Strip Footing - Ultimate Bearing Capacity
Outline

1 Description 3
  1.1 Material Properties ................................................................. 4
  1.2 Modeling Approach ................................................................. 5

2 Finite Element Model 7
  2.1 Geometry .................................................................................. 9
  2.2 Properties .................................................................................. 10
    2.2.1 Concrete ............................................................................... 10
    2.2.2 Soil ..................................................................................... 11
  2.3 Boundary Conditions ................................................................. 12
    2.3.1 Supports ............................................................................. 12
    2.3.2 Prescribed Displacement ..................................................... 14
  2.4 Loads ......................................................................................... 16
    2.4.1 Prescribed Displacement ..................................................... 16
    2.4.2 Self-weight ......................................................................... 16
  2.5 Mesh .......................................................................................... 17

3 Structural Nonlinear Static Analysis 21
  3.1 Commands ................................................................................ 21
    3.1.1 Phase 0 - K0 ....................................................................... 22
    3.1.2 Phase 1 - Footing ................................................................. 25
    3.1.3 Phase 2 - Displacement ....................................................... 28
  3.2 Results ....................................................................................... 31
    3.2.1 Vertical Displacements ........................................................ 31
    3.2.2 Total Displacements .............................................................. 33
    3.2.3 Load vs. Settlement Curve .................................................. 35

Appendix A Additional Information 36
1 Description

In this tutorial we investigate the ultimate bearing capacity $q_u$ of an infinite strip footing on a pure cohesive (frictionless) no-weight dry soil [Fig. 1]. The bearing capacity of the soil obtained from the finite element simulation in DAINAIE is compared with that from the Limit Equilibrium Analysis proposed by Terzaghi [Fig. 2]:

$$q_u = c N_c$$

where $c$ is the soil cohesion while the bearing dimensionless capacity factor $N_c = 5.14^1$.

---

^1Das and Sivakugan, *Fundamentals of geotechnical engineering*, 2016
1.1 Material Properties

The footing, made of concrete, is assumed isotropic and linear elastic. The mechanical response of the soil is simulated according to the Mohr-Coulomb model. The material properties for the footing *Concrete* and the soil *Soil-Clay* are summarized in Table 1.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Concrete</th>
<th>Soil-Clay</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density $\rho$</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>Young's modulus $E$</td>
<td>35000</td>
<td>35</td>
</tr>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.15</td>
<td>0.35</td>
</tr>
<tr>
<td>Friction angle $\phi$</td>
<td>0.02</td>
<td>°</td>
</tr>
<tr>
<td>Dilatancy angle $\psi$</td>
<td>0.0</td>
<td>°</td>
</tr>
<tr>
<td>Cohesion $c$</td>
<td>30</td>
<td>kN/m$^2$</td>
</tr>
<tr>
<td>Porosity</td>
<td>0.53</td>
<td></td>
</tr>
<tr>
<td>Lateral pressure ratio $K_0$</td>
<td>0.55</td>
<td></td>
</tr>
</tbody>
</table>

To enhance the numerical convergence of the analysis we assume a very small value for the soil friction angle $\phi$ (although a frictionless soil was considered). For the Mohr-Coulomb model, the cut-off based on principal axes is considered.
1.2 Modeling Approach

The following aspects are considered:

- two-dimensional plane strain is used for the modeling of this problem
- due to symmetry, only half of the geometry is modeled in DianaIE
- the lateral boundaries of the model (denoted by gray dashed lines in Figure 3) are constrained in normal displacements. Since only half of the model is considered, the normal displacement is constrained also along the symmetry axis ($X = 0$)
- a vertical downward displacement is incrementally applied on top of the concrete foundation and the corresponding reaction force is calculated
- as illustrated in Figure 4, to get the equivalent reaction force directly from the output, the vertical displacement is imposed only at point A. Tyings are used to force the displacements of the upper boundary of the footing (line A) to be to be equal to the displacement at point A (such a procedure is based on the master-slave method)
- akin to all geotechnical examples, a phased analysis is performed to account for stress initialization in the soil (although, this could be avoided due to the null weight of the soil and the footing). The three phases are:
  - phase 1: stress initialization in the soil without footing
  - phase 2: stress initialization in the soil due to the footing
  - phase 3: application of the incremental vertical load on the footing
- the load versus settlement curve is derived from the analysis results to assess the ultimate bearing capacity of the foundation
Figure 3: Geometry of the model (half of the geometry)

\[ c = 30 \text{ kPa} \]

\[ \phi = 0^\circ \]

\[ B = 8 \text{ m} \]

\[ 5B \]

\[ 5.5B \]

Figure 4: Scheme of the displacements applied at the footing using the master-slave method

\[ \Delta u_{y}^{\text{Point A}} = \Delta u_{y}^{\text{Line A}} \]
2 Finite Element Model

For the modeling session we start a new project in which structural analysis can be performed [Fig. 5] and plane strain conditions are imposed. The dimensions of the domain are set equal to 100 m. Quadratic triangular finite elements are used in the analysis.

Figure 5: New project dialog
We choose meter for the length unit, kilonewton for force unit and degree for the angle.
2.1 Geometry

To model the geometry of the soil and the strip foundation in DIANAIE, we create two polygon sheets with the size and the dimensions of the two shapes (see Figure 3) [Fig. 10].

![Add polygon sheet - Footing](image1)
![Add polygon sheet - Soil](image2)

**Figure 8: Add polygon sheet – Footing**

**Figure 9: Add polygon sheet – Soil**

**Figure 10: Top view – geometry**
2.2 Properties

2.2.1 Concrete

We assign the material properties to the footing.
2.2.2 Soil

We assign the material properties to the soil.
2.3 Boundary Conditions

2.3.1 Supports

We constrain the normal displacement along the right and bottom side of the soil [Fig. 19].
Similarly, we constrain the normal displacement of the soil and footing edges along $x = 0$ to impose the symmetry boundary conditions [Fig. 21].

**Figure 20:** Attach support – *Symmetry*

**Figure 21:** Top view - supports
2.3.2 Prescribed Displacement

In DianaIE, in order to apply a controlled displacement at point \( A \) (see Figure 3 and Figure 4), we first need to constrain its vertical translation and, then, apply a prescribed vertical deformation. Accordingly, point \( A \) is the *master node* for imposing a uniform vertical translation equal to \(-0.01\) m at the top of the footing.

Main menu ➔ Geometry ➔ Assign ➔ Add supports 🛡️ [Fig. 22]
Main menu ➔ Geometry ➔ Assign ➔ Add loads 🤖 [Fig. 23]

Figure 22: Attach support – *Point A*
Figure 23: Attach load – *Point A*
Figure 24: Top view - imposed displacement
Having defined the boundary conditions at point $A$, we apply the same conditions on the points along line $A$ (see Figure 4). In DianaIE, this is performed by using the tying tool. For the present case, the vertical displacement of the nodes along line $A$ (slave nodes) is set equal to that at point $A$ (master node).

**Main menu** → Geometry → Assign → Add tyings  🌻  [Fig. 25] – [Fig. 27]
2.4 Loads

2.4.1 Prescribed Displacement

The prescribed displacement was defined in Figure 23 along with the required support condition in Figure 22.

2.4.2 Self-weight

We create a load case for the self-weight.

![Main menu ➔ Geometry ➔ Assign ➔ Add global loads](https://dianafea.com)

Figure 28: Global load - self-weight
2.5 Mesh

We define the mesh in the footing by specifying the characteristic size of the elements equal to 0.5 m.

Figure 29: Mesh seeding for the *Footing*

Figure 30: Selection of the *Footing*
To discretize the soil, we use a graded mesh such that the elements close to the footing are smaller than those at the border of the domain. Therefore, we need to specify the mesh seeding of the top and left edges of the Soil shape.

We start with the top edge.

**Figure 31:** Mesh seeding for the top edge of the Soil

**Figure 32:** Selection of the top edge of the Soil

Main menu ➔ Geometry ➔ Assign ➔ Mesh properties  🍀 Fig. 31  🍀 Fig. 32
And now the left edge.

Figure 33: Mesh seeding for the left edge of the Soil

Figure 34: Selection of the left edge of the Soil
Finally, we can generate the finite element mesh.

Figure 35: Finite element mesh
3 Structural Nonlinear Static Analysis

3.1 Commands

Before the application of the load on the strip foundation, we need to initialize the stresses in the soil, both before and after the placing of the foundation on the soil. Thus, we use a phased analysis. We define three phases Table 2: i) initialization of the stresses in the soil due to its self weight; ii) emplacement of the footing on the soil; iii) application of the load on the footing.

<table>
<thead>
<tr>
<th>Phase number</th>
<th>Phase name</th>
<th>Description</th>
<th>Analysis type</th>
<th>Block type</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>K0</td>
<td>stress initialization (without footing)</td>
<td>nonlinear analysis</td>
<td>start step</td>
</tr>
<tr>
<td>1</td>
<td>Footing</td>
<td>stress initialization (with footing)</td>
<td>nonlinear analysis</td>
<td>start step</td>
</tr>
<tr>
<td>2</td>
<td>Displacement</td>
<td>the vertical settlement at the footing (point A) is increased up to failure</td>
<td>nonlinear analysis</td>
<td>start step &amp; displacement load</td>
</tr>
</tbody>
</table>

To set-up a phase we follow the following steps:

1. create a new phase (rename accordingly)
2. open the phase Edit properties dialog box to:
   - select the Element sets to be active during the phase
   - select the element material properties (if required)
   - select the Support sets and Tying sets to be active during the phase
3. add the required analysis commands (in this tutorial we always perform Structural nonlinear analysis):
   - add a Start step if new elements are included in the model
   - set up a Load step (or remove it if not needed in the phase)
   - set up the details for the calculations (e.g., solver, convergence criteria, superposition, ...)

---

2 Due to the negligible density of the soil and the foundation this phased analysis could be avoided. Nevertheless, this is performed in the present tutorial since a typical geotechnical simulation would required a phased analysis.
3.1.1 Phase 0 - K0

We start by creating the first phase to initialize the stresses in the model without the footing. Since the footing is not considered in this phase, we do not take into account the boundary conditions attached to it (these settings are specified in the phase properties [Fig. 39]).

---

**Main menu** → Analysis → Add analysis

**Analysis browser** → Analysis1 → Rename → PhasedAnalysis → Phased

**Analysis browser** → PhasedAnalysis → Add command → Phased

**Analysis browser** → PhasedAnalysis → Phased → Rename → K0

**Analysis browser** → PhasedAnalysis → K0 → Edit phases

---

**Figure 36**: Analysis browser

**Figure 37**: Command menu

**Figure 38**: Analysis browser

**Figure 39**: Edit phase K0 properties
We add a *Structural nonlinear* analysis command to the *K0* phase to perform the stress initialization in the soil with a *Start step* execution block (here, we have to remove the default *new execution block*).

Figure 40: Add *Structural nonlinear 0*

Figure 41: Remove the default *new execute block*

Figure 42: Add *Start step 0*
We need to set up the details of the *Start step 0* execution block.

The stress initialization in soil is achieved by balancing the internal stress state in the soil with the external load (the self-weight). Therefore, this step is used to create a stress state in the soil that is in equilibrium with its self-weight and such that the displacement field is null in the entire soil model. Here, we assume that the stress state in the soil due to its weight is elastic (this is admissible due to the soil zero density). Accordingly, only one iteration is required for the calculation of the stress field in the soil.
3.1.2 Phase 1 - Footing

We create a new phase to place the footing [Fig. 47].

**Figure 45: Command menu**

**Figure 46: Analysis browser**

**Figure 47: Edit phase Footing properties**
We add a *Structural nonlinear* analysis command to the *Footing* phase.

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Add command* ➔ *Structural nonlinear*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear* ➔ *Rename* ➔ *Structural nonlinear 1*  

*Fig. 48*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 1* ➔ *new execute block* ➔ *Remove*

*Fig. 49*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 1* ➔ *Add...* ➔ *Execute steps* ➔ *Start steps*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 1* ➔ *new execute block* ➔ *Rename* ➔ *Start-step 1*

*Fig. 49*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 0* ➔ *Footing*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 0* ➔ *Evaluate model* ➔ *Nonlinear effects* ➔ *new execute block* ➔ *Solution method* ➔ *Output*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 0* ➔ *Footing*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 0* ➔ *Evaluate model* ➔ *Nonlinear effects* ➔ *Solution method* ➔ *Output*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 1* ➔ *Evaluate model* ➔ *Nonlinear effects* ➔ *Solution method* ➔ *Output* ➔ *Start step 1*

**Analysis browser** ➔ *PhasedAnalysis* ➔ *Structural nonlinear 1* ➔ *Evaluate model* ➔ *Nonlinear effects* ➔ *Solution method* ➔ *Output* ➔ *Start steps* ➔ *Equilibrium iteration* ➔ *Logging information* ➔ *Solution method* ➔ *Output*

**Figure 48**: Add *Structural nonlinear 1*

**Figure 49**: Remove the default *new execute block*

**Figure 50**: Add *Start step 1*
We set up the *Start step 1*. Differently from the previous case, we do not need to specify a new load set: we use the load from the previous phase (the self-weight). Only one iteration is needed to calculate the stress field in equilibrium with the external load.
3.1.3 Phase 2 - Displacement

We create a last phase to add the incremental vertical displacement on the footing. Here, all sets of elements, constraints and tyings are considered in the analysis [Fig. 55].

Figure 53: Command menu

Figure 54: Analysis browser

Figure 55: Edit phase Displacement properties
We add a Structural nonlinear analysis command for the Displacement phase. Here, we need a start step (named Start step 2) to account for the stresses from the previous phase and a load execute block (named Displacement) to apply the displacement load to the footing.

Then, we can run the analysis.
For the Displacement execution block we set up load steps, convergence criteria and solution method. For the imposed displacement we define the User specified sizes [Fig. 61] equal to 1.0(30); \( n(m) \) means that a load factor \( n \) for the load set Displacement is repeated \( m \) times consecutively. Then, we set the maximum number of iterations equal to 40 and use the Secant (Quasi-Newton) iterative method.
3.2 Results

3.2.1 Vertical Displacements

We make a contour plot of the total vertical displacement $TD_{tY}$. To compare the contour plots at different load steps, we set the minimum and maximum values of the color scale to -0.3 and 0.01 m, respectively, that correspond to the extreme results of $TD_{tY}$ in the analysis.

Results browser ➔ PhasedAnalysis ➔ Output ➔ Nodal results ➔ Total displacements ➔ $TD_{tY}$  [Fig. 63]

Property Panel ➔ Result ➔ Contour plot settings  [Fig. 71]

Figure 63: Results browser

Figure 64: Output settings
We can inspect the vertical displacement in the soil for the different load cases. Here, we show the contour plot of TDtY for load step case 11, 21 and 31, which correspond to -0.1, -0.2 and -0.3 m of imposed displacement, respectively.

As expected, the vertical displacement of the footing increases with the imposed displacement. In particular, it is possible to observe a triangular region of soil (blue region) where all points have similar vertical displacement.

![Figure 65: Displacements TDtY – Load case 11 (imposed displacement = -0.1 m)](image1)

![Figure 66: Displacements TDtY – Load case 21 (imposed displacement = -0.2 m)](image2)

![Figure 67: Displacements TDtY – Load case 31 (imposed displacement = -0.3 m)](image3)
3.2.2 Total Displacements

It is also interesting to display the vector plot of the total displacement field $TDtXYZ$ after failure (in this case at load step 31 that corresponds to -0.30 m to imposed displacement).

The vector plot of the total displacement field provides considerable insights regarding the failure mechanism in the soil. As shown in Figure 70, it is possible to identify the three regions denoted by the failure surfaces in the soil. The first triangular region (in red) moves vertically with the footing. The second region, or radial shear zone (in green), is subjected to a rotation. Finally, the third region, or triangular Rankine passive zone (in blue), is pushed towards the top-right area.
We can use the mirroring option in DianaIE to see the results in the complete geometry.

**Property Panel ➔ Common ➔ Mirroring [Fig. 71]**

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Common</td>
<td></td>
</tr>
<tr>
<td>View Points</td>
<td></td>
</tr>
<tr>
<td>Workplane</td>
<td></td>
</tr>
<tr>
<td>Node size</td>
<td>4</td>
</tr>
<tr>
<td>Edge size</td>
<td>1</td>
</tr>
<tr>
<td>Fixed symbol size</td>
<td>✓</td>
</tr>
<tr>
<td>Shrink factor</td>
<td>0.6</td>
</tr>
<tr>
<td>Label color</td>
<td>#00ffff</td>
</tr>
<tr>
<td>Label font size</td>
<td>9</td>
</tr>
<tr>
<td>Selections</td>
<td></td>
</tr>
<tr>
<td>Highlights</td>
<td></td>
</tr>
<tr>
<td>Supports</td>
<td></td>
</tr>
<tr>
<td>Loads</td>
<td></td>
</tr>
<tr>
<td>Axes</td>
<td></td>
</tr>
<tr>
<td>Ruler settings</td>
<td></td>
</tr>
<tr>
<td>Phase</td>
<td></td>
</tr>
<tr>
<td>Water level</td>
<td></td>
</tr>
<tr>
<td>Mirroring</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 71: Common settings**

**Figure 72: Mirroring results: displacement TDtXYZ – Load case 31**
3.2.3 Load vs. Settlement Curve

We create a load versus settlement curve in DianaIE.

The value of the ultimate bearing capacity multiplied by 2 (since we considered only half of the model) is approximately 1280 kN. This result is in good agreement with that from Terzaghi ($q_u = 1237$ kN, as shown in Section 1): the difference is around 3.4%.
Appendix A  Additional Information

Folder: Tutorials/StripFoundation

Number of elements  $\approx 400$

Keywords:
- ANALYS: nonlin phase physic.
- CONSTR: suppor tying.
- ELEMEN: ct12e pstrai.
- LOAD: deform weight.
- MATERI: consta crack cutoff elastif full harden isstro mohco multil plasti porosi retent smear soften soil strain.
- OPTION: bfgs direct newton nonsym regula secant units.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy displa extern force green reacti strain stress total.

References:

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.