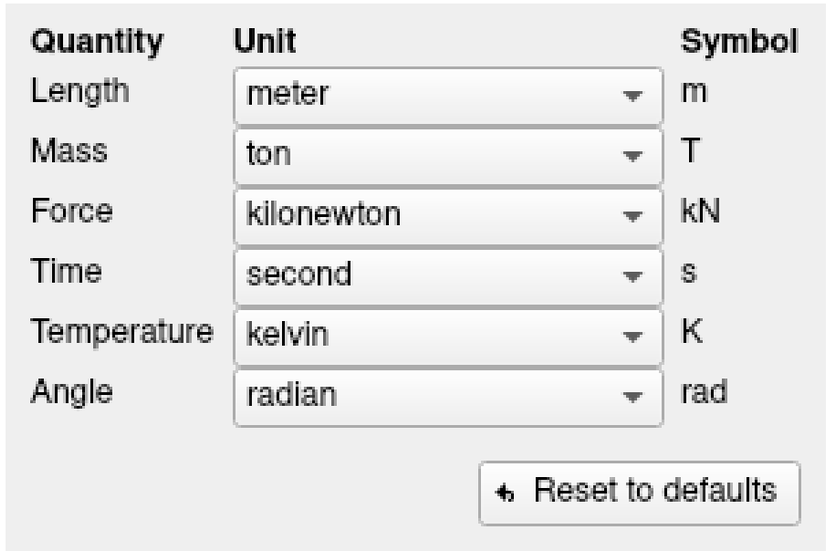


Figure 5: Property panel - units

In the mesher preferences we define quadratic tetra/triangle elements.

Main menu → Mesh → Mesher preferences [Fig. 6]

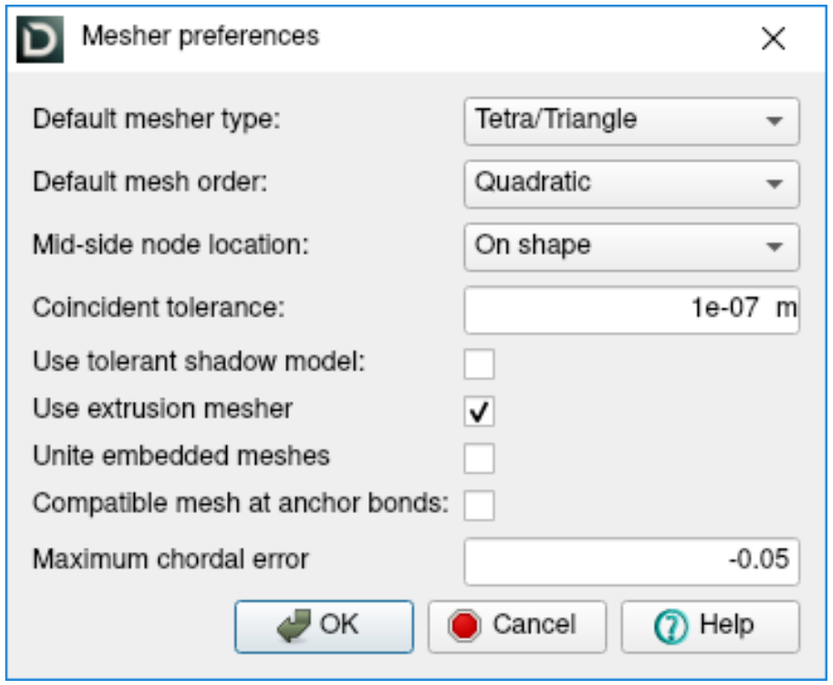


Figure 6: Mesher preferences

2.1 Geometry

We create two sheets, one for the soil and the other for the footing.

DIANAIE

Main menu → Geometry → Create → Polygon sheet [Fig. 7] [Fig. 8]
< *Repeat 2*× >

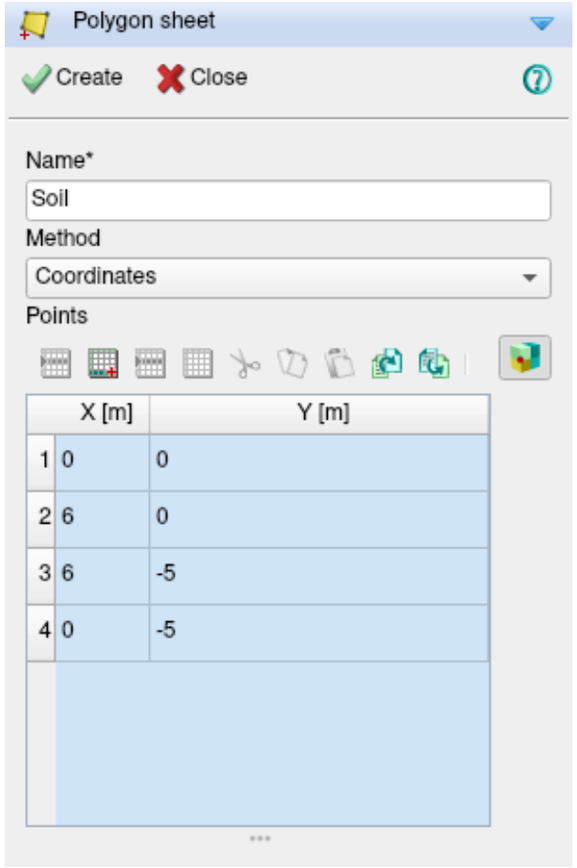


Figure 7: Add sheet *Soil*

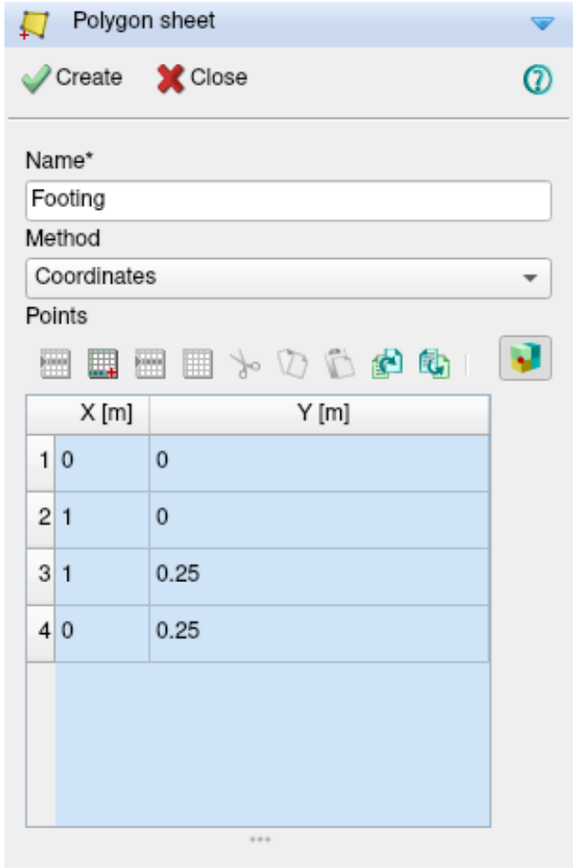


Figure 8: Add sheet *Footing*

We fit the geometry in the workspace.

Main menu → Viewer → Fit all

Figure 9: Geometry view of the soil and footing

We create a circle line where we want to have a finer mesh.

Main menu → Geometry → Create → Circle [Fig. 10] [Fig. 11]

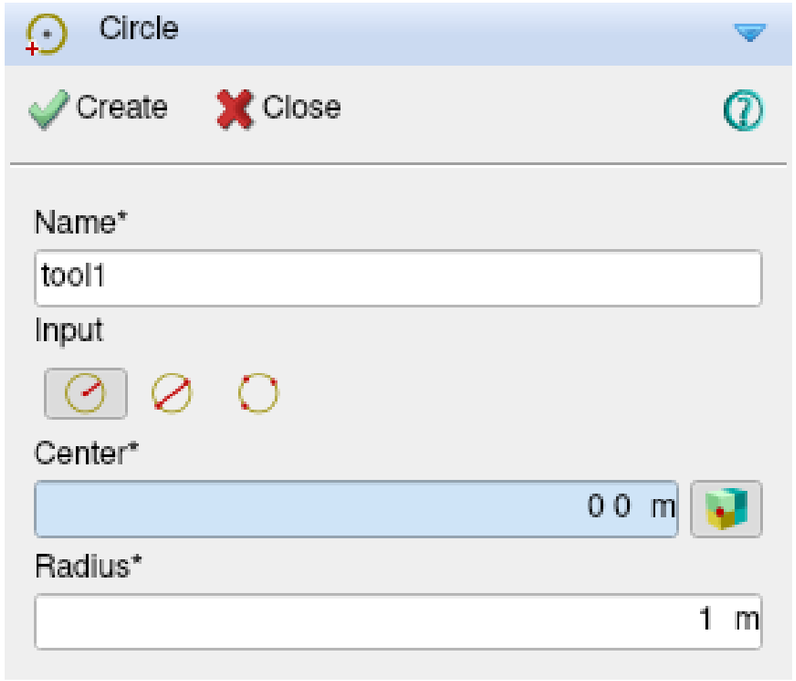


Figure 10: Geometry - add circle line

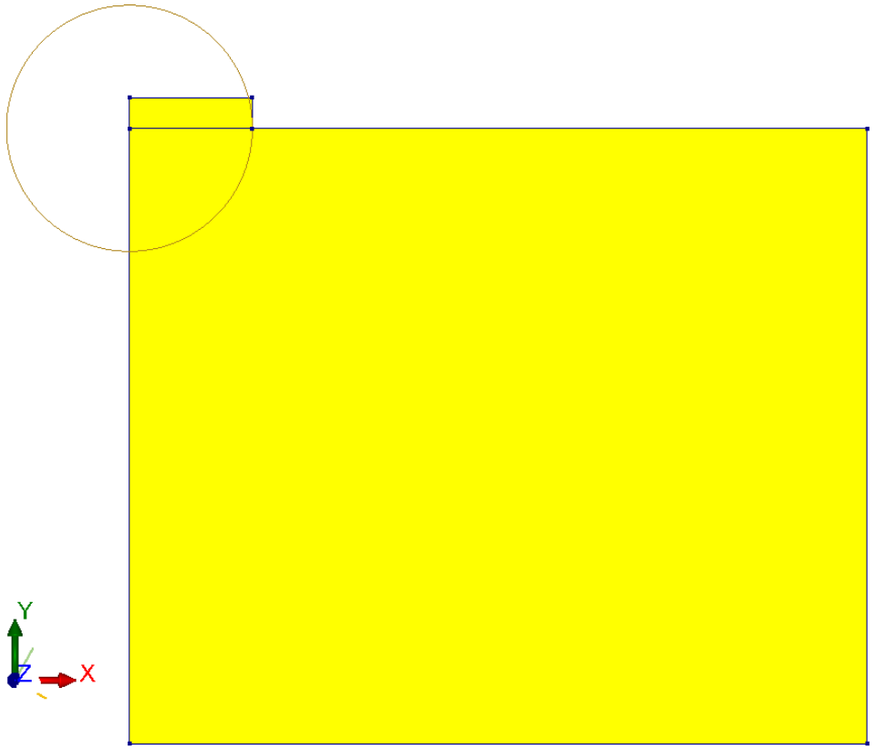


Figure 11: Geometry view - circle line

We imprint the circle line in the soil shape and the geometry definition is complete.

Main menu → Geometry → Modify → Projection [Fig. 12] [Fig. 13]

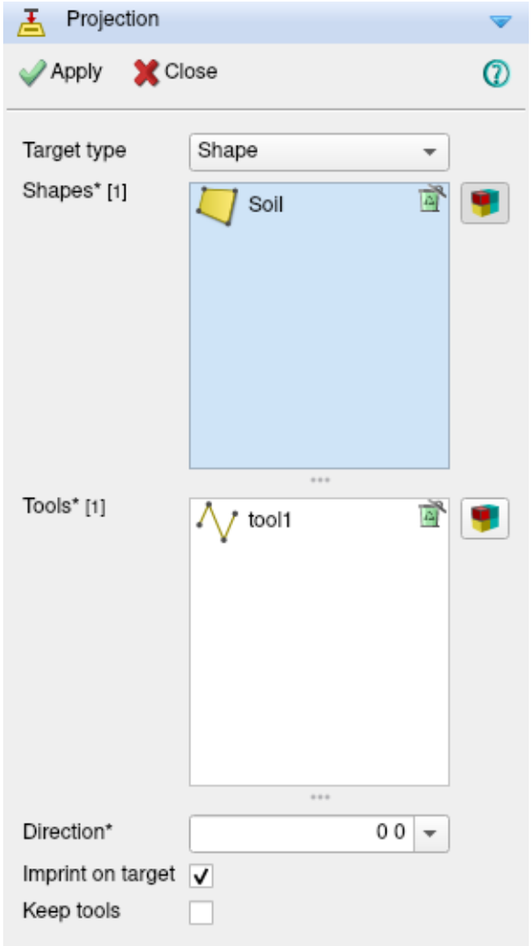


Figure 12: Imprint circle line

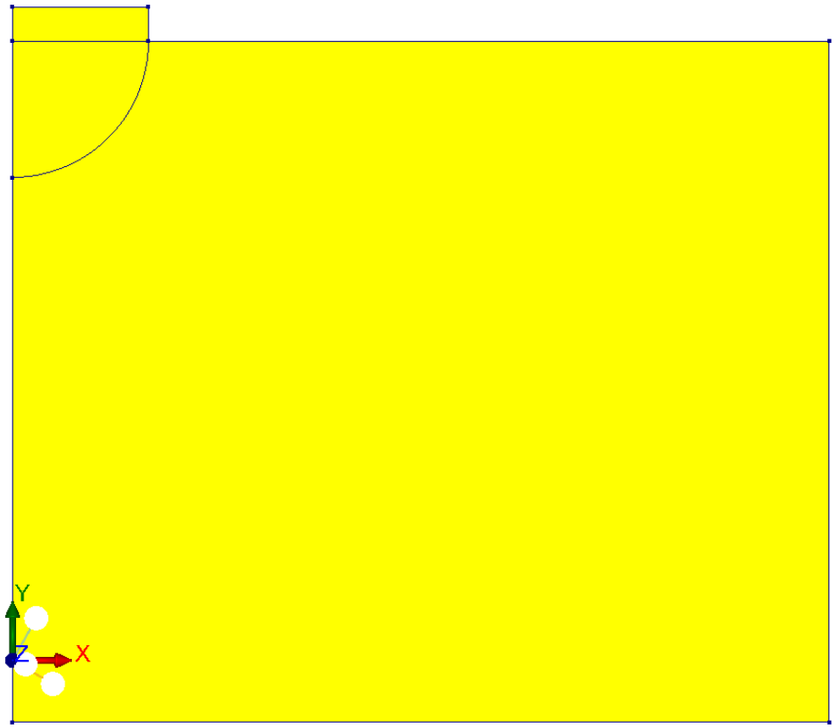


Figure 13: Geometry of the model

2.2 Properties

We define the properties of the soil and footing using the parameters listed in Table 2. As we use two-dimensional plane strain elements and we do not need to define the geometry properties. We use the default integration scheme, so no data group is required.

We start by assigning the properties to the soil.

DIANAIE

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

Shape properties → Material → Add material [Fig. 15] → Edit material [Fig. 16]

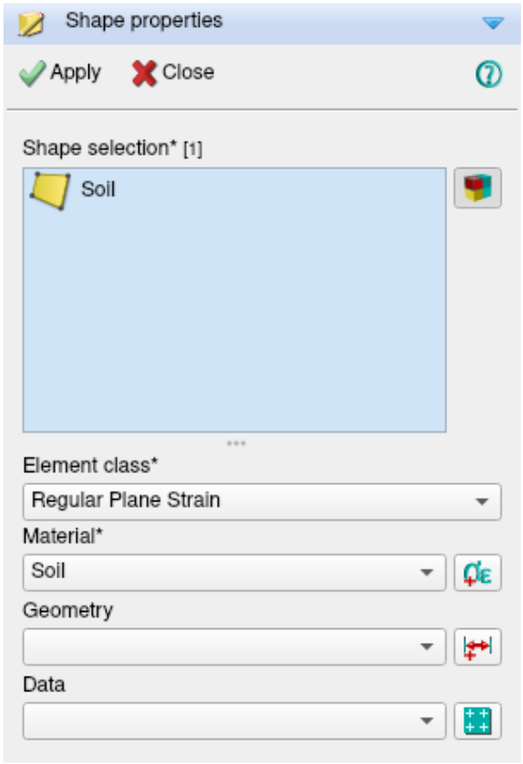


Figure 14: Assign soil properties

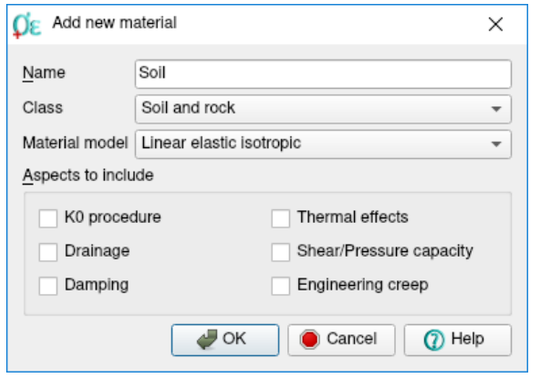


Figure 15: Soil - add new material

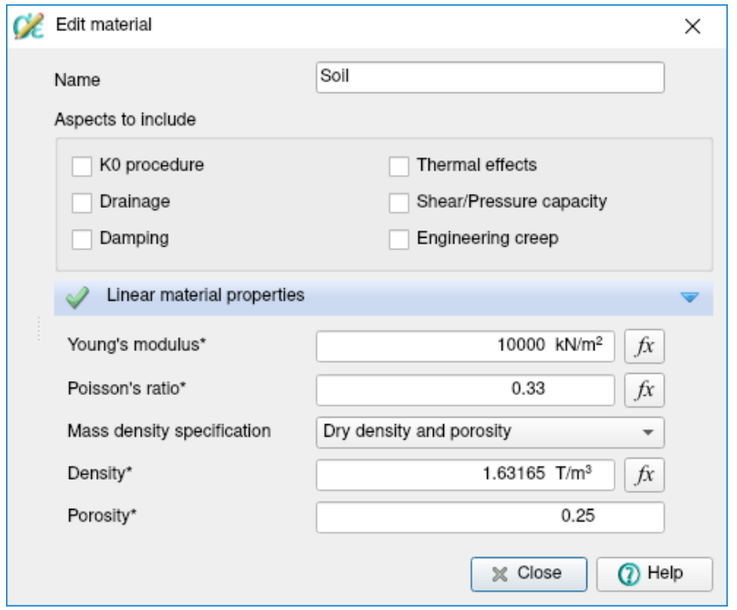


Figure 16: Soil - material properties

And we assign the properties to the concrete footing.

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

Shape properties → Material → Add material [Fig. 18] → Edit material [Fig. 19]

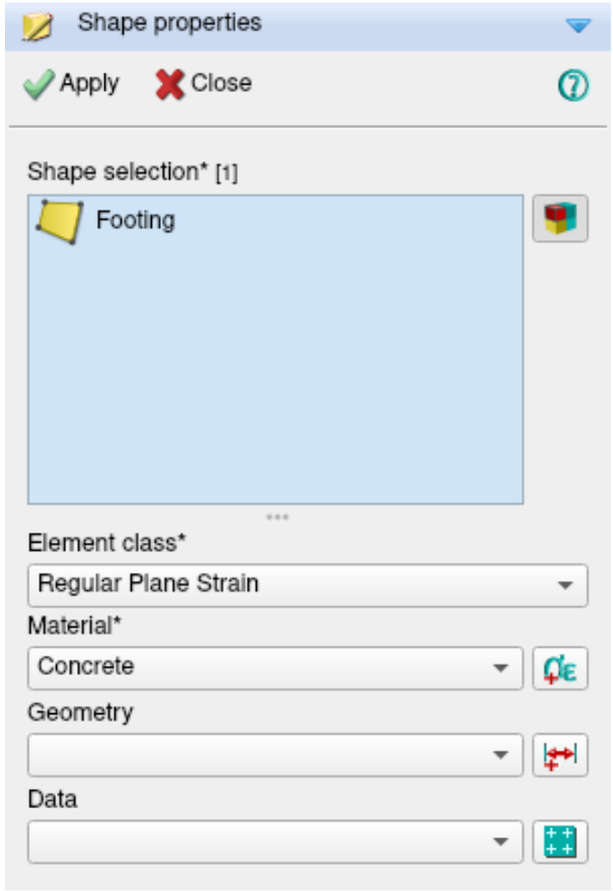


Figure 17: Assign footing properties

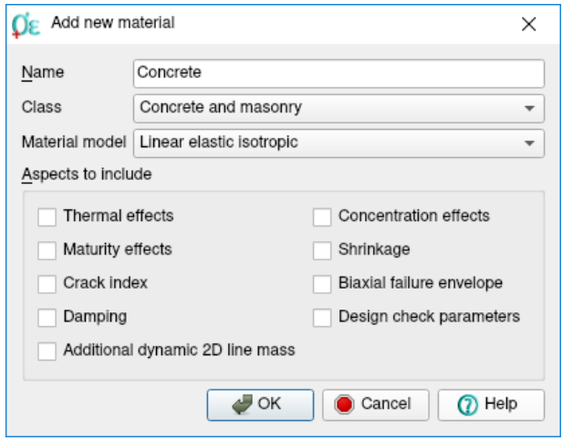


Figure 18: Concrete - add new material

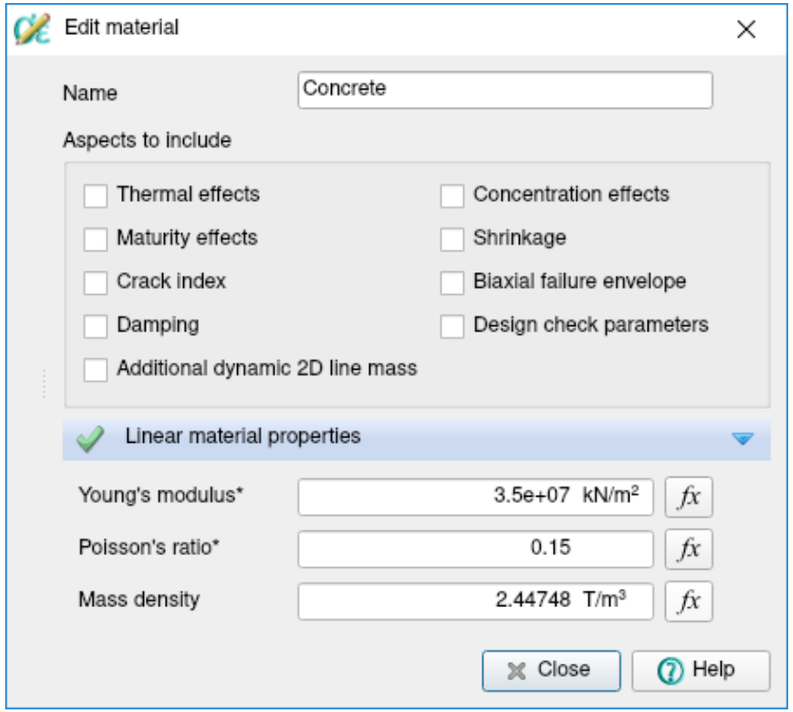


Figure 19: Concrete - material properties

2.3 Boundary Conditions

For the boundary conditions we defined two support sets: *i*) *Ground support* that includes fixing the translations in the X direction of the side edges and fixing the translations in the X and Y directions in the bottom edge of the soil; *i*) *Foundation support* where we fix the translations in the X direction in the symmetry edge of the footing.

We start by defining the supports that belong to the support set *Ground support*.

DIANAIE

Main menu → Geometry → Assign → Supports [Fig. 20] [Fig. 21]

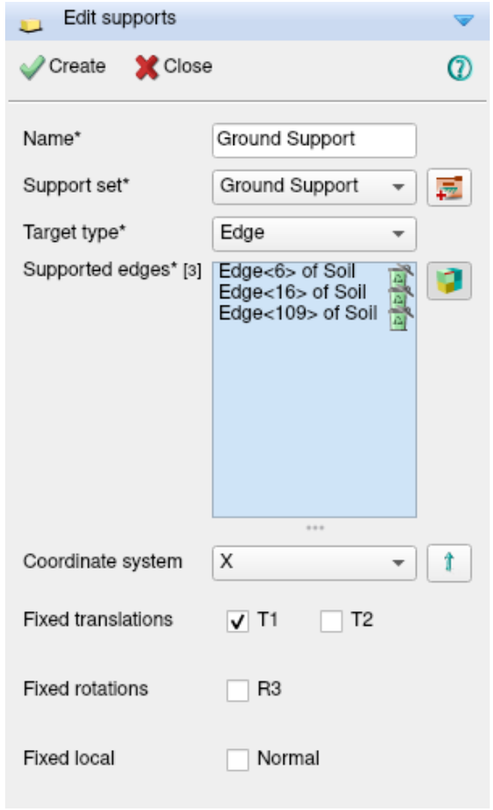


Figure 20: Add edge supports to the soil

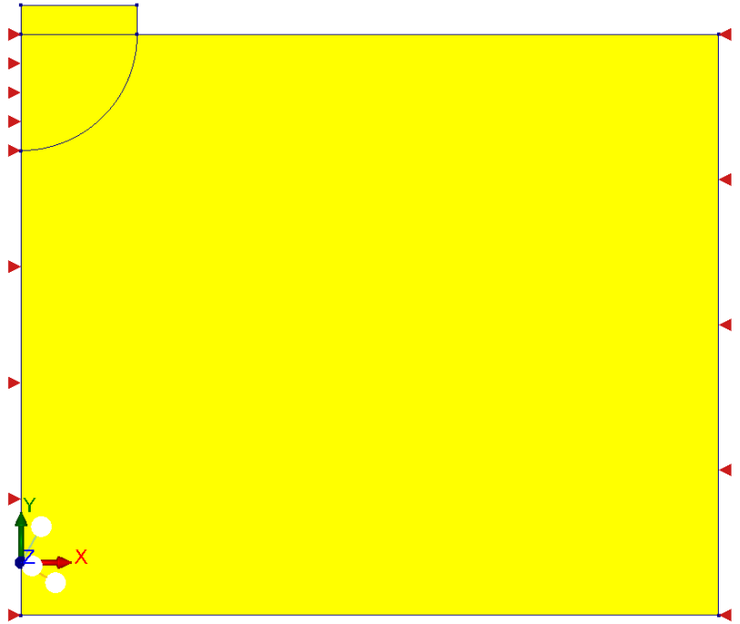


Figure 21: View of the model - supports

Main menu → Geometry → Assign → Supports [Fig. 22] [Fig. 23]

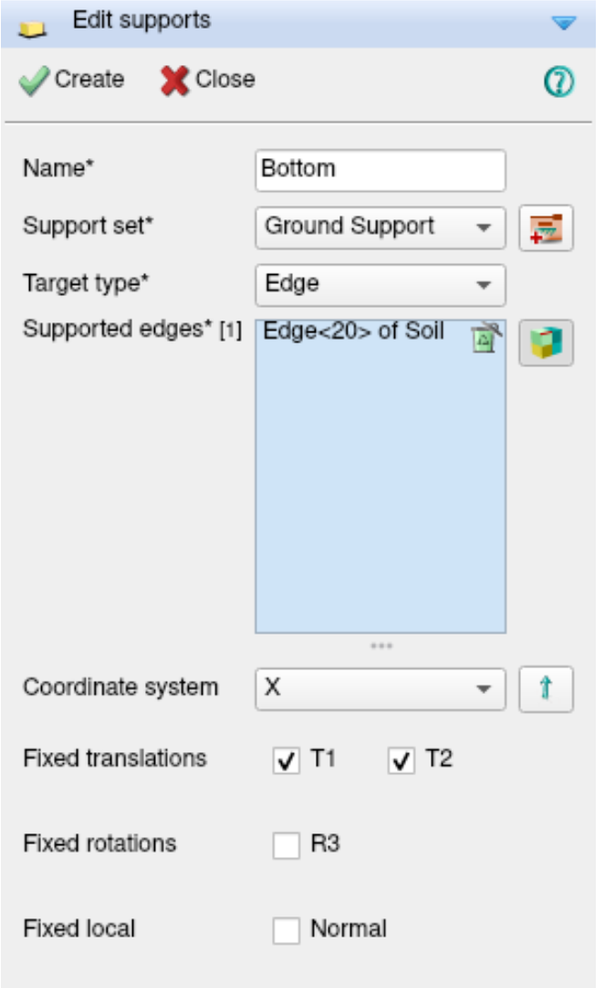


Figure 22: Add bottom supports to the soil

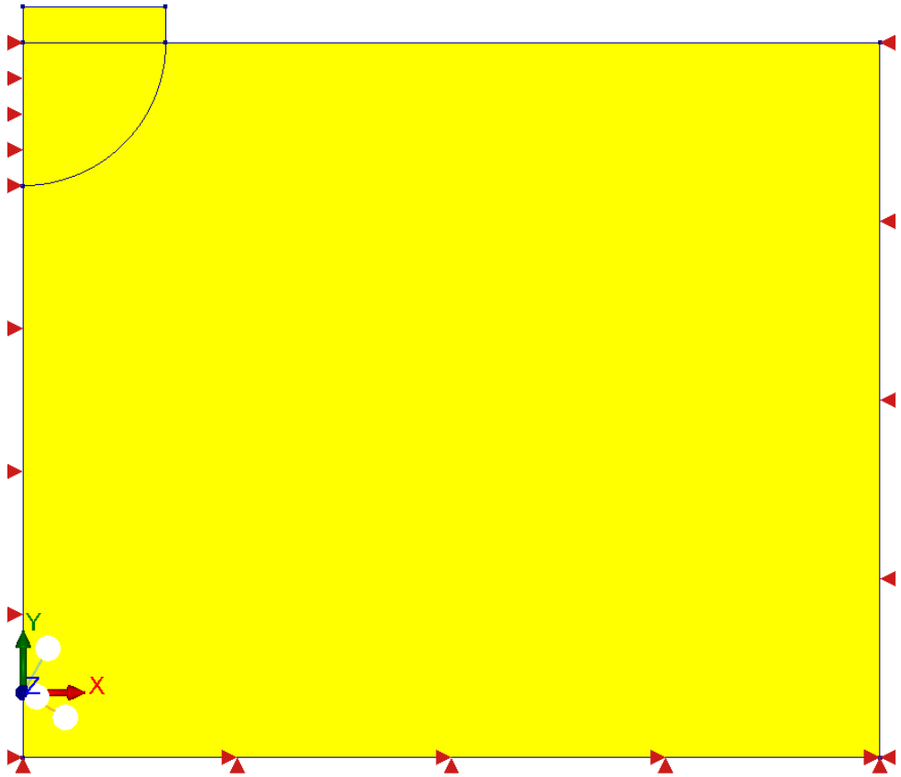


Figure 23: View of the model - supports

We now define the support set *Foundation support*.

Main menu → Geometry → Assign → Supports [Fig. 24] [Fig. 25]

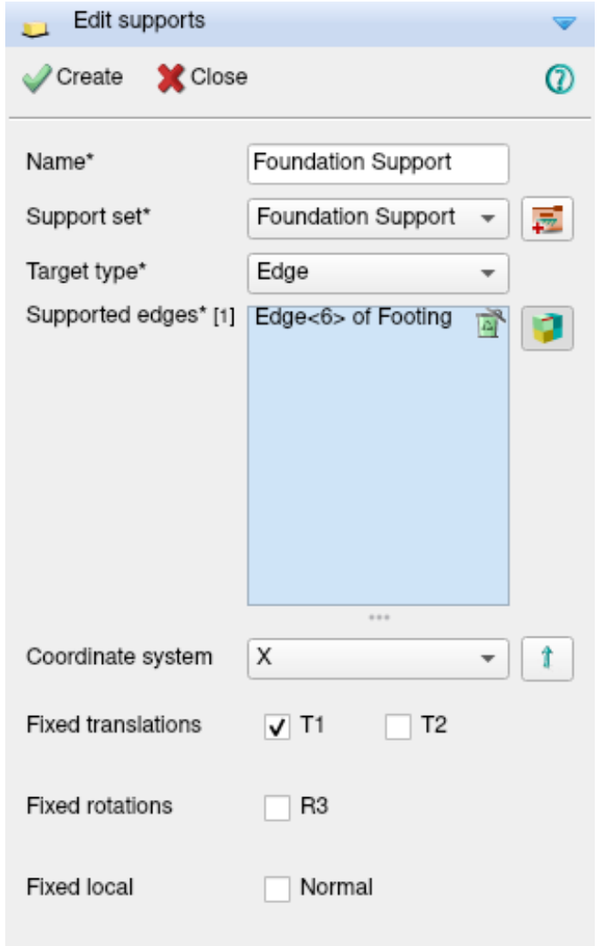


Figure 24: Add edge support to footing

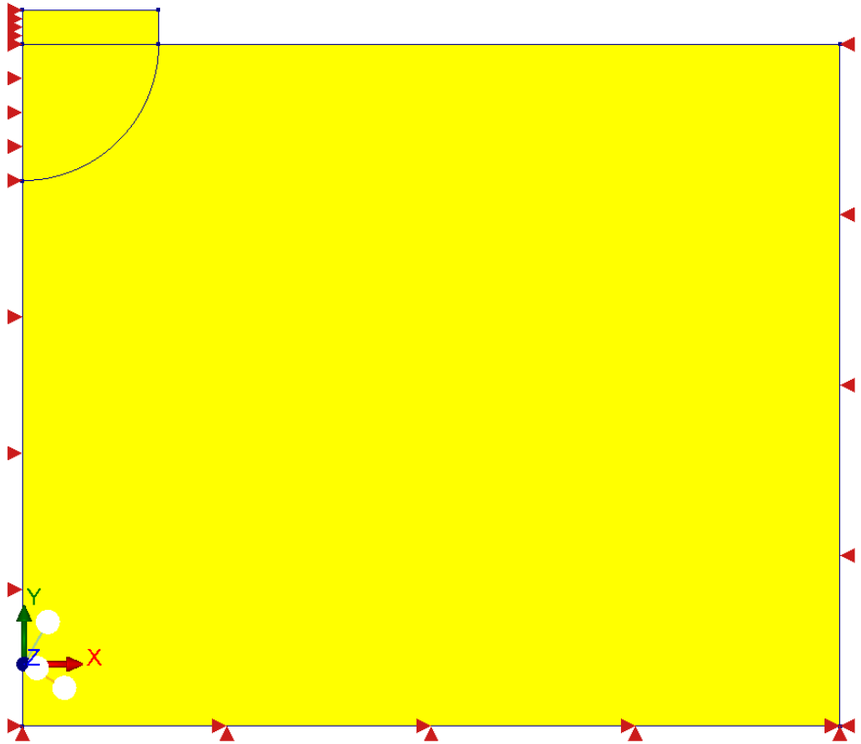


Figure 25: View of the model - supports

2.4 Loads

We define the pressure load acting on the footing (see Figure 1).

DIANAIE

Main menu → Geometry → Assign → Loads [Fig. 26] [Fig. 27]

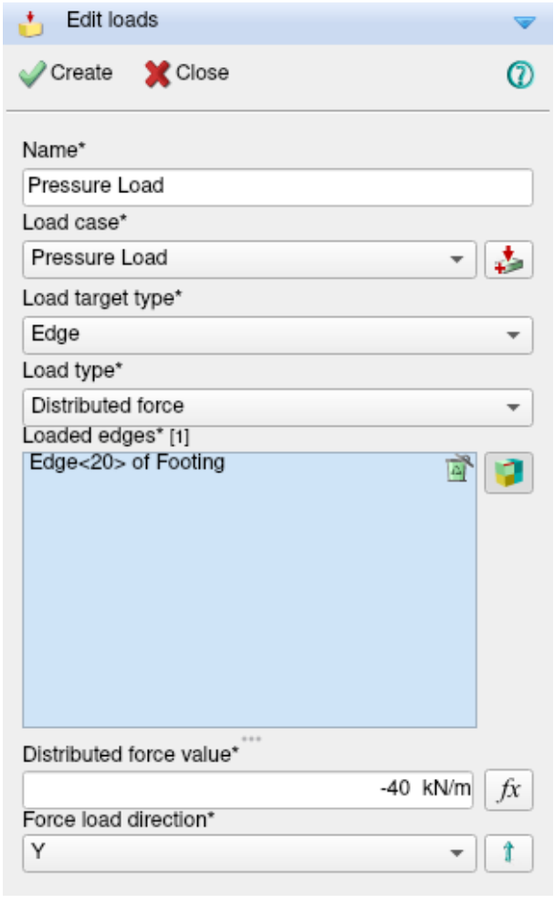


Figure 26: Add pressure load

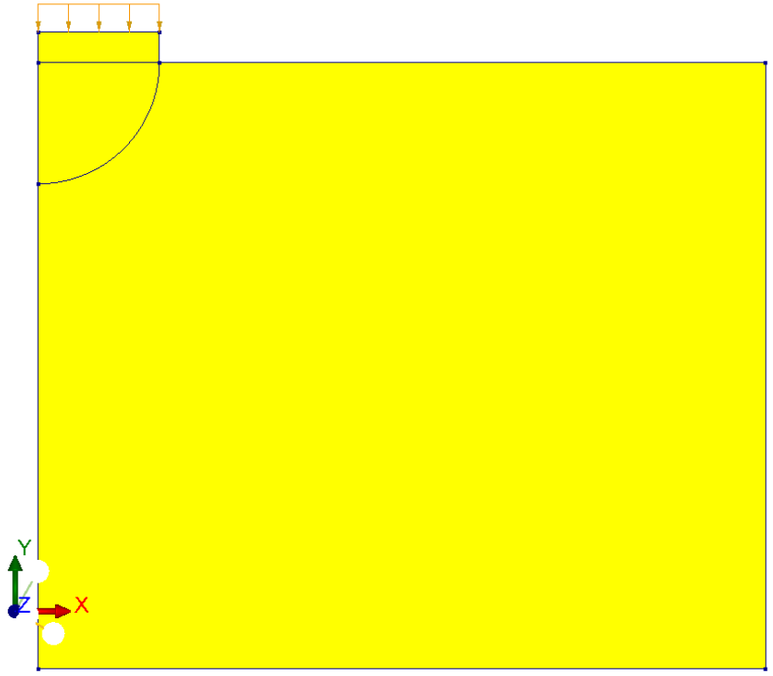


Figure 27: View of the model - pressure load

2.5 Mesh

We define the mesh properties. We use edge divisions as depicted in Figure 29.

DIANAIE

Main menu → Geometry → Mesh → Mesh properties [Fig. 28]
< Repeat this action for all the edges as presented in [Fig. 29] >

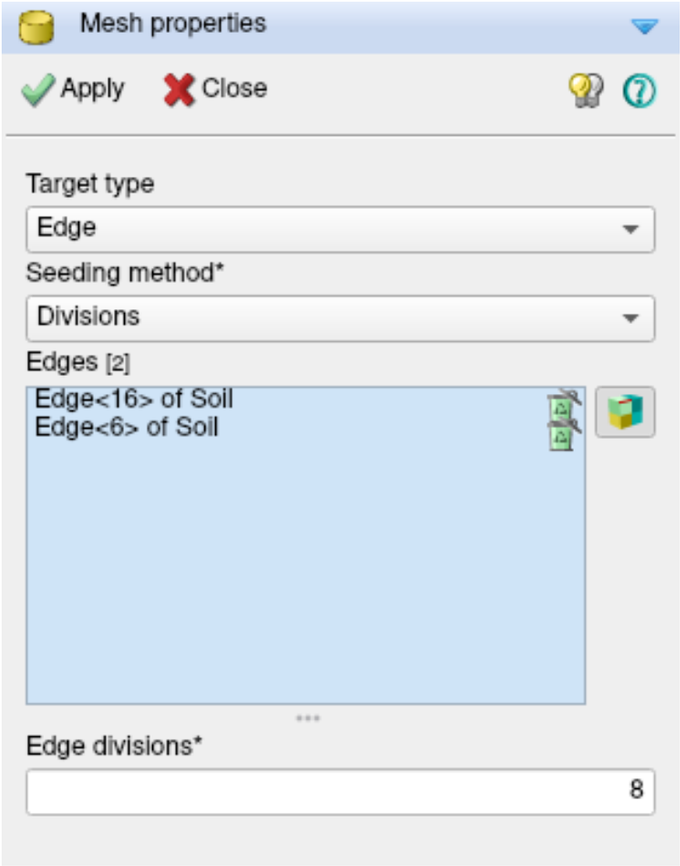


Figure 28: Set mesh properties

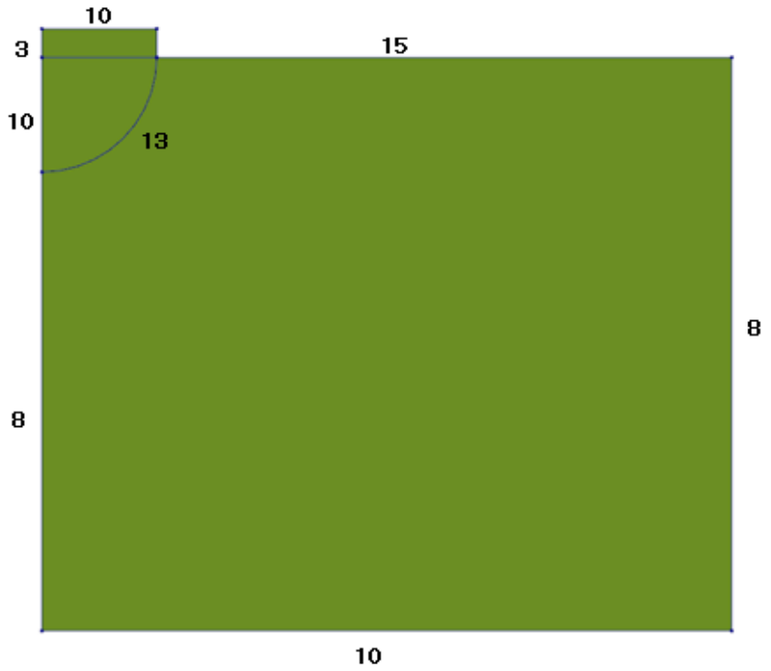


Figure 29: Mesh properties - edge divisions

And we generate the mesh.

Main menu → Geometry → Mesh → Generate mesh [Fig. 30]

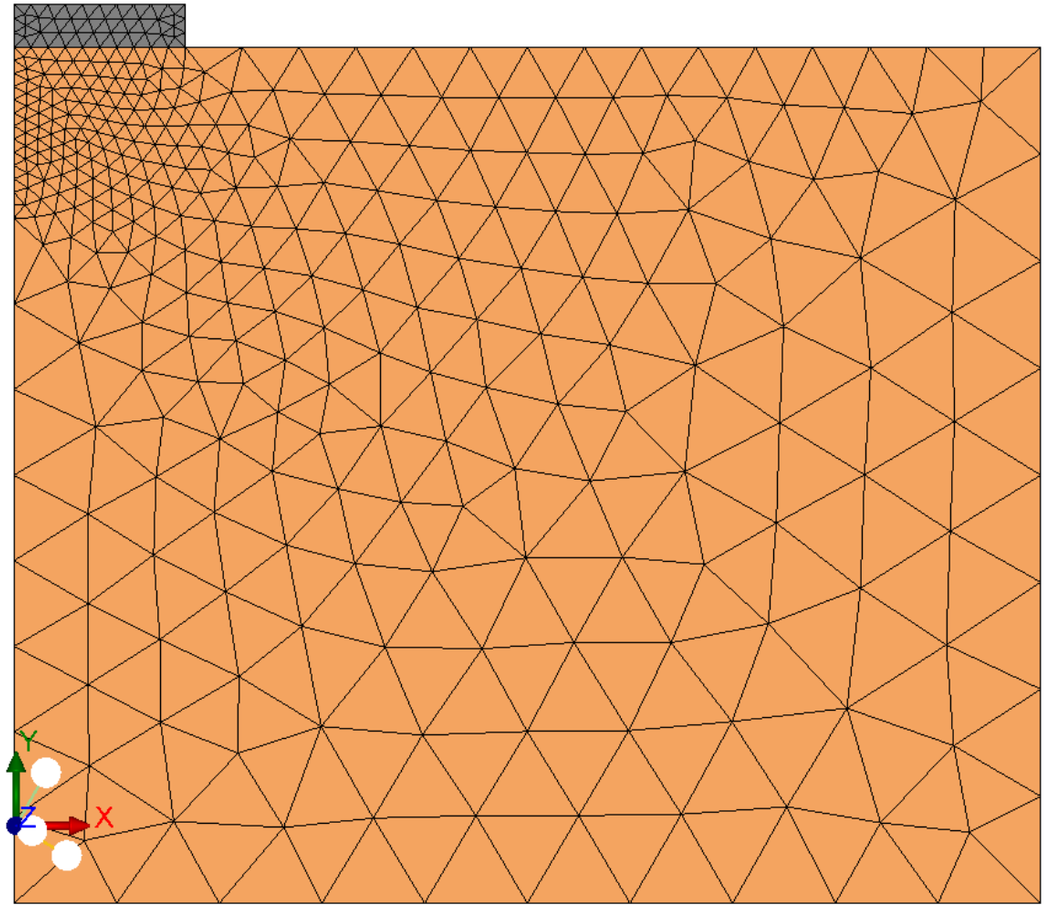


Figure 30: Finite element mesh

3 Structural Linear Static Analysis

3.1 Commands

We define the commands for a linear static analysis.

DIANAIE

Main menu → Analysis → Add analysis

Analysis browser → Analysis1 → Rename → Linear [Fig. 31]

Analysis browser → Linear → Add command → Structural linear static [Fig. 32] [Fig. 33]

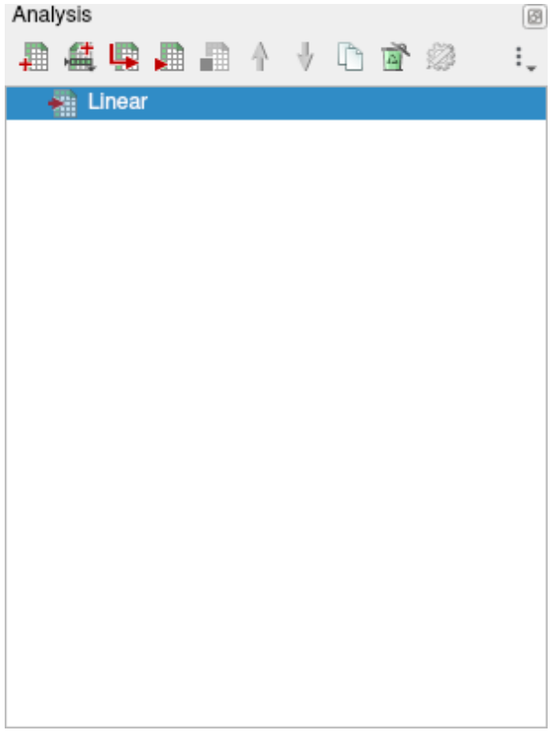


Figure 31: Analysis browser

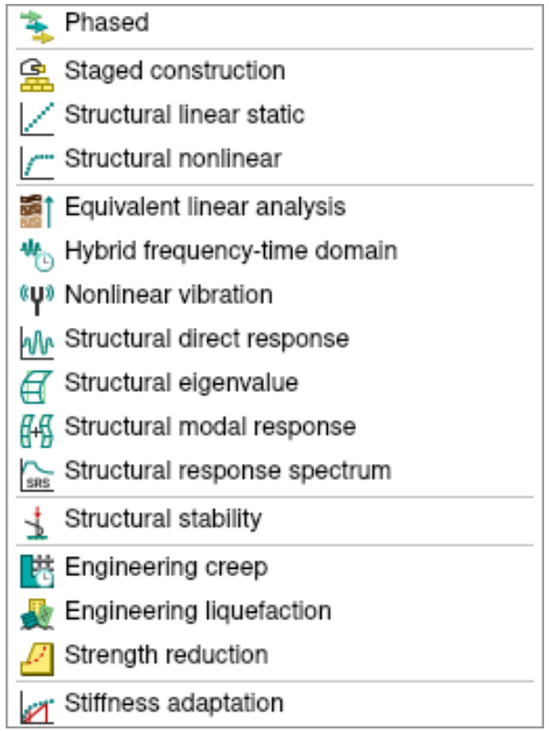


Figure 32: Add command

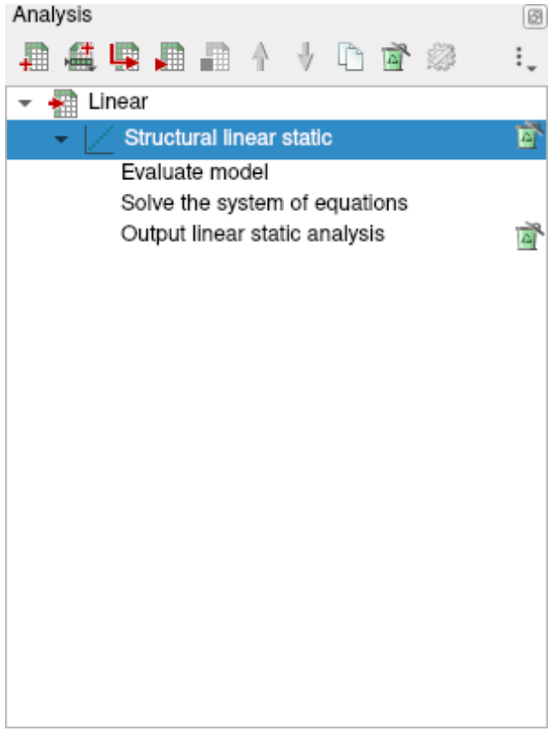


Figure 33: Analysis browser

In the output properties we select the results of displacements, strains and stresses. We change the output file name to "*Linear*". Finally we run the analysis.

DIANAIE

Analysis browser → Linear → Structural linear static → Output linear static analysis → Edit properties [Fig. 34]
Properties - Output → Properties → Output Device Properties [Fig. 35]
Properties - Output → User selection → Results Selection [Fig. 36]

Main menu → Analysis → Run selected analysis

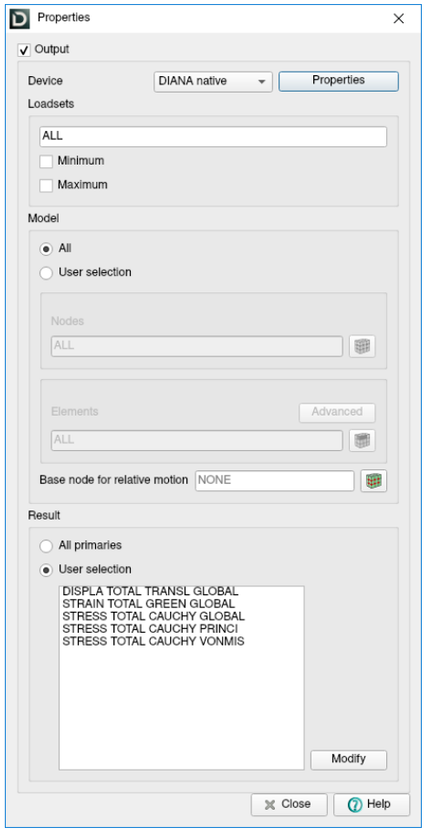


Figure 34: Output properties

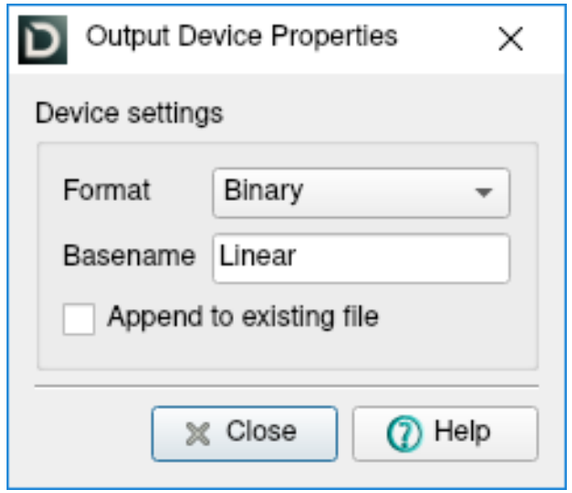


Figure 35: Output file

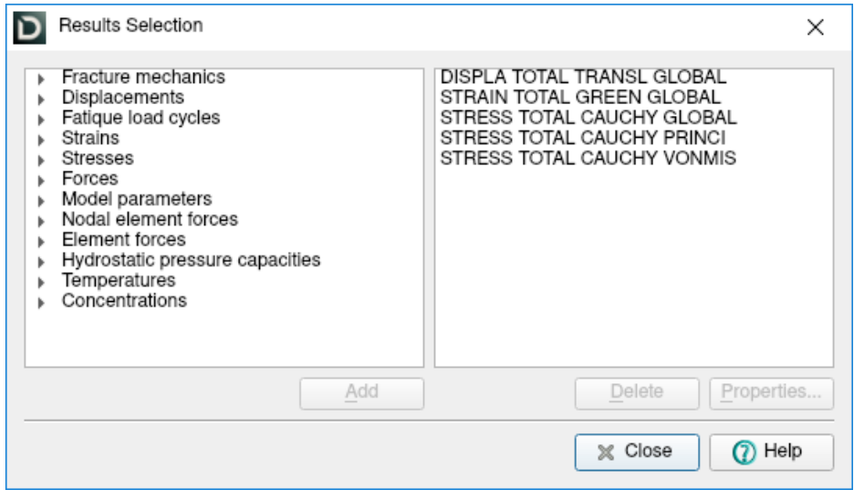


Figure 36: Results selection

3.2 Results

We show the contour plot of the vertical displacements [Fig. 39]. We can increase the number of levels of the contour plot in the properties panel [Fig. 38].

DIANAIE

Results browser → Linear → Output linear static analysis → Nodal results → Displacements → DtY [Fig. 37] [Fig. 39]
Property Panel → Result → Contour plot settings → Number of contour levels → 15 [Fig. 38]

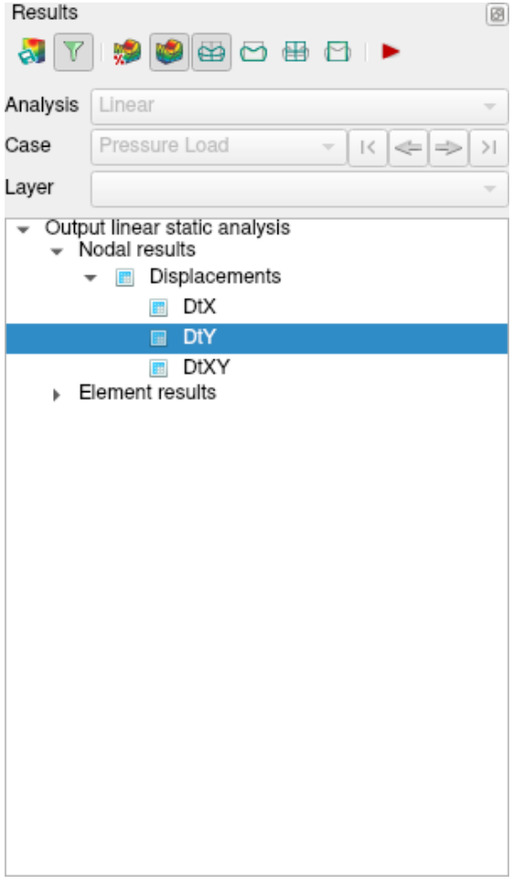


Figure 37: Results browser

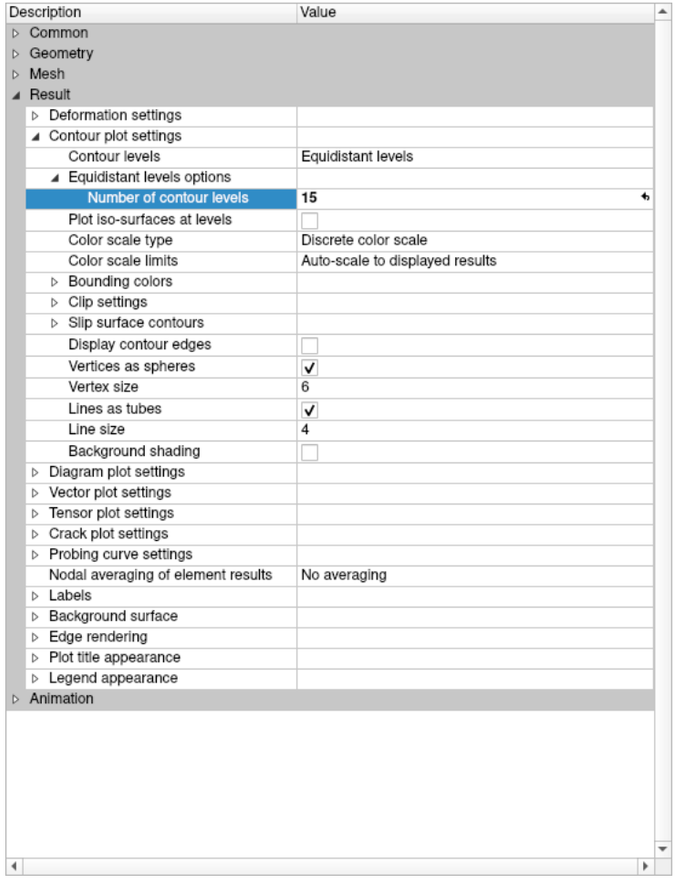


Figure 38: Contour plot properties

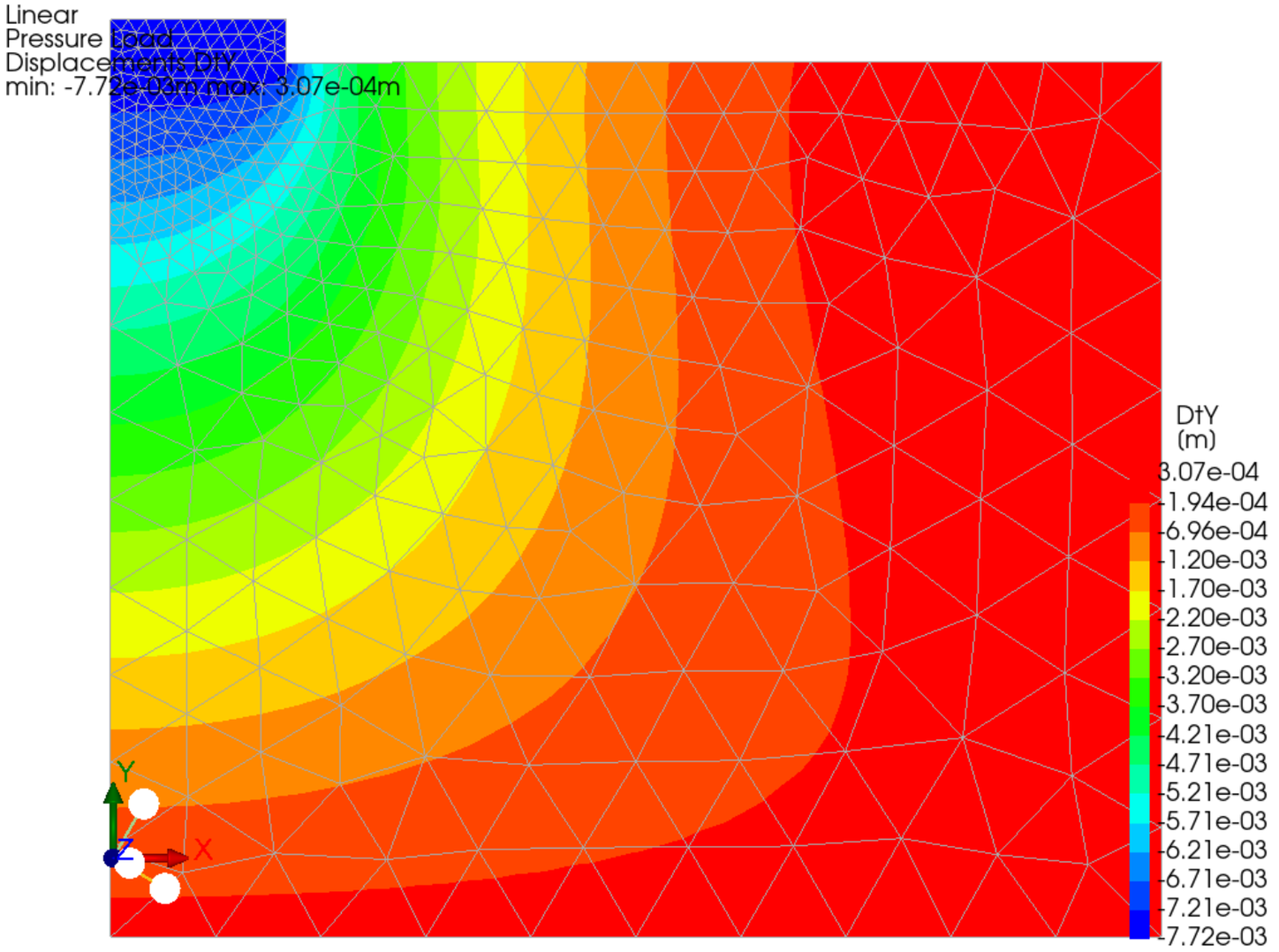


Figure 39: Displacements DtY

We select the nodes at the interface between the soil and the footing. We tabulate the vertical displacements for these nodes. We observe that the displacements in this area are approximately 7.77 mm.

DIANAIE

Main menu → Viewer → Selection mode → Node selection [Fig. 41]
< *Drag the mouse around the nodes at the connection between* Soil *and* Footing >

Results browser → Linear → Output linear static analysis → Nodal results → Displacements → DtY → Show table [Fig. 40] [Fig. 42]

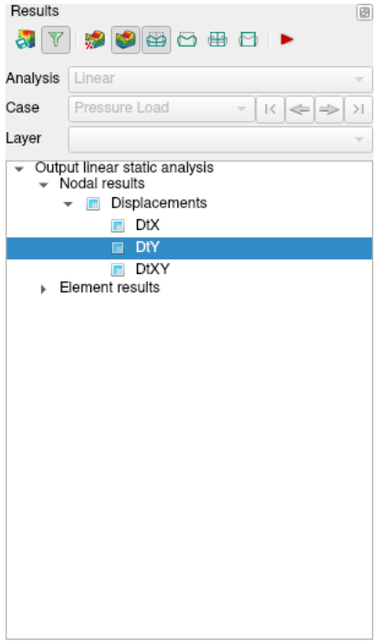


Figure 40: Results browser

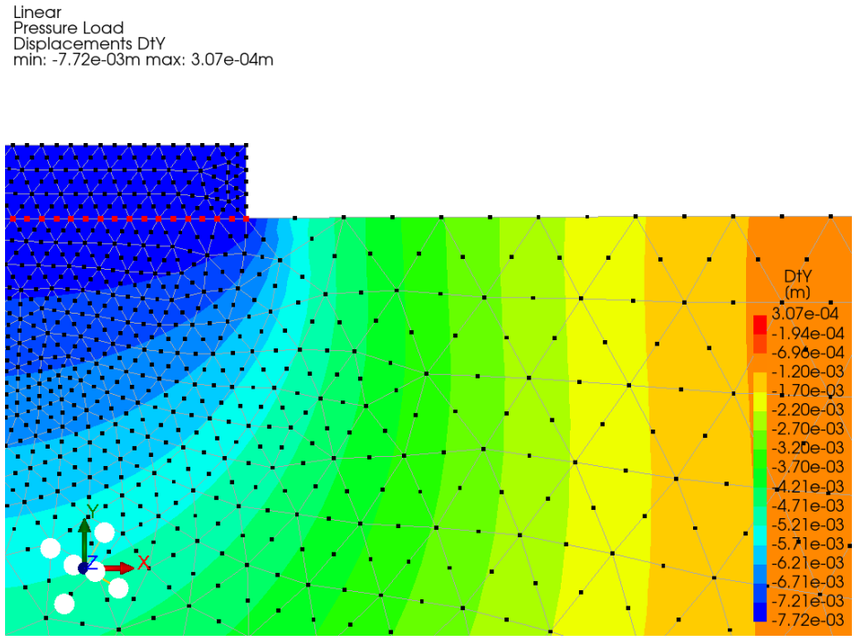
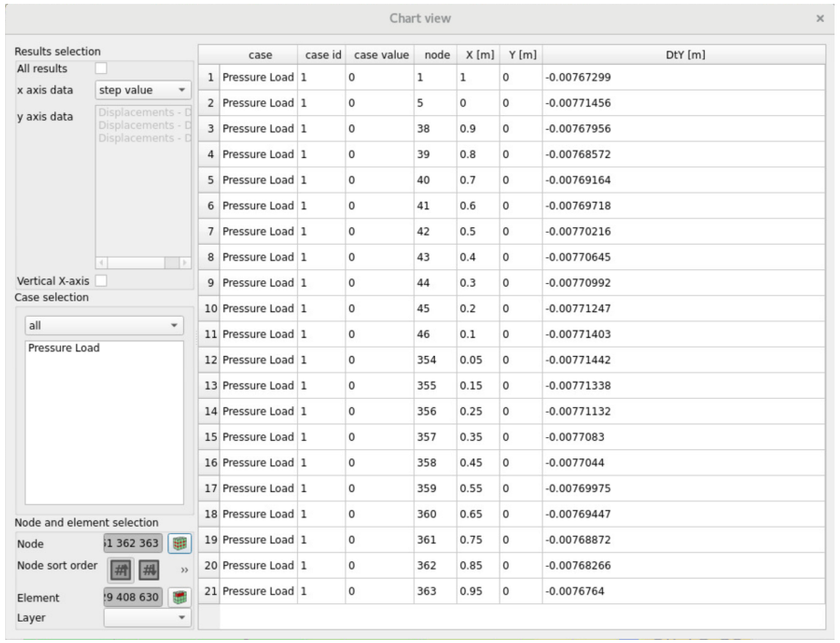


Figure 41: Select nodes



	case	case id	case value	node	X [m]	Y [m]	DtY [m]
1	Pressure Load	1	0	1	1	0	-0.00767299
2	Pressure Load	1	0	5	0	0	-0.00771456
3	Pressure Load	1	0	38	0.9	0	-0.00767956
4	Pressure Load	1	0	39	0.8	0	-0.00768572
5	Pressure Load	1	0	40	0.7	0	-0.00769164
6	Pressure Load	1	0	41	0.6	0	-0.00769718
7	Pressure Load	1	0	42	0.5	0	-0.00770216
8	Pressure Load	1	0	43	0.4	0	-0.00770645
9	Pressure Load	1	0	44	0.3	0	-0.00770992
10	Pressure Load	1	0	45	0.2	0	-0.00771247
11	Pressure Load	1	0	46	0.1	0	-0.00771403
12	Pressure Load	1	0	354	0.05	0	-0.00771442
13	Pressure Load	1	0	355	0.15	0	-0.00771338
14	Pressure Load	1	0	356	0.25	0	-0.00771132
15	Pressure Load	1	0	357	0.35	0	-0.0077083
16	Pressure Load	1	0	358	0.45	0	-0.0077044
17	Pressure Load	1	0	359	0.55	0	-0.00769975
18	Pressure Load	1	0	360	0.65	0	-0.00769447
19	Pressure Load	1	0	361	0.75	0	-0.00768872
20	Pressure Load	1	0	362	0.85	0	-0.00768266
21	Pressure Load	1	0	363	0.95	0	-0.0076764

Figure 42: Results table

3.3 Comparison with Analytical Solution

The settlement of an infinitely long rigid footing on smooth finite homogeneous isotropic ground is determined analytical through the expression from Giroud (1972)[2]:

$$\delta = \frac{P \times B}{E' \times \bar{p}_H} = \frac{F}{E' \times \bar{p}_H} \qquad (1)$$

$\delta =$ Settlement

$P =$ Normal pressure $= 40\ kN/m^2$

$E' =$ Young modulus $= 10\ kN/m^2$

$\bar{p}_H =$ parameter dependent on $\frac{H}{0.5B} = 0.99$ (for $\nu = 0.33$)

$F =$ pressure on footing

Analytical solution

$$\delta = \frac{40 \times 2}{10000}.99 = 7.92\ mm$$

Diana solution = 7.70 mm
The error in the FE solution is ≈ **3%**

[2] Giroud, *Settlement of rectangular foundation on soil layer*, 1972

We also show the principal minimum stresses for the soil.

Mesh browser → Mesh → Shapes → *Footing* → Hide
Results browser → Linear → Output linear static analysis → Element results → Cauchy Total Stresses → S3 [Fig. 43] [Fig. 44]

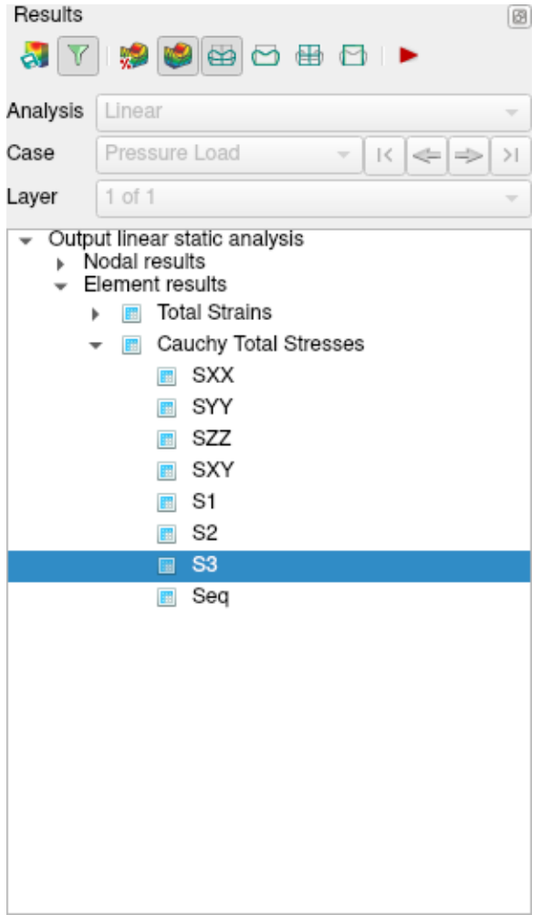


Figure 43: Results browser

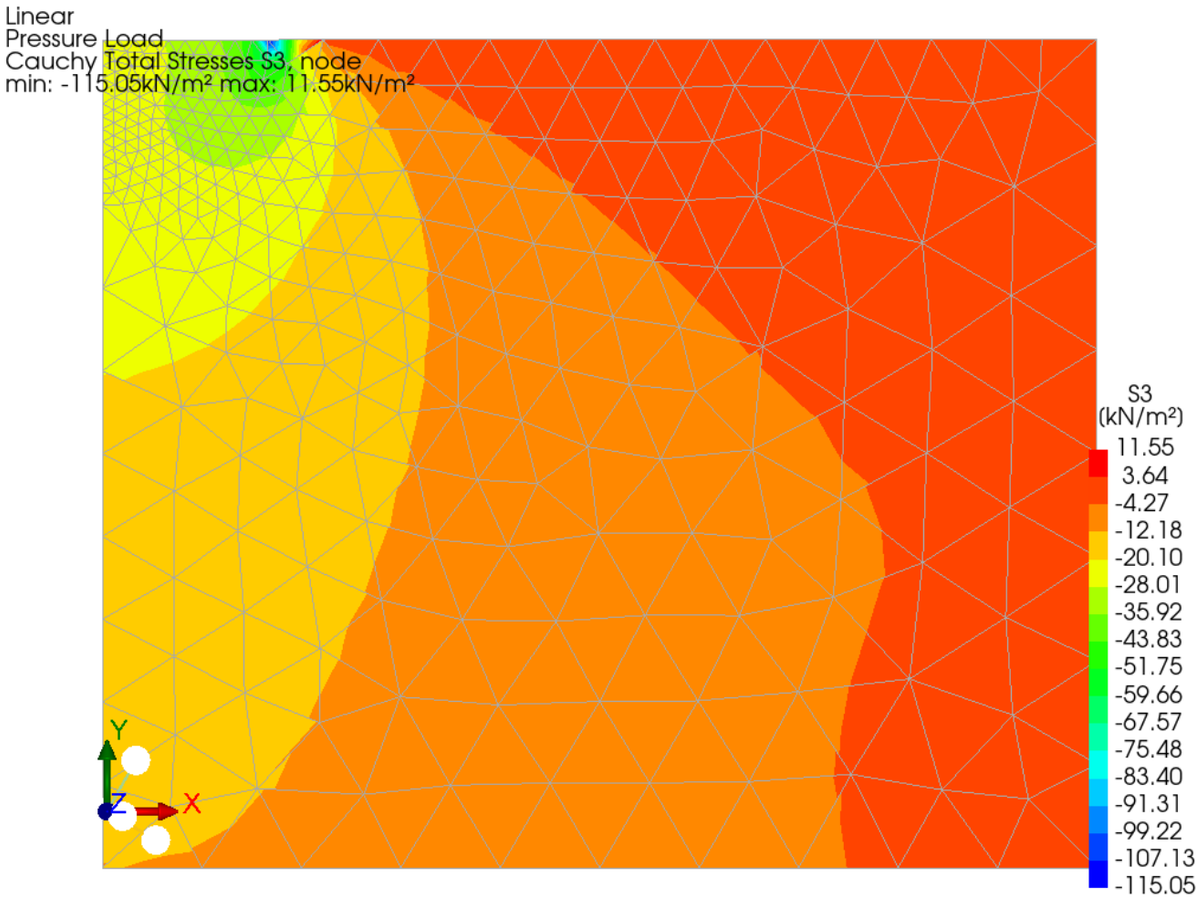


Figure 44: Stresses S2

Appendix A Additional Information

Folder: Tutorials/SettleStripFootingLinear

Number of elements ≈ 694

Keywords:
ANALYS: linear static.
CONSTR: suppor.
ELEMEN: ct12e pstrai.
LOAD: edge elemen force.
MATERI: elasti isotro porosi.
OPTION: direct units.
POST: binary ndiana.
PRE: dianai.
RESULT: cauchy displa green princi strain stress total vonmis.

References:

[1] J.P. Giroud. Settlement of rectangular foundation on soil layer. *Journal of the Soil Mechanics and Goundations Division, ASCE*, 98(1):149–154, 1972.

Laan van Waalhaven 462
2497 GR, The Hague
The Netherlands

DIANAFEA.COM INFO@DIANAFEA.COM +31(0)88 34 262 00

DIANA