LS-G-AN5
Software: LimitState:GEO 2.0f
Date: February 2011

Application Note - Footing Analysis

1 Introduction

This note describes the typical steps followed in setting up and analysing a footing problem with LimitState:GEO.

All files used in this note are available in a zip file that can be downloaded from http://www.limitstate.com/files/application-notes/LSGAN5/footing_analysis.zip

Familiarity with the use of LimitState:GEO is assumed. The reader is referred to the user manual for further information on any features discussed in this note.

A movie with all the steps undertaken in setting up and solving the problem is available at http://www.limitstate.com/files/videos/flash/footingApplicationNote.htm

2 Problem definition

The specified problem depicted in Figure 1 involves analysis of a 1.0 m wide strip footing required to carry a vertical load of 200 kN/m. The footing is founded at a depth of 0.8 m in a silty clay underlain by a layer of sandy clay. The problem is to be checked against Eurocode 7 Design Approach 1 requirements for long term conditions. In this example it is assumed that the strength of the soil above foundation level *does* contribute to the bearing capacity (it is often ignored).

The main problem geometry is available as a series of co-ordinates. The modelled soils and associated parameters are listed in Tables 1 and 4 in Section 3.

Details of how to set up the problem manually are given in Section 3. To skip direct to the analysis stage, please go to Section 4.

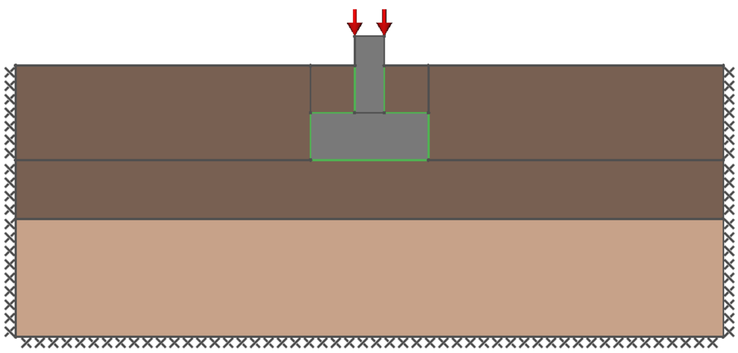

Figure 1: Main problem geometry

3 Setting up the model and solving the problem

3.1 Footing: Setting up problem geometry, materials and loading

In this example, the LimitState:GEO model was created using the built in footing wizard:

1. Start the wizard using **File/New.../Vertically Loaded Footing Project**. You will be asked to complete tables with geometry, properties etc. in order to define the problem. In the **Project** tab, enter any general information about the project. Then click **Next**> .
2. In the **Geometry** tab, define the problem geometry using the parameters shown in Figure 2. Click **Next**>.

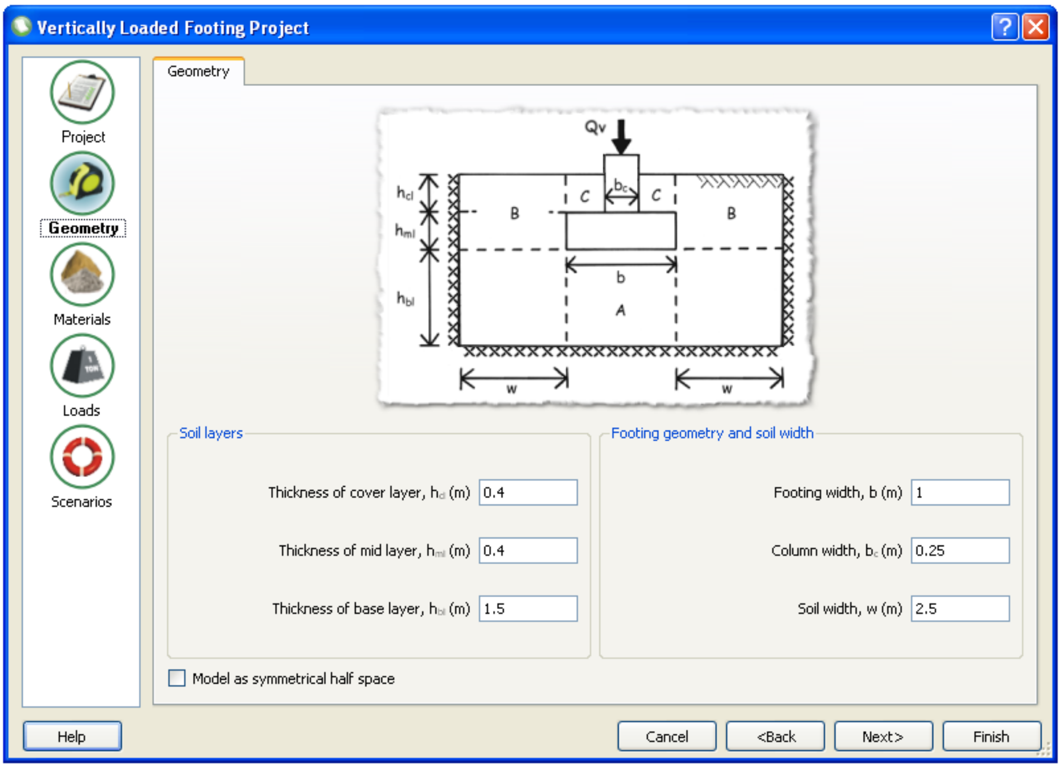

Figure 2: Gravity wall wizard - geometry section

3. Specify the properties of the soil layers with the values listed in Table 1 and click **Next**>. When specifying properties of Layer A use the **Create new material** option. For Layer B and Layer C choose the **Use Layer A material** and **Use Layer B material** option respectively. Note that soil property ϕ' in Table 1 and elsewhere in this Application Note is set as the peak state angle of shearing resistance i.e. $\phi' = \phi'_{\textbf{peak}}$.

Soil layer	Soil name	Material drainage behaviour	c_u (kN/m^2)	c$'$ (kN/m^2)	ϕ'	γ (kN/m^3)	γ_{sat} (kN/m^3)
A	Silty Clay	Drained/undrained	80	2	27°	21	21
B	Silty Clay	Drained/undrained	80	2	27°	21	21
C	Silty Clay	Drained/undrained	80	2	27°	21	21

Table 1: Soil properties

4. Use the default material properties of the footing (as listed in Table 2) and click **Next**>.
The footing itself is modelled as a Mohr-Coulomb material where c_u represents the shear strength of the footing and allows LimitState:GEO to model internal footing failure by shearing. However, if internal structural failure is not of interest, then for computational efficiency use of a *Rigid* material is recommended. This can be applied to the wall using drag and drop after finishing the initial set up with the wizard. For more information on modelling internal structural failure in LimitState:GEO, see Technical Note: **Modelling Structural Resistance LS-G-TN4**.

The values of the multipliers on the soil/wall interface properties specified in the Wizard are not relevant to this example since a different approach for their creation is used. See box below for more information.

Material Name	$\mathbf{c_u}$ (kN/m^2)	γ (kN/m^2)	Multiplier on tan ϕ	Multiplier on $\mathbf{c', c_u}$
Footing	5000	24	1	1

Table 2: Footing and soil-footing interface properties

Key modelling concept: *Soil/structure interface properties in LimitState:GEO*

Often when dealing with soil/structure interaction problems, it is desirable to define a soil/structure interface property that is a function of the adjacent soil. A typical example is a retaining wall where the interface adhesion and friction will be some multiplier (≤ 1) of the adjacent soil strengths. This may be set using **Derived Mohr Coulomb Material** type and entering multipliers for cohesion (c', c_u) and shear resistance ($\tan\phi'$) in the **Property Editor**. If the Wizard is used, this material is automatically applied to all the soil-structure interfaces using the specified multipliers.

Note that:

- If the interface friction property is to be a function of ϕ' rather than $\tan\phi'$, a standard material type (e.g. **Mohr Coulomb**) instead of a derived material type should be used.
- If the interface friction property is to be a function of ϕ_{crit} but the actual value used for the adjacent soil in LimitState:GEO is not a critical state angle of shearing resistance but e.g. a peak value, then it will be necessary either to make a modification to the multiplier or to use a standard material as above.

5. In the **Loads** tab, set the **Variable vertical load** to 200 kN/m and click **Next**>.
6. In the **Scenarios** tab, application of partial factor sets and long/short term analysis modes can be specified. In this example Eurocode 7 Design Approach 1 Combinations 1 and 2 (EC7 DA1/1 and DA1/2) for long term analyses are to be checked. Select **Multiple scenarios** and create the two Scenarios as defined in the first two columns of Table 3. Click **Finish**. (NB For simplicity, the pore pressures in the clay are conservatively assumed to be zero in the drained analyses, assuming the water table lies a short distance beneath the problem geometry.)

Scenario	Scenario details	Adequacy Factor
1	EC7 DA1/1, long term analysis	1.50
2	EC7 DA1/2, long term analysis	1.19

Table 3: Scenarios and analysis results

7. The Wizard automatically sets the **Loading Type** of the applied load and the self weight of the footing to *Unfavourable*. Check these settings in the **Property Editor** by selecting the relevant **Solids** or **Boundary**. The wizard does not automatically set the weight of the soil above the footing as *Unfavourable*. However in this example this will be done. Click on the relevant **Solids** and set the **Loading Type** to *Unfavourable*.
8. Switch on the **Grid** using the **View/Show Grid** function or .

9. Draw a horizontal line at level $y = 1.0$ m from the left to the right boundary of the model using **Draw/Line** or (the y-axis with heights is displayed on the left side of the model).
10. Create a *Sandy Clay* material using **Tools/Create New Material**. Use the properties as listed in Table 4.
11. Apply the created material to the lowest soil layer solid by drag and drop. (Whilst dropping the material, the **combined material** dialog will appear. Select the **Replace** option.)
12. Soil/structure interface properties were determined using the following assumptions:
    - Soil/wall friction angle $\delta = \phi'_{crit}$ (here it is assumed that the value of ϕ'_{crit} for the *Silty Clay* is 24°).
    - Soil/wall base adhesion $c_w = c_u$ (short term analysis).
    - Soil/wall base adhesion $c'_w = 0c'$ (long term analysis).

    Interface materials automatically created by the Wizard therefore have to be replaced as explained in the box above. Create the required material for the soil-wall interfaces using **Tools/Create New Material**. Use the properties as listed in Table 4.

Name	Material type	Material drainage behaviour	c' (kN/m^2)	ϕ' (kN/m^2)	c_u	γ (kN/m^3)	γ_{sat} (kN/m^3)
Sandy Clay	Mohr-Coulomb	Drained/undrained	2	24°	50	20	20
Silty Clay/Footing interface	Mohr-Coulomb	Drained/undrained	0	24°	80	0	0

Table 4: Soil/footing interface and Sandy Clay properties

13. Apply the created material to the footing boundaries by drag and drop.
14. Set the **Nodal Density** to **Coarse** for initial checking runs and click **Analysis/Solve** or .

    The returned failure mode shown in Figure 3 is clearly restrained by the model boundaries. This indicates that the achieved result will be inaccurate and that the size of the problem domain has to be increased, as specified in the next step, so that the mechanism is not restricted by the boundaries (in general this may require some trial and error).

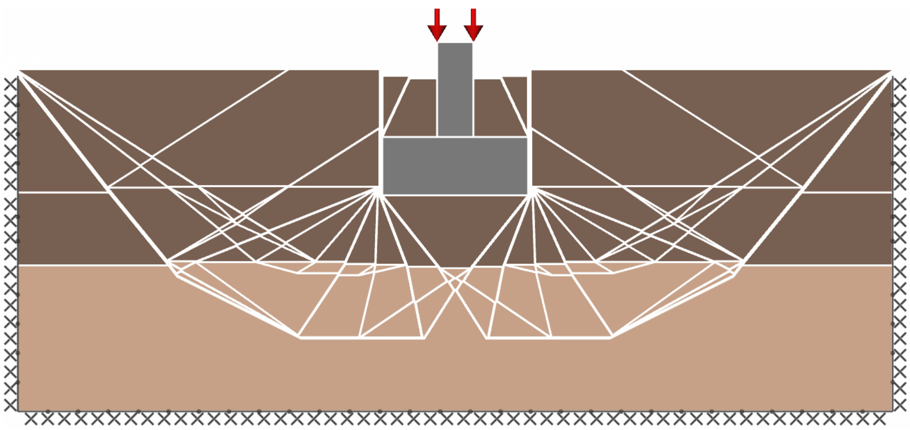

Figure 3: Main problem restrained failure mode

15. Increase the horizontal size of the problem domain. Click . Before increasing the size, change the grid settings using **Draw/Settings**. Set the extents to minimum **x(m)**= -6 and maximum **x(m)**= 6. Then, pressing CTRL (on the keyboard) select all the boundaries forming the global left boundary of the model and drag them to the $x = -5.0$ (the x-axis is displayed below the model). Repeat the same step with the right global boundary of the model moving it to the $x = 5.0$. It is recommended to use the **ORTHO** mode whilst performing this step. Be sure that the "click mode" is set up, before dragging the boundaries.
16. Set the **Nodal Density** to **Fine**.

4 Analysis

The LimitState:GEO model may be set up manually as described in Section 3. Alternatively the complete model may be loaded by opening the file footing.geo.

Click **Analysis/Solve** or [icon] to obtain the **Adequacy Factors** listed in Table 3. The critical scenario is found to be the **Long term analysis** with *DA1/2* which has an **Adequacy Factor** of 1.19 and the failure mechanism depicted in Figure 4 . A value above 1.0 means that in Eurocode 7 terms the design is safe. The additional margin of safety (or overdesign factor) of 1.19 is given in terms of the load to which the **Adequacy Factor** was assigned.

If analysis results are required in terms of an alternative factor of safety (F.O.S.) such as factor on soil strength see the LimitState:GEO manual Section 8.2.3 and/or Application Note: **Gabion Wall Analysis LS-G-AN2** for more information. Note that the Eurocode 7 DA1/1 analysis is carried out with factors on actions applied at source by LimitState:GEO which is appropriate for a standard footing problem.

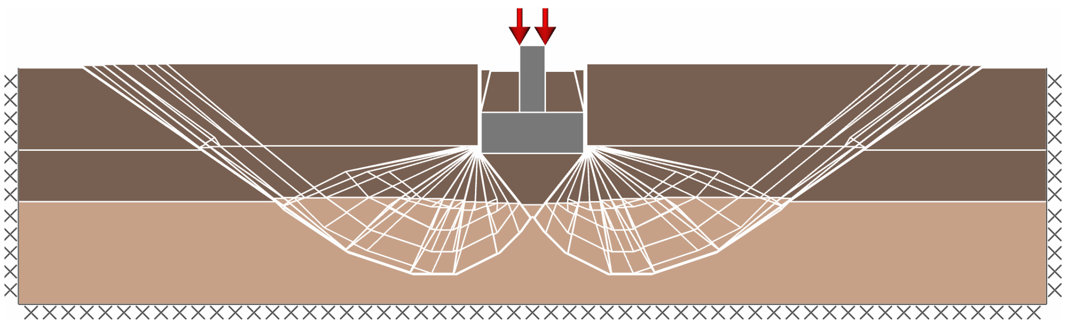

Figure 4: Main problem failure mode

5 Further analysis - trench excavation

The model is now altered by the excavation of a 1.8 m deep trench, 1.0 m to the left from the footing as depicted in Figure 5.

1. Unlock the project using [icon].
2. Draw a trench using **Draw/Solid/Polygon** or [icon] function. The co-ordinates of the diagonally opposite vertices used in the example are: x=1.5, y=2.3 and x=2.0, y=0.5 . It is recommended to use **Grid** when drawing.
3. Left click on the drawn rectangles in turn and press Delete (on the keyboard).
4. In order to allow the modelling of a possible tension crack, apply the default *No-Tension Cutoff* material (available in the **Material Explorer**) to the vertical edges of the footing and also the vertical edge of the soil blocks sitting immediately above the footing. This will give no tensile strength to those boundaries. Whilst dropping the material on the footing edges, the **combined material** dialog will appear. Select the **Add** option. Note that in this case it is assumed that the blocks of soil above the footing will essentially act as part of the foundation in the failure mode as shown in Figure 5.

Key modelling concept: *Cohesive soil/structure interface tensile strength in LimitState:GEO*

When a cohesive material is modelled using the Mohr-Coulomb failure envelope, the mathematical representation may give the material an unrealistically large tensile strength. For many problems dominated by compressive forces this is not an issue. However for e.g. slope stability or retaining wall problems, unrealistic tensile stresses may arise. To model such cases, it is possible to model a **combined material** with both cohesive and tension cutoff properties. The latter is specified using a *Cutoff* material. The property defined is the normal stress at which tensile failure occurs and may be specified in the **Property Editor**. The predefined *No-Tension Cutoff* material has a zero tensile cutoff.

5. Left click on the horizontal boundary connecting the footing base and the trench and press Delete (on the keyboard). This will allow for even nodal distribution within the failure mode domain (by default boundaries are initialised with a higher nodal density than solids).
6. In this problem it is assumed that only the short term analysis is of interest. In the **Analysis/Scenario manager ...** deselect the **Long term analysis*** options for both scenarios.
7. Set the **Nodal Density** to **Coarse** for initial checking and click **Analysis/Solve** or ▶.

   The domain of the returned failure mode is significantly smaller than the specified problem domain. In order to increase solution accuracy and computational efficiency in such case it is recommended to decrease the problem domain size so it just houses the expected failure mode. Additionally for the DA1/1 analysis it might be considered that the weight of all the soil is now unfavourable for the given failure mode.

8. Decrease the horizontal size of the problem domain. Pressing CTRL (on the keyboard) select all the boundaries forming the global left boundary of the model and drag them to $x = -1.0$ (the x-axis is displayed below the model). Repeat the same step with the right global boundary of the model moving it to $x = 2.5$. It is recommended to use the **ORTHO** mode whilst performing this step. (Alternatively the coordinates of each individual boundary can be edited using the **Geometry Editor**).
9. Set the **Loading Type** of all soil **Solids** to *Unfavourable*.
10. Set the **Nodal Density** to **Fine** and click **Analysis/Solve** or ▶.

This returns an **Adequacy Factor** of 1.035 and 0.9896 for the DA1/1 and DA1/2 scenarios respectively. Since the **Adequacy factor** for DA1/2 is below 1.0 the structure is unsafe.

The complete altered model as defined above is provided in footing_trench.geo.

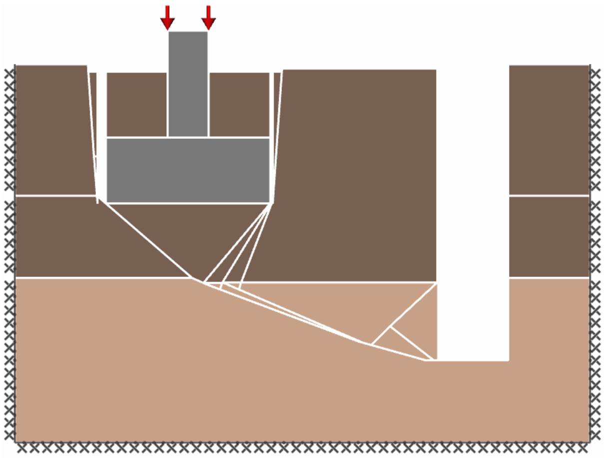

Figure 5: Failure mechanism for the problem after trench excavation

For more information: www.limitstate.com/geo