



LimitState:GEO Manual

VERSION 3.6.1

LimitState Ltd

May 5, 2021

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

LimitState:GEO 3.6.1

© LimitState Ltd

All rights reserved. No parts of this work may be reproduced in any form without the written permission of LimitState Ltd.

While every precaution has been taken in the preparation of this document, LimitState Ltd assumes no responsibility for errors or omissions. LimitState Ltd will not be liable for any loss or damage of any kind, including, without limitation, indirect or consequential loss (including loss of profits) arising out of the use of or inability to use this document and/or accompanying software for any reason.

This document is provided as a guide to the use of the software. It is not a substitute for standard references or engineering knowledge. The user is assumed to be conversant with standard engineering terminology and codes of practice. It is the responsibility of the user to validate the software for the applications for which it is to be used.

Contents

I	Introduction and Quickstart	15
1	Introduction	17
1.1	General Description	17
1.2	Program Features	17
1.3	LimitState:GEO Terminology	19
1.4	About LimitState	20
1.5	Using Help	20
1.6	System Requirements	20
1.7	Program Limits	20
1.8	Contact Details	21
1.8.1	Sales	21
1.8.2	Software Support	21
1.8.3	Website	21
2	Getting Started	23
2.1	Installation and Licensing	23
2.1.1	Thumbnails, Tags and File Search	23
2.2	Starting LimitState:GEO	24
2.3	Guidance Available in this Manual	25
3	Quick Start Tutorial	27
3.1	Introduction	27
3.2	Getting Started	27
3.3	Analysis Modes and the Adequacy Factor	29
3.4	Solving a Problem	29
3.5	Viewing Mechanism Deformation	31
3.6	Viewing Stresses and Bending Moments	32
3.7	Zooming In and Out	33
3.8	Pan, Rotate and 3D Views	33
3.9	Trying Different Problems and Parameter Sets	34
3.10	The Property Editor	34
3.10.1	Changing Material Properties	35
3.10.2	Changing the Material Used in a Soil Layer (Solid Object)	35
3.10.3	Changing the Material used in an Interface (Boundary Object)	37
3.10.4	Modifying Loads	38
3.10.5	Resizing a Block of Soil	39
3.10.6	Changing a Boundary Condition	40
3.11	Solving a Collapse Problem by Factoring Soil Strength	41
3.12	Modelling Layered Soils	42
3.13	Solution Accuracy	43
3.14	Conclusion	44

4 Wizards	45
4.1 Introduction	45
4.2 Using a New Project Wizard	46
4.2.1 Introduction	46
4.2.2 General Project Settings	47
4.2.3 Geometry	48
4.2.4 Materials	48
4.2.5 Loads	49
4.2.6 Scenarios	50
4.2.7 Analysis Options	51
II Theory	53
5 Discontinuity Layout Optimization	55
5.1 Introduction	55
5.2 Limit Analysis and Limit Equilibrium	57
5.3 DLO: How Does it Work?	57
5.3.1 Basic Principles	57
5.3.2 Adaptive Solution Procedure	59
5.3.3 Rotational Failure Mechanisms	60
6 Model components	67
6.1 Material Models	67
6.1.1 Introduction	67
6.1.2 Mohr-Coulomb Material	67
6.1.3 Cutoff Material (Tension and/or Compression)	68
6.1.4 Rigid	69
6.1.5 Engineered Element	69
6.1.6 Combined Materials	71
6.2 Representation of Water Pressures	72
6.2.1 Modelling of Water Pressures using DLO	72
6.2.2 Water Tables	73
6.2.3 Water Regimes	75
6.2.4 Water Pressure Hierarchy	76
6.3 Seismic Loading	76
6.3.1 Modelling of Seismic Loading Using DLO	76
6.3.2 Modelling of Water Pressures During Seismic Loading	76
6.4 Soil Reinforcement	77
7 Limit Analysis: Advantages and Limitations	79
7.1 Introduction	79
7.2 Simplicity	79
7.3 Stress States in Yielding and Non-yielding Zones	80
7.4 Yield Surface	80
7.5 Strain Compatibility	81
7.6 Non-associativity and Kinematic Constraints	82

III Modelling	85
8 Generic Principles	87
8.1 Model Definition and Solver	87
8.1.1 Model Definition	87
8.1.2 Solver Specification	87
8.2 Adequacy Factor and Factors of Safety	88
8.2.1 Introduction	88
8.2.2 Method 1 - Factor of Safety on Load(s)	90
8.2.3 Method 2 - Factor of Safety on Material Strength(s)	91
8.2.4 Method 3 - Factor of Safety on Ratio of Forces	94
8.2.5 Application of the Adequacy Factor on Load	96
8.2.6 Adequacy Factor Sensitivity	97
8.2.7 Adequacy Factor Direction	98
8.3 Use of Partial Factors	100
8.3.1 Introduction	100
8.3.2 Factoring of Actions (Loads)	101
8.3.3 Factoring of Material Properties	103
8.4 Solution Accuracy	103
8.4.1 Introduction	103
8.4.2 Benchmarking Results	103
8.4.3 Interfaces	104
8.4.4 Small Solid Areas	104
8.4.5 Singularities	105
8.4.6 Model Extent	105
8.4.7 Failure Mechanisms Dominated by Rotations	105
8.5 Adapting Plane Strain Results to 3D	105
8.6 Troubleshooting	107
8.6.1 Insoluble Problems	107
8.6.2 Troubleshooting Insoluble Problems	108
8.6.3 Problems Giving Solutions That Appear Incorrect	108
9 Example Problems	111
10 Slope Stability Problems	113
10.1 Principles of Slope Stability Limit Analysis	113
10.2 Factor of Safety on Strength	113
10.2.1 Slopes in Cohesionless Soils	114
10.2.2 Slopes in Cohesive Soils	115
10.3 Factor of Safety on Load	115
10.4 Unusual Failure Surfaces for Slopes in Cohesive Soils	116
11 Reinforced Soil	117
11.1 Introduction	117
11.2 Soil Reinforcement with Specified Interface Materials	118
11.3 Soil Reinforcement - Predefined Types	119
11.3.1 Predefined types	119
11.3.2 Soil Nail - general parameters	119
11.3.3 Soil Nail (Rigid)	120
11.3.4 Soil Nail (Flexible)	120
11.3.5 Soil Nail (Can Yield)	120
11.3.6 Sheet Pile Wall - general parameters	120

11.3.7 Sheet Pile Wall (Rigid)	121
11.3.8 Sheet Pile Wall (Can Yield)	121
11.3.9 Geotextile	121
11.3.10 Other	121
11.4 Modelling soil nails	122
11.5 Modelling flexible geosynthetic reinforcement	122
11.5.1 General principles	122
11.5.2 Geotextiles - Method 1	122
11.5.3 Geotextiles - Method 2	123
11.5.4 Geotextiles - Method 3	124
11.6 Anchor tendons	125
11.7 Example 1 - Simple Reinforcement Pullout	125
11.8 Example 2 - Simple Reinforcement Tensile Rupture	127
11.9 Example 3 - Simple Reinforcement Lateral Displacement	128
11.10 Example 4 - Simple Reinforcement Bending	129
12 Seismic Problems	131
12.1 Modelling of Seismic Loading Using DLO	131
12.2 Example 1 - Modelling a Simple Mechanics Problem	131
12.3 Example 2 - Checking the Mononobe-Okabe Equation	132
12.4 Example 3 - Correct use of Adequacy Direction	134
IV User Guide	137
13 Introduction to the User Guide	139
14 The Graphical Interface	141
14.1 Introduction	141
14.2 Title Bar	142
14.3 Menu Bar	142
14.4 Toolbars	143
14.5 Material Explorer	144
14.6 Water Regime Explorer	145
14.7 Viewer Pane	146
14.8 Property Editor	147
14.9 Geometry Editor	148
14.10 Output Pane	148
14.11 Status Bar	149
14.12 Vertex, Boundary and Solid Explorers	150
14.12.1 Introduction	150
14.12.2 Vertex Explorer	150
14.12.3 Boundary Explorer	150
14.12.4 Solid Explorer	151
14.13 Calculator	151
15 Specifying the Problem Geometry	153
15.1 Terminology	153
15.2 Setting the Units	153
15.3 Starting with an Empty Project	154
15.3.1 Project Details	155
15.3.2 Draw Settings	155

15.4 Starting with a DXF Imported Geometry	157
15.4.1 Permitted DXF Commands	159
15.4.2 Tips for Modifying a DXF in an External CAD Package	159
15.4.3 Permitted DXF Types	160
15.4.4 Following DXF Import	160
15.4.5 Export to DXF	160
15.5 Construction Lines	161
15.5.1 Horizontal Construction Lines	161
15.5.2 Vertical Construction Lines	161
15.5.3 Custom Construction Lines	161
15.6 Drawing Functions	162
15.6.1 Rectangle	162
15.6.2 Polygon	162
15.6.3 Line	162
15.6.4 Vertex	162
15.7 Selecting Objects	163
15.7.1 Single Click Selection	163
15.7.2 Rectangle Selection	163
15.7.3 Multiple Selection	164
15.8 Snapping to Other Objects	164
15.9 Modifying the Geometry	164
15.9.1 Using the Mouse	164
15.9.2 Connecting Geometry Objects	165
15.9.3 Overlapping Geometry Objects	166
15.9.4 Changing the End Vertex of a Boundary	167
15.9.5 Creating One Object Inside Another	167
15.9.6 Using the Geometry Editor	168
15.10 Undo/Redo	168
15.11 Editing Geometry Object Properties	168
16 Setting the Analysis Mode	171
16.1 Short and Long Term Stability	171
16.2 Translational and Rotational Solution Mode	172
16.3 Factor Load(s) vs Factor Strength(s)	172
16.3.1 Changing Analysis Type	172
16.3.2 Factor Strength(s) Settings	172
17 Setting Material Properties	175
17.1 Material Types	175
17.1.1 Converting Yield Stress to Cohesion	176
17.2 Assigning a Material to a Solid or a Boundary	177
17.3 Post-Solve Diagrams	178
17.4 Standard Mohr-Coulomb Material	179
17.4.1 Setting Soil/Material Strength Properties	179
17.4.2 Setting Short Term/Long Term Behaviour	181
17.4.3 Mohr-Coulomb Material with Linear Variation of Strength with Depth	181
17.4.4 Mohr-Coulomb Material with Spatial Variation of Strength	183
17.5 Derived Mohr-Coulomb Material	185
17.6 Cutoff Material	186
17.7 Rigid	187
17.8 Setting Soil/Material Unit Weight (Weight Density)	187

17.9 Engineered Element	188
17.9.1 Introduction	188
17.9.2 Defining Engineered Element Geometry	190
17.9.3 Post Solve Information	192
17.10 Creating and Deleting User Defined Materials	194
17.10.1 Materials Explorer Context Menu	194
17.10.2 Creating a New Material	194
17.10.3 Creating a Duplicate Material	196
17.10.4 Deleting a Material	196
17.11 Exporting and Importing Materials	197
17.12 Previewing Strength Distributions	197
18 Setting Boundary Conditions	199
19 Setting Applied Loads	201
19.1 Specifying Boundary Loads	201
19.2 Specifying Self Weight Loading	202
19.3 Adequacy Factors	202
19.4 Favourable/Unfavourable Setting	203
20 Setting Water Pressures	205
20.1 Introduction	205
20.2 Enabling and Disabling Water Pressures	205
20.3 Drawing a Water Table	206
20.4 Use of an r_u Value	207
20.5 Localized Water Pressures (Regimes)	208
20.5.1 Defining a New Water Regime	209
20.5.2 Creating a Duplicate Water Regime	214
20.5.3 Deleting a Water Regime	214
20.5.4 Exporting and Importing Water Regimes	214
20.5.5 Viewing Water Regime Affected Objects	214
20.6 Previewing Modelled Water Pressures	214
21 Setting Seismic Parameters	217
22 Analysis	219
22.1 Overview	219
22.2 The Solver	219
22.2.1 Pre-Solve Checks and the Diagnostics Tool	220
22.3 Analysis Settings	224
22.3.1 Overview	224
22.3.2 Setting and Previewing the Nodes	225
22.3.3 Setting Nodal Distribution within Geometry Objects	225
22.3.4 Optimizing Nodal Layout	226
22.4 Solving	226
22.5 Analysis Results	226
22.5.1 Collapse Adequacy Factor Found	226
22.5.2 No Solution Found	227
22.5.3 Aborting an Analysis	229
22.5.4 Lock and Unlock	229

23 Post-Analysis Functions	231
23.1 Adequacy factor	231
23.2 Animation	231
23.3 Slip-lines	233
23.4 Pressure Distributions around Blocks	234
23.4.1 Post-Solve Display Dialog	234
23.4.2 Block Context Menu	235
23.5 Force Distributions along Engineered Elements	236
23.5.1 Post-Solve Display Dialog	238
23.5.2 Engineered Element Context Menu	238
24 Report Output	241
24.1 Viewing the Report Output	241
24.2 Saving the Report Output	243
24.3 Customizing the Header or Footer	243
25 Exporting Graphical Output	245
25.1 Overview	245
25.2 Geometry Export	245
25.3 Image Export	246
25.4 Animation Export	247
25.5 Animations and Multiple Scenario Problems	248
26 Opening and Saving Projects	249
26.1 Opening and Saving Projects	249
26.2 Auto-recovery Files	249
27 Thumbnails, Tags and Searching	251
27.1 Thumbnails	251
27.2 Tags and Searching	252
27.2.1 Tags	252
27.2.2 Searching	252
28 Preferences	255
28.1 General	255
28.2 Units	256
28.3 Startup	256
28.4 Report	257
28.5 Solve	258
28.6 Export	260
29 License Information Dialog	261
30 Scenario Manager and Partial Factors	263
30.1 Introduction	263
30.2 Specification of Partial Factors for a Single Scenario	264
30.3 Specification of Long and Short Term Analysis Mode	265
30.4 Specifying Multiple Scenarios	265
30.5 Using the Partial Factor Set Manager	265
30.6 Exporting and Importing Scenarios	266
30.7 Solving with Multiple Scenarios	266

V Appendices	269
A Verification	271
A.1 Verification Tests	271
A.2 Academic Papers	271
B Menu and Toolbar Reference	273
B.1 General	273
B.1.1 Scrollbars	273
B.1.2 Current Mouse Position	273
B.1.3 Scrolling Wheels	273
B.2 Menus	274
B.2.1 File Menu	274
B.2.2 Edit Menu	275
B.2.3 Select Menu	275
B.2.4 View Menu	276
B.2.5 Draw Menu	278
B.2.6 Tools Menu	278
B.2.7 Analysis Menu	279
B.2.8 Help Menu	280
B.3 Toolbars	281
B.3.1 Default Toolbars	281
B.3.2 Optional Toolbars	282
B.4 Context Menus	283
B.4.1 Viewer Pane Context Menu	283
B.4.2 Toolbar / Property Editor Context Menu	284
B.4.3 Geometry Object Explorer Context Menu	285
C Accessing Example Files	287
D Derivation of Theory	289
D.1 Work Done by Cohesion by Rotation of a Log Spiral	289
E Interpolated Grid	291
E.1 Grid Square Interpolation	291
F Command-Line Interface	293
F.1 Introduction	293
F.2 License Requirements	293
F.3 File Format	294
F.3.1 Uncompressing Files	294
F.4 Input	294
F.5 Units	295
F.6 Output	295
F.7 Running	297
F.8 Help	297
F.9 Syntax	297
F.9.1 Options Syntax	298
F.9.2 Object Keys	299
F.10 Properties	301
F.10.1 Project Properties	301
F.10.2 Scenario Properties	301

F.10.3	Material Properties	302
F.10.4	Water Regime Properties	304
F.10.5	Boundary Object Properties	305
F.10.6	Solid Object Properties	306
F.10.7	Examples	307
F.11	Creating and Running a Batch File	308
G	Benchmark Solutions	309
G.1	Introduction	309
G.2	Simple Soil Nail in Retaining Wall (Undrained)	309
H	Frequently Asked Questions	311
	Bibliography	315

Part I

Introduction and Quickstart

Chapter 1

Introduction

1.1 General Description

LimitState:GEO is a general purpose software program which is designed to rapidly analyse the ultimate limit state (or ‘collapse state’) for a wide variety of geotechnical problems.

The software can be used to model 2D problems of any geometry specified by the user (including slopes, retaining walls, foundations, pipelines, tunnels, anchors etc. and any combination of these).

It directly determines the ultimate limit state (ULS) using the computational limit analysis technique Discontinuity Layout Optimization (DLO, see Section 5.3), and is designed to work with modern design codes such as Eurocode 7 by providing full support for partial factors and the ability to solve multiple scenarios.

1.2 Program Features

LimitState:GEO is designed to be general, fast and easy to use. The main features of LimitState:GEO are summarized below:

- LimitState:GEO utilizes Discontinuity Layout Optimization (DLO) to directly identify the critical collapse mechanism. The DLO procedure effectively relies on the familiar ‘mechanism’ method of analysis originally pioneered several centuries ago by workers such as [Coulomb \(1776\)](#), but posed in a modified form to allow modern-day computational power to be applied to the problem of finding the critical solution from billions of possibilities.
- The **solution** is presented as an ‘adequacy factor’ (applied to one or more loads or material strengths in the problem). This in effect allows the software to determine either a factor on load or a factor on strength for any problem. The solution is also displayed visually as a failure mechanism involving a number of blocks which will slide and/or rotate relative to one another. To facilitate rapid interpretation of the mode of response the failure mechanism can be animated. The distribution of stresses around the edges of any con-

stituent block can also be displayed. In addition the variation of tensile force, shear force and bending moment can be displayed along reinforcing elements or sheet pile walls.

- Many types of problems can be solved, including those involving slopes, foundations, gravity walls, soil reinforcement and any combination of these. The **problem geometry** can be specified using Wizards for common problems or alternatively by:
 - drawing the geometry on-screen using the mouse,
 - importing from an AUTOCAD DXF file.

The geometry can subsequently be edited using the mouse or by editing coordinates.

- The following **Materials** are available:
 - Mohr-Coulomb (strength defined in terms of cohesion intercept and angle of shearing resistance). A linear variation of undrained shear strength c_u with depth may also be specified.
 - Rigid (i.e. no yield conditions are applied).
 - Cutoff (allows modelling of a tension and/or compression cutoff).
 - Engineered element (e.g. for modelling a soil nail or sheet pile wall).
- A wide range of **Wizards** are provided to permit rapid modelling of common geotechnical problems, including slope stability, retaining wall, and foundation problems.
- The user may switch between **long term** and **short term** (drained and undrained) analysis modes by changing a single setting.
- The **ground water** conditions may be defined using:
 - a water level (phreatic surface) with hydrostatic pore pressures,
 - an overall value of the pore pressure coefficient r_u .
 - constant potential or constant pressure
 - a spatially varying function based on user defined grid values
- **Soil reinforcement/Engineered elements** and **pre-existing slip surfaces** may be modelled at any location within the geometry.
- **Seismic** loadings may be modelled using the pseudo static method by specifying **Horizontal** and **Vertical** accelerations.
- Choice of **Translational** or **Translational and Rotational** analysis modes.
- LimitState:GEO provides extensive support to users wishing to use **Partial Factors**:
 - user specified partial factor sets can be defined,
 - commonly used partial factor sets are built-in (e.g. Eurocode 7 Design Approach 1 partial factor sets),
 - different partial factors can be defined for permanent, variable, accidental and favourable / unfavourable loads (see Section 19),
 - automatic post-analysis checks are performed to verify that Favourable/Unfavourable load specifications are in fact correct.

- LimitState:GEO provides a comprehensive and easy to use GUI interface with fully selectable geometry objects, a Property Editor, Materials Explorer and drag and drop facilities. It also provides users with full Undo/Redo facilities and ability to recover lost work via an auto-saved recovery file.
- A comprehensive **Report** can be generated, with user control over what is included, and the unique ability to output free body diagrams for all blocks of material identified in the critical failure mechanism, with equilibrium equations to permit easy hand validation (see Section 24).
- Comprehensive guidance within the program in the form of messages, warnings and text descriptions. Where appropriate these are hyperlinked direct to relevant sections within the online help file.
- Choice of working in Metric or Imperial units.

1.3 LimitState:GEO Terminology

LimitState:GEO is designed to rapidly identify the critical failure mechanism in any geotechnical stability analysis problem. The annotated image in Figure 1.1 highlights the most important objects the user will encounter when using LimitState:GEO.

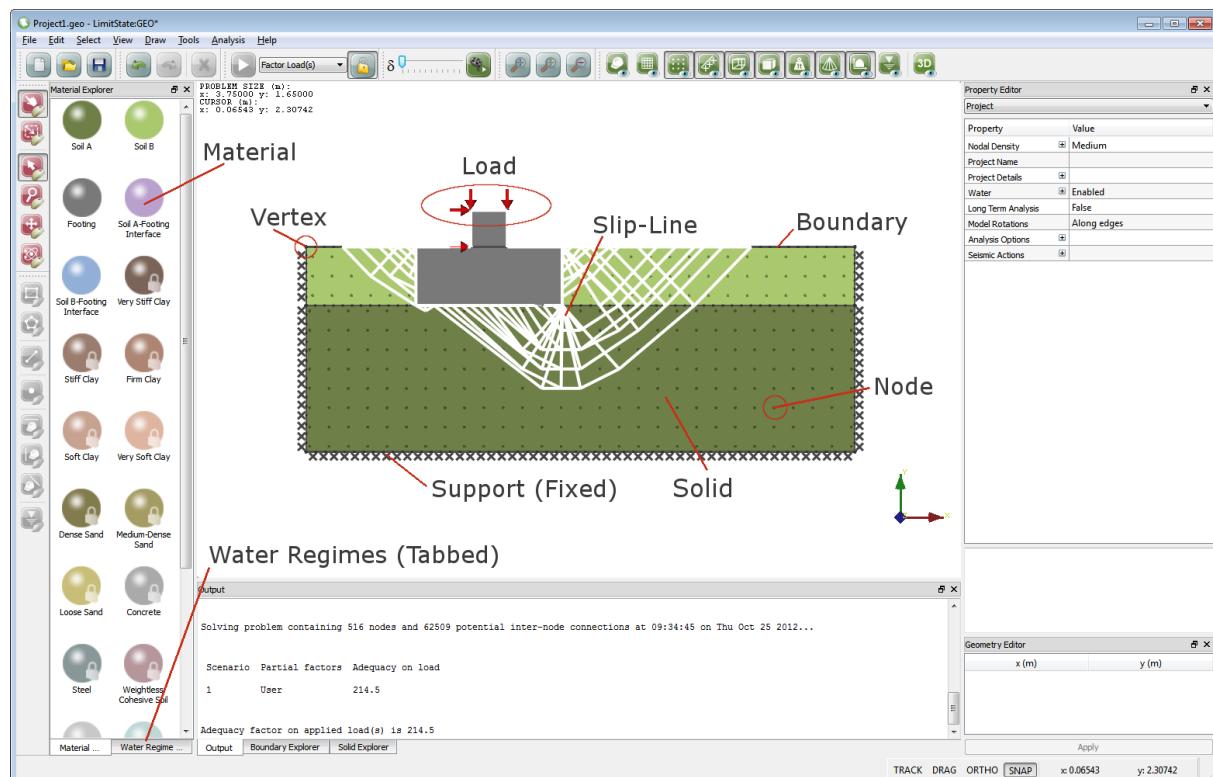


Figure 1.1: The main objects encountered in LimitState:GEO

1.4 About LimitState

[LimitState Ltd](#) was spun out from the [University of Sheffield](#) in 2006 to develop and market cutting edge ultimate analysis and design software for engineering professionals. These include the LimitState:RING, LimitState:GEO, and LimitState:YIELD products, with applications in the structural, geotechnical and mechanical engineering sectors. Our aim is to be a world leading supplier of computational limit analysis and design software. LimitState maintains close links with the University of Sheffield, enabling it to draw on and rapidly implement the latest innovations in numerical and theoretical limit state analysis.

1.5 Using Help

Pressing F1 at any time, or pressing the **Help** button, gives users access to the online help facility, providing users with a convenient means of accessing material contained within the manual whilst using the software. The software also includes hyperlinks which link directly to relevant parts of the online help material (e.g. from within messages, dialogs and text descriptions), to provide users with rapid access to relevant explanatory material.

1.6 System Requirements

LimitState:GEO runs on the Windows 7, 8.1 and 10 operating systems (support for Mac OSX and Linux operating systems is available on request, subject to demand). Recommended minimum system specifications are as follows:

- Intel or compatible processor (i3 or better recommended)
- 200 MB free hard disk space
- 4 GB RAM (8 GB recommended)

Note that the use of emulated platforms such as VMware is not supported.

1.7 Program Limits

The program uses a ‘Single Document Interface’ which means that one project file can be open in LimitState:GEO at any given time. However, several instances of LimitState:GEO can be opened simultaneously if required and each of these may contain a separate project file.

When using LimitState:GEO with a ‘full’ license, problem size is limited only by available computer power.

1.8 Contact Details

1.8.1 Sales

To request information on pricing, a formal quotation, or to purchase the software please contact [LimitState](mailto:sales@limitstate.com), at sales@limitstate.com.

1.8.2 Software Support

Software support for LimitState:GEO is available to all users with a valid support and maintenance contract. Additionally we are happy to help users with time-limited ‘trial’ or ‘evaluation’ licenses. All queries should be directed to support@limitstate.com.

1.8.3 Website

For the most up-to-date news about LimitState:GEO, please visit the LimitState:GEO website: limitstate.com/geo.

Chapter 2

Getting Started

2.1 Installation and Licensing

Full details regarding installation and licensing issues are provided in the '**LimitState:GEO Installation and Licensing Guide**'.

2.1.1 Thumbnails, Tags and File Search

For users running Windows 7 or above, the ability to include thumbnails and tags with the saved project is available¹. Careful use of these features allows different LimitState:GEO files to be categorized and later searched for on a computer, without the need to open and inspect each one individually. Additionally, other properties from the **Project details** dialog can be searched if the system properties are set appropriately.

2.1.1.1 Enabling Thumbnails

To allow a system to take advantage of the thumbnail feature it may be necessary to set it to display these instead of the program icon. To do this:

1. Open **Control Panel** on the PC.
2. Select **Folder Options**.
3. Select the **View** tab.
4. Ensure that **Always show icons, never thumbnails** is NOT selected.
5. Click OK.

For further information see Chapter 27. Note that uninstalling LimitState:GEO may require a reboot to remove the thumbnails feature from Windows registry.

¹For Windows Vista users, thumbnails may be available depending on the system configuration.

2.1.1.2 Enabling File Property Searching

To allow a system to search within the **Project details** it may be necessary to change the default search settings. To do this:

1. Open **Control Panel** on the PC.
2. Select **Folder Options**.
3. Select the **Search** tab.
4. Ensure that **Always search file names and contents...** IS selected.
5. Click OK.

For further information see Chapter 27. Note that searching for file properties in a folder that is included in the Windows "Index" may not work at first. This is because the system requires time to register the file properties in the index and, during this period, will not return matches. The expected behaviour will return once indexing has occurred.

2.2 Starting LimitState:GEO

To start LimitState:GEO, on the **Start** menu, point to **All Programs**, then click on LimitState:GEO. On starting LimitState:GEO the following welcome screen should appear (Figure 2.1)



Figure 2.1: LimitState:GEO welcome screen

You then have three options:

1. **Create a new project** - select this option and click **OK** to bring up the **New Project Dialog**. You may then select either an **Empty project** (which provides you with full flexibility to define the geometry of your problem), or one of the many application-specific predefined projects (e.g. the **Simple Footing Project**), each of which will activate a wizard to guide you through the process of specifying your problem.
2. **Open an existing project** - select this option and click **OK** to display the open file dialog.
3. **Open a recently accessed project** - select this option, choose a file from the list and click **OK** to return to a recent project.

2.3 Guidance Available in this Manual

Guidance is available as follows:

1. For a quickstart introduction to LimitState:GEO refer to Chapter 3.
2. For an explanation of how to use the Wizards to rapidly set up and solve common problems refer to Chapter 4.
3. For a description of the theory underlying the solutions generated by LimitState:GEO refer to the Theory chapters in Part II. It is assumed that users are fully familiar with the advantages and limitations of limit analysis which is the basis of the analysis technique utilised by LimitState:GEO. A short summary of key points is given in Chapter 7.
4. For guidance on using LimitState:GEO to model a range of problem types refer to the Modelling chapters in Part III.
5. For full instructions on use of the program user interface and description of its features refer to the User Guide in Part IV.

Chapter 3

Quick Start Tutorial

3.1 Introduction

This chapter gives an overview of the capabilities of LimitState:GEO. It is recommended reading for new users and is designed to give users the confidence to subsequently make use of some of the more sophisticated features of the program.

However, for the sake of brevity some important issues are not discussed in this chapter and the reader is referred to the Modelling Guide (Part III) and User Guide (Part IV) for fuller information.

Note that for sake of simplicity all examples in this tutorial involve undrained (cohesion only) problems.

3.2 Getting Started

It is assumed that the user is starting from the **Welcome to LimitState:GEO** dialog.

- ▷ Select **Create a new project** and click **OK** to bring up the New Project dialog (see Figure 3.1).

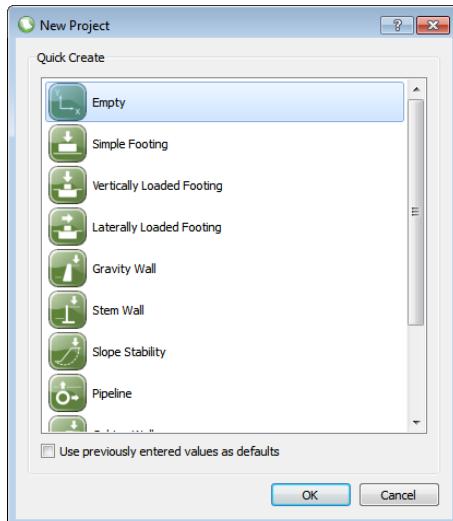


Figure 3.1: The New Project dialog

If **Cancel** is selected, the user is free to define their own problem geometry. However it is easier to learn how the program works by initially using one of the predefined problem wizards. The wizards permit rapid definition of common problem geometries. The geometry can be easily amended subsequently.

▷ Select **Simple Footing** and click **OK**.

The wizard then appears. Project data is entered in six stages as follows (the icons in the navigation bar on the left hand side of the wizard will be highlighted during each stage of the problem definition):

Project Background details to the project may be entered here. Click **Next** to accept defaults.

Geometry The problem geometry may be amended here. Click **Next** to accept the defaults.

Materials Material properties may be entered here. Note the default value of undrained shear strength (c_u) for the soil of 50.0 kN/m². Click **Next** twice to accept defaults.

Loads Loads may be entered here. Note the default value, Q_V (the applied line load on the foundation) is set to a unit value of 1.0 kN/m. Change this to 200.0 kN/m. Click **Next** to accept this.

Scenarios Scenarios describe the different partial factor sets that are applied during analysis and whether each is a *short* or *long term analysis*. The partial factors, which are treated by LimitState:GEO as multipliers on applied loads or divisors on material properties, may be modified to suit different design codes. Click **Next** to accept the default **User** factors, which are all set to unity.

Analysis The type of analysis and the solution accuracy (defined by the **Nodal Density**) may be modified here. Click **Finish** to accept the default settings, which specify that the analysis determines a *Factor on Loads* and uses a *Medium* density of nodes (500).

The problem geometry will be displayed in the main viewer (see Figure 3.2).

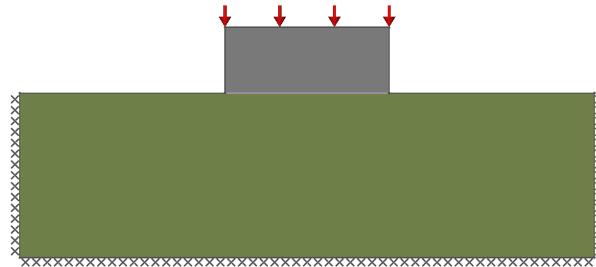


Figure 3.2: Geometry generated by the Simple Footing Wizard. Note that all visual objects are fully interactive, and their geometry can be changed by selecting these with the mouse.

3.3 Analysis Modes and the Adequacy Factor

Following an analysis the critical collapse **Adequacy factor** will be displayed in the **Output** pane. The **Adequacy factor** is the factor by which specified loads must be increased, or material strengths decreased, in order for the system under consideration to reach a collapse state. There are two types of **Adequacy factor** that can be reported:

- **Adequacy factor** on load
- **Adequacy factor** on soil strength

This quickstart starts with a number of examples that report the **Adequacy factor** for loads before moving on to examples where material strength is factored.

Further information regarding the **Adequacy factor** and different modes of analysis is available in Section 8.2.

3.4 Solving a Problem

With the default values set in the wizard, the defined problem for a short term (undrained) analysis is equivalent to the simple problem of a footing of width $B = 1.0\text{m}$ on a semi-infinite layer of weightless cohesive soil of undrained shear strength $c_u = 50.0 \text{ kN/m}^2$.

While it is not possible to model a semi-infinite layer of soil, it is sufficient to model a finite size layer providing it is large enough (generally the critical mechanism should not touch a fixed boundary unless it is known that failure is unlikely to occur beyond this point).

To determine whether the specified load of 200.0 kN/m^2 can be carried by the foundation,

▷ either use the menu: Analysis / Solve, click on the icon or press F5.

Upon **Solve**, LimitState:GEO will carry out a number of pre-solve diagnostic checks to ensure that the model geometry definition is valid. Errors or settings that may lead to invalid or non-credible solutions will be displayed in the **Diagnostics window** (see Section 22.2.1).

The program will first display a series of nodes, superimposed on the geometry objects in the problem, and will then rapidly try out all possible combinations of slip-lines interconnecting the nodes to find the optimum solution. The program will gradually refine the failure mechanism until an optimal slip-line mechanism is found (see Figure 3.3), together with an associated **Adequacy factor** (margin of safety) on the specified load.

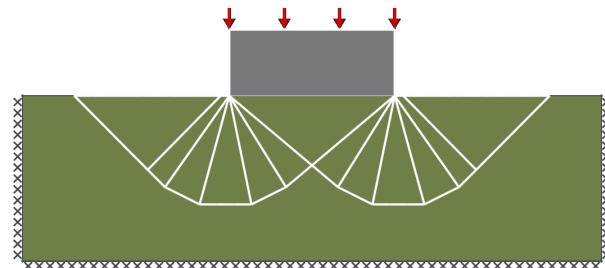


Figure 3.3: Simple Footing problem displayed in the main viewer (after analysis and before animation)

Note that occasionally a problem can be found where several equally critical failure mechanisms exist. The **Simple Footing** is one such problem and a slip line mechanism involving the right or left side, or both, can be returned. However the numerical solution remains the same in all cases.

With the specified parameters, an **Adequacy factor** of 1.301 should be obtained (displayed in the **Output** window at the bottom of the screen). This means that the foundation is safe against collapse by a factor of 1.301. The actual load that would cause failure is $200 \times 1.301 = 260.2$ kN/m.

In general,

Adequacy factor > 1.0 The problem is safe against collapse

Adequacy factor < 1.0 The problem is unsafe against collapse.

However, note that there is not necessarily any margin of safety in the above definition.

In order to analyse the ultimate limit state, at least one load in the problem must be increased until the collapse state is reached. This may be an applied load or the self-weight of a solid body. In order to indicate to LimitState:GEO which load is to be increased, the **Adequacy** property must be set for at least one load.

For the **Simple Footing Project**, the wizard automatically sets the **Adequacy** property for the load applied to the footing. The returned **Adequacy factor** is the factor by which that load must be increased in order to cause collapse. Further details may be displayed by clicking on the displayed **Adequacy factor**. This displays the dialog shown in Figure 3.4. For further information on the **Adequacy factor** and its usage, see Section 8.2.

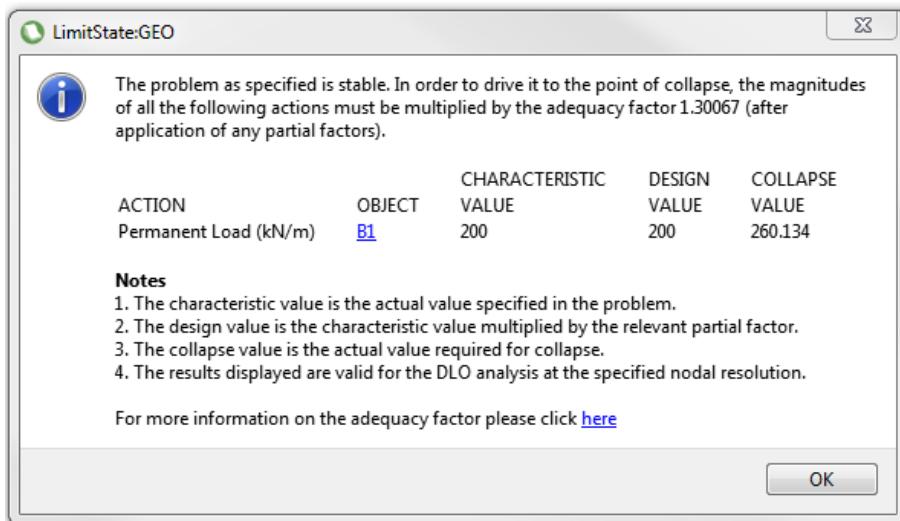


Figure 3.4: Detailed information on the solution relating the adequacy factor to applied loadings.

This example problem is the well known Prandtl punch solution for which the true analytical collapse load Q_{ULS} is given by:

$$\frac{Q_{ULS}}{B} = (2 + \pi)c_u + q \quad (3.1)$$

where B is the width of the footing, c_u is the undrained shear strength, and q is the surcharge load on the adjacent soil surface. Note that the solution is unaffected by the soil self weight.

The adequacy factor F is given by:

$$F = Q_{ULS}/Q_d \quad (3.2)$$

where Q_d is the specified design load.

In this example, with $q = 0$, the theoretical adequacy factor is 1.285. The overestimate by LimitState:GEO is thus $\sim 1.2\%$, utilizing the default *Medium* nodal refinement.

3.5 Viewing Mechanism Deformation

By default the software will automatically animate the solution after solve (see Figure 3.5), by magnifying the instantaneous displacements at failure. Note that though the solution is strictly only valid for infinitesimal displacements, large displacements are displayed to assist visualization of the collapse mode. For direct control over magnification of the mechanism

displacements, the slider bar  can also be used. The Play Animation  button can be clicked to replay the animation.

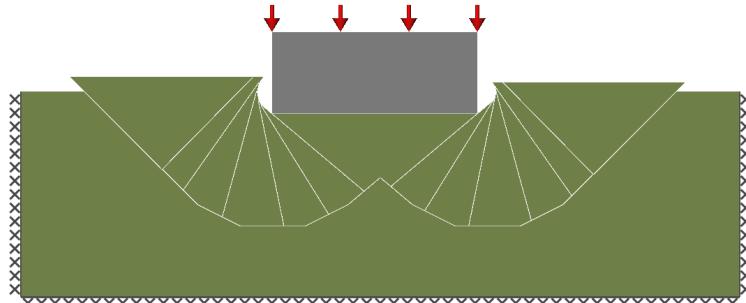


Figure 3.5: Deformation of the Simple Footing problem (after deformation)

3.6 Viewing Stresses and Bending Moments

After the software has solved a problem, the user is able to select any of the **Solids** (rigid block identified in the collapse mechanism) or **Engineered Elements** (e.g. soil reinforcement) in the failure mechanism (by clicking with the mouse). (Note that Engineered Elements are not further discussed or used in the examples in this Quick Guide chapter. The theory behind **Engineered Elements** is discussed in Section 6.1.5. The practical use of **Engineered Elements** in LimitState:GEO is discussed in Chapter 11).

For **Solids**, clicking will display a diagram of the *Normal Stress* (default) or *Shear stress* acting upon that object. To switch between types, right-click on the solid to bring up the context menu and then select the desired diagram type.

For **Engineered Elements**, clicking will display the default diagram type specified in the corresponding material property. *Tensile Force*, *Shear Force* and *Bending Moment* can all be displayed. To switch between types, right-click on the appropriate boundary element to bring up the context menu and select the desired diagram type.

In all cases, hovering over part of the diagram will display the magnitude of these stresses next to the mouse cursor (see Figure 3.6). For further information on displaying boundary stresses, see Section 23.4.

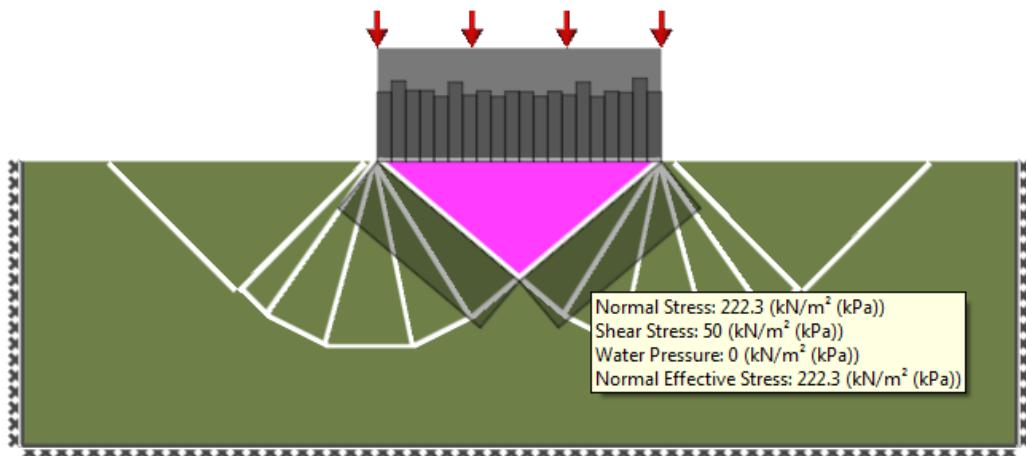


Figure 3.6: Viewing normal stress information after solving, plotted as a bar chart, together with specific values for a selected bar.

3.7 Zooming In and Out

To zoom in and out, use the magnifying glass toolbar buttons (Zoom In , Zoom Out , Zoom All). If a scroll wheel mouse is being used, the wheel can be used to zoom in and out. Alternatively the keyboard buttons +, - and * will zoom in, out and to the extents of the problem geometry respectively.

Note that with the mouse, zooming takes place centred on the current position of the mouse pointer (the Select button must be on for this feature to work). Zoom All resizes the image to display optimally in the viewer. This is useful if the image has become too large or small.

3.8 Pan, Rotate and 3D Views

To pan, select the cross arrows toolbar button and hold the left mouse button while dragging inside the **Viewer Pane**. Alternatively, if a scroll wheel / 3 button mouse is being used, click and hold the central button. The arrow keys on the keyboard will also change the view. Note that panning takes place centred on the current position of the mouse pointer.

To rotate the view in 3D, select the rotate toolbar button and hold the left mouse button whilst moving the cursor in the **Viewer Pane**. To return the view back to the default, right-click in the **Viewer Pane** to bring up the context menu. From here, select **View >Front** to return the display to the default viewing angle. Using the context menu, or the **View 3D** toolbar (Appendix B.3.2), it is also possible to snap the viewer to a number of pre-defined 3D viewing angles. These can be particularly effective when examining the solved problem.

3.9 Trying Different Problems and Parameter Sets

Perhaps the simplest way to vary problem parameters is to simply re-run the wizard (File>New or ) and enter different parameters; alternatively a different wizard can be tried.

However, the current problem may be modified in any way required after a wizard has been run. Making use of this capability will give the user a good feel for what the software is capable of, and how problems can be created without using the wizards.

Note that if there is a previously solved problem, the Unlock icon  must be clicked to allow modification of any of the parameters (this is to prevent inadvertent alterations being made once a solution has been obtained).

3.10 The Property Editor

The **Property Editor (PE)**, displayed on the right hand side of the display, allows the user to quickly read and / or modify the attributes of one or more objects within the current project. Figure 3.7 shows the typical parameters displayed in the Property Editor when a **Material** is selected.

Property Editor	
Property	Value
ID	Soil A
Type	Mohr-Coulomb
Color	
c' (kN/m ² (kPa))	0
ϕ (degrees)	30
c _u (kN/m ² (kPa))	50
Unit Weight (kN/m ³)	20
Sat. Unit Weight (kN/m ³)	20
Drainage Behaviour	Always undrained
Advanced	
c _u Gradient (kN/m ² (kPa)/m depth)	0
Datum c _u Elevation (m)	0
Override Strength	False
Post-Solve Display	Click: Shear

Figure 3.7: Typical parameters displayed in the LimitState:GEO Property Editor when a **Material** object is selected.

The **Property Editor** is visible by default when LimitState:GEO is started for the first time, but can be hidden and shown using the **View** menu described in Appendix B.2.4.

Some of the functions in LimitState:GEO are only accessible via the Property Editor. These are described in more detail in this section. Other functions and attributes can be accessed and modified elsewhere, but are shown for convenience in the Property Editor.

Generally when an object is selected on the screen it will be highlighted and its properties will be displayed in the **PE** - where they can be viewed or edited as required. Single clicking on any item in the Property column of the **PE** gives an expanded explanation of the parameter in the window at the base of the **PE**. A  sign next to an item in the **PE** indicates that there are additional sub-parameters relating to that item that may be viewed. Click on the  sign to

access these. Clicking on a value in the **PE** allows it to be modified by typing or selecting the required choice, unless it is a read only value or the project is *Locked*.

3.10.1 Changing Material Properties

By default the **Materials Explorer** is located on the left hand side of the screen. This contains a list of available material types. Click on any icon to view the properties of the material in the **Property Editor**. The soil material used in the Simple Footing Project is called ‘Footing Soil’. When selected, its (default) properties are displayed as follows:

- Shear strength parameters:
 - Drained cohesion, $c' = 0 \text{ kN/m}^2$
 - Drained phi, $\phi' = 30^\circ$
 - Undrained shear strength, $c_u = 50 \text{ kN/m}^2$
- Unit weights (weight densities):
 - Unit weight, $\gamma = 20.0 \text{ kN/m}^3$
 - Unit weight (saturated), $\gamma_{sat} = 20.0 \text{ kN/m}^3$.

Change the value of the Undrained cohesion, c_u to 75.0 kN/m^2 (click on the box with the value 50.0 in it, enter the new value then press Enter or click elsewhere in the **Property Editor** to accept the value) and then Solve again. A new solution of **Adequacy factor** = 1.951 (approximately 1.5 times the previous solution discussed in Section 3.4) should be obtained.

Now try changing the unit weight to 10 kN/m^3 and click **Solve** (remember to click **Unlock** first). The solution should remain as 1.951. Self weight does not affect bearing capacity problems involving cohesion only when the soil surface is horizontal.

3.10.2 Changing the Material Used in a Soil Layer (Solid Object)

Unlock and click anywhere in the soil mass below the footing in the viewer frame (to use single selection, ensure that toolbar buttons  and  are set to on).

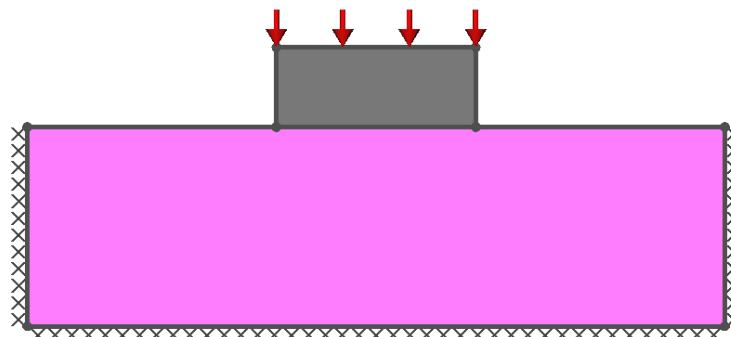


Figure 3.8: Problem geometry with lower soil body highlighted following selection

Once selected with the mouse the **Solid Object** representing the soil body or layer will be highlighted in pink (as shown in Figure 3.8), and its properties displayed in the **Property Editor**. To change the allocated material three options are available:

1. If a material defined by the user (e.g. in a wizard) has been selected, its properties may be freely edited directly in the **Property Editor** in the same way as described in Section 3.10.1. (It is necessary to click the sign next to the caption 'Footing soil' in the Property column of the **Property Editor** to access the material properties.)
2. If it is necessary to allocate an already defined material to the **Solid Object** representing the soil layer, 'drag and drop' may be used. Using the mouse, drag a material from the **Materials Explorer** onto the **Solid Object** representing the soil layer. A dialog box will be displayed and the user asked whether the user wished to **Add** to the material(s) already present, or **Replace** them.

Click **Replace** and the new material will be used in place of the previous material. The colour of the solid object should also change to reflect the change in material (this will not be seen while the soil mass is still selected). Try replacing the default soil with *Very Stiff Clay*. After solving, the same failure mechanism as before should be obtained, but an adequacy factor of 3.902 will be obtained (this is 3.0 times the original value obtained with $c_u = 50 \text{ kN/m}^2$ since the cohesion of the *Very Stiff Clay* is 150 kN/m^2).

3. The third option, which is an alternative to drag and drop, is to click on the **Value** cell in the **Materials** row in the **Property Editor** (this should read *1 Material assigned...*). A **Change** button will appear. Click this and the **Edit Object Material(s)** dialog will allow the selection of a material (or set of materials) that can be used in the zone.

Note that if the material was changed, prior to solving the problem, a message 'Diagnostic information messages: [1 warning\(s\)](#)' is displayed. Clicking on the hyperlink opens the Diagnostics dialog shown in Figure 3.9. The warning indicates that the interface material assigned to the base of the footing may be inconsistent with the soil in the adjacent solid body. (In this case, the interface material *Footing Soil - Footing Interface* was assigned a value that would always be 0.5 of its parent material, the original *Footing Soil*. Having changed the material below the footing to e.g. *Very Stiff Clay*, the material *Footing Soil - Footing Interface* may no longer provide correct interface material properties. Hence the warning.) However for the purposes of this QuickStart tutorial the value will be left unchanged.

An important point here is that the Diagnostics facility provides a number of useful items of information and warnings that are well worth examining prior to running the model and can help identify potential issues in the set up of the model. The Diagnostics dialog can be displayed at any time from the main menu, by selecting Analysis - Diagnostics...

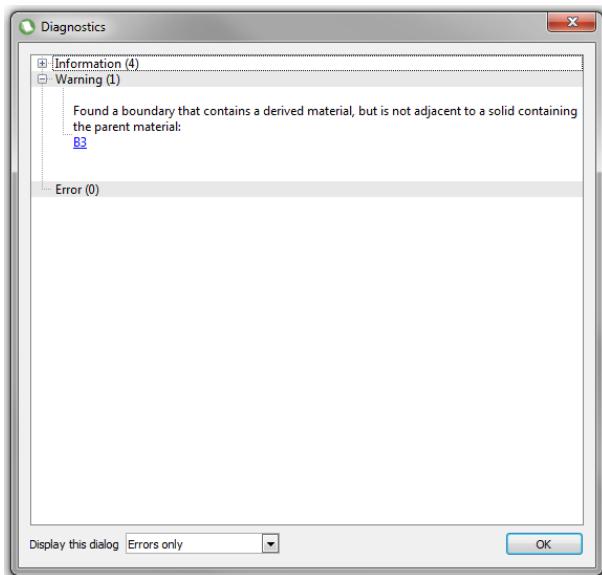


Figure 3.9: Pre-solve diagnostic displayed after changing the material in the main soil body for the Simple Footing problem.

Note also that the system defined material properties (indicated by a padlock symbol on the material:) are read only and may not be changed. User editable copies may however be made or new materials created (see Section 17.10).

3.10.3 Changing the Material used in an Interface (Boundary Object)

As well as being able to assign material types to a **Solid Object** (soil layer), it is also possible to add materials to a boundary or interface (termed a **Boundary Object**). With a few exceptions (e.g. foundation / soil interfaces or elements such as soil nails) the wizards will assign no materials to boundaries by default.

Material properties assigned to a **Boundary Object** are often related to the material allocated to an adjacent **Solid Object**. Thus if the material allocated to the **Solid Object** is changed, it is usually necessary to change the material in the adjacent boundary (if set), otherwise an unexpected solution may be obtained. For example, **Unlock** the project and drag *Very soft clay* onto the bottom boundary of the soil (whilst retaining *Very stiff clay* in the main soil layer). The boundary should be highlighted in pink when the mouse is exactly over it, and will change colour to the material colour if the material has been successfully assigned to it. After solving, you will see that the solution changes to utilize this weak layer of soil - the mechanism is attracted to the base of the soil (see Figure 3.10) and a lower adequacy factor of 3.546 is obtained. To remove this material from the line, click **Unlock**, select the boundary line and in the **Property Editor** click the cell opposite **Materials** to show the **Change** button. Click this button, uncheck *Very Soft Clay* in the **Edit Zone Material(s)** dialog and click **OK**.

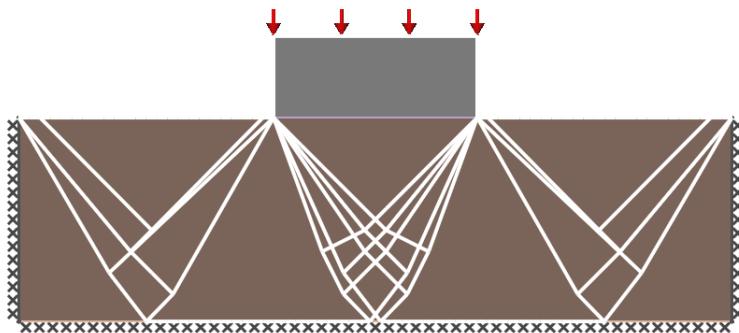


Figure 3.10: Change in critical failure mechanism caused by modification of boundary material in the Simple Footing problem. (Compare with the mechanism in Figure 3.5.)

Note that the default list of materials contains a *Frictionless* material that has zero shear strength and zero self weight. This is particularly useful for modelling smooth interfaces between solid zones.

3.10.4 Modifying Loads

Unlock the project and remove the *Very soft Clay* material from the bottom surface. Now select the upper right hand surface of the soil. View the load parameters by clicking the adjacent to the **Loading** parameter label in the **Property Editor**, and then click the adjacent to the **Loads** parameter label. The three load types **Permanent**, **Variable** and **Accidental** will be displayed. Loads for each of these load types may be defined and they can attract different partial factors depending on the partial factor set chosen (recall that the default **User** set in which all parameters were set to be unity was chosen earlier).

Now click on the adjacent to the **Permanent** parameter label to see the **Shear** and **Normal** applied loads. These should both be zero. Change the Normal value to 10.0, ensure **Adequacy** is set to *false*, and then repeat for the upper left hand surface. It can be seen that non-zero loads (with no adequacy factor applied to them) are displayed graphically in the viewer as green arrows. Click **Solve**, and a solution of 3.952 should be obtained. This is higher than the solution with zero surface load (3.902) as expected.

Note that if **Adequacy** had been applied to these surface surcharge loads, the program would try to find the factor by which these loads *and* the load on the footing would have to be simultaneously increased to cause collapse and would generate a very different factor to that which might be initially expected.

In this example, with $q = 10 \text{ kN/m}^2$ and $c_u = 150 \text{ kN/m}^2$, the theoretical adequacy factor is given by:

$$F = \frac{Q_{ULS}}{Q_d} = \frac{((2 + \pi)c_u + q)B}{Q_d} = 3.91 \quad (3.3)$$

The overestimate by LimitState:GEO in this case is now $\approx 1.06\%$, utilizing the default *Medium* nodal refinement.

3.10.5 Resizing a Block of Soil

Changing the geometry of a problem is straightforward. For example, re-run the Simple Footing wizard, (click **New**  to display the **New Project** dialog again) check the **Model as symmetrical half space** box on the **Geometry** page and accept all the other defaults (or simply click **Finish** after checking the half space box). This will produce a half space model. **Solve** to obtain the baseline Adequacy factor of 258.5. Now **Unlock** and click on the top right hand **Vertex** of the soil block and with the left hand mouse button held down, drag this **Vertex** downwards to form a slope (see Figure 3.11). Note that pressing **F8** or clicking **ORTHO** will toggle ‘Ortho mode’, which restrains the movement of selected items to the Cartesian directions.

Now **Solve** and a smaller value of adequacy factor should be obtained. Points may also be moved more precisely by typing in their coordinates. The **Geometry Editor** (located below the **Property Editor**) can be used for this purpose. Again click on the same point and the x and y coordinates will be displayed in the **Geometry Editor**. Set $x = 1.75$ and $y = 0.4$ and **Solve** again. An **Adequacy factor** of 208.6 should be obtained.

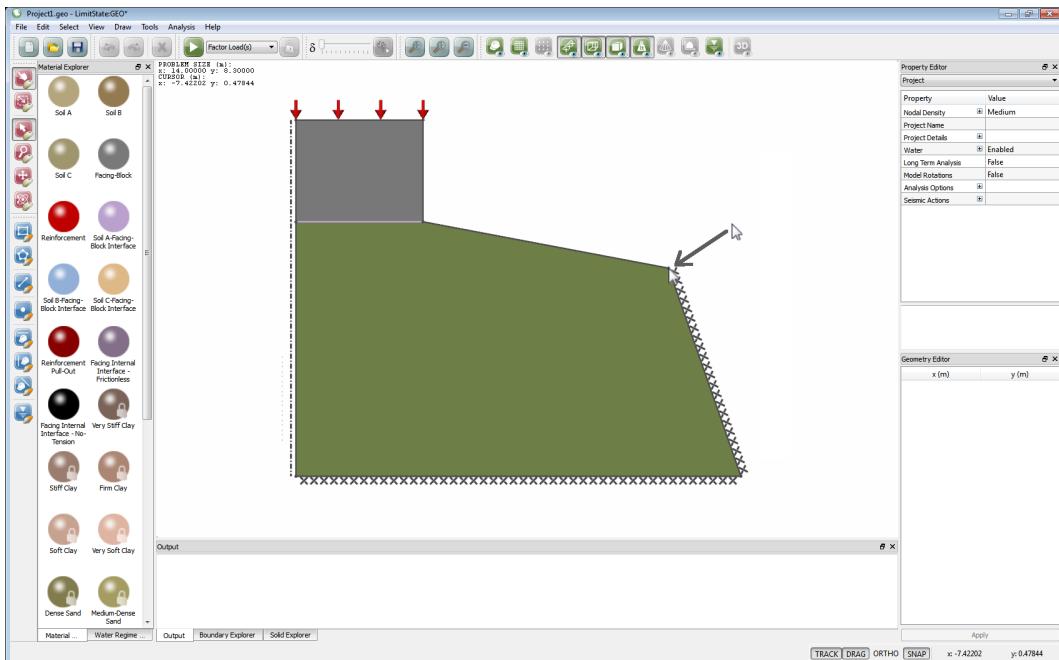


Figure 3.11: Dragging a vertex by clicking and dragging in the viewer

3.10.6 Changing a Boundary Condition

The current boundary conditions for the problem are displayed graphically. They may also be viewed in the **Property Editor** by selecting any boundary. In the **Property Editor** it is possible to set the boundary type, and the permanent, variable and accidental loads independently. Try selecting the upper right hand surface of the soil. It will be seen in the **Property Editor** that this has a Support Type of *Free*. This can be changed to *Fixed* or *Symmetry* (though this latter setting may not be appropriate here).

Change the Support Type on this boundary (upper right hand soil surface) to *Fixed*. Then select the vertical right hand edge of the soil zone and set its Support Type to *Free*. Solve again to obtain a radically altered failure mechanism (depicted in Figure 3.12) in which the soil can only flow out of the right hand side free edge.

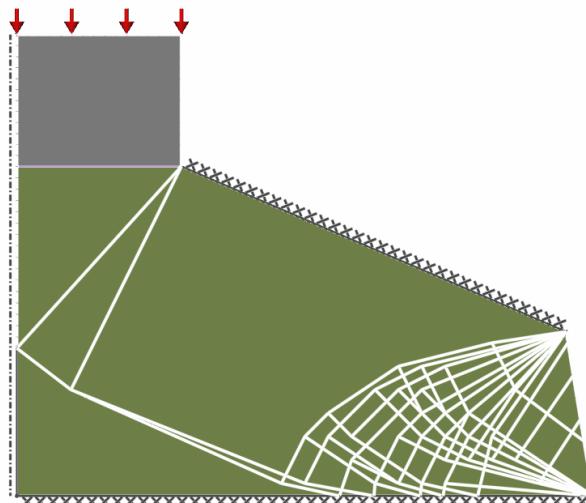


Figure 3.12: Solution mechanism caused by modification of support types in the Simple Footing problem (soil is ‘extruded’ through the opening at the right hand side free edge)

3.11 Solving a Collapse Problem by Factoring Soil Strength

When conducting a slope analysis and other similar situations it is often preferable to determine the adequacy factor on soil strength rather than loading. This can be achieved simply in LimitState:GEO:

Run the slope stability wizard (click File>New or and choose **Slope Stability** and click OK). Accept the defaults and click **Finish**. Note that the analysis mode is set by default to *Factor Strength(s)*. This setting will cause the software to automatically determine the factor by which the soil strengths need to be reduced in order to initiate collapse.

Click on the icon to solve. The failure mechanism depicted in Figure 3.13 and a solution of **Adequacy factor = 2.471** should be obtained. This means that the material strength needs to be reduced by a factor of 2.471 to cause failure.

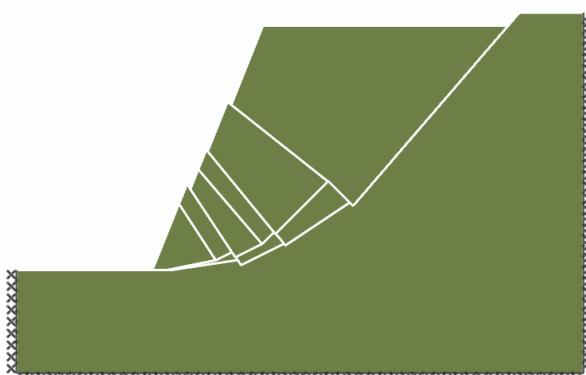


Figure 3.13: Deformed slope solution using default parameters in the Slope Stability Wizard

This solution may be compared with that derived by Taylor (1948). For a 68° slope the Taylor stability chart gives the stability number $N = c_u/F\gamma H \approx 0.2$. For this problem with $c_u/\gamma H = 0.5$, $F \approx 2.5$.

Note that for some problems such as a slope of frictional soil, it is only possible to bracket the Factor on Strength solution, and a collapse mechanism cannot be directly identified. For more information on solving problems using a factor on material strength, see Section 8.2.

3.12 Modelling Layered Soils

Frequently it is required to model several layers of soil in a problem. Layers may be built up individually or existing bodies of soil split into layers. The latter will be covered in this example, for further information on the former, see Section 15.6.

Using the slope from Section 3.11, click on the **Line** icon  on the left hand tool bar. Then click on the vertex at the toe of the slope (A in Figure 3.14) and then drag the line that appears across to the far right hand vertical boundary (B in Figure 3.14) and click here. This adds a new **Boundary** object and splits the existing **Solid** into two layers. By default these new layers receive the same properties as the original one, however these may be modified as required.

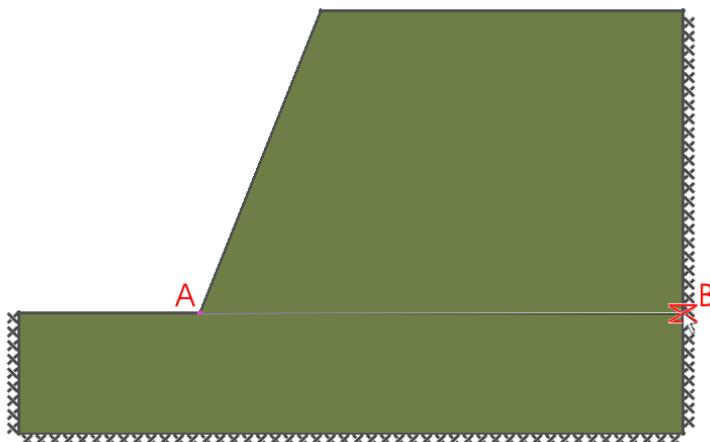


Figure 3.14: Splitting a **Solid** by adding a line (**Boundary**)

For example, change the material in the lower layer to a *Soft Clay* (e.g. by using drag and drop as described in Section 3.10.2). Clicking solve should generate a significantly changed failure mechanism similar to that depicted in Figure 3.15.

An important point to note here is that the critical mechanism intersects the boundaries of the problem domain (in this case in four places). It is thus unlikely to be the correct solution for a problem where the soil in reality extends much further to the left, right and below. In general it is necessary to move the boundaries outwards until the mechanism lies fully within the problem

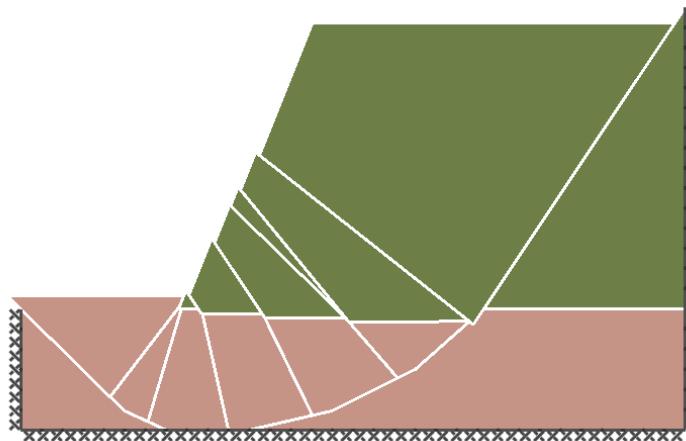


Figure 3.15: Failure mechanism for a two layer slope problem

domain. However, for some classes of problem, the mechanism is in theory semi-infinite and will always touch one or more boundaries.

3.13 Solution Accuracy

As with any numerical method, solution accuracy is dependent on the resolution of the underlying model. With DLO this relates to the distribution of nodes within **Solid** and **Boundary** objects. The method will provide the most critical sliding block mechanism that can be generated using slip-lines connecting any of the nodes present. In many cases a sufficiently accurate solution will be generated for a *Coarse* or *Medium* distribution of nodes (250 and 500 nodes respectively).

The number of nodes in the problem¹ may be set using the **Nodal Density** setting in the Project level parameters accessible in the Property Editor. The basic settings are *Coarse* (250 nodes), *Medium* (500 nodes), *Fine* (1000 nodes) and *Very Fine* (2000 nodes). In addition, a *Custom* option allows the user to assign a specific number of nodes (although care should be taken not to provide a number that requires excessive computational resources).

To investigate the effect of nodal distribution on the basic slope problem shown in Section 3.11, run the **Slope Stability** wizard once more. Click anywhere in an empty part of the **Viewer** pane (the part of the screen where the problem geometry is displayed). This will display the project level properties in the **Property Editor**. Change the value in the **Nodal Density** entry to *Fine*, and then solve the problem again. This time a similar but more detailed collapse mechanism is found with a slightly lower adequacy factor of 2.456.

Further information on solution accuracy may be found in Section 8.4 and on setting the **Nodal Density** in Section 22.3.

¹Due to the way in which nodes are distributed by the program, the nodal density is actually a target number rather than a set value.

3.14 Conclusion

This brief quick start tutorial has been designed to familiarize users with the basic functionality of LimitState:GEO. It is recommended that users experiment with the various wizards, and modify the various parameters involved, before constructing problems from scratch.

Chapter 4

Wizards

4.1 Introduction

LimitState:GEO provides a number of Wizards which allow the user to rapidly create models of commonly encountered problems. Variants on these geometries are easily generated by modifying the basic geometry created by the Wizard (see Section 15.9). To use a Wizard, select **Create a new project** in the **Welcome to LimitState:GEO** dialog and click **OK**. The **New Project** dialog (Figure 4.1) will then start. A number of different project types are available, all of which initiate a Wizard.

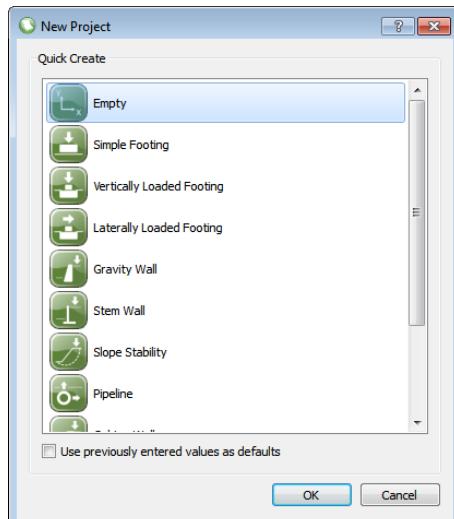


Figure 4.1: The **New Project** dialog

Empty Creates an empty project for the user to draw their own model. Includes options for specifying an underlying grid. Solves for factor on applied loads.

Simple Footing Creates a model of a basic rectangular footing, loaded vertically and sitting on a single layer of soil. Solves for factor on applied loads.

Vertically Loaded Footing Creates a model of a vertically loaded footing embedded in a layer of soil and overlying a second layer of soil. Solves for factor on applied loads.

Laterally Loaded Footing Creates a model of a footing embedded in a layer of soil and overlying a second layer of soil. Both horizontal and vertical loading can be applied. Solves for factor on applied loads.

Gravity Wall Creates a model of a trapezoidal gravity retaining wall holding back a slope with surcharge load. Solves for factor on applied loads.

Stem Wall Creates a model of a T-shaped stem wall holding back a slope with surcharge load. Solves for factor on applied loads.

Slope Stability Creates a model of a simple, single material slope with a potential surcharge. Solves for factor on material strengths.

Pipeline Creates a model of a buried pipeline subject to horizontal and vertical applied loading. Solves for factor on applied loads.

Gabion Wall Creates a model of a gabion wall, optionally tilted, holding back a slope with surcharge load. Solves for factor on applied loads.

Reinforced Soil Wall Creates a model of a wall constructed from facing blocks and strip reinforcement, retaining backfill and soil with a surcharge load. Solves for factor on applied loads.

Sheet Pile Wall Creates a model of an embedded sheet pile wall retaining a body of soil with potential surcharge load. Solves for factor on applied loads.

Select the required Project Wizard and click **OK**. If the Wizard has been run before and the previously entered values are to be used as defaults, ensure the **Use previously entered values as defaults** box is checked. Otherwise the system defaults will be used.

For further information on starting with an *Empty Project* see Section 15.3.

4.2 Using a New Project Wizard

4.2.1 Introduction

The Wizards in LimitState:GEO are designed to help the user quickly generate a model of their problem.

Each Wizard guides the user through the process of defining the model:

- **Project** description (Section 4.2.2)
- **Geometry** (Section 4.2.3)
- **Materials** (Section 4.2.4),
- **Loads** (Section 4.2.5)
- **Scenarios** (Section 4.2.6)

- **Analysis Options** (Section 4.2.7)

It should be noted that, at any point whilst running a Wizard, it is possible to click **Finish**. LimitState:GEO will automatically fill in any information that has not been explicitly supplied (by using default values together with information already provided by the user up until that point).

In most cases, information is entered in a sequential manner, the user clicking **Next** after each step. However, it is possible to move backwards through the various steps by using the **Back** button. The left-hand pane of the Wizard dialog serves as a reference point, with the current section being highlighted in blue as shown for the Project description stage in Figure 4.2.

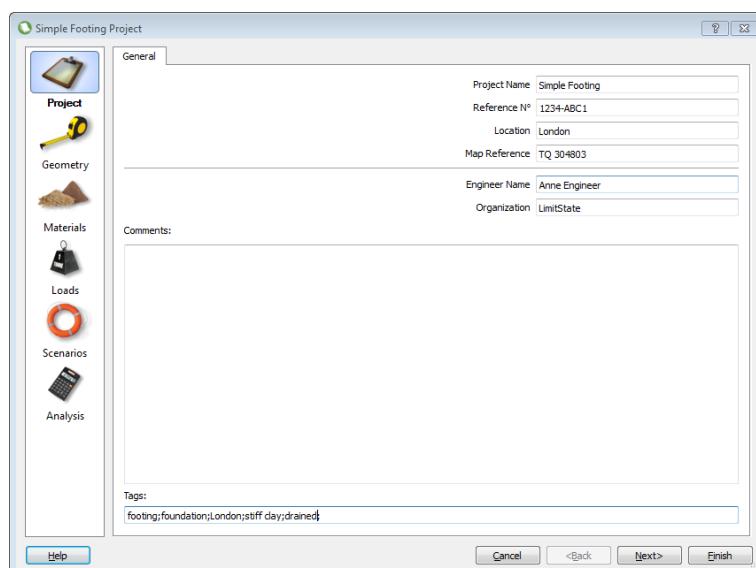


Figure 4.2: Wizard dialog (Project tab)

All parameters with dimensions should be entered in either Metric or Imperial units depending on the program preferences set by the user (see Section 15.2). When a data entry box is selected, data entry via a calculator is also available by clicking on the calculator button . The Calculator (see Section 14.13) also incorporates a unit converter for many commonly used units, including Metric and Imperial units.

4.2.2 General Project Settings

This dialog allows entry of the **Project name**, **Reference number**, **Location**, **Map reference**, **Engineer name**, **Organization**, general **Comments** and **Tags**. This information may be included in the **Report** output (Section 24). For further details on the **Tags** functionality, see Chapter 27.

4.2.3 Geometry

The next stage is to describe the geometry of the problem. Figures in the dialog clearly indicate the meaning of the parameters to be entered, e.g. as shown in Figure 4.3.

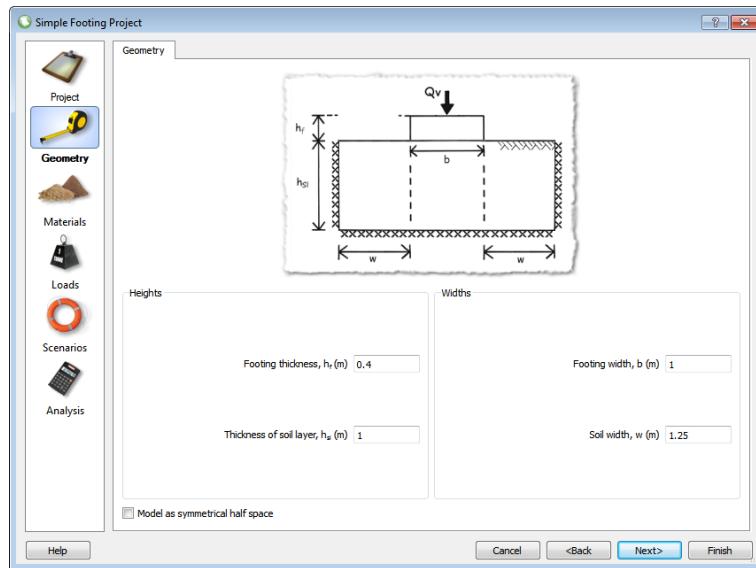


Figure 4.3: A typical wizard geometry dialog

On some Geometry dialogs, it is possible to select ***Model as symmetrical half space***. This option is available for problems that have a natural plane of symmetry. Modelling only half the problem increases computational efficiency and accuracy of the solution. However the option to model the full problem is available to a user who may wish to subsequently add asymmetrical loading or make the geometry asymmetrical.

The geometry data entry is normally designed to prevent unrealistic/impossible geometries being entered. However this is not feasible in all cases. If a combination of parameters is entered that would result in an unrealistic geometry being generated, a warning message will be displayed on pressing ***Finish*** and the software will adjust the geometry to produce a feasible result. The user may then either edit the final geometry directly (see Section 15.9), or, alternatively re-run the wizard and adjust the initial settings appropriately. To ensure the data entered previously is retained, ensure the ***Use previously entered values as defaults*** box is checked in the **New Project** dialog (see Section 4.1)

4.2.4 Materials

Each tab in this dialog allows the user to specify the basic properties of the materials used in the problem or to select a material from a predefined list as shown in Figure 4.4.

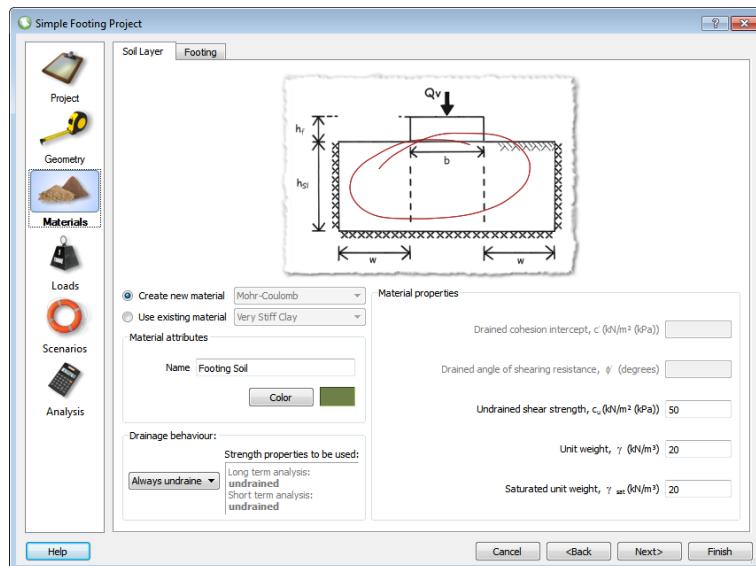


Figure 4.4: A typical Wizard materials dialog

For materials such as soils, the shear strength parameters c_u , c' , ϕ' , and the unit weights (weight densities) may be entered as well as the material drainage behaviour (in LimitState:GEO **Unit Weight** and **Saturated Unit Weight** designate the unit weight of the material above and below the water table respectively).

For materials representing engineered ‘mass’ structural elements such as footings or walls, they are assumed to behave as a rigid material by default and generally only the unit weight will be requested, together with multipliers that define the soil/structure interface properties. These multipliers are applied to any adjacent soils to define a **Derived** material. For further information on **Derived** materials see Section 17.5.

For materials representing other types of engineered element, such as soil nails or sheet pile walls, the properties requested will relate to the material strength (e.g. *Plastic Moment* and *Rupture Strength*) and interaction with the surrounding soil (e.g. *Pullout* and *Lateral* factors). For further information on **Engineered Element** materials see Chapter 11.

In order to keep the Wizards straightforward to use, only basic material properties are entered, and default values are used for the other properties. To edit other available properties it is necessary to use the Property Editor (see Chapter 17) once the Wizard has been exited.

4.2.5 Loads

Specify the required loads here as shown in Figure 4.5. Loads specified as **Variable** will generally by default have a margin of safety or **Adequacy factor** computed for them. For such loads, you may either enter the actual applied load and a margin of safety will be computed. Alternatively a value of 1.0 may be entered and the **Adequacy factor** will equal the collapse load. For further discussion of the **Adequacy factor** see Chapter 19.

Line loads (in kN/m) are typically specified for loading on structural elements (such as a footing or pipeline) while surface surcharge pressures (in kN/m²) are specified for distributed surface

loads. Note that LimitState:GEO internally represents loads on **Boundary objects** as stresses. Line loads will thus automatically be converted to stresses acting over the width of the relevant boundary and will be displayed in the **Property Editor** as both a stress and a line load (see Chapter 19).

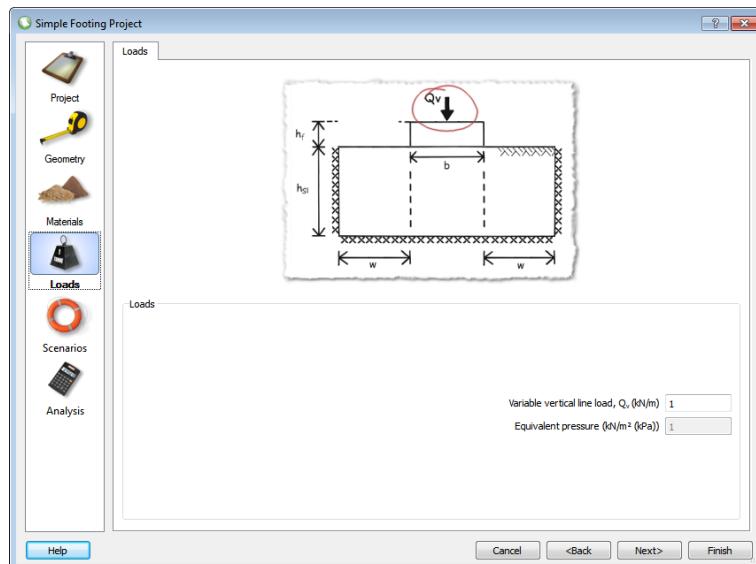


Figure 4.5: A typical Wizard loads dialog

4.2.6 Scenarios

This stage displays the **Scenario Manager** as shown in Figure 4.6. For a description of the full functionality of the **Scenario Manager** and for guidance on defining **Partial Factors**, see Chapter 30 and Section 8.3. By default a **Single scenario** is active and the required partial factor set (multipliers on applied loads or divisors on material properties) can be defined manually, or a set of pre-defined factors can be selected using the drop down box.

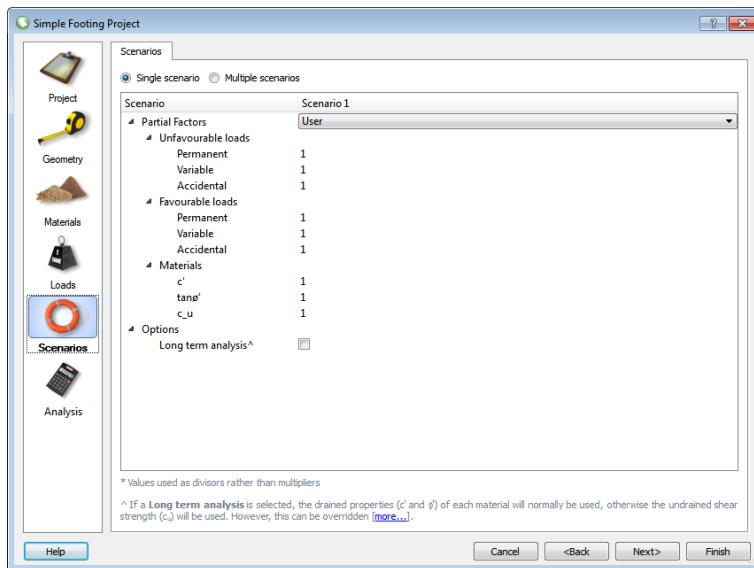


Figure 4.6: A typical Wizard scenario manager dialog

4.2.7 Analysis Options

This stage displays the **Analysis** tab, as shown in Figure 4.7. Here the user can set the **Analysis Type** to either *Factor Load(s)* or *Factor Material(s)* (see e.g. Section 3.3) and alter the nodal density of the problem.

By default the **Analysis Type** is generally set to *Factor Load(s)* and the **Nodal Density** is set to *Medium* (500 nodes). The **Target Number** field will display the number of nodes associated with the selected density and, when a *custom* density is specified, the value can be set manually.

For particular types of problem (e.g. Gravity Retaining Walls) an option to **Optimize nodal layout** is provided. This allows for increased accuracy when rotational failure mechanisms are likely to be present and is set to *true* by default. More information regarding this setting can be found in Section 22.3.4.

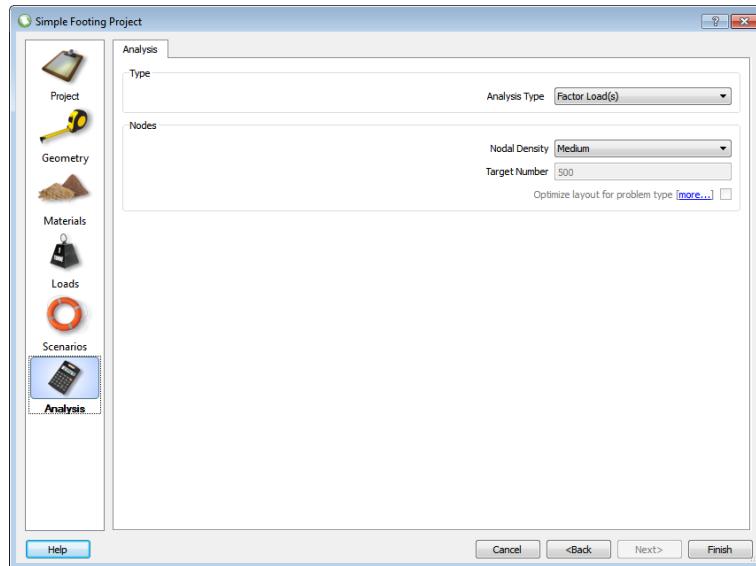


Figure 4.7: A typical Wizard analysis options dialog

Part II

Theory

Chapter 5

Discontinuity Layout Optimization

5.1 Introduction

At the heart of LimitState:GEO is a solution engine which uses the Discontinuity Layout Optimization (DLO) numerical analysis procedure to find a solution. The procedure was developed at the University of Sheffield and was first described in a paper published in the Proceedings of the Royal Society ([Smith & Gilbert 2007a](#)). In essence DLO can be used to identify critical translational sliding block failure mechanisms, output in a form which will be familiar to most geotechnical engineers (for example see Figure 5.1 for a Coulomb wedge analysis of a retaining wall, or for a method of slices solution for a slope stability analysis problem see Figure 5.2). However while traditional methods can typically only work with mechanisms involving a few sliding blocks, DLO has no such limitations. It can identify the critical translational failure mechanism for any geotechnical stability problem, to a user specified geometrical resolution. This can be illustrated by examining the critical solution to the vertical anchor uplift problem in cohesive soil problem, as shown in Figure 5.3.

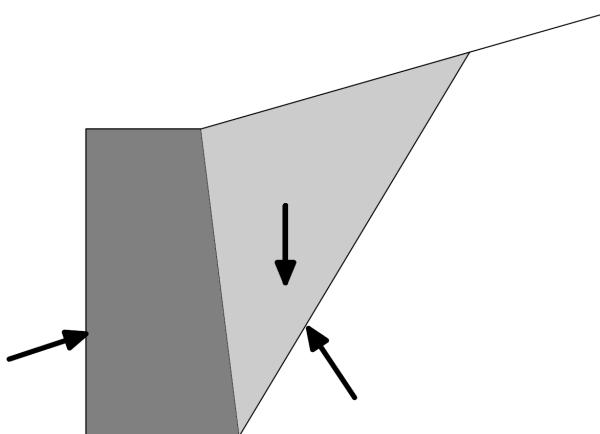


Figure 5.1: Example of Coulomb wedge analysis of a gravity retaining wall

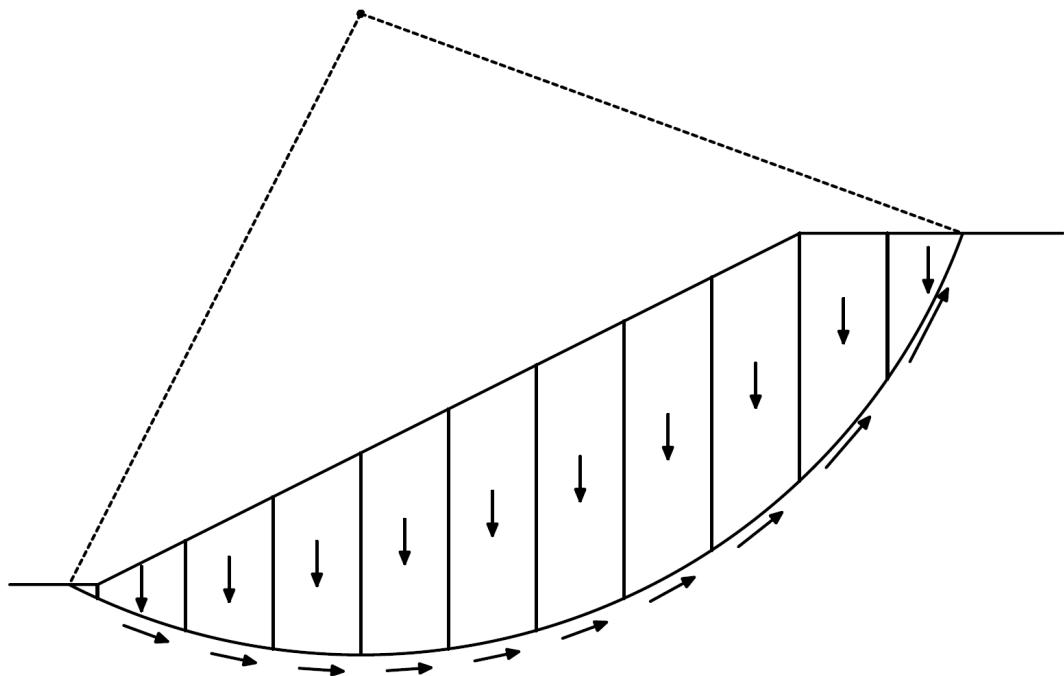


Figure 5.2: Example of slope stability by method of slices

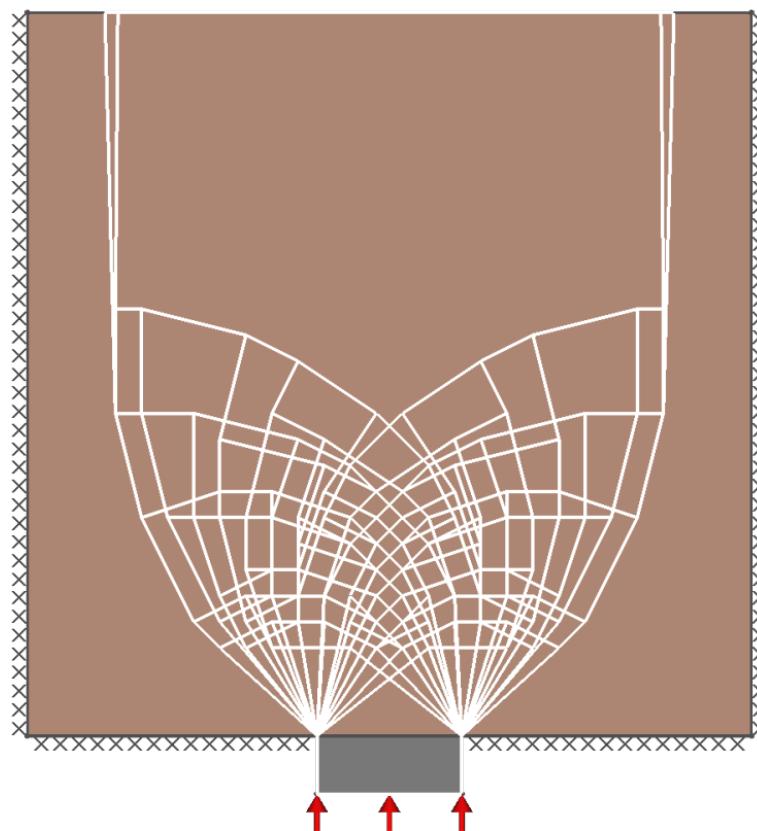


Figure 5.3: Example of the DLO solution of vertical anchor uplift in a cohesive soil

5.2 Limit Analysis and Limit Equilibrium

Determination of the ultimate limit state (ULS) or ‘collapse state’ of any geotechnical problem has traditionally been carried out using a range of approaches. The most commonly used approaches are Limit Equilibrium and Limit Analysis.

Limit Analysis procedures are rigorously based upon the theorems of plasticity while Limit Equilibrium typically involves a similar form of calculation but relaxes one or more of the conditions of plasticity theory to make the problem easier to solve. For example the method of slices used to analyse slope stability problems is a Limit Equilibrium method. The sliding mass is divided up into a number of independent blocks. Equilibrium of each block is not fully determined; instead an assumption is made concerning inter-slice forces. This may result in a solution that does not fully satisfy equilibrium, or the yield condition for all blocks, but is typically found to give reasonable results.

Limit Analysis is used to determine many of the bearing capacity and retaining wall formulae typically used by geotechnical engineers.

DLO is a limit analysis method that effectively allows free choice of slip-line orientation, and the critical solution identified may involve the failing soil mass being divided into a large number of sliding blocks. Accuracy can be assessed by determining the influence of nodal refinement. DLO also readily handles variation of soil parameters, and heterogeneous bodies of soil.

5.3 DLO: How Does it Work?

5.3.1 Basic Principles

Discontinuity Layout Optimization (DLO), as its name suggests, involves the use of rigorous mathematical **optimization** techniques to identify a critical **layout** of lines of **discontinuity** which form at failure. These lines of discontinuity are typically ‘slip-lines’ in planar geotechnical stability problems and define the boundaries between the moving rigid blocks of material which make up the mechanism of collapse. Associated with this mechanism is a collapse load factor, which will be an upper bound on the ‘exact’ load factor according to formal plasticity theory. Thus in essence the procedure replicates and automates the traditional ‘upper bound’ hand limit analysis procedure which has been used by geotechnical engineers for many years. DLO is general, and can be applied to a wide range of geotechnical stability problems involving cohesive and/or frictional soils.

In order to allow a wide range of different failure mechanisms to be identified, a large number of potential lines of discontinuity must be considered. In order to achieve this, closely spaced nodes are distributed across the problem domain and potential lines of discontinuity are created to connect each node to every other node, thereby providing a very large search space. In numerical terms, if there are n nodes, there are approximately $n(n - 1)/2$ possible slip-lines and approximately $2^{n(n-1)/2}$ possible slip-line mechanism topologies. Thus for example 500 nodes give rise to $\approx 125,000$ possible slip-lines and of the order of 10^{38000} possible mechanism topologies.

A simple example involving the bearing capacity of a footing is given in Figure 5.4. The thin

black lines indicate the set of potential discontinuities to be considered and these connect an initial set of nodes (for sake of clarity very coarse nodal refinement has been used, and only near-neighbour nodes have been inter-connected). Whilst there is no restriction on the pattern of nodal distribution utilized, square or triangular grids are generally most convenient. In LimitState:GEO uniform square grids aligned to the global x and y axes are utilized. The solution will clearly be restricted to sets of discontinuities that connect these nodes, and is thus the solution will be influenced by the starting positions of the nodes. However when fine nodal refinement is used, the exact positions of individual nodes will have relatively little influence on the solution generated.

In the DLO procedure the problem is formulated in entirely in terms of the relative displacements along discontinuities, e.g. each potential line of discontinuity can be assigned a variable that defines the relative slip displacement along that discontinuity. When relative displacements are used, compatibility can be straightforwardly checked at each node by a simple linear equation involving these variables. Finally an objective function may be defined in terms of the total energy dissipated in sliding along all discontinuities. This is a linear function of the slip displacement variables. A linear optimization problem is thus defined, the solution of which identifies the optimal subset of discontinuities that produce a compatible mechanism with the lowest energy dissipation (thick lines in Figure 5.4). The accuracy of the solution obtained depends on the prescribed nodal spacing. As is evident from Figure 5.4, a key benefit of the procedure, compared with comparable ‘element based’ procedures, is that singularities can be identified without difficulty (potential fan zones centred on a given node can be identified if critical by linking multiple lines of discontinuity to that node). Finally, while the fact that discontinuities are clearly free to ‘cross over’ one another might appear problematic, it can be shown that compatibility is implicitly enforced at ‘cross over’ points, and their presence is very beneficial as the search space is dramatically increased. Further discussion of this issue and a detailed description of the DLO procedure is given in [Smith & Gilbert \(2007a\)](#).

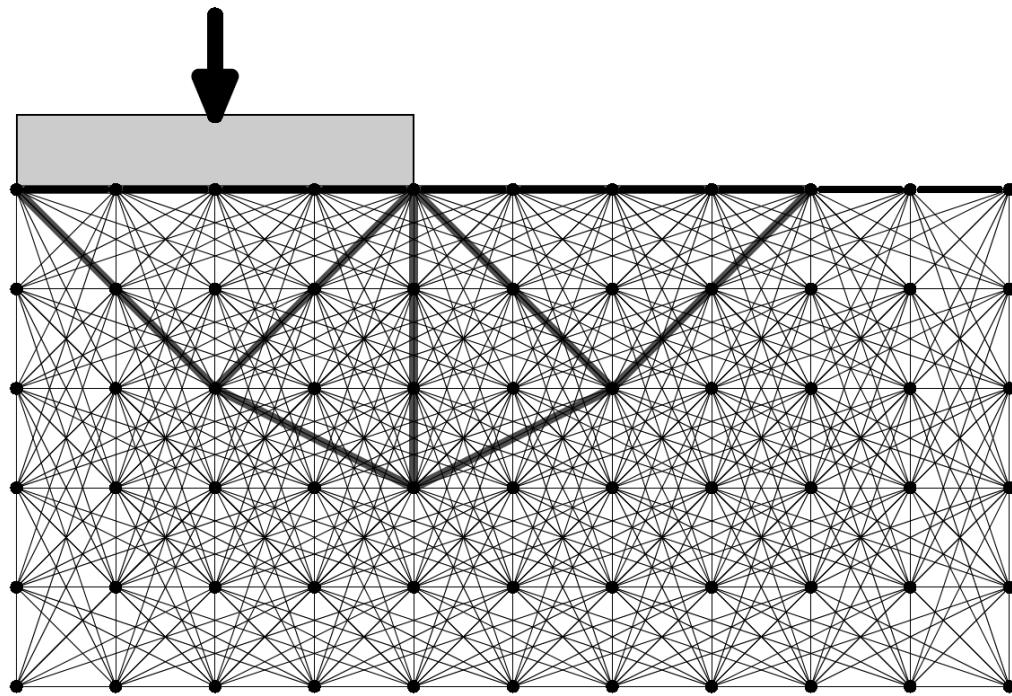


Figure 5.4: DLO analysis of the undrained stability of a footing

5.3.2 Adaptive Solution Procedure

While LimitState:GEO utilizes the core methodology described above, it also makes use of an adaptive solution procedure described by [Gilbert & Tyas \(2003\)](#) in order to significantly reduce memory requirements and also reduce the time needed to obtain a solution. The procedure makes use of the fact that the solution identified by the linear programming solver can be used to determine the forces on any slip-line connecting any pair of nodes, even if the solver did not explicitly consider that slip-line in determining its solution. In essence the procedure operates as follows:

1. Set up an initial ‘ground state’ in which nodes are connected by slip-lines only to nearest neighbour nodes vertically, horizontally and diagonally.
2. Solve this problem utilising the DLO procedure. This will give a valid upper bound solution, but of relatively low accuracy, utilising only these short slip-lines.
3. Utilizing this solution, search through all potential slip-lines connecting every node to every other node and determine whether yield is violated on any potential slip-line.
4. If yield is violated on a potential slip-line, add this slip-line to the linear optimization problem set and re-solve. Repeat from Stage 3.
5. If yield is not violated on any slip-line then the correct solution has been arrived at, as if all possible slip-lines had been included in the original linear programming problem.

5.3.3 Rotational Failure Mechanisms

5.3.3.1 Implementation

The current implementation of DLO in LimitState:GEO generates solutions based on translational mechanisms. In order to model mechanisms involving rotation of structural elements such as cantilever retaining walls, it is necessary to set the **Model Rotations** value in the **Project** level **Property Editor** to *Along Edges*. (If this value is reset to *False* then a purely translational solution will be found.)

With *Along Edges* set, LimitState:GEO allows rotations to be modelled along **Boundaries**. This allows **Solids** to rotate as rigid bodies and to transmit these rotations into translational deforming bodies by modelling localized rotational deformations along the boundary. This is an approximation to modelling rotational and translational failure everywhere, but is generally accurate enough for most problems while keeping the size of the numerical problem smaller. The approach is depicted in Figure 5.5, where a series of small rotational elements are modelled along the length of the boundary. This maintains the upper bound status of the solution. However in some cases modification to the boundary properties are required to ensure collapse loads are not overestimated (see following subsection).

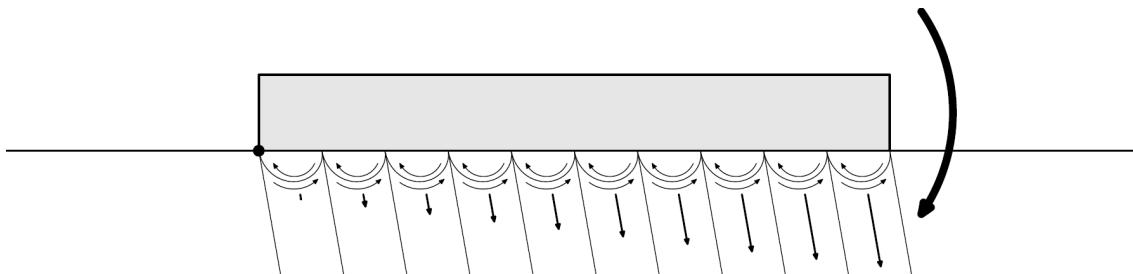


Figure 5.5: Modelling of rotational elements along a rotating boundary

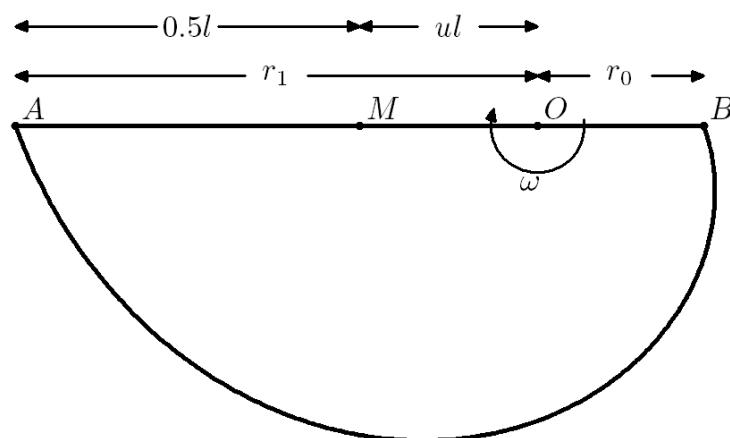


Figure 5.6: Geometry of log-spiral

In the following analysis c and ϕ may represent either c' and ϕ' for drained parameters or c_u

(and $\phi_u = 0$) for undrained parameters. The geometry of each rotational element is depicted in Figure 5.6. In general the shape of the slip surface in this element will be a log-spiral. From the geometry of the log-spiral, the following expression can be obtained for r_0 :

$$r_0 = \frac{l}{(1 + e^{\pi \tan \phi})} \quad (5.1)$$

where l is the length of segment along the boundary. If the boundary rotates by an angle ω , then the resultant effect is an equivalent rotation about the segment midpoint M by ω , accompanied by an additional dilation:

$$n_\omega = ul\omega \quad (5.2)$$

where

$$u = 0.5 - \frac{1}{(1 + e^{\pi \tan \phi})} \quad (5.3)$$

Outside the small rotational element, movement is purely translational and equal to the translation of the segment midpoint. If there is also a cohesion c present, then the work done W on the arc AB overcoming the cohesion c for a relative body rotation of ω is given by the following expression:

$$W = \frac{c\omega ul^2}{\tan \phi} \quad (5.4)$$

The derivation of this equation may be found in Appendix D.1.

Examination of equation 5.4 and equation 5.2 indicates that l gets smaller, both the dilation and work done associated with the rotational elements tends to zero. A reduction in l corresponds to an increase in nodal density on the rotating boundary. However any sliding on the boundary is not coupled to the rotations, thus a relative slip s will give rise to an additional dilation component of $n = s \tan \phi$, and the overall dilation or normal displacement is given by the following equation:

$$n = s \tan \phi + ul\omega \quad (5.5)$$

Only the material properties in the solid are used for parameters in the (edge) rotational equations. If a separate material is defined on the boundary then LimitState:GEO utilizes the weakest of the solid and boundary material to determine dilation and energy dissipation due to sliding along this boundary.

It should be noted that LimitState:GEO assumes that the boundary segments and thus rotational elements are small. LimitState:GEO makes no checks to see whether the modelled rotational elements remain within the adjacent solid and do not extend into adjacent solids which may have different properties. Hence this mode should only be used with high nodal densities. It should also be noted that in circumstances where the actual true failure mechanism happens to be similar to the modelled rotational mechanism for a few nodes, then increasing the

nodal density may cause a increase in collapse load rather than decrease as would normally be expected.

Additionally any work done against gravity by the soil within the rotational elements is neglected. As the size of the elements reduces (as nodal density increases), this work term tends to zero.

5.3.3.2 Accuracy of edge rotations model

The edge rotations model is an approximation to modelling rotational and translational failure everywhere, but is generally accurate enough for most problems while keeping the size of the numerical problem smaller. It works well for problems that are dominated by translational displacements in the soil (even if rigid elements are undergoing significant rotation, see for example the case in Figure 5.7).

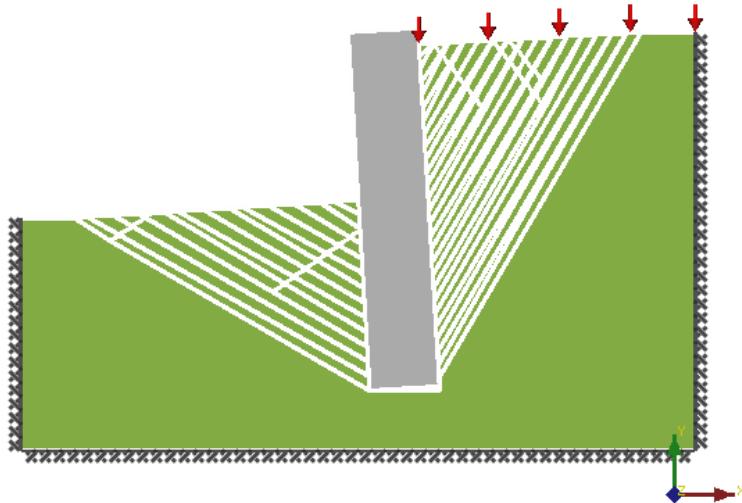


Figure 5.7: Mechanism involving rigid body rotation, but translation (shear) deformation of soil

However for problems where strong rotational soil displacements occur such as in eccentrically loaded footings, the method tends to overestimate collapse loads because in the process of modelling the collapse mechanism beneath the footing, it models additional sliding between the soil and the boundary interface. This leads to extra work either against cohesion or against gravity (due to additional dilation). Examples of measures that may be used to remove this effect in the **Edge Rotations** model are described below:

1. For problems with internal corners on the rotating solid, the solver can be assisted by linking the adjacent vertices either side of the corner as shown in Figure 5.8. This allows all the soil ‘internal’ to the corner to fully rotate. A common problem where this should be undertaken is that of a stem wall analysis as shown in Figure 5.9. The LimitState:GEO Stem Wall Wizard automatically generates these lines.

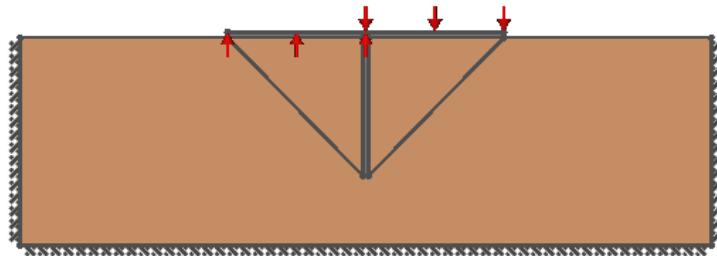


Figure 5.8: Modelling rotation of a footing with rigidly connected sheet pile below. Note connection across internal corners by internal **Boundaries** in order to allow soil adjacent to the footing/pile to rotate freely with the footing/pile when using the edge rotations model.

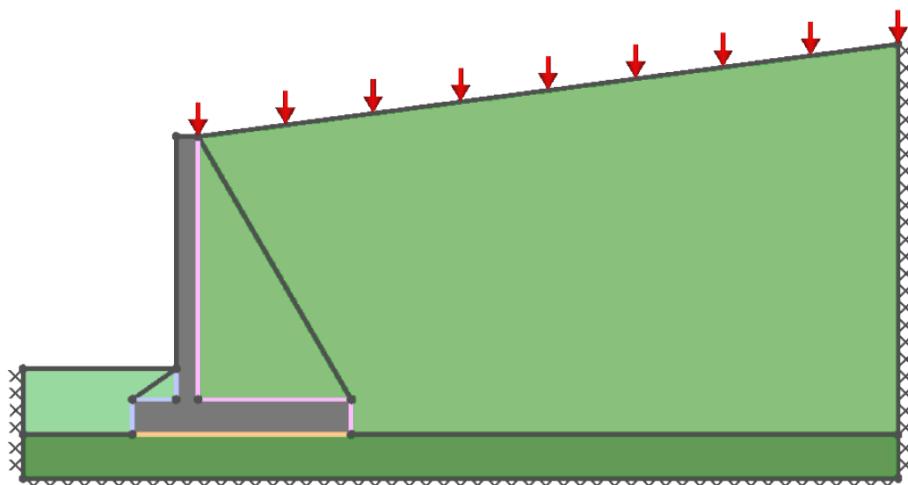


Figure 5.9: Modelling rotation of a stem wall. Note connection across internal corners by internal **Boundaries**, in order to allow soil adjacent to the wall to rotate freely with the wall when using the edge rotations model.

2. Delineation of a block of soil immediately adjacent to a rotating rigid body has been found to significantly improve results. An example of an eccentrically loaded footing is shown in Figure 5.10. The additional boundaries allow rotation within the material and permit a ‘rigid’ block of material adjacent to the foundation to rotate. Such a situation could also arise for example with a slope stabilised with a concrete facing and a similar approach is recommended.

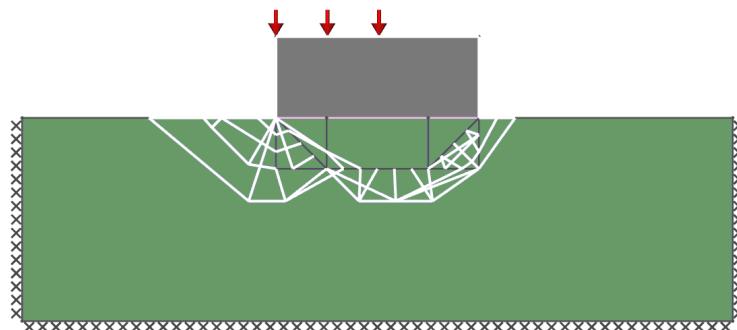


Figure 5.10: Modelling rotation of an eccentrically loaded footing. Provision of delineated boundaries below the footing allows rotational mechanism to be more closely modelled when using the edge rotations model.

3. For problems where structural elements such as beams or walls are modelled using a **Soild** block rather than an **Engineered Element**, then shear failure only would normally be modelled within the block (assuming a 'cohesion' only Mohr-Coulomb model has been used to represent the beam/wall material). To enable failure by bending it is necessary to introduce a number of boundary divisions perpendicular to the main axis within the beam/wall as shown for example in Figure 5.11, which has employed 5 divisions. In this simply supported example only a central hinge is identified as shown. However if the beam had fixed supports at the ends then hinges would also be identified at the end divisions. The more vertical division employed, the more possible hinge locations can be modelled.

The edge rotation model will closely represent simple engineering beam bending theory and will correctly account for the effects of axial force on the bending capacity. However to capture this to a good accuracy it is necessary to ensure that sufficient nodes are employed in the beam. Normally it would only be required to place nodes around the beam boundaries (including the vertical divisions) and it is not generally necessary to use nodes within the beam itself unless more complex failure is anticipated.

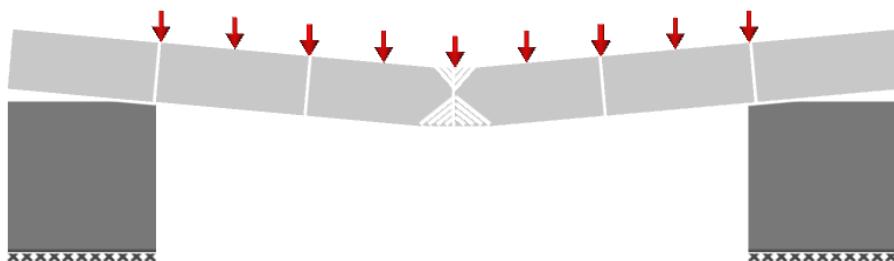


Figure 5.11: Modelling bending failure of a beam using a rigid block. Provision of vertical boundary divisions along the beam provide positions where hinges can be modelled when using the edge rotations model.

In all cases it is recommended that calibration tests are carried out where strong rotational displacements are dominant.

The verification tests (see Appendix A) available on the LimitState website may assist in assessing this issue for a range of problem types. solution for a slope stability analysis problem

see Figure 5.2.

Chapter 6

Model components

6.1 Material Models

6.1.1 Introduction

LimitState:GEO is an ultimate limit state analysis program. Thus only the parameters defining the yield surface are required for the material models used by LimitState:GEO. In LimitState:GEO the following models are available:

- Mohr-Coulomb
- Tension and/or compression cut off
- Rigid
- Engineered Element

In addition, material models may be combined to generate more complex yield surfaces. Note that while **Engineered Elements** (e.g. soil nails) are defined by setting a material property, they are a special case. **Engineered Elements** are discussed further in Section 6.1.5.

6.1.2 Mohr-Coulomb Material

The yield surface is defined as depicted in Figure 6.1 in terms of parameters c and ϕ where c is the intercept on the shear stress (τ) axis and ϕ the angle of shearing resistance.

For drained analysis the values c' and ϕ' are used, while for undrained analysis the parameter c_u is used. A linear variation of c_u with depth may be specified, as can strengths interpolated using a regular grid of points and associated multipliers on the cohesive material properties. While it is possible to derive a value of ϕ_u from a shear box or triaxial test, interpretation and use of this parameter requires care. The parameter ϕ_u is typically taken as zero, and this is assumed in LimitState:GEO. If an analysis using a non-zero value of ϕ_u is required then it

is recommended that the user defines the problem in LimitState:GEO as if it was a drained problem but used the undrained parameters c_u and ϕ_u .

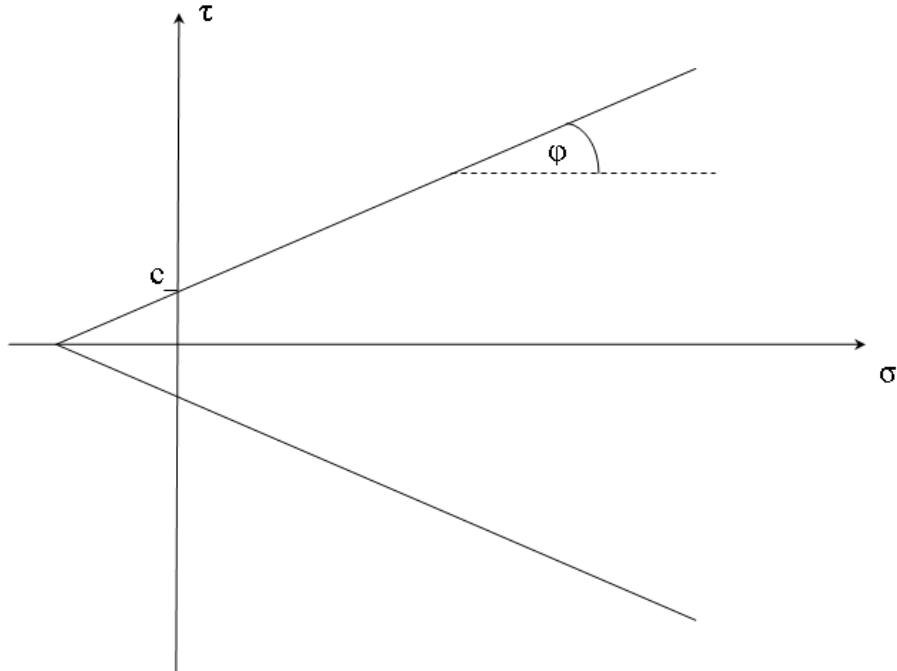


Figure 6.1: Mohr-Coulomb yield surface

6.1.3 Cutoff Material (Tension and/or Compression)

The yield surface is defined as depicted in Figure 6.2 in terms of a limiting tensile stress σ_T and a limiting compressive stress σ_C . The limiting tensile stress is typically used to model tension cutoff so that tension cracks may be modelled by LimitState:GEO or to indirectly represent tensile yield in adjacent structural elements without fully modelling the structural element. The limiting compressive stress is typically used to represent crushing of materials, or to indirectly represent compressive yield in adjacent structural elements (e.g. a retaining wall prop) without fully modelling the structural element.

No distinction is made in these values for drained or undrained analyses. This material model may be utilized on its own, or in conjunction with another material e.g. Mohr-Coulomb (see Section 6.1.6). Additionally no partial factors are defined for these parameters.

Note that if the yield condition lies on this surface, then a shear stress may still be maintained. It is only the normal stress that is limited. To avoid this, a more rounded yield surface may be defined using multiple materials (see Section 6.1.6).

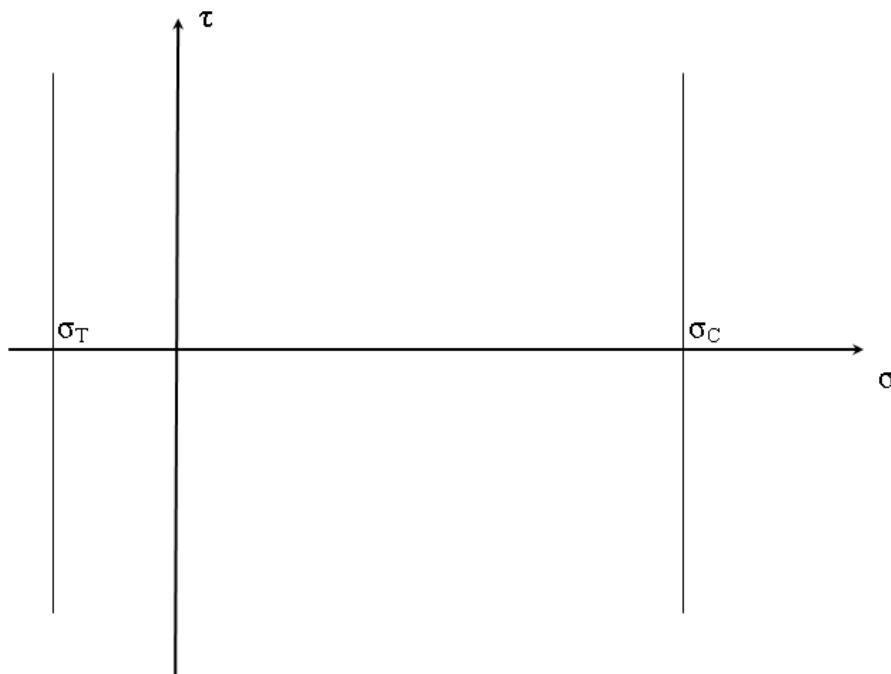


Figure 6.2: Tension cutoff and crushing yield surface

6.1.4 Rigid

This simple material type is fully rigid. No slip-line can pass through it at any load. Use of this material type for solids for which it is known that they will not deform, should enhance the solver efficiency, since it reduces the number of nodes and slip-lines that need to be considered.

6.1.5 Engineered Element

An Engineered Element is a special one dimensional element that may be assigned to a **Boundary** object. It is commonly used to model items such as sheet pile walls, soil nails, and geotextiles. They have the following attributes:

1. When an Engineered Element is assigned to a **Boundary**, three parallel components are generated: a top boundary interface, the Engineered Element itself and a bottom boundary interface. The interfaces may be assigned Mohr Coulomb or Cutoff materials. The Engineered Element itself is sandwiched between these boundary interfaces. Note that depending on the orientation of the Engineered Element the top and bottom boundary interfaces may be designated right and left.
2. The Engineered Element may translate and/or rotate relative to the top and bottom boundary interfaces. This is controlled by pullout and lateral factors. The top and bottom boundary interfaces themselves are locked to each other.
3. Within the model formulation, the relative slip displacement (s_i) and the normal displacement (n_i) between the Engineered Element and the top and bottom boundary interfaces

for each segment i of the Engineered Element is computed. These values may vary along its length due to variations in soil displacement and or element displacement.

4. The Engineered Element itself is treated as a rigid object between each **Vertex** along the element, but is allowed to rotate (bend), rupture (break) and fail in compression at each vertex. The rotation (θ_j) and axial force at each vertex j is computed.

If the user sets **Subdivide at nodes** to *True* in the **Material Properties** then additional vertices will be added at nodes along the Engineered Element during solve. If this value is set to *False* then the only the original Vertices defined in the problem geometry are used. The default value is *True*.

5. The additional work done within the system, due to the movement of an Engineered Element relative to the top and bottom boundaries is calculated from $\sum_{i=1}^g (Ts_i + Nn_i) + \sum_{j=1}^v M_p\theta_j$, where T is the pullout resistance per unit length per unit width, N is the resistance per unit length per unit width to lateral displacement, g is the number of segments, v is the number of vertices and M_p is the plastic moment resistance of the element per unit width.
6. If an Engineered Element lies on a soil surface, then one half of the relative slip s is used in the energy equation.
7. Where the relative normal displacement n is different on either side of an element segment, then the average of the two values is used in the energy equation. This circumstance arises only when the Engineered Element lies on the surface of a soil.
8. The optimizer will determine the absolute displacement of the Engineered Element that minimizes the energy dissipated within the problem.

Note that if an **Engineered Element** is used to represent an effectively one dimensional object such as a soil nail that has a pullout resistance T_1 per unit length, a resistance N_1 per unit length to lateral displacement and a plastic moment of M_{p1} , then if there are m objects present per unit width the following values should be used. $T = mT_1$ is the pullout resistance per unit length per unit width, and $N = mN_1$ is the resistance per unit length per unit width to lateral displacement of the objects and $M_p = mM_{p1}$.

Suitable values for T and N will normally be selected based on experimental evidence, or on appropriate theory for the situation. In certain circumstances, the resistances T and N will be a function of the normal stress and pore pressure around the element. In common with many approaches in the literature, LimitState:GEO allows T and N to be computed as linear functions of the nominal vertical effective stress at the element as follows:

$$T = T_c + T_q\sigma'_v \quad (6.1)$$

$$N = N_c + N_q\sigma'_v \quad (6.2)$$

where T_c , T_q , N_c , and N_q are constants defined in the **Engineered Element** material. The vertical effective stress σ'_v is computed by LimitState:GEO prior to solving based on the weight of overburden per unit area above the element midpoint minus the pore pressure at the element midpoint. This is computed separately for each straight line portion of element (i.e. section between vertices).

Note that if a water table is present that intersects the element, then the solver automatically adds a vertex at the intersection and will therefore compute separate stresses for the parts of the element above and below the water table.

Also note that it is very likely that the vertical effective stress along the element is modified by the overall failure mechanism. While the vertical effective stress and pore pressure may be derived from the final solution determined by LimitState:GEO, they cannot be used in the solution procedure due to the indirect representation of the element in the numerical model. If required the element parameters could be refined iteratively using the results of a previous solution.

Finally if element rotation occurs, it is necessary to employ are relatively high nodal density along the element to ensure accuracy.

If an **Engineered Element** is used to represent a two dimensional object such as a sheet pile or geotextile, then the special features of the element (pullout and lateral resistance) that are used to allow soil to flow past 1D elements are not required. In this case they may be modelled by setting the pullout and lateral resistances T_c and N_c per unit length per unit width to infinity (in LimitState:GEO any number $> 1 \times 10^{30}$ is regarded as infinity). This prevents any flow through or past the Engineered Element as required for e.g. a sheet pile.

Instead of being determined by T , the pullout resistance and surface sliding properties of the element are determined by the boundary interface properties that can be independently defined above and below the element.

More about the use of Engineered Elements can be found in Chapter 11, and about how they are specified in Section 17.9.

6.1.6 Combined Materials

Any number of materials may be combined by assigning them simultaneously to soil layers (**Solid objects**) or interfaces (**Boundary objects**).

It is therefore effectively possible to build up any convex yield surface as a series of individual linear yield surfaces. Typical example uses of this facility are:

1. To model an undrained material with zero tension cutoff. Specify the undrained shear strength c_u in a Mohr-Coulomb model and combine with a Tension cutoff material with $\sigma_T = 0.0$ and a limiting compressive stress $\sigma_C = 1.0E30$. (It is not possible to specify infinity in LimitState:GEO so a very large value is specified instead.)
2. To model a material with a non-linear but convex yield surface. Such a yield surface may be approximated by two or more linear yield surfaces. LimitState:GEO will always work with the yield surface that produces the lowest adequacy factor. An example of such an application is in modelling the non-linear Hoek-Brown yield surface for rock masses.
3. To model uncertainty in soil parameters. In many circumstances the properties of a given soil will be uncertain. Normally the most conservative shear strength values would be selected. However for soils with both c and ϕ values, the most conservative shear strength mobilized will depend on the prevalent normal stress. To be sure of a conservative result, the lowest values of c and of ϕ might separately be selected, which may result in

an unnecessary degree of conservatism. With LimitState:GEO this is unnecessary and multiple soil models may be assigned to a body. LimitState:GEO will automatically utilize the weakest material consistent with the prevalent normal stresses.

Note that when combined materials are used which have different specified unit weight, it will be necessary to indicate which values are to be used (minimum, average or maximum).

6.2 Representation of Water Pressures

6.2.1 Modelling of Water Pressures using DLO

The effect of water pressures is readily incorporated into the DLO computations. As shown by [Smith & Gilbert \(2007b\)](#), the presence of water requires an additional work term in the Linear Programming formulation equal to the average water pressure multiplied by the dilation occurring on any slip-line. This has the same effect as implementing the principle of effective stress in the analysis.

In LimitState:GEO, the presence of water can be represented by one or both of:

1. **A Water Table** which affects the whole model, or
2. **Water Regimes** which can be assigned on a per-zone basis

It is generally advisable that a model containing water exhibits continuity of water pressures over the boundaries separating solid zones. Without this, the results obtained in the analysis stage will be unreliable. Thus, a pre-solve diagnostic check (see Section 22.2.1) is conducted by the software to ensure that there is no mismatch at each node along the separating boundary and a warning message is displayed if the check is failed. This may also be determined by using the **Preview Water Pressure** menu item (see Section 20.6).

However, this does not prevent the model from being solved. Where some discontinuity in water pressures is to be expected (e.g. in problems containing retaining walls or similar) then an *Aquiclude* pressure regime can be applied to a particular zone to suppress the continuity check in that area (see Section 6.2).

Where a water table and one or more water pressure regimes have been specified in the same model, the water regime will take precedent in the analysis.

The water pressure on a free surface is always assumed to be due to a water table (and is thus zero if the water table is not present or is below the free surface).

6.2.2 Water Tables

6.2.2.1 Water Tables with Hydrostatic Water Pressures

A water table (or phreatic surface) combined with an assumption of hydrostatic water pressures below this provides a commonly used representation of the water pressure conditions within a body of soil, without resorting to full seepage computations. With this idealization a water table is defined for the problem and the water pressure u at any point is given by the hydrostatic pressure as follows:

$$u = \gamma_w d \quad (6.3)$$

where γ_w is the unit weight (weight density) of water and d is the depth of the point vertically below the water table. The water pressure is taken as zero above the water table. This method of determining water pressures provides a reasonable approximation for engineering purposes, especially for water tables of shallow gradient.

The water pressures are thus governed by the rule given in Table 6.1.

Position	Pressure (kN/m ²)
Above water table	0 (dry)
Below water table	$\gamma_w d$

Table 6.1: Determination of water pressure (where γ_w is the unit weight of water, and d is the vertical depth below the piezometric surface)

Note that in the software it is possible to draw a water table with a vertical edge. This requires careful interpretation. With a water table of any gradient less than vertical, the water pressure on the inclined surface would be taken as zero. The software maintains this assumption at the limit where the water table has a vertical edge. Thus the water pressure is zero on the vertical edge, but a small distance to the side it is computed as normal according to the depth of water vertically above. Exactly below the vertical edge the water pressure is computed according to the depth below the base of the edge. These definitions are illustrated in Figure 6.3. If it is required to model a water pressure on for example a concrete dam face, but not continue the same water pressures beyond the dam, then the vertical (or steeply inclined) face of the water table should be drawn inside the dam itself as illustrated in Figure 6.4. The software will give a warning on Solve if a near vertical water table has been drawn.

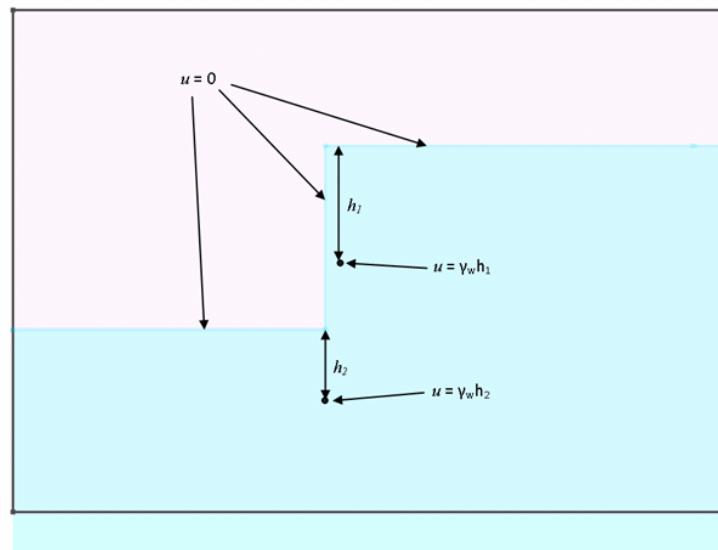


Figure 6.3: Computation of water pressures at a vertical edge

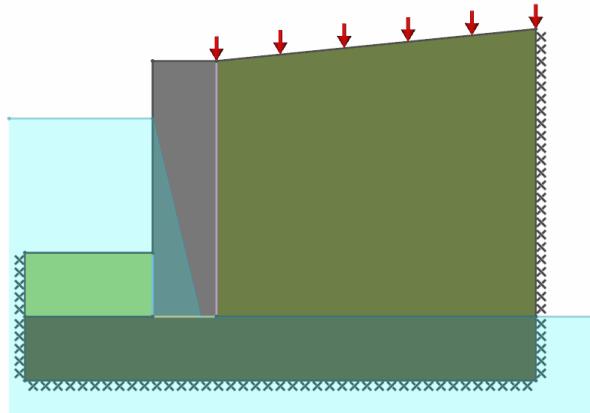


Figure 6.4: Example of how to define water table to act with hydrostatic pressure on the outer face of a dam.

6.2.2.2 Water Tables Using r_u Values

An alternative approach to representing water pressures is to utilize the average pore pressure ratio r_u . This may be defined as:

$$r_u = \frac{u}{\gamma h} \quad (6.4)$$

and thus the water pressure u at any point may be given by

$$u = r_u \gamma h \quad (6.5)$$

where γh is the overburden pressure which may be represented by defining h as the vertical depth of the point in question below the ground surface.

6.2.3 Water Regimes

Where a more detailed water table is required, LimitState:GEO is able to assign water pressure distributions on a per-**Solid** basis. In zones where a regime has been assigned, the properties of the regime will always be used in preference to those of any water table present in the problem.

The following water regimes are available:

Constant Potential The affected zone is subject to a water pressure defined by a specified constant water potential h . This potential is defined relative to a global datum level y_0 which is set in the **Property Editor** (default value = 0.0m). A multiplication factor f on the unit weight of water can also be specified.

The water pressure u at any point is therefore given by

$$u = \gamma_w f(h - (y - y_0)) \quad (6.6)$$

where y is the height of the point.

Using this regime type it is also possible to define areas containing negative pressures (e.g. to represent capillary rise). However note that at a free surface the water pressure is assumed to be zero (or defined by a water table if present above the surface) and thus there would be a mismatch of pressures if capillary rise was modelled in the soil below. To model the effect of a water meniscus at a surface would require the additional application of a surcharge (see [Smith & Gilbert 2007b](#)).

Constant Pressure The affected zone is subject to a specified constant water pressure.

This regime is typically used to assign areas of zero pressure or constant suction.

Interpolated Grid The affected zone is subject to spatially varying water pressures defined by a series of user-specified points and associated pressure values.

Intermediate pressures are calculated using a bilinear interpolation algorithm (see Appendix E). The points comprising an interpolated grid are not restricted to any particular zone, allowing a single grid to be applied to multiple areas or the entire model.

Aquiclude The affected zone is subject to zero water pressure.

The assignment of an aquiclude regime (a) prevents the software from checking the continuity of pressures at the boundaries of that zone (b) allows the full water pressures in adjacent zones to act on the boundary of the body as a mass.

This provides a convenient approach for modelling undrained or impermeable objects (e.g. mass concrete) subject to differential water pressures, without having to specifically define a gradient in water pressure across the body in question.

6.2.4 Water Pressure Hierarchy

As there are several methods by which water pressures can be defined in LimitState:GEO, an order of precedence is enforced. With the **Property Editor** setting **Water** set to *True*, the hierarchy by which pressures are used in the calculations is as follows:

1. Water regime
2. Water table (or r_u surface)
3. None (zero water pressure)

For example, if a water table is defined but a solid below the water table also contains a water regime, the pressures of the latter will be assumed when calculating a solution (even if these are lower than the pressures that would be provided by the water table).

6.3 Seismic Loading

6.3.1 Modelling of Seismic Loading Using DLO

In common with most Limit Analysis procedures, the DLO method can be extended to handle pseudo-static accelerations to enable analysis of problems subject to seismic or earthquake loading. If the accelerations are as follows:

$$k_h = \text{horizontal acceleration} / g$$

$$k_v = \text{vertical acceleration} / g$$

where g is the acceleration due to gravity (9.81 m/s^2), then additional inertial body forces of magnitude $k_h W$ and $k_v W$ are imposed on a body of weight W .

The adopted sign convention is that positive k_h acts in the negative x -direction (i.e. right to left) and positive k_v acts in the negative y -direction (i.e. downwards).

6.3.2 Modelling of Water Pressures During Seismic Loading

The effects of horizontal and vertical accelerations on free and pore water pressures in pseudo static calculations often require consideration of other factors in addition to k_h and k_v , and it is up to the engineer to decide how to incorporate these appropriately into a limit analysis approach as taken by LimitState:GEO.

Within LimitState:GEO the effect of seismic accelerations are only assumed to modify the unit weight of water used in the water pressure calculations to $(1 + k_v)\gamma_w$. The horizontal acceleration k_h is assumed not to affect water pressures. This has the effect of correctly modelling lateral earth pressures on a retaining wall according to the method of [Matsuzawa et al. \(1985\)](#), for low permeability backfill soils. To address highly permeable soils by the method of [Matsuzawa et al.](#)

(1985) it would be necessary to modify k_h by the ratio of γ_{dry} to γ_{sat} of the soil being modelled. If more than one soil is modelled then a representative or average k_h factor would need to be utilised. In addition an additional horizontal dynamic water pressure cannot be modelled directly. Note also that the 'static' water pressure would be computed as $u = (1 + k_v)\gamma_w z$ where z is the depth below the water table.

If required, water pressures can be additionally modified using the approaches described in Section 6.2.

6.4 Soil Reinforcement

In LimitState:GEO soil reinforcement may be modelled directly as e.g. a long thin **Solid** or using the special material type **Engineered Element**. The latter is recommended in LimitState:GEO.

The advantage of the latter approach over the former, for a plane strain analysis program such as LimitState:GEO, is that it is implemented such that it can allow soil to flow around or 'through' the reinforcement - as would be expected for e.g. a soil nail. This behaviour can, however, be suppressed for sheet reinforcement such as a geotextile fabric (if appropriate). Modelling of bending and / or tensile failure is also much more straightforward for an **Engineered Element** than for a solid zone.

The theory behind **Engineered Elements** is discussed in Section 6.1.5. The practical use of **Engineered Elements** in LimitState:GEO is discussed in Chapter 11.

Chapter 7

Limit Analysis: Advantages and Limitations

7.1 Introduction

The analytical method utilised in LimitState:GEO is based on the core principles of limit analysis. Limit analysis is a powerful analytical technique with a long history of application in geotechnical engineering, and has been used to determine many of the ultimate limit state (ULS) bearing capacity and retaining wall formulae typically used by geotechnical engineers. It features in many standard undergraduate geotechnical engineering textbooks.

In common with all models, the model of soil behaviour utilised in limit analysis is an approximation of the true behaviour of soil. Utilised by engineers who have a clear understanding of soil mechanics and the nature of the limit analysis approximation, it provides a powerful, yet simple analytical tool for the determination of the collapse state of a wide range of geotechnical constructions.

The following subsections discuss some of the advantages and limitations of limit analysis. This is not intended to be an exhaustive list. For a more in depth treatment, the reader is referred to the literature e.g. [Chen & Liu \(1990\)](#) and [Chen \(2008\)](#).

7.2 Simplicity

Unlike elasto-plastic FE analysis which typically requires many iterations in order to arrive at a ULS solution, numerical limit analysis seeks out the solution directly by coupling optimization techniques and rigorous plasticity theory. Limit analysis thus has the advantage that it can directly determine a solution and normally suffers from no numerical instabilities. Formