



the complete stability analysis solution

Product Description

Version 3.6.1 - May 2021

LimitState

The Innovation Centre
217 Portobello
Sheffield S1 4DP
United Kingdom
+44 (0) 114 224 2240
info@limitstate.com
limitstate.com

Summary



About LimitState:GEO

LimitState:GEO identifies in seconds failure mechanisms that cannot be identified using conventional ultimate limit state analysis software and which might take a lifetime to identify manually. To achieve this, the powerful and efficient numerical analysis procedure 'Discontinuity Layout Optimization' (DLO) is used to provide an automatic means of identifying very accurate limit analysis solutions:

- An easy to use and generally applicable ultimate limit analysis tool for geotechnical stability problems.
- No need to pre-determine the problem type or likely failure mechanism. Model the problem as it is and the DLO analysis procedure establishes the critical failure mechanism, meaning that it is suitable for a wide range of geotechnical circumstances including foundations, slopes, retaining walls or mixed problem types.
- View and animate the failure mechanism on-screen. Query stresses, forces and moments on solid bodies and reinforcement elements.
- Analyse the ultimate limit state by factoring loads or material strengths – solutions are obtained with a single click, without the need to spend time manually factoring parameters etc.
- Designed for use with Eurocode 7 and other design codes of practice.

Underlying Technology

- LimitState:GEO uses the Discontinuity Layout Optimization (DLO) procedure to obtain solutions.
- DLO automatically identifies the critical configuration of sliding soil blocks at failure, which can then be visualized.
- The general formulation means that it can be applied to standard and non-standard problems alike.
- No need to independently consider different failure modes as all possible modes (anticipated or not anticipated by the engineer) are simultaneously considered, thus significantly reducing the time required to undertake a stability analysis.
- The DLO method is underpinned by rigorous theory, published in the leading peer reviewed journal, Proceedings of the Royal Society A.

Modelling Capabilities

- Wizards for quick generation of standard problem types including slopes, retaining walls and footings.
- Intuitive GUI allows user defined problems to be rapidly constructed.
- Import / export geometries from and to CAD (AutoCAD DXF files).
- Quick and simple to modify geometry, boundary conditions, materials, water pressures and loading.

- Multiple scenarios can be handled easily, with the critical case always highlighted (no need to manually analyse different potential failure modes).
- Handle uncertain ground conditions by modelling multiple soil types in a single soil body (LimitState:GEO will always pick the critical case).
- Directly apply Partial Factors in the analysis, making the software straightforward to use with design codes as such as Eurocode 7.
- LimitState:GEO ships with a number of predefined partial factor sets for common design codes / approaches; however, the user can define (and save) an unlimited number of their own partial factor sets.
- Switch between “factor of safety on load” and “factor of safety on strengths” analysis modes.

Materials

- Eurocode 7 standard material models include cohesive and frictional soils (Mohr-Coulomb). Rigid materials, materials with tension and compression cut-off and engineered elements (such as soil nails and sheet pile walls) can also be modelled.
- Soil-structure interfaces can easily be defined and modelled.
- Switch between drained and undrained analysis with a single mouse click.
- Use the built-in library of materials or define (and save) an unlimited number of custom materials.

Eurocode 7

- Built-in Eurocode 7 partial factor sets.
- Specify favourable or unfavourable loading and be alerted following analysis if this is incorrect for the critical mechanism.
- Use the inbuilt 'Scenario manager' to define and solve multiple scenarios in one go (e.g. EC7 Design Combinations 1 & 2, drained and undrained cases etc.).

Output

- The solution can be reported as a factor on load or material strength.
- Results are based on rigorous and proven limit analysis methods.
- Failure mechanisms are represented as slip-lines (a familiar concept to geotechnical engineers).
- Simplified solutions can be generated that are amenable to hand checking.
- Animated displacements provide a highly useful visual interpretation of the failure mechanism.
- Shear and normal stresses along slip-lines and forces and moments along reinforcement elements are displayed both graphically and numerically.
- The analysis report output is easily tuned to user preferences.

Interactive Viewer

Define, alter and experiment with the model:

- Quickly define or modify a model by drawing the geometry directly in the viewer.
- Examine or edit object properties using the Property and Geometry editors.
- Add to / change materials or water regimes in a zone using drag and drop functionality.
- Switch between different partial factor scenarios using the tabbed viewer window.
- Benefit from a comprehensive context sensitive help system.

Accessibility

Many features designed to help the user:

- Wizards to allow standard problem types (*e.g.* slopes, retaining walls, footings) to be rapidly analysed.
- Convenient inbuilt library of standard material types.
- Customized reports of the analysis findings can be automatically generated, printed and saved in pdf, rtf and html formats
- Integrated calculator with comprehensive unit conversion functionality.

Validation

- Underlying technology published in leading peer-reviewed science journal.
- Validated against a large, and ever-growing, number of standard geotechnical test problems ([limitstate.com/geo/verification](https://www.limitstate.com/geo/verification)).

This Appendix contains a detailed breakdown of the various features available in LimitState:GEO:

Contents

Summary	1
About LimitState:GEO	1
System Requirements	Error! Bookmark not defined.
Underlying Technology	1
Modelling Capabilities	1
Materials	2
Eurocode 7	2
Output	2
Interactive Viewer	2
Accessibility	2
Validation	3
User Interface	6
Property Editor	7
Project	8
Project (contd...)	9
Materials	9
Materials (contd...)	10
Water Regimes	10
Vertices	11
Boundaries	11
Solids	12
Nodes	12
Slip-lines	12
Water Pressure Table	13
Blocks	13
Geometry Editor	14
Scenario Manager	14
Calculator	15
Units	15
Functions	15
Explorers	16
Vertex, boundary and solid explorers	16
Material explorer	16
Water regime explorer	17
Viewer Pane	19
Wizards	20
Sample Wizard – Simple Footing Project	21
Project Details	21

Geometry.....	21
Materials.....	22
Soil Layer:.....	22
Footing Layer:	23
Loads.....	23
Scenarios.....	24
Analysis	24
Solve	25
Analysis Details	25
Analysis Engine	25
Results	25
Report Output.....	26
Report Contents.....	27
Menus.....	29
File	29
Edit.....	29
Select	30
View	30
Draw	31
Tools	32
Analysis	32
Analysis (contd...).....	33
Help	33
Dialogs	34
Selection Filter	34
Draw Settings.....	34
Create New Material.....	35
Create New Material (contd...)	36
Create New Water Regime	36
Project Details.....	36
Preferences.....	37
Preferences (contd...).....	38
Scenario Manager	38
Analysis Options	39
Report.....	39
Toolbars.....	40
File	40
Edit.....	40
Cursor	40
Draw	40
Draw (contd...)	41
Analysis	41

Animation	41
Zoom	41
Show	41
Show (contd...)	42
Select	42
Rotate 3D	42
View 3D	42
View 3D (contd...)	43
Help	43
Context Menus	44
Toolbars and Property Editor	44
Viewer Pane (General)	45
Viewer Pane (General) (contd...)	45
Viewer Pane (Post-Solve Solid)	46
Viewer Pane (Post-Solve Engineered Element)	46
Material Explorer	46
Other Explorers	46
Geometry Editor	47
Output Pane	47

User Interface

Figure 1 illustrates the main features of the LimitState:GEO user interface:

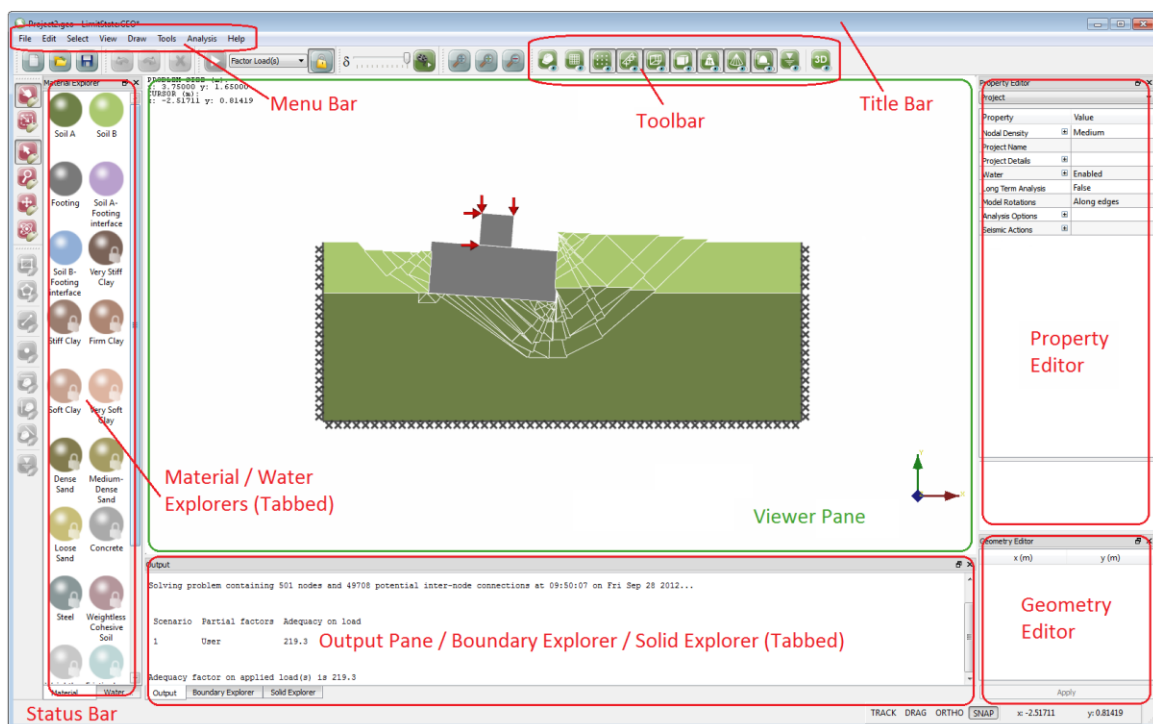
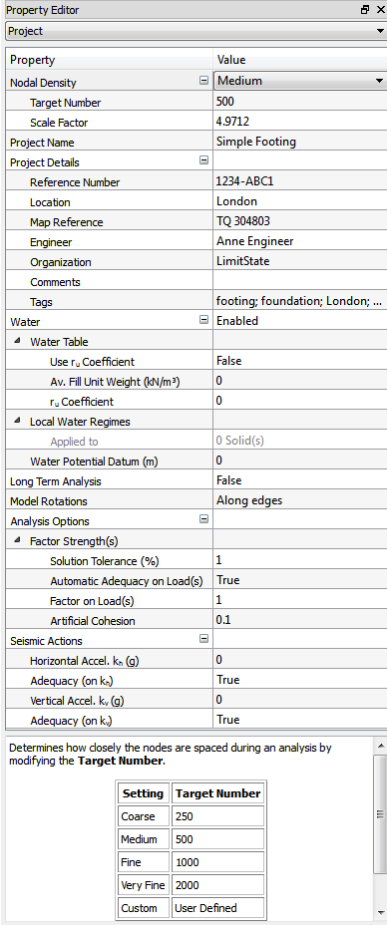


Figure 1 - The LimitState:GEO User Interface

Property Editor

The Property Editor allows the user to quickly query and / or modify the attributes of one or more objects within the current project.

- Select objects on screen and modify their properties to better reflect the real-life circumstances of the model.
- Change a property for a single object or modify the common properties of multiple objects en masse.
- Access some of the more specialized functions.
- Changes made in the property editor can be undone / redone using 'undo' and 'redo'.



The screenshot shows the 'Property Editor' window for a 'Project'. It contains a table of properties and values, with expandable sections for 'Project Details', 'Water', and 'Analysis Options'.

Property	Value
Nodal Density	Medium
Target Number	500
Scale Factor	4.9712
Project Name	Simple Footing
Project Details	
Reference Number	1234-ABC1
Location	London
Map Reference	TQ 304803
Engineer	Anne Engineer
Organization	LimitState
Comments	
Tags	footing; foundation; London; ...
Water	Enabled
Water Table	
Use r_u Coefficient	False
Av. Fill Unit Weight (kN/m^3)	0
r_u Coefficient	0
Local Water Regimes	
Applied to	0 Solid(s)
Water Potential Datum (m)	0
Long Term Analysis	False
Model Rotations	Along edges
Analysis Options	
Factor Strength(s)	
Solution Tolerance (%)	1
Automatic Adequacy on Load(s)	True
Factor on Load(s)	1
Artificial Cohesion	0.1
Seismic Actions	
Horizontal Accel. k_a (g)	0
Adequacy (on k_a)	True
Vertical Accel. k_v (g)	0
Adequacy (on k_v)	True

Determines how closely the nodes are spaced during an analysis by modifying the **Target Number**.

Setting	Target Number
Coarse	250
Medium	500
Fine	1000
Very Fine	2000
Custom	User Defined

Figure 2 - The LimitState:GEO Property Editor

Using the property editor it is possible to examine and / or modify the general properties of the **Project** and each of the **Materials** and **Water Regimes**. In addition, the following objects may be examined and / or modified:

- **Vertices** – points where two or more Boundary objects meet. A vertex may lie on a straight line. A Vertex is shared by the Boundary objects that meet at that vertex.
- **Boundaries** – straight lines that link two Vertices and defines the edge or boundary of a Solid. If a Boundary object separates two Solids then it is shared by those solids and forms an interface.
- **Solids** – 2D polygons defining a body of soil or other material. Their extents are defined by the surrounding Boundary objects.
- **Nodes** – potential intersection-points of slip-lines.
- **Slip-lines** – lines coinciding with the direction of failure for regions of material stressed to the limiting or yield condition
- **Water Pressure Table** – a phreatic surface denoted by a line, or series of connected lines, below which there is an assumption of hydrostatic water pressure.

The following tables describe, in detail, the properties found in the Property Editor for each of the described features:

Project

Property	Description	Editable?
Nodal Density	Determines how closely nodes are spaced during an analysis by modifying the Target Number . Values can be coarse (250), medium (500), fine (1000), very fine (2000) or custom defined by the user.	✓
Target Number	The total number of nodes that the software will attempt to use when solving the problem. Altering this value will cause the Scale Factor on the Baseline Nodal Spacing to change, thus altering the Actual Nodal Spacing : Actual Nodal Spacing = Baseline Nodal Spacing / Scale Factor Note: The Actual Nodal Spacing is calculated by the software to determine the spacing required to achieve the Target Number of nodes and cannot be directly modified by the user.	✓
Scale Factor	A global factor used to scale the spacing between nodes in the project to achieve (or alter) the Target Number of nodes). e.g. doubling the Scale Factor causes the number of nodes used along boundaries to double and the number of nodes used in solids to quadruple.	✓
Project Name	A title for the project.	✓
Project Details	General information about the project.	
Reference Number	A reference number.	✓
Location	The physical location of the project.	✓
Map Reference	A map reference number for the project location.	✓
Engineer	Name of the engineer responsible.	✓
Organization	Name of the organization responsible.	✓
Comments	Any additional comments.	✓
Tags	Add tags to the file which can be used when searching outside of LimitState:GEO	✓
Water	Indicates whether water is taken into account during analysis.	✓
Water Table	Define properties for the water table (or r_u reference surface), if defined.	
Use r_u Coefficient	Use an r_u coefficient in pore pressure calculations rather than hydrostatic pressures.	✓
Av. Fill Unit Weight	Average unit weight (weight density) of fill to be used in r_u calculations.	✓
r_u Coefficient	Value of r_u coefficient. Water pressure = $r_u \times$ Average Fill Unit Weight \times Depth Below r_u Reference Surface The r_u Reference Surface is defined by the water table and generally this would be drawn coincident with the soil surface.	✓
Local Water Regime	Display where local water regimes are applied.	
Water Potential Datum	Level (y-coordinate) of water potential datum. This is used as the reference datum for water regime models involving water potentials. Default is 0.0	✓
Long Term Analysis	Determine whether the problem is modeled as taking place over a long or short period of time.	✓
Model Rotations	Enable or disable the modelling of rotational failure mechanisms. Can be set to 'False' or 'Along edges'	✓
Analysis Options	Set general analysis options.	
Factor Strength(s)	Set parameters that control how the factor on strength(s) is computed.	
Solution Tolerance	Relative percentage tolerance on factor of strength solution when in "Factor Strength(s)" solution mode (default value: 1.0). e.g. set the value to 1.0 to terminate the analysis when the solution is bracketed such that the upper solution is within 1% of the lower solution.	✓
Automatic Adequacy on Load(s)	ADVANCED FEATURE (Default value: True). When set to true, the software will use the automatic global "adequacy on load" settings to search for a factor of strength. When set to false, the software will use the existing user-defined "adequacy on load" settings.	✓
Factor on Load(s)	ADVANCED FEATURE (Default value: 1.0). This is the target "adequacy on loads" value that is used when searching for the "adequacy on strength" when in Factor Strength(s) mode. Normally this parameter would be set at 1.0 so that the factored strength that requires no change to loading is determined.	✓

Project (contd...)

Property	Description	Editable?
Artificial Cohesion	ADVANCED FEATURE (Default value: 0.1) Adds a (generally small) cohesion to purely frictional materials when undertaking a Factor Strength(s) analysis. This has the effect of improving convergence towards a solution and ensuring that a failure mechanism is identified.	✓
Seismic Actions	Set horizontal and vertical ground acceleration values (units: g).	✓
Horizontal Accel. k_h	Specify a positive value for a left-to-right base acceleration (i.e. likely to lead to right-to-left soil movements).	✓
Adequacy (on k_h)	State whether or not the adequacy factor is to be applied to the specified horizontal base acceleration.	✓
Vertical Accel. k_v	Specify a positive value for upward base acceleration (i.e. likely to lead to downward soil movements).	✓
Adequacy (on k_v)	State whether or not the adequacy factor is to be applied to the specified vertical base acceleration.	✓

Materials

Property	Description	Editable?												
Object ID	A unique identifier given to all objects in the project.	✓												
Type	<div>The underlying material type, can be:</div> <table><thead><tr><th>Type</th><th>Description</th></tr></thead><tbody><tr><td>Mohr-Coulomb</td><td>The yield surface is defined in terms of effective stress parameters c' and ϕ', or undrained parameter c_u where c' or c_u is the intercept on the shear stress axis and ϕ' the angle of shearing resistance.</td></tr><tr><td>Mohr-Coulomb (Derived)</td><td>A Mohr-Coulomb material derived from one already existing in the materials library. Adopts the same material properties as the parent but has multipliers on the cohesion and $\tan \phi'$.</td></tr><tr><td>Cutoff</td><td>The yield surface is defined in terms of a limiting tensile stress σ_t and a limiting compressive stress σ_c.</td></tr><tr><td>Rigid</td><td>This simple material type is fully rigid. No slip-line can pass through it at any load.</td></tr><tr><td>Engineered Element (1D)</td><td>This material type may be used to represent 1D engineered materials such as soil reinforcement and sheet piles. It is defined in terms of resistance factors to pullout (T) and lateral displacement (N), Rupture Strength (R) and plastic moment (M_p).</td></tr></tbody></table>	Type	Description	Mohr-Coulomb	The yield surface is defined in terms of effective stress parameters c' and ϕ' , or undrained parameter c_u where c' or c_u is the intercept on the shear stress axis and ϕ' the angle of shearing resistance.	Mohr-Coulomb (Derived)	A Mohr-Coulomb material derived from one already existing in the materials library. Adopts the same material properties as the parent but has multipliers on the cohesion and $\tan \phi'$.	Cutoff	The yield surface is defined in terms of a limiting tensile stress σ_t and a limiting compressive stress σ_c .	Rigid	This simple material type is fully rigid. No slip-line can pass through it at any load.	Engineered Element (1D)	This material type may be used to represent 1D engineered materials such as soil reinforcement and sheet piles. It is defined in terms of resistance factors to pullout (T) and lateral displacement (N), Rupture Strength (R) and plastic moment (M_p).	✓
Type	Description													
Mohr-Coulomb	The yield surface is defined in terms of effective stress parameters c' and ϕ' , or undrained parameter c_u where c' or c_u is the intercept on the shear stress axis and ϕ' the angle of shearing resistance.													
Mohr-Coulomb (Derived)	A Mohr-Coulomb material derived from one already existing in the materials library. Adopts the same material properties as the parent but has multipliers on the cohesion and $\tan \phi'$.													
Cutoff	The yield surface is defined in terms of a limiting tensile stress σ_t and a limiting compressive stress σ_c .													
Rigid	This simple material type is fully rigid. No slip-line can pass through it at any load.													
Engineered Element (1D)	This material type may be used to represent 1D engineered materials such as soil reinforcement and sheet piles. It is defined in terms of resistance factors to pullout (T) and lateral displacement (N), Rupture Strength (R) and plastic moment (M_p).													
Color	The color of a material as shown in the viewer.	✓												
Drained cohesion intercept (c')	Drained cohesion intercept.	✓												
Drained Angle of Shearing Resistance (ϕ')	Drained angle of shearing resistance (degrees).	✓												
Undrained shear Strength (c_u)	Undrained shear strength.	✓												
Dry Unit Weight	Unit weight (weight density) of material located above the water table. In general this is synonymous with dry unit weight. However if for example a saturated clay lies above the water table, then its saturated unit weight should be entered here.	✓												
Sat. Unit Weight	Unit weight of material located below the water table. In general this is synonymous with saturated unit weight. However if for example a saturated clay lies above the water table, then its saturated unit weight should be entered under 'Dry' Unit Weight.	✓												
Drainage Behaviour	Determines the behaviour of the material in long and short term analysis modes. Can be 'Drained', 'Undrained' or 'Drained / undrained'. If the behaviour is set to 'Drained / undrained', then drained properties will be used in long term analyses and undrained properties used in short term analyses.	✓												
Limiting Stress - Comp.	CUTOFF ONLY - Limiting compressive stress.	✓												
Limiting Stress - Tens.	CUTOFF ONLY - Limiting tensile stress.	✓												
Multiplier on c' , c_u	DERIVED ONLY - A multiplier on the cohesion values of a parent material that is used to determine the properties of the derived material.	✓												
Multiplier on $\tan \phi'$	DERIVED ONLY - A multiplier on the angle of shearing resistance of a parent material that is used to determine the properties of the derived .	✓												
Pullout Factor (T_c)	ENGINEERED ELEMENT ONLY - Pullout factor, T_c used to compute the pullout resistance ($T = T_c + T_q \sigma'_v$) of the element.	✓												
Pullout Factor (T_q)	ENGINEERED ELEMENT ONLY - Pullout factor, T_q used to compute the pullout resistance ($T = T_c + T_q \sigma'_v$) of the element .	✓												

Materials (contd...)

Property	Description	Editable?
Lateral Factor (N_c)	ENGINEERED ELEMENT ONLY - Lateral Factor, N_c used to compute the resistance (N) of the element to lateral displacement $N = N_c + N_q \sigma'_v$	✓
Lateral Factor (N_q)	ENGINEERED ELEMENT ONLY - Lateral Factor, N_q used to compute the resistance (N) of the element to lateral displacement $N = N_c + N_q \sigma'_v$	✓
Plastic Moment (M_p)	ENGINEERED ELEMENT ONLY - Plastic moment of resistance M_p at end vertices. Note that between vertices the element is assumed to be rigid	✓
Rupture Strength (R)	ENGINEERED ELEMENT ONLY - Maximum tensile force that can be carried (kN, per m width). If a notional infinitely strong material is being modelled then this parameter should be set to 1E+30	✓
Compression Strength (C)	ENGINEERED ELEMENT ONLY - Maximum compression force that can be carried (kN, per m width). If a notional infinitely strong material is being modelled then this parameter should be set to 1E+30	✓
Advanced		
Subdivide at Nodes?	ENGINEERED ELEMENT ONLY - Specify whether or not to automatically subdivide the Engineered Element at intermediate nodes. This allows failure or articulation at intermediate positions between end vertices (e.g. tensile failure and/or plastic hinge formation). This also ensures any displayed diagrams show values at intermediate positions. Note: this function is provided primarily for compatibility with LimitState:GEO 1.0 & 2.0 files (parameter is 'False' in these files).	✓
Datum c_u elevation	MOHR-COULOMB ONLY - The elevation (y coordinate) at which c_u is equal to its datum value	✓
c_u Gradient	MOHR-COULOMB ONLY - The change in c_u value per unit distance increase in depth below the datum	✓
Override Strength	MOHR-COULOMB ONLY - Override the strength properties assigned for use in this zone using a spatially varying grid function multiplier. Click in the Value field to show the Change button. Click this to define the override grid parameters using the Grid Interpolation dialog. In override strength mode, the cohesion parameters c' and c_u are multiplied by the spatially varying factor defined in the grid. The drained angle of shearing resistance is unaffected. This setting may be reset to False using the 'Override strength?' checkbox in the Grid Interpolation dialog.	✓
Post-Solve Display	Options for setting the diagrams to display automatically and when clicking on an object, once an analysis has been conducted. Mohr-Coulomb / derived / rigid material: Normal stress, shear stress, none Cutoff material: Normal stress, shear stress, none Engineered element material: Tensile force, shear force, moment, none	✓

Water Regimes

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Type	The underlying type of the water regime used. Can be 'Constant Potential', 'Constant Pressure', 'Interpolated Grid' or 'Aquiclude (Dry)'.	
Color	The color of a water regime as shown in the viewer.	✓
Water Potential	CONSTANT POTENTIAL ONLY - Potential of water relative to Datum. Datum is defined in the Project level Property Editor.	✓
Factor on Unit Weight of Water	CONSTANT POTENTIAL ONLY - Factor on unit weight of water. This can be used to model different fluids or simply model for example vertical flow of water.	✓
Constant Pore Pressure	CONSTANT PORE PRESSURE ONLY - Constant pore pressure to be applied to a zone. This can be used to model e.g. dry soil (pressure = 0.0) or a fixed soil suction (-ve pressure).	✓
Function	INTERPOLATED GRID ONLY - Type of spatial function used to determine pore pressure at any point. Provides access to the interpolated grid data.	✓

Vertices

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
x	The x coordinate of a vertex.	Use Geometry Editor
y	The y coordinate of a vertex.	Use Geometry Editor

Boundaries

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Baseline Nodal Spacing	The baseline spacing (m) between nodes. Used when calculating the actual spacing between nodes to achieve the Target Number: Actual Nodal Spacing = Baseline Nodal Spacing / Scale Factor.	✓
Loading	Define the applied loads and loading type.	
Loads	Individual loads applied to a 'Free' boundary.	
Permanent	Permanent actions, as defined by Eurocode 1.	
Shear	Permanent shear stress applied to a boundary.	✓
Normal	Permanent normal stress applied to a boundary.	✓
Adequacy	Set this to 'true' to include this permanent load during the computation of the global problem 'adequacy factor'.	✓
Accidental	Accidental actions, as defined by Eurocode 1.	
Shear	Accidental shear stress applied to a boundary.	✓
Normal	Accidental normal stress applied to a boundary.	✓
Adequacy	Set this to 'true' to include this accidental load during the computation of the global problem 'adequacy factor'.	✓
Variable	Variable actions, as defined by Eurocode 1.	
Shear	Variable shear stress applied to a boundary.	✓
Normal	Variable normal stress applied to a boundary.	✓
Adequacy	Set this to 'true' to include this variable load during the computation of the global problem 'adequacy factor'.	✓
Loading Type	Defines whether the load on a boundary acts to make the adequacy factor lower (favourable) or higher (unfavourable).	✓
Materials	A list of the materials assigned for use in a boundary. Where more than one material is assigned, the material giving the lowest factor of safety will be used in the solution. Selecting this field brings up a button that opens a dialog allowing materials to be assigned / removed from a zone.	✓
Top / Right	A list of the materials assigned for use on the top / right edge of this zone. Where more than one material is assigned, the material giving the lowest factor of safety will be used in the solution.	✓
Engineered Element	The engineered material assigned for use in this zone.	✓
Bottom / Left	A list of the materials assigned for use on the bottom / left edge of this zone. Where more than one material is assigned, the material giving the lowest factor of safety will be used in the solution.	✓
Support Type	Defines the type of support given to this boundary. Can be fixed, free or symmetry.	✓

Solids

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Baseline Nodal Spacing	The baseline spacing (m) between nodes in a solid. Used when calculating the actual spacing between nodes to achieve the Target Number.	
x spacing	Baseline Nodal Spacing in the x direction.	✓
y spacing	Baseline Nodal Spacing in the y direction.	✓
Self-Weight Loading	Define how the self-weight of a solid is treated during analysis.	✓
Loading Type	Defines whether the self-weight of a solid acts to make the adequacy factor lower (favourable) or higher (unfavourable).	✓
Adequacy	Set this to 'true' to include the self-weight of this solid during the computation of the global problem 'adequacy factor'.	✓
Materials	A list of the materials assigned for use in a zone. Where more than one material is assigned, the material giving the lowest factor of safety will be used in the solution.	✓
Water Regime	Details of the water regime present in the zone.	✓

Nodes

Property	Description	Editable?
x	The x coordinate of a node.	
y	The y coordinate of a node.	

Slip-lines

Property	Description	Editable?
ID	A unique identifier given to all objects in the project.	
Start Node	The node at the start of a slip-line.	
x	The x coordinate of the point at the start of a slip-line.	
y	The y coordinate of the point at the start of a slip-line.	
End Node	The node at the end of a slip-line.	
x	The x coordinate of the point at the end of a slip-line.	
y	The y coordinate of the point at the end of a slip-line.	
Length	The Straight-line distance between 'Start Node' and 'End Node'.	
Normal Displacement	The instantaneous relative normal displacement across a slip-line.	
Shear Displacement	The instantaneous relative shear displacement across a slip-line.	
Normal Force Per Unit Width	The total normal force acting along a slip-line.	
Shear Force Per Unit Width	The total shear force acting on a slip-line.	
Normal Stress	The total shear stress acting along the slip-line.	
Shear Stress	The total shear stress acting along the slip-line.	

Water Pressure Table

Property	Description	Editable?
ID	A unique identifier given to all objects in the project	✓

Blocks

Blocks can be selected once the problem is in the solved and displaced state.

Property	Description	Editable?
ID	A unique identifier given to all objects in the project	✓
Point X		
ID	The identifier of the point	✓
x	The x coordinate of the point.	
y	The y coordinate of the point.	

Geometry Editor

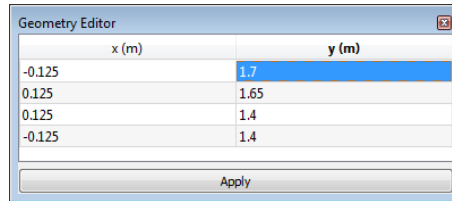


Figure 3 - The LimitState:GEO Geometry editor

- Examine and / or modify the x and y coordinates of a vertex, boundary, solid or water table
- Make rapid alterations to the problem geometry

Scenario Manager

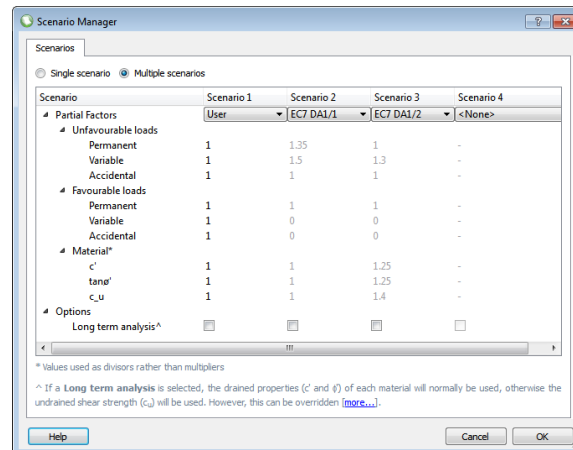


Figure 4 - The LimitState:GEO Scenario Manager

Define partial factor sets to be considered during analysis. If multiple scenarios are defined, the critical case will be chosen automatically:

- Specify whether a single or multiple scenario analysis is carried out.
- Editable partial factors:
 - Unfavourable loads (permanent, accidental and variable).
 - Favourable loads (permanent, accidental and variable).
 - Materials (c' , $\tan\phi'$, c_u).
- Default available partial factor sets:
 - User.
 - Unity.
 - EC7 DA1/1 (Eurocode 7 Design Approach 1, Combination 1).
 - EC7 DA1/2 (Eurocode 7 Design Approach 1, Combination 2).
- Import partial factor sets from .csv file.
- Export partial factor sets to .csv file.
- Define new partial factor sets.
- Delete a partial factor set from a multiple scenario problem.
- Use the 'manage' dialog to rename or delete individual partial factor sets from the file.
- Specify whether an long term analysis is to be carried out.

Calculator

- Quickly calculate values for entry into any field that requires numerical input (simply click in the field and select the calculator button).
- Convert a wide variety of types into the default LimitState:GEO units:

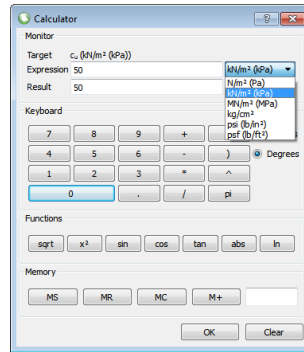


Figure 5 - The LimitState:GEO Calculator

Units

Unit	Metric	Imperial	Converter Options
Force / Weight	kN	lbf	N, kN, MN, lbf
Unit weight	kN/m ³	pcf (lbf/ft ³)	kN/m ³ , MN/m ³ , lbf/ft ³ , lbf/yd ³
Length	m	ft	mm, cm, m, ft, yd
Area	m ²	ft ²	mm ² , cm ² , m ² , ft ² , yd ²
Volume	m ³	ft ³	mm ³ , cm ³ , m ³ , ft ³ , yd ³
Cohesive strength / Stress	kN/m ² (kPa)	psf (lbf/ft ²)	N/m ² (Pa), kN/m ² (kPa), MN/m ² (MPa), kg/cm ² , psi (lbf/in ²), psf (lbf/ft ²)
Angle of friction	Degrees	Degrees	Radians, Degrees
Water pressure	kN/m ² (kPa)	psf (lbf/ft ²)	N/m ² (Pa), kN/m ² (kPa), MN/m ² (MPa), kg/cm ² , psi (lbf/in ²), psf (lbf/ft ²)

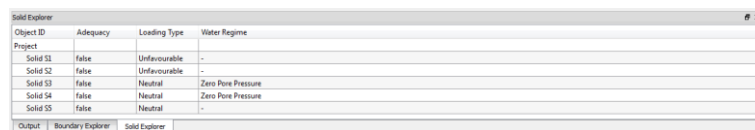
Functions

- +, -, *, / - add, subtract, multiply, divide
- **sqrt** - square root
- **x²** - square
- **sin**, **cos**, **tan** - trigonometric functions
- **abs** - absolute
- **ln** - natural logarithm
- **(x)** - parenthesis
- **^** - exponent
- **pi** - 3.14159
- **MS** – store in memory
- **MR** – recall memory
- **MC** – clear memory
- **M+** - add to number in memory

Explorers

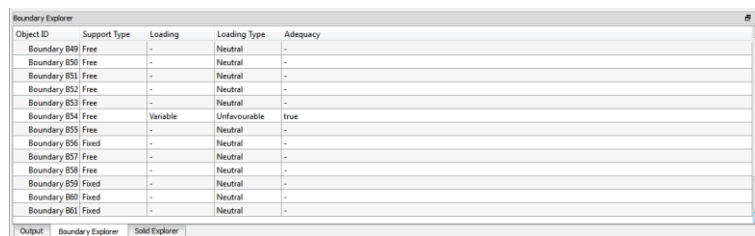
Vertex, boundary and solid explorers

- When the project is unlocked, objects selected in the **Vertex**, **Boundary** and **Solid** explorers are highlighted in the viewer pane (and *vice versa*).
- The **Vertex Explorer** displays the Object ID.
- The **Boundary Explorer** displays the Object ID, Support Type, Loading Type, Loading and Adequacy.
- The **Solid Explorer** displays the Object ID, Adequacy, Loading Type and Water Regime.



Object ID	Adequacy	Loading Type	Water Regime
Project			
Solid S1	false	Unfavourable	-
Solid S2	false	Unfavourable	-
Solid S3	false	Neutral	Zero Pore Pressure
Solid S4	false	Neutral	Zero Pore Pressure
Solid S5	false	Neutral	-

Figure 6 - The LimitState:GEO Solid Explorer



Object ID	Support Type	Loading	Loading Type	Adequacy
Boundary B88	Free	-	Neutral	-
Boundary B90	Free	-	Neutral	-
Boundary B91	Free	-	Neutral	-
Boundary B92	Free	-	Neutral	-
Boundary B93	Free	-	Neutral	-
Boundary B94	Free	Variable	Unfavourable	true
Boundary B95	Free	-	Neutral	-
Boundary B96	Fixed	-	Neutral	-
Boundary B97	Free	-	Neutral	-
Boundary B98	Free	-	Neutral	-
Boundary B99	Fixed	-	Neutral	-
Boundary B60	Fixed	-	Neutral	-
Boundary B61	Fixed	-	Neutral	-

Figure 7 - The LimitState:GEO Boundary Explorer

Material explorer

- Drag and drop from the **Material Explorer** to assign materials to boundaries and solids.
- Create user-defined materials, with full control over properties for each type (see the Dialogs section).
- Import pre-defined materials from file.
- Export user-defined materials to file.
- Assign multiple material types to a boundary or solid – the most critical one will always be used.



Figure 8 - The LimitState:GEO Material Explorer

The Material Explorer contains a library of built-in materials of different types (metric defaults shown):

Name	Type	c'	ϕ'	c_u	Dry Unit Weight (kN/m ³)	Saturated Unit Weight (kN/m ³)	Limiting Stress – Comp. (kN/m ²)	Limiting Stress – Tens. (kN/m ²)	Pullout Factor T_c (kN/m ²)	Pullout Factor T_q (kN/m ²)	Lateral Factor N_c (kN/m ²)	Lateral Factor N_q (kN/m ²)	M_p (kNm)	Rupture Strength (kN/m)	Subdivide at nodes?	Drainage Behaviour	Post-solve Display
Very Stiff Clay	Mohr-Coulomb	5	25	150	22	22	-	-	-	-	-	-	-	-	-	Drained/ Undrained	Click: Shear Solve: N/A
Stiff Clay	Mohr-Coulomb	4	22	120	21	21	-	-	-	-	-	-	-	-	-	Drained/ Undrained	Click: Shear Solve: N/A
Firm Clay	Mohr-Coulomb	3	20	60	20	20	-	-	-	-	-	-	-	-	-	Drained/ Undrained	Click: Shear Solve: N/A
Soft Clay	Mohr-Coulomb	2	18	30	19	19	-	-	-	-	-	-	-	-	-	Drained/ Undrained	Click: Shear Solve: N/A
Very Soft Clay	Mohr-Coulomb	1	16	10	18	18	-	-	-	-	-	-	-	-	-	Drained/ Undrained	Click: Shear Solve: N/A
Dense Sand	Mohr-Coulomb	0	45	0	18	21	-	-	-	-	-	-	-	-	-	Always drained	Click: Shear Solve: N/A
Medium Dense Sand	Mohr-Coulomb	0	37.5	0	16	20	-	-	-	-	-	-	-	-	-	Always drained	Click: Shear Solve: N/A
Loose Sand	Mohr-Coulomb	0	30	0	14	19	-	-	-	-	-	-	-	-	-	Always drained	Click: Shear Solve: N/A
Concrete	Mohr-Coulomb	0	0	10,000	23	23	-	-	-	-	-	-	-	-	-	Always undrained	Click: Shear Solve: N/A
Steel	Mohr-Coulomb	0	0	125,000	78	78	-	-	-	-	-	-	-	-	-	Always undrained	Click: Shear Solve: N/A
Weightless Cohesive Soil	Mohr-Coulomb	0	0	1	0	0	-	-	-	-	-	-	-	-	-	Always undrained	Click: Shear Solve: N/A
Weightless Concrete	Mohr-Coulomb	10,000	0	10,000	0	0	-	-	-	-	-	-	-	-	-	Always undrained	Click: Shear Solve: N/A
No-Tension Cutoff	Cutoff	-	-	-	0	0	1E+30	0	-	-	-	-	-	-	-	-	Click: Shear Solve: N/A
Rigid	Rigid	-	-	-	0	0	-	-	-	-	-	-	-	-	-	-	Click: Normal Solve: N/A
Soil Nail (Rigid)	Engineered Element	-	-	-	-	-	-	-	50	0	5	0	1E+30	1E+30	y	-	Click: N/A Solve: Tension
Sheet Pile Wall (Rigid)	Engineered Element	-	-	-	-	-	-	-	1E+30	0	1E+30	0	1E+30	1E+30	y	-	Click: N/A Solve: Moment

Water regime explorer

- Drag and drop from the **Water Regime Explorer** to assign water pressure schemes to solids.
- Create user-defined regimes, with full control over properties for each type.
- Import pre-defined regimes from file.
- Export user-defined regimes to file.

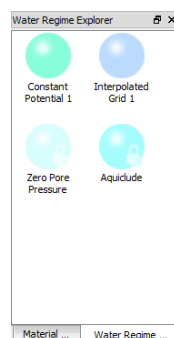


Figure 9 - The LimitState:GEO Material Explorer

The Water Regime Explorer contains two built-in regimes:

- **Zero Pore Pressure** – Constant pore pressure = 0
- **Aquiclude** - Water pressures in the solid = 0 (intended to be used for impermeable materials that are internally unaffected by water pressure).

Viewer Pane

- A fully interactive graphical modelling environment.
- Total freedom to define or modify problem geometries.
- View the model from any angle (including in 3D) and zoom into areas of interest.
- Select any part of the model with the mouse to view or modify its properties.

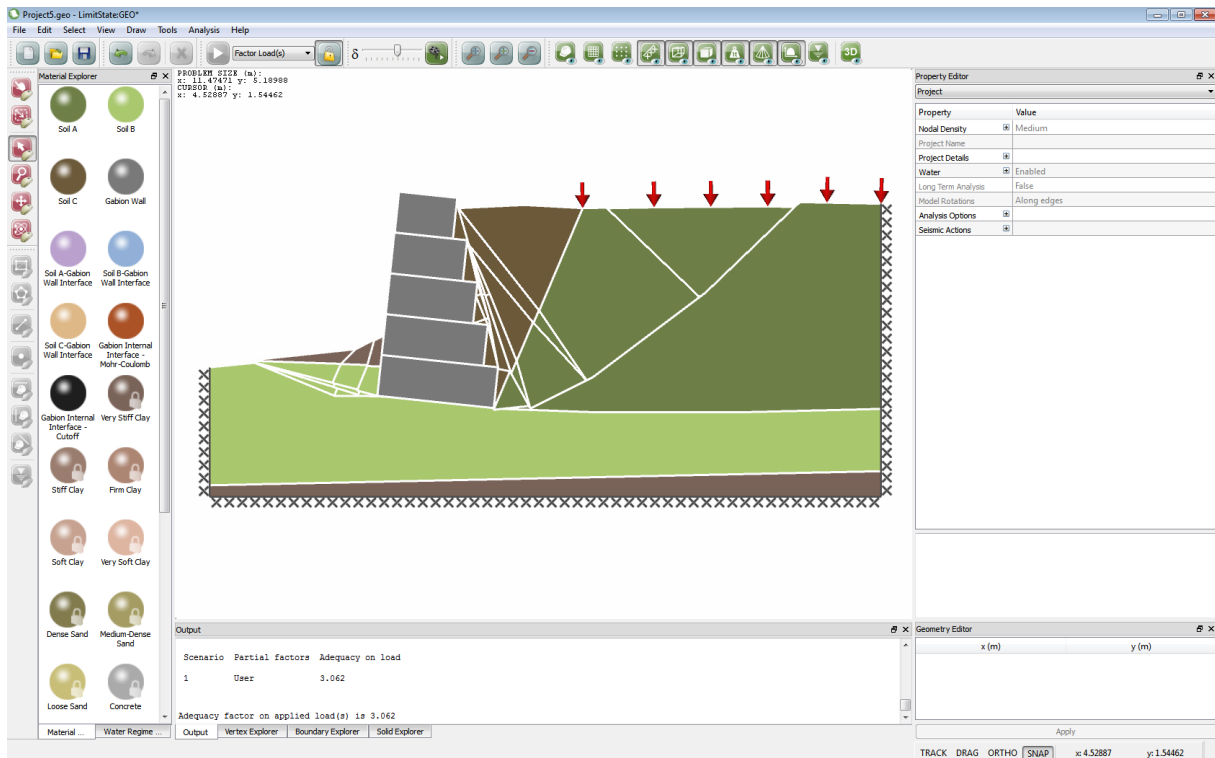


Figure 10 - The LimitState:GEO Modelling Environment

Wizards

- Rapidly define a standard problem type using the built in wizards, which guide users through the process of defining the geometry, materials, loading etc.

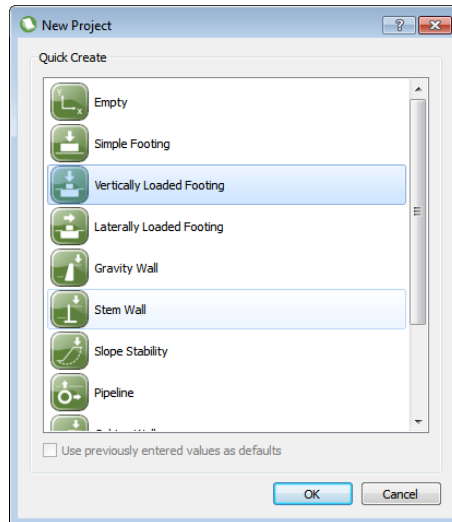


Figure 11 - The LimitState:GEO Project Selector Dialog

- Choose from the following wizard types:
 - Empty
 - Simple Footing
 - Vertically Loaded Footing
 - Laterally Loaded Footing
 - Gravity Wall
 - Stem Wall
 - Slope Stability
 - Pipeline
 - Gabion Wall
 - Reinforced Soil Wall
 - Sheet Pile Wall
- Click 'Finish' at any point during the wizard process: LimitState:GEO will automatically fill in any information that has not been explicitly supplied by assuming default values and using information already given.
- Move backwards and forwards through the various steps of the wizards and edit data as necessary; a very useful feature should any detail accidentally be omitted initially.

The dialogs found within most wizards are as follows:

- **Project** – defines the general properties of the project.
- **Geometry** – defines the form of geotechnical feature to be analyzed.
- **Materials** – defines the material properties of the different solid zones and interfaces.
- **Loads** – defines a standard loading configuration relevant to the particular problem.
- **Scenarios** – defines the number of different scenarios and corresponding partial safety factor set(s) applied to the loads and materials.
- **Analysis** – define the nodal resolution of the problem and the type of analysis to carry out, Factor Load(s) or Factor Strength(s).

Sample Wizard – Simple Footing Project

- Some of the dialogs shown in other wizards will be different to those shown here (*e.g.* those relating to geometry). This example wizard is however indicative of the many different wizards available in LimitState:GEO.
- In all wizard dialogs, default data is assumed unless otherwise specified by the user.

Project Details

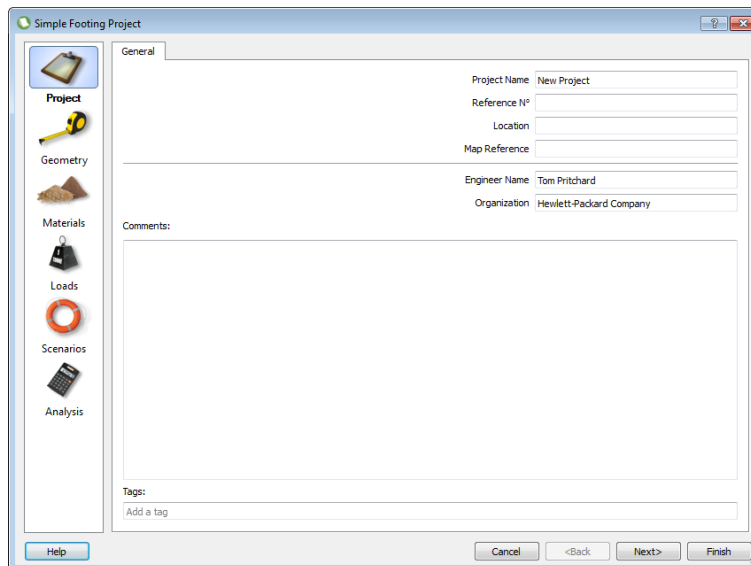


Figure 12 - The LimitState:GEO Simple Footing Project 'Project Details' Dialog

- Deals with 'general' aspects of the model
- All of the details that can be specified here are optional:
 - Project Name
 - Reference number
 - Location
 - Map reference
 - Engineer name
 - Organization
 - Comments
 - Tags

Geometry

Define the key dimensions of the footing (the figure in the dialog clearly indicates the meaning of the parameters to be entered):

- Footing thickness
- Thickness of soil layer
- Footing width
- Soil width
- Model as symmetrical half space

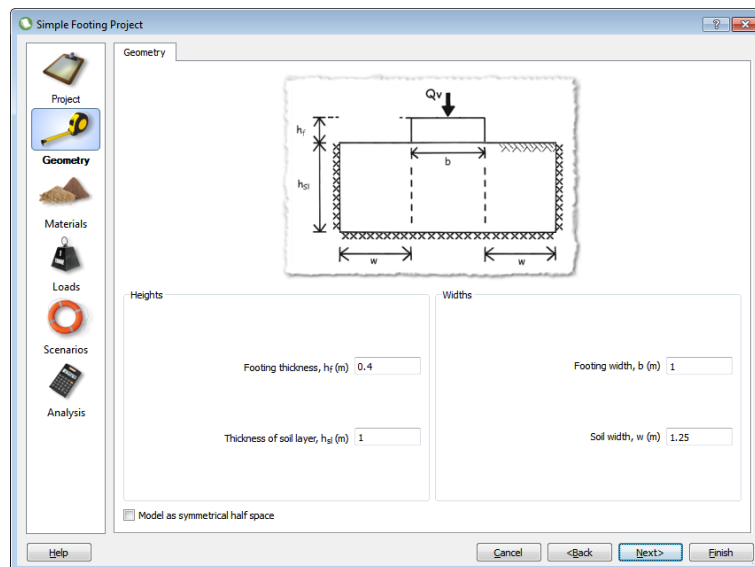


Figure 13 - The LimitState:GEO Simple Footing Project 'Geometry' Dialog

Materials

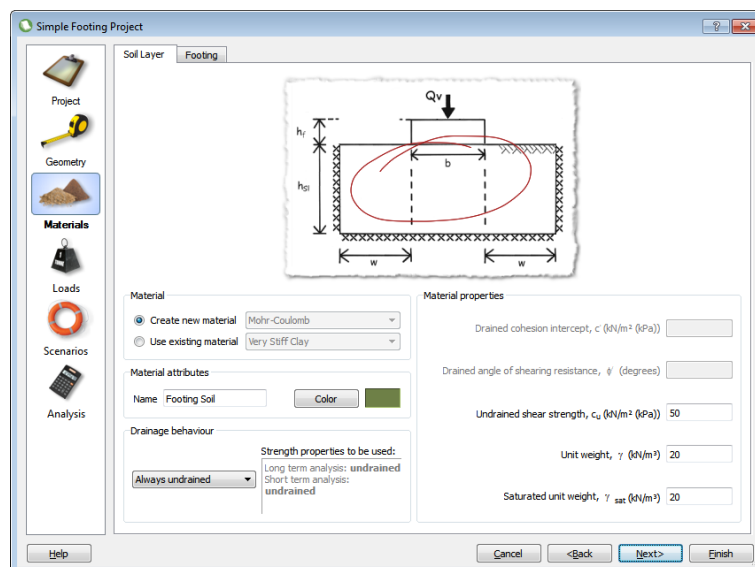


Figure 14 - The LimitState:GEO Simple Footing Project 'Materials – Soil Layer' Dialog

Define the material properties of the soil and footing:

Soil Layer:

- Create new material
 - Material attributes – Name
 - Material attributes – Color
 - Material drainage behaviour – Always undrained / drained-undrained / always drained
 - Material properties – c_u
 - Material properties – c'
 - Material properties – ϕ'
 - Material properties – γ_{dry}
 - Material properties – γ_{sat}

- Use existing material

Footing Layer:

- Create new material
 - Material attributes – Name
 - Material attributes - Color
 - Material properties – c_u
 - Material properties – Unit weight, γ
 - Soil-footing interface properties – Multiplier on $\tan \phi$
 - Soil-footing interface properties – Multiplier on c' , c_u
- Use existing material

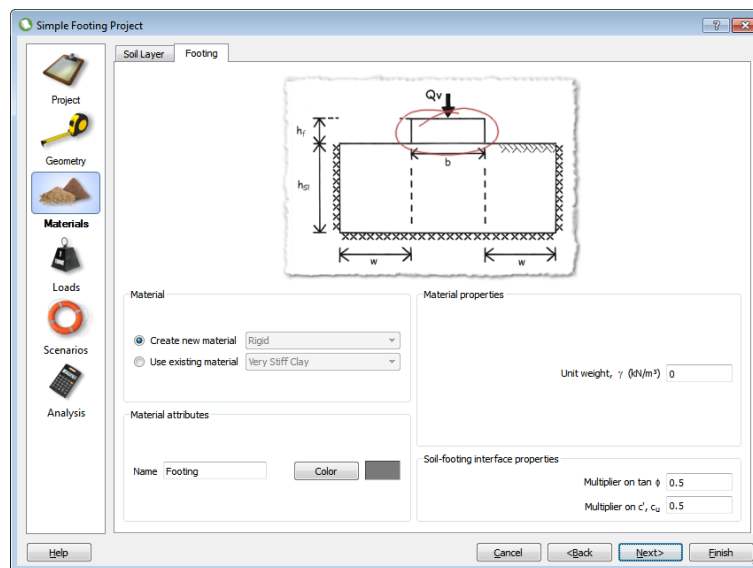


Figure 15 - The LimitState:GEO Simple Footing Project 'Materials – Footing' Dialog

Loads

Define the vertical load applied to the footing:

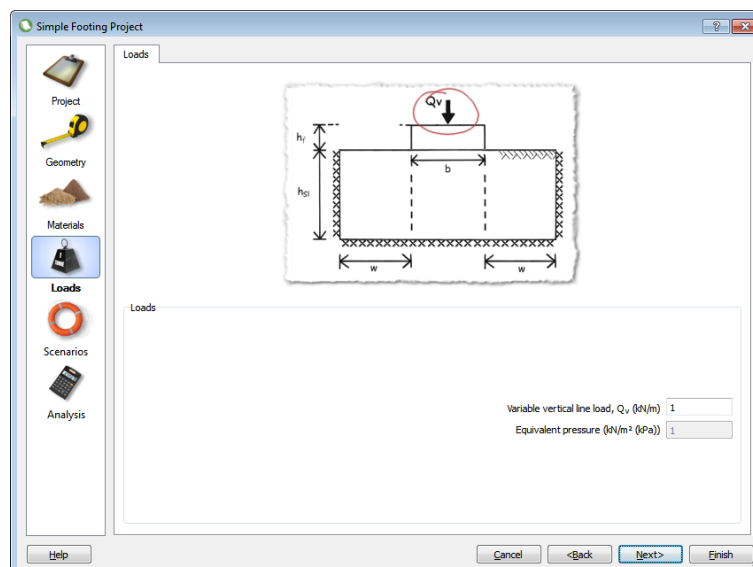


Figure 16 - The LimitState:GEO Simple Footing Project 'Loads' Dialog

Scenarios

Define the different scenarios and partial factor sets to be considered during analysis:

- Specify whether a single or multiple scenario analysis is carried out.
- Editable partial factors:
 - Unfavourable loads (permanent, accidental and variable).
 - Favourable loads (permanent, accidental and variable).
 - Materials (c' , $\tan\phi'$, c_u).
- Default available partial factor sets:
 - User.
 - Unity.
 - EC7 DA1/1 (Eurocode 7 Design Approach, combination 1).
 - EC7 DA1/2 (Eurocode 7 Design Approach, combination 2).
- Import partial factor sets from .csv file.
- Export partial factor sets to .csv file.
- Define new partial factor sets.
- Delete a partial factor set from a multiple scenario problem.
- Specify whether a long term analysis is to be carried out.

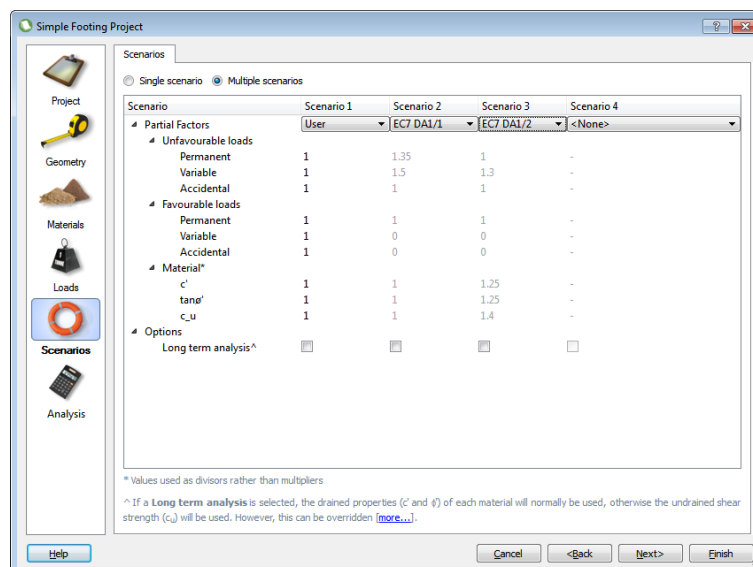


Figure 17 - The LimitState:GEO Simple Footing Project 'Scenarios' Dialog

Analysis

Define the nodal resolution and type of analysis to undertake.

- Specify the type of analysis (Factor Loads or Factor Strengths).
- Specify the target nodal density.

Solve

Analysis Details

- Performs a kinematic or 'mechanism' analysis based on plasticity theory, and assuming small displacements.
- Provides a solution in terms of an adequacy factor on either loads or material.
- For a Project with a several scenarios the adequacy factor for each is computed and the lowest (most critical) is identified.

Analysis Engine

- Uses MOSEK, a powerful interior point linear programming solver (see www.mosek.com).
- For maximum efficiency all problem data is passed to the solver via memory.
- More detailed information on the progress towards a solution is available in the Preferences dialog.

Results

- Critical mechanism automatically identified and computed adequacy factor displayed in output window.
- If no solution can be found, or if model is unstable, output reports this and gives possible reasons for this behaviour.
- Critical mechanism can be animated for enhanced user understanding (instantaneous velocities magnified).
- Shear and normal stresses along slip-lines displayed in graphical and numerical formats when solids are selected after solve.
- Axial / shear forces and moments displayed in graphical and numerical formats when engineered elements are selected after solve.

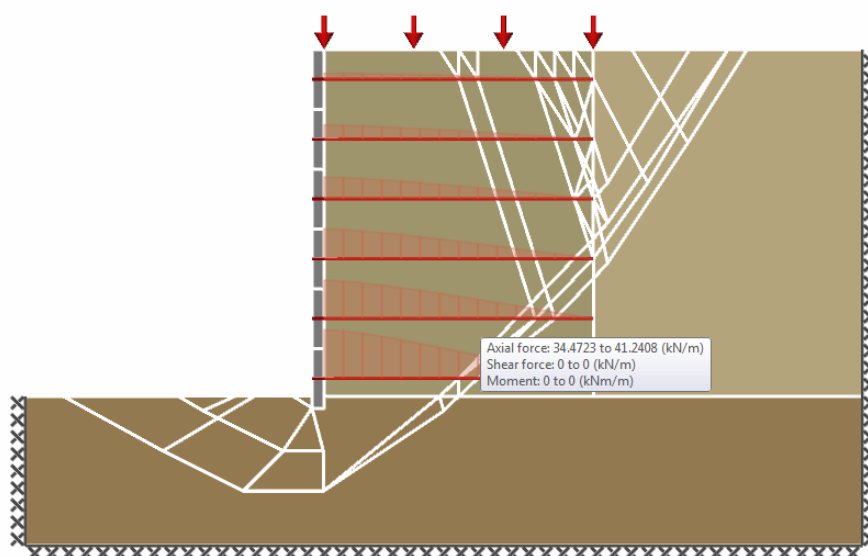


Figure 18 - Post-solve diagrams displayed by LimitState:GEO (Axial Force)

Report Output

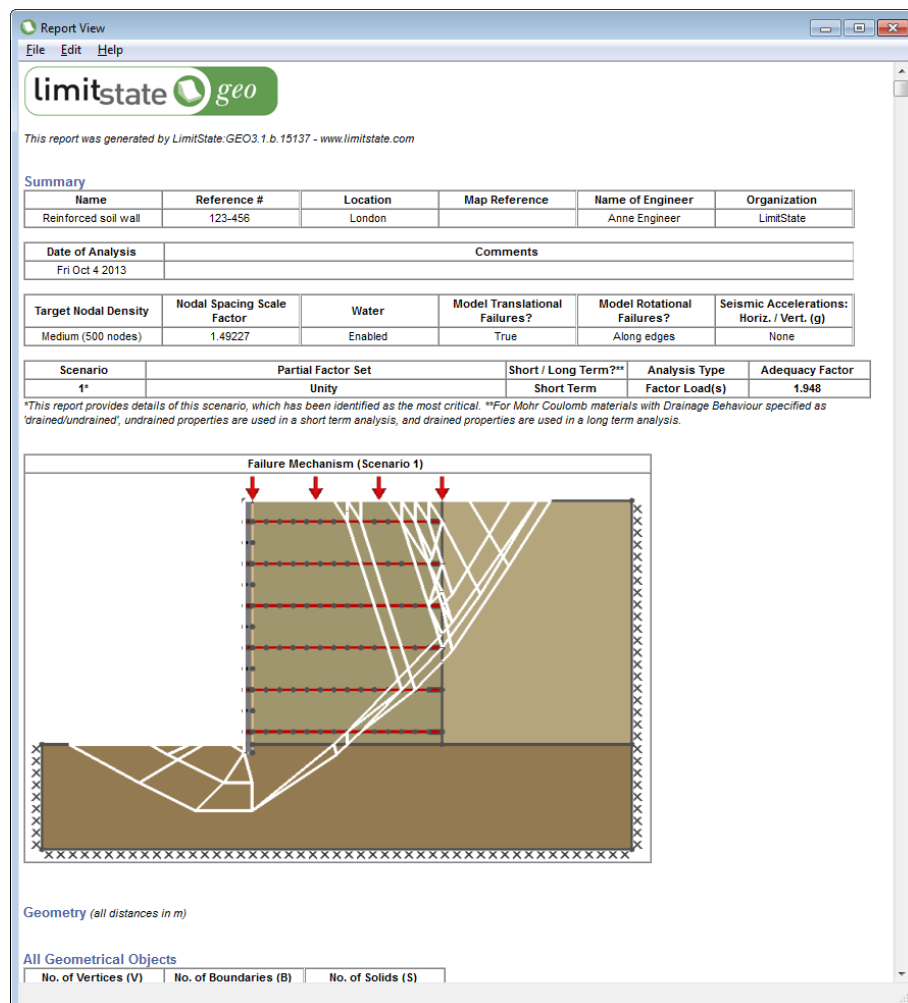


Figure 19 - The LimitState:GEO Report View

- Chose the sections that are included in the report:
 - About
 - Summary
 - Geometry
 - Water
 - Materials
 - Partial Factors
 - Loads
 - Free Body Diagrams
- Customize the document with user-defined headers and footers.
- Print the report directly from LimitState:GEO or save the file in pdf, rtf or html formats.

Report Contents

By default, each report contains all the sections mentioned above, containing the following details:

Section	Contents
About	<ul style="list-style-type: none"> A paragraph of text outlining the method of analysis (DLO) used by LimitState:GEO
Summary	<ul style="list-style-type: none"> Name Reference N°. Location Map reference Name of engineer Organization Date of analysis Comments Tags Target nodal density Nodal spacing scale factor Water Model transitional failures? Model rotational failures? Seismic accelerations: Horizontal / Vertical Solution tolerance Automatic adequacy on load(s) Factor on load(s) Artificial cohesion Analysis result: <ul style="list-style-type: none"> Critical scenario Partial factor set Short / long term analysis Analysis type Adequacy factor The critical scenario is identified and a graphical representation of the critical failure mechanism is shown
Geometry	<ul style="list-style-type: none"> Global <ul style="list-style-type: none"> Number of vertices Number of boundaries Number of solids Boundary objects <ul style="list-style-type: none"> ID (any boundaries with loads applied are highlighted by an *) Start vertex ID End vertex ID Baseline nodal spacing Support type Material(s) Solid objects <ul style="list-style-type: none"> ID (any solids with loads applied are highlighted by an *) Vertex IDs Boundary IDs Baseline nodal spacing (x / y) Material(s)
Water Table	<ul style="list-style-type: none"> Status Vertices (x, y)
Water Regimes	<ul style="list-style-type: none"> All properties associated with the water regimes used in the model: <ul style="list-style-type: none"> Constant pressure regimes: <ul style="list-style-type: none"> Key Name Pressure Constant potential regimes: <ul style="list-style-type: none"> Key Name Potential Factor on unit weight of water Interpolated grid regimes: <ul style="list-style-type: none"> Key Name Origin (top left) x Origin (top left) y Spacing, dx Spacing, dy No. of x divisions No. of y divisions Aquiclude regimes: <ul style="list-style-type: none"> Key Name

Section	Contents
Materials (contd...)	<ul style="list-style-type: none"> All properties associated with the materials used in the model: <ul style="list-style-type: none"> Color Mohr-Coulomb materials: <ul style="list-style-type: none"> Name Unit weight (dry / saturated) c' / ϕ' c_u (datum) (gradient) Mohr-Coulomb (Derived) materials: <ul style="list-style-type: none"> Name Unit weight (dry / saturated) c' / ϕ' c_u (datum) (gradient) Cutoff materials: <ul style="list-style-type: none"> Name Unit weight (dry / saturated) σ_t σ_c Rigid materials: <ul style="list-style-type: none"> Name Unit weight (dry / saturated) Engineered Element materials: <ul style="list-style-type: none"> Name Pullout factors, T_c (T_d) Lateral factors, N_c (N_d) Plastic moment, M_p Rupture strength, R Compression strength, C Subdivide at nodes?
Partial factors	<ul style="list-style-type: none"> For each scenario: <ul style="list-style-type: none"> Unfavourable: permanent Unfavourable: variable Unfavourable: accidental Favourable: permanent Favourable: variable Favourable: accidental c' $\tan\phi'$ c_u The partial factor set used in the critical scenario is also identified.
Loads	<ul style="list-style-type: none"> Boundary objects: <ul style="list-style-type: none"> Loaded object ID Type of load Favourable load? (true / false) Adequacy applied? (true / false) Normal load Shear load Solid objects: <ul style="list-style-type: none"> Loaded object ID Type of load Favourable load? (true / false) Adequacy applied? (true / false)
Free body diagrams	<ul style="list-style-type: none"> For each solid block: <ul style="list-style-type: none"> Free body diagram with shear and normal values labeled. For each face of the block: <ul style="list-style-type: none"> start and end points angle to horizontal normal and shear force values horizontal equilibrium term vertical equilibrium term Additional applied loading (e.g. self-weight). Sum of all forces per unit width.

Menus

File

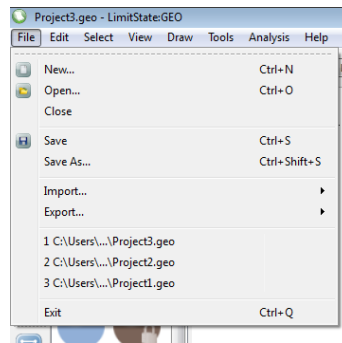


Figure 20 - The LimitState:GEO File Menu

Action	Result
New...	Start a new project from scratch.
Open...	Open a previously saved project.
Close	Close the current project.
Save	Save the current project using the current filename.
Save As...	Save the current project under a different filename.
Import...	Import predefined Materials (.csv), Water Regimes (.csv) or problem Geometry (.dxf).
Export...	Export problem Materials (.csv), Water Regimes (.csv), Geometry (.dxf, .eps or .asy), an Image (.png, .jpg, .tiff, .eps or .ps) or Animation (.avi or .gif).
Recent Files	Open a recently saved project.
Exit	Exit LimitState:GEO.

Edit

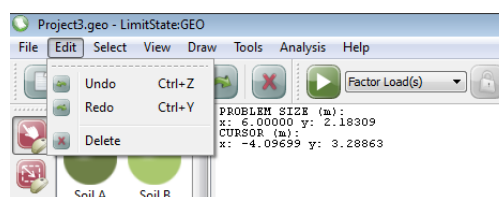


Figure 21 - The LimitState:GEO Edit Menu

Action	Result
Undo	Undo the last action.
Redo	Redo the last undo.
Delete	Delete the currently selected object(s).

Select

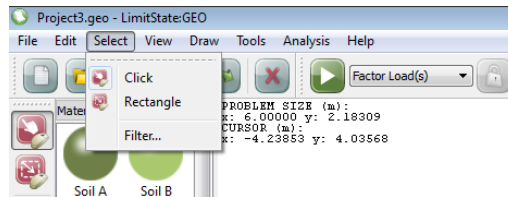


Figure 22 - The LimitState:GEO Select Menu

Action	Result
Click	Select objects by clicking them with the mouse pointer.
Rectangle	Select objects by drawing a rectangle around them.
Filter...	Select only objects of a particular type (or types): <ul style="list-style-type: none"> Vertex Boundary Solid Water Pressure Table Slip-line All

View

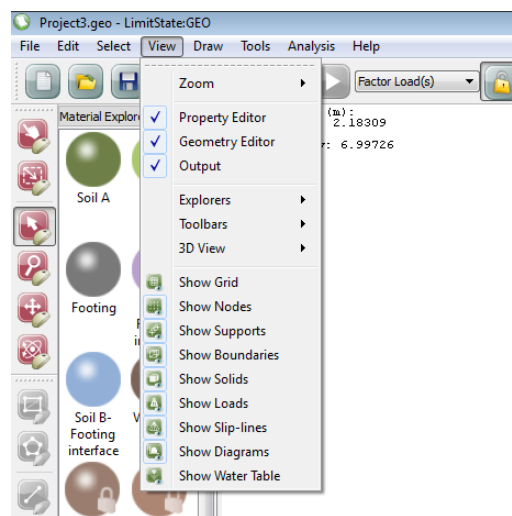


Figure 23 - The LimitState:GEO View Menu

Action	Result
Zoom	Zoom towards or away from the model: <ul style="list-style-type: none"> Zoom All (Display the entire model) Zoom In (Zoom towards the centre of the model) Zoom Out (Zoom away from the centre of the model)
Property Editor	Toggle the display of the Property Editor .
Geometry Editor	Toggle the display of the Geometry Editor .
Output	Toggle the display of the Output pane .

Action	Result
Explorers	Toggle the display of the Explorers : <ul style="list-style-type: none"> ○ Vertex ○ Boundary ○ Solid ○ Material
Toolbars	Toggle the display of the Toolbars : <ul style="list-style-type: none"> ○ File ○ Edit ○ Select ○ Show ○ View 3D ○ Analysis ○ Animation ○ Zoom ○ Draw ○ Cursor ○ Rotate 3D ○ Help
3D View	View the model from various 3D viewpoints : <ul style="list-style-type: none"> ○ Top ○ Bottom ○ Left ○ Right ○ Front (Default) ○ Back ○ Show Global Axis ○ Perspective
Show Grid	Toggle the display of the Grid .
Show Nodes	Toggle the display of the Nodes .
Show Supports	Toggle the display of the Supports .
Show Boundaries	Toggle the display of the Boundaries .
Show Solids	Toggle the display of the Solid objects.
Show Loads	Toggle the display of the Loads .
Show Slip-lines	Toggle the display of the Slip-lines .
Show Post-Solve Diagrams	Toggle the display of post-solve 'on solve' diagrams.
Show Water Table	Toggle the display of the Water Pressure Table (Note: the water table will still be enabled during analysis).

Draw

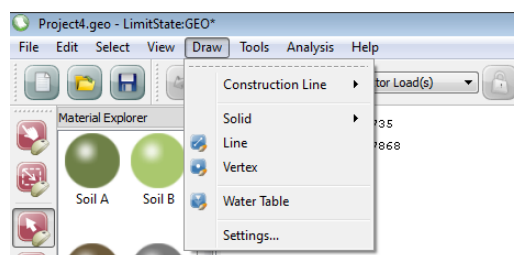


Figure 24 - The LimitState:GEO Draw Menu

Action	Result
Construction Line	Draw a construction line on the viewer pane (these are to aid in the generation of the model and are not counted during analysis): <ul style="list-style-type: none"> Vertical / Horizontal / Custom / Clear All
Solid	Draw a solid object: <ul style="list-style-type: none"> Rectangle (a 4 sided solid defined by two corners) Polygon (a multi-sided solid defined by all of its corners)
Line	Draw a line using two points.
Vertex	Add a vertex to a line.
Water Table	Draw a water table (defined by its vertices).
Settings...	Open the draw settings dialog.

Tools

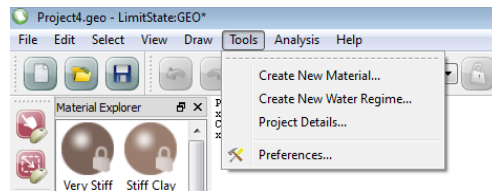


Figure 25 - The LimitState:GEO Tools Menu

Action	Result
Create New Material...	Open the Create New Material dialog.
Create New Water Regime...	Open the Create New Water Regime dialog.
Project Details...	Open the Project Details dialog.
Preferences...	Open the Preferences dialog.

Analysis

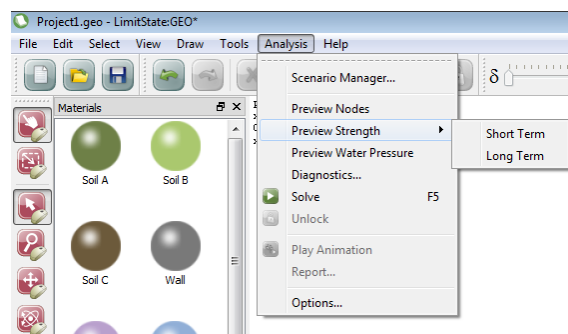


Figure 26 - The LimitState:GEO Analysis Menu

Action	Result
Scenario Manager...	Open the Scenario Manager.
Preview Nodes	Display the nodes that will be used during analysis (these are the potential end-points of slip-lines).
Preview Strength	Display the short or long-term strength of the soil bodies in the model. Interpolated grid materials will be displayed with a contour and gradient plot.

Analysis (contd...)

Action	Result
Unlock	Allow the problem to be edited following an analysis or previewing of the nodes.
Preview Water Pressure	Display the water pressures within the soil bodies in the model. Interpolated grid water regimes will be displayed with a contour and gradient plot.
Diagnostics	Open the Diagnostics dialog to display information, warnings and errors associated with the problem.
Solve	Analyse the problem.
Play Animation	Animate the failure mechanism for the selected scenario.
Report...	Generate a report following analysis.
Options...	Open the Analysis Options dialog.

Help

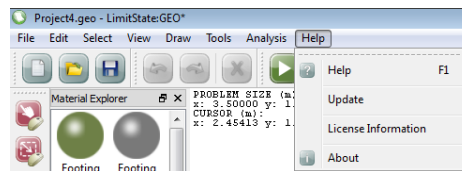


Figure 27 - The LimitState:GEO Help Menu

Action	Result
Help	Open the LimitState:GEO help system.
Update	Check for (and download) updates for LimitState:GEO.
License Information	View information about the current license being used and swap to a different license if required.
About	Display information regarding the version of LimitState:GEO currently being used.

Dialogs

Aside from standard Save / Open / Import / Export dialogs, the following dialogs are available in LimitState:GEO:

Selection Filter

- Accessed from the Select menu.
- Set which object types can be selected in the viewer pane:

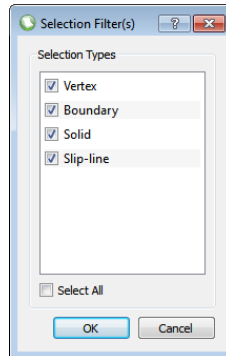


Figure 28 - The LimitState:GEO Selection Filter Dialog

Draw Settings

- Accessed from the Draw menu.
- Specify grid dimensions and snap settings:

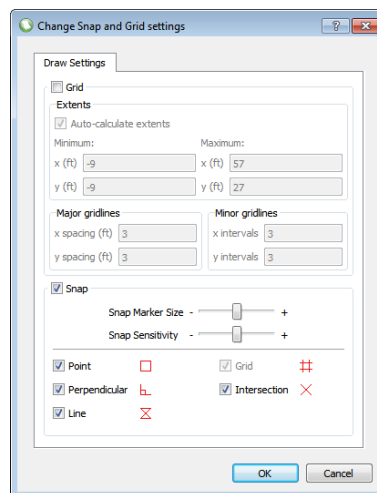


Figure 29 - The LimitState:GEO Draw Settings Dialog

Grid / Snap	Option	Description
Grid	Extents : Auto calculate extents	Adaptively resize the drawing extents during modelling.
	Extents : Minimum x	Specify an initial minimum coordinate along the x axis.
	Extents : Minimum y	Specify an initial minimum extent along the y axis.
	Extents : Maximum x	Specify an initial maximum extent along the x axis.
	Extents : Maximum y	Specify an initial maximum extent along the y axis.
	Major gridlines: x spacing	Specify the spacing between major x axis gridlines.
	Major gridlines: y spacing	Specify the spacing between major y axis gridlines.
	Minor gridlines: x intervals	Specify the number of intervals between major x axis gridlines.
	Minor gridlines: y intervals	Specify the number of intervals between major y axis gridlines.
Snap	Snap Marker Size	Increase or decrease the size of the snap marker as displayed in the viewer.
	Snap Sensitivity	Increase or decrease the required distance between the cursor and a feature before a snap marker is displayed.
	Point	Snap to Points.
	Perpendicular	When drawing a line or polygon, snap to a point on an existing line that forms a right angle between that and the line being drawn.
	Line	Snap to Line objects.
	Grid	Snap to the Grid intersection points.
	Intersection	Snap to Intersections between line objects.

Create New Material

- Accessed from the Tools menu.
- Define a new material of any of the standard types:

Type	User-Defined Variables
Mohr-Coulomb	<ul style="list-style-type: none"> • Material name • Material color • Undrained shear strength, c_u • Drained cohesion intercept, c' • Drained angle of shearing resistance, ϕ' • Drainage behaviour • Dry unit weight, γ_{dry} • Saturated unit weight, γ_{sat} • Post-Solve Display
Mohr-Coulomb (Derived)	<ul style="list-style-type: none"> • Material name • Material color • Parent material • Cohesion multiplier • $\tan\phi'$ multiplier • Drainage behaviour • Post-Solve Display
Cutoff	<ul style="list-style-type: none"> • Material name • Material color • Limiting tensile stress • Limiting compressive stress • Dry unit weight, γ_{dry} • Saturated unit weight, γ_{sat} • Post-Solve Display

Create New Material (contd...)

Type	User-Defined Variables
Rigid	<ul style="list-style-type: none"> Material name Material color Dry unit weight, γ_{dry} Saturated unit weight, γ_{sat} Post-Solve Display
Engineered Element (1D)	<ul style="list-style-type: none"> Material name Material color Pullout resistance (T_c / T_q) Lateral resistance (N_c / N_q) Plastic moment (M_p) Rupture strength (R) Subdivide at nodes? Post-Solve Display

Create New Water Regime

- Accessed from the Tools menu.
- Define a new water regime of any of the standard types:

Type	User-Defined Variables
Constant Potential	<ul style="list-style-type: none"> Water regime name Color Water potential Factor on unit weight of water
Constant Pressure	<ul style="list-style-type: none"> Water regime name Color Water pressure
Interpolated Grid	<ul style="list-style-type: none"> Water regime name Color Set grid (opens the grid interpolation dialog)
Aquiclude	<ul style="list-style-type: none"> Water regime name Color

Project Details

- Accessed from the Tools menu.
- Specify general details about the current project:

Type	User-Defined Variables
Details	<ul style="list-style-type: none"> Project Name Reference No Location Map Reference Engineer Name Organization Comments Tags

Preferences

- Accessed from the Tools menu.
- The Preferences dialog is the main place to alter the default behaviour of LimitState:GEO. Within the dialog there are 6 tabs, each controlling a different aspect of the software:

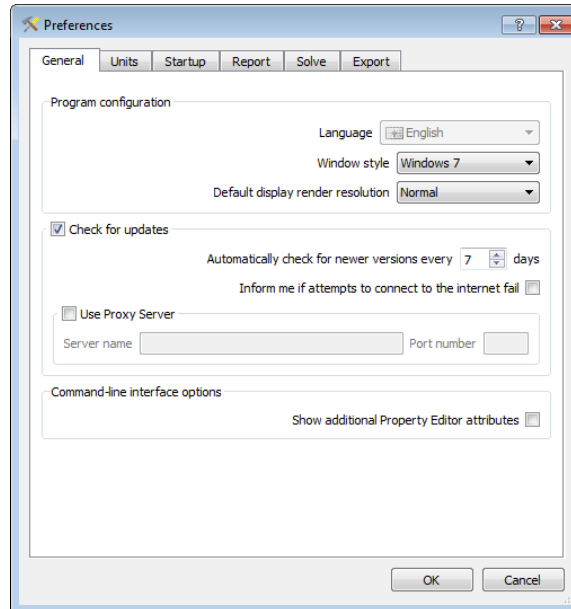


Figure 30 - The LimitState:GEO Preferences Dialog

Tab	Setting	Description
General	Language	Change the interface language (generally fixed).
	Window Style	Change the appearance of LimitState:GEO (7 styles available).
	Default display render resolution	Change the quality of the rendering in the viewer pane (low / normal / high).
	Check for updates	Cause LimitState:GEO to automatically check the LimitState server for updates to the software (requires internet connection).
	Automatically check for new versions every <i>n</i> days	Determines how often the software checks for updates.
	Use Proxy Server	Use a proxy server to access the internet (requires Server Name and Port Number).
	Show additional Property Editor attributes	Display "Object key" information, which can be used in conjunction with the command line interface.
Units	Metric	Use metric units (kN, m etc.) in the project.
	Imperial	Use imperial units (lb, ft etc.) in the project. Also option to use defaults tailored for Imperial Units (in Wizards and built-in materials).
Startup	Show 'Welcome' dialog next time	Display the welcome dialog the next time the software is started.
	Show analysis tip after wizards completed	Display the analysis tip each time a wizard is completed.
	Clear recent files list	Clear the list of recently accessed files.
Report	Header image	Change the default report header image for one of your own choice (requires png format image).
	Footer image	Change the default report footer image for one of your own choice (requires png format image).

Preferences (contd...)

Tab	Setting	Description
Solve	Display detailed information in output window	Cause the output window to display detailed information about the analysis: <ul style="list-style-type: none"> • Scenario * • Partial factors * • Iteration • Variables • Constraints • Solution * • Violation% • Variables added • Constraints added • Time (sec) * Shown by default
	Number of significant figures to display in the answer	Increase or decrease the precision of the answer.
	Animate after solve for problems with less than 1000 nodes	Automatically animate the solution if the problem contains less than 1000 nodes.
	Animate if no more than X nodes	Automatically animate the solution after solve if the target number of nodes is less than X.
	Do not animate after solve	Do not animate the failure mechanism after a solution is found.
Export	Width (pixels)	Change the width of exported images and animations.
	Show initial geometry	Display the initial problem geometry in exported animations.
	Show interim solutions	Display the interim solutions from each iteration step of the analysis.
	Show final solution (undeformed)	Display the final solution (slip-line mechanism) in an undeformed state.
	Show deformation animation	Deform the final solution.
	Show nodes	Display the problem nodes at the start of the animation.

Scenario Manager

Define the partial factor sets to be considered during analysis:

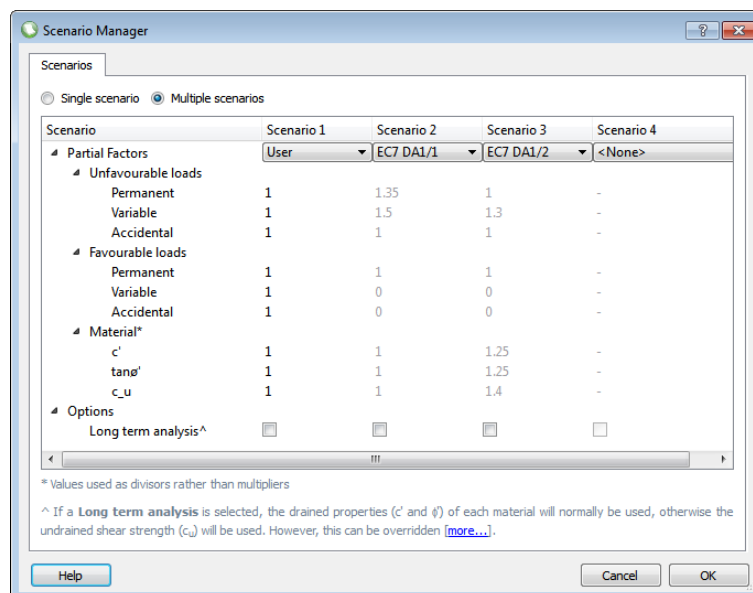


Figure 31 - The LimitState:GEO Scenario Manager Dialog

Option	Option	Description
Number of scenarios	Single scenario	Carry out a single scenario analysis.
	Multiple scenarios	Carry out a multiple scenario analysis.
Editable partial factors	Unfavourable loads	Define partial factors (multipliers) for unfavourable permanent, accidental and variable loads.
	Favourable loads	Define partial factors (multipliers) for favourable permanent, accidental and variable loads.
	Materials	Define partial factors (divisors) for c' , $\tan\phi'$ and c_u .
Default available partial factor sets	User	All partial factors are user defined.
	Unity	All partial factors set to 1.0
	EC7 DA1/1	Partial factors set according to Eurocode 7 Design Approach, combination 1.
	EC7 DA1/2	Partial factors set according to Eurocode 7 Design Approach, combination 2.
Other options	{Import...}	Import partial factor sets from .csv file.
	{Export...}	Export partial factor sets to .csv file.
	<New...>	Define new partial factor sets.
	<Delete...>	Delete a partial factor set from a multiple scenario problem.
	Long Term Analysis	Specify whether a long term analysis is to be carried out.

Analysis Options

Set the options for a Factor Strength(s) analysis:

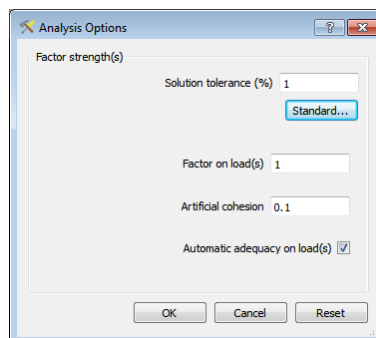


Figure 32 - The LimitState:GEO Analysis Options Dialog

Report

Chose the sections that are included in the report:

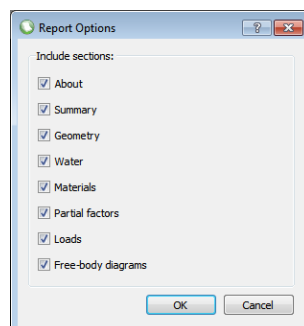





Figure 33 - The LimitState:GEO Report Options Dialog




Toolbars

The following tables detail the buttons to be found on each of the toolbars within LimitState:GEO:







File

Icon	Description
	Start a New project.
	Open an existing project.
	Save the current project.




Edit

Icon	Description
	Undo an action.
	Redo an action.
	Delete an object.






Cursor

Icon	Description
	Click Select .
	Rectangle Select .
	Select Cursor .
	Zoom Cursor .
	Pan .
	Rotate .


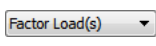

Draw

Icon	Description
	Draw Rectangle .
	Draw Polygon .
	Draw Line .


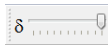
Draw (contd...)

Icon	Description
	Draw Vertex .
	Draw Horizontal construction line.
	Draw Vertical construction line.
	Draw Angled construction line.
	Draw Water pressure table.




Analysis

Icon	Description
	Solve .
	Change Analysis type .
	Lock / Unlock .



Animation

Icon	Description
	Play animation.
	Magnify displacements.










Zoom

Icon	Description
	Zoom All .
	Zoom In .
	Zoom Out .



Show

Icon	Description
	Show Construction lines.
	Show the Grid .





Show (contd...)

Icon	Description
	Show Nodes .
	Show Supports .
	Show Boundaries .
	Show Solids .
	Show Loads .
	Show Slip-lines .
	Show Post-solve diagrams .
	Show Water table .
	View in 3D mode.




Select

Icon	Description
	Select a Single object.
	Select objects using a Rectangle .




Rotate 3D

Icon	Description
	Rotate around the x axis.
	Rotate around the y axis.
	Rotate around the z axis.
	Rotate 3D .



View 3D

Icon	Description
	View from the Top .
	View from the Bottom .
	View from the Left .

View 3D (contd...)

Icon	Description
	View from the Right .
	View from the Back .
	View from the Front .

Help

Icon	Description
	Help.
	Information.

Context Menus

The following details the context menus to be found when right-clicking the mouse in various areas of the LimitState:GEO user interface:

Toolbars and Property Editor

Toggle the display of various GUI objects:

Option	Description
Vertex Explorer	Toggle the display of the Vertex explorer.
Boundary Explorer	Toggle the display of the Boundary explorer.
Solid Explorer	Toggle the display of the Solid explorer.
Water Regime Explorer	Toggle the display of the Water Regime explorer.
Material Explorer	Toggle the display of the Material explorer.
Property Editor	Toggle the display of the Property editor.
Geometry Editor	Toggle the display of the Geometry editor.
Output	Toggle the display of the Output pane.

Toggle the display of various toolbars:

Option	Description
File	Toggle the display of the File toolbar.
Edit	Toggle the display of the Edit toolbar.
Cursor	Toggle the display of the Cursor toolbar.

Option	Description
Draw	Toggle the display of the Draw toolbar.
Rotate 3D	Toggle the display of the Rotate 3D toolbar.
Analysis	Toggle the display of the Analysis toolbar.
Animation	Toggle the display of the Animation toolbar.
Zoom	Toggle the display of the Zoom toolbar.
Show	Toggle the display of the Show toolbar.
View 3D	Toggle the display of the View 3D toolbar.
Help	Toggle the display of the Help toolbar.

Viewer Pane (General)

Option	Description
Clear Water Table	Delete the water table (if present).
Exit	Exit the context menu.
Select	Change the cursor to select objects.
Click	Select objects by clicking on them.
Rectangle	Select objects by drawing a rectangle around them.
Pan	Change the cursor to allow panning of the model in the viewer pane.
Rotate	Change the cursor to allow rotation of the model in the viewer pane.
Rotate 3D	Rotate the model around all 3 main Cartesian axes.
Rotate About x	Rotate the model around the x axis.
Rotate About y	Rotate the model around the y axis.
Rotate About z	Rotate the model around the z axis.
Zoom	Change the cursor to allow zooming into and out of the viewer pane.
Zoom In	Zoom into the image by 1 increment.
Zoom Out	Zoom out of the image by 1 increment.
Zoom All	Zoom out to the extents of the problem.
View	Change the view of the model.
Top	View the model from the top.
Bottom	View the model from the bottom.
Right	View the model from the right.
Left	View the model from the left.
Front	View the model from the front.
Back	View the model from the back.
View 3D	View the model in 3D.
Zoom All	Zoom out to the extents of the problem.
Show Global Axis	Toggle the display of the global axis.
Perspective	Toggle perspective view (needs to be in conjunction with 3D view).

Viewer Pane (General) (contd...)

Option	Description
Render	
High	View the model at high quality (maximum antialiasing).
High Resolution Wireframe	View the model as a high resolution wireframe.
Normal	View the model at normal quality (moderate antialiasing).
Wireframe	View the model as a wireframe.
Low Resolution	View the model in low resolution (minimal antialiasing).
Low Resolution Wireframe	View the model as a low resolution wireframe.
Save Image	Save the current view of the model as a png, jpg, tiff, eps or ps image.

Viewer Pane (Post-Solve Solid)

Option	Description
Normal Stress	View normal stress diagrams for selected solids.
Shear Stress	View shear stress diagrams for selected solids.
Export Selected Solid Forces	Export the stresses on the selected solid to a CSV file.

Viewer Pane (Post-Solve Engineered Element)

Option	Description
Tensile Force	View axial force diagrams for engineered elements.
Shear Force	View shear force diagrams for engineered elements.
Moment	View bending moment diagrams for engineered elements.
Export All Engineered Element Forces	Export the forces on the selected engineered element to a CSV file.

Material Explorer

Option	Description
Delete User Defined Material	Delete the selected material.
Duplicate Material	Copy all the properties of the selected material to a new material.
New Material...	Create a new material from scratch.
Export Materials...	Save the current materials library as a csv file.
Import Materials...	Import a pre-saved csv materials library into LimitState:GEO.

Other Explorers

Option	Description
Copy Details	Copy all the properties of the selected object.
Paste Details	Paste all the properties of a previously copied object.
Copy	Copy the current cell.
Paste	Paste the properties of a previously copied cell.
Select All	Select all cells in the table.
Deselect All	Deselect all selected cells.

Geometry Editor

Option	Description
Copy	Copy the current cell.
Paste	Paste the properties of a previously copied cell.

Output Pane

Option	Description
Copy	Copy the currently highlighted text.
Select All	Select all text.
Clear All	Clear all text.

LimitState Ltd

The Innovation Centre
217 Portobello
Sheffield
S1 4DP
UK

+44 (0) 114 224 2240
info@limitstate.com
limitstate.com