

Application Note - Reinforced Soil Analysis

1 Introduction

This note describes the typical steps followed in setting up and analysing a reinforced soil 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/LSGAN6/reinforced_soil_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.

2 Problem definition

The specified problem depicted in Figure 1 involves the construction of a reinforced soil wall to a height of 5.7m in a stratum of sandy silt. The wall is formed of compacted fill with the placement of 4m long geotextile layers at 1 m vertical intervals. The facing skin is formed of concrete panels seated on a strip foundation. The geotextile has a design tensile strength of 30 kN/m and pullout coefficient $C_{po} = 0.8$ for the fill material. A 2m wide variable surface load of 20 kN/m² is applied 1m back from the top of the wall.

The main problem geometry is available as a dxf file ([reinforced_soil.dxf](#)). The modelled soils and associated parameters are listed in Table 1.

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.

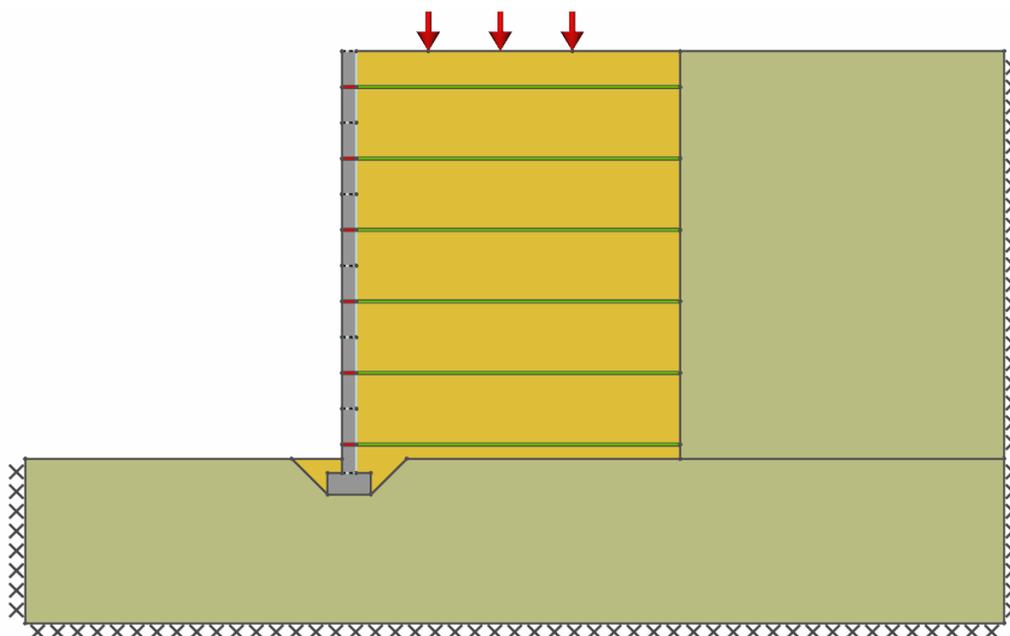


Figure 1: Problem geometry

3 Setting up problem geometry, materials and loading

Creation of the LimitState:GEO model was undertaken in the following stages:

- 1) Import the geometry from [reinforced_soil.dxf](#) using the **File/Import.../Geometry...** function. Remove any residual construction lines using **Draw/Construction Line/Clear All Construction Lines** since these will not be needed.
- 2) Define soil properties as listed in Table 1. Create materials using **Tools/Create New Material...** and apply to the appropriate **Solids** by drag and drop.

Material name	Material type	Material drainage behaviour	c_u (kN/m ²)	c' (kN/m ²)	ϕ'	γ (kN/m ³)	γ_{sat} (kN/m ³)
Compacted backfill	Mohr-Coulomb	Always drained	-	0	32°	18	21.5
Sandy silt	Mohr-Coulomb	Always drained	-	0	28°	19	21

Table 1: Soil properties

Alternatively for this example, these and other relevant pre-defined materials may be imported from [reinforced_soil_material.csv](#) using the **File/Import.../Materials...** function. Materials are named by the type of **Solid** or **Boundary** they should be applied to.

Key modelling concept: *modelling facing skin in LimitState:GEO.*

It is a common practice in design of reinforced earth structures to assume no contribution of the facing skin to the resistance. To allow for this in LimitState:GEO, the horizontal interfaces between facing units have to be defined as **combined material** consisting of **Frictionless** and **No-Tension Cutoff** material. This will allow for free relative sliding and rotation of facing blocks.

To create a **combined material** two materials can be dragged and dropped onto the same object. Whilst dropping the second one, the **combined material** dialog will appear. Select the **Add** option.

- 3) Define properties of the facing skin. Create the material with the properties as listed in Table 2 using **Tools/Create New Material...** and apply together with default materials *Frictionless* and *No-Tension Cutoff* to the appropriate **Solids** and **Boundaries** by drag and drop. The backfill/facing panel interface is to be modelled as *Frictionless*, and the horizontal interfaces between adjacent panels are to be modelled with the combined material *Frictionless* and *No-Tension Cutoff*. Note that as an alternative to drag and drop it can be quicker to select multiple **Boundaries** or **Solids** using the rectangle select tool (see the LimitState:GEO manual Section 15.7.2)
- 4) Define properties of the geotextile as listed in Table 3 using **Tools/Create New Material...** Apply created materials to the appropriate **Boundaries** by drag and drop. The *Geotextile* material is to be assigned to the reinforcement within the backfill, while the *Facing geotextile* material is to be assigned to the reinforcement within the facing panels. Connecting Engineered Elements will transmit loads to each other. Hence this is

Material name	Material type	c_u (kN/m ²)	c' (kN/m ²)	ϕ'	γ (kN/m ³)	γ_{sat} (kN/m ³)
Facing panel	Rigid	-	-	-	24	24

Table 2: Facing panel material properties

a method of connecting the *Geotextile* to the facing panel via the *Facing geotextile*. For the *Geotextile*, under **Post Solve Display** select *After Solve for Tensile force*.

Material name	Material type	Application	T_c (kN/m ²)	T_q (-)	N_c (kN/m ²)	N_q (-)	M_p (kNm/m)	R (kN/m)	C (kN/m)
Geotextile	Engineered Element (1D)	Other	0	1	0	0	1E30	30	0
Facing geotextile	Engineered Element (1D)	Other	1E30	0	1E30	0	1E30	1E30	1E30

Table 3: Geotextile material properties

Key modelling concept: *modelling soil reinforcement in LimitState:GEO*

Soil reinforcement is typically modelled using the **Engineered Element** material in LimitState:GEO. This is a material that can be applied to **Boundaries** to provide essentially one dimensional (in section) objects such as geotextiles or reinforcing strips. The special properties of an **Engineered Element** allow soil to flow through or past it if required as would be expected of e.g. reinforcing strip. Conventional analysis assumes zero resistance of reinforcement to lateral flow and this can be easily modelled, representing a good approximation to the fully flexible behaviour of geotextiles.

The parameters defining the behaviour of an Engineered element are the Pullout Factors T_c and T_q , the Lateral factors N_c and N_q , the plastic moment M_p and the tensile (rupture) and compressive strengths R and C . The theoretical background to these parameters is given in the LimitState:GEO manual Section 6.1.5.

The pullout resistance T will normally be selected based on experimental evidence, or on appropriate theory for the situation. LimitState:GEO allows T to be computed as linear function of the nominal vertical effective stress at the element as follows:

$$T = T_c + T_q \sigma'_v \tag{1}$$

To model zero lateral resistance, the parameters N_c and N_q should both be set to zero. However in some circumstances this can lead to the reinforcement 'floating' out of the soil and nominal small values for N_c and N_q should be used instead of zero.

In this note it was assumed that the pullout resistance of the geotextile within the backfill is expressed in the form of:

$$T = 2\sigma'_v \times C_{po} \times \tan \phi' \tag{2}$$

Thus the pullout parameters to be specified in LimitState:GEO are: $T_c = 0$ and $T_q = 2 \times C_{po} \times \tan 32 = 1.0$. To model a design tensile strength of 30 kN/m, the rupture strength R is set to 30 kN/m. The compressive strength is set to 0. To ensure the geotextile is fully bonded to the facing panel, the value 1E30 is used to represent an 'infinite' value for all relevant parameters.

Note that all engineered element parameters are unaffected by any partial factors specified in the **Scenario Manager**.

- 5) Specify the applied variable surface load of 20 kN/m² by selecting the relevant **Boundary** and by using **Loads - Variable - Normal** via the **Property Editor**. Set the **Loading Type** for this to *Unfavourable*.
- 6) Set the boundary conditions on the base and sides of the model to *Fixed*.
- 7) Apply the **Adequacy factor** to an appropriate surcharge load and/or body force. (Click on the load or body and set **Adequacy** to *True* in the **Property Editor**). In this example it is recommended that the **Adequacy Factor** be applied to the surcharge load.
- 8) Choose the required Partial factor set using **Scenario Manager**. In this example the problem will be investigated using Unit partial factors. Select *Unity* from the drop down box. Also tick the **Long term analysis** box to specify a drained analysis. (N.B. since all materials have been defined as *Always drained* this is not strictly necessary but is good practice).
- 9) Set the **Nodal Density** to **Fine**.

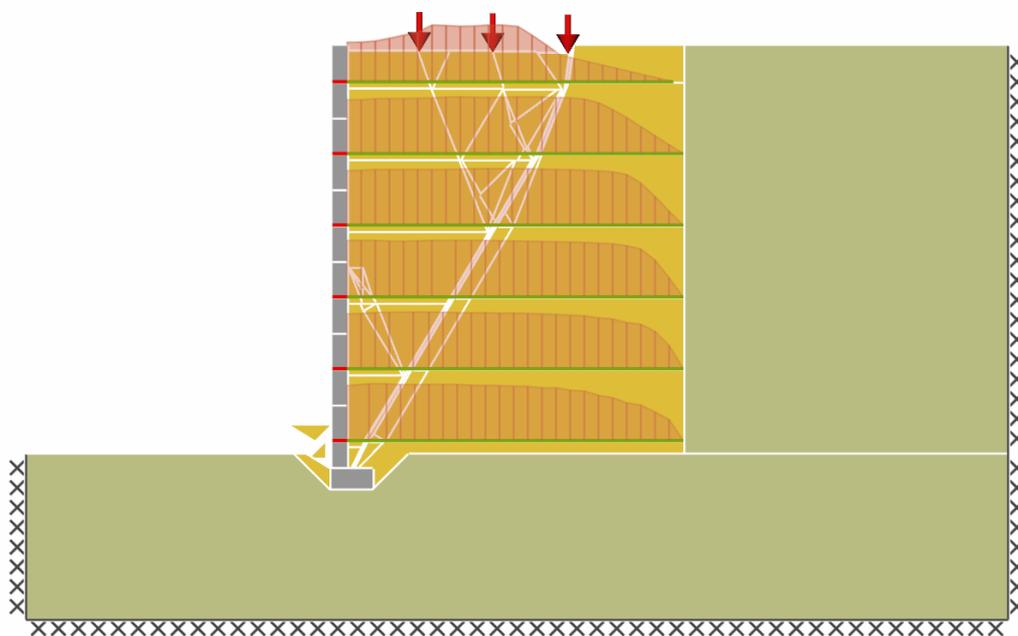


Figure 2: Failure mode for the problem (Factor Load(s) mode)

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 [reinforced_soil.geo](#).

Click **Analysis/Solve** or  to obtain an **Adequacy Factor** of 3.102. This means that the surface load must be multiplied by a factor of 3.102 to cause collapse. The identified critical failure mode (depicted in Figure 2) involves pullout failure of the top reinforcing element and tensile failure of the bottom five reinforcement elements while the wall is failing as a rigid body by sliding. The tensile force built up along the reinforcement is plotted in red and plateaus as would be expected at the value of the rupture strength. Note that in non-yielding bodies of soil the stress state is not fully defined in a limit analysis and so it is possible for the displayed tensile force to reduce towards the facing panel as seen in the top layer of reinforcement. This is just one of a range of possible stress states and does not affect the collapse analysis.

Note also that the analysis in LimitState:GEO is fully general and can handle multiple wedges as shown in this example. An **Adequacy factor** above 1.0 means that the structure is safe. The additional margin of safety (or overdesign factor) of 3.102 is given in terms of the surface 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, this can be obtained by changing the **Analysis mode** to *Factor strength(s)*. In this case a factor of 1.2 is obtained. This means that the soil strengths need to be reduced by a factor of 1.2 to cause collapse (note that this factor is *not* applied to the Engineered Element properties). The identified critical failure mode (depicted in Figure 3) involves displacement of the reinforcement with soil wedges for the top four reinforcing elements and tensile failure of the bottom two reinforcement elements while the whole wall undergoes a combined sliding and bearing failure with some rotation.

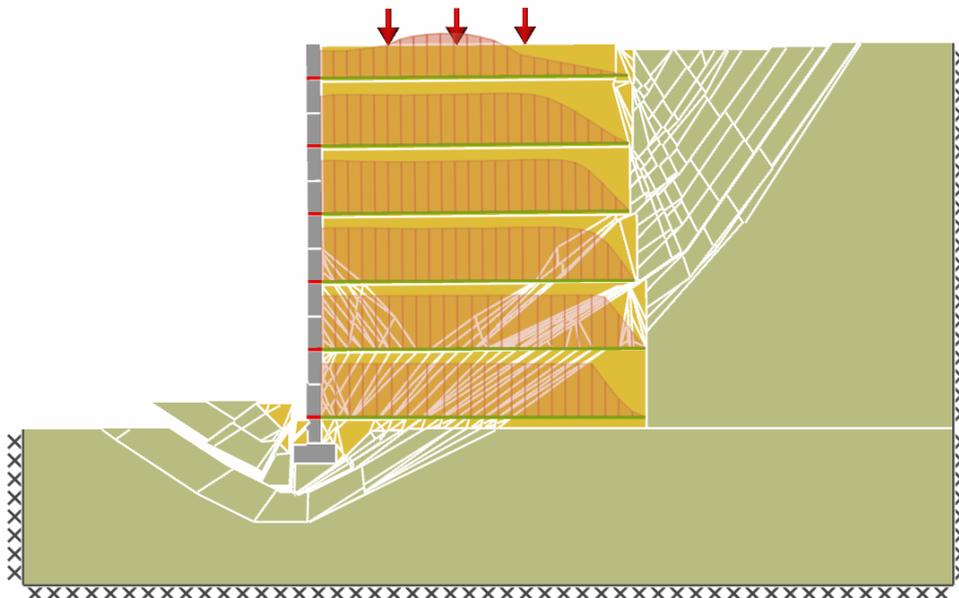


Figure 3: Failure mode for the problem (Factor Strength(s) mode)

5 Frequently Asked Questions

- 1) *Why does LimitState:GEO provide only one failure mode and an associated **Adequacy Factor** whilst in conventional analysis several different standard failure modes are checked?*

LimitState:GEO identifies the critical failure mode (which may not be a standard mode) from millions of potential mechanisms instead of checking a few pre-specified collapse modes. Thus only the critical solution is returned.

- 2) *How do I model reinforcing strips in LimitState:GEO?*

Reinforcing strips can be modelled in the same way as geotextiles. Note however that properties of the strip are to be specified per m width. Thus instead of real strip properties, the average properties per m width have to be specified.

- 3) *How accurate will the solution be?*

The accuracy of the solution depends on the resolution of the nodes at the failure mechanism. The accuracy can be refined either by increasing the **Nodal Density** or by optimizing the nodal distribution. For more information on how to improve solution accuracy see the LimitState:GEO manual Section 8.4.

For more information: www.limitstate.com/geo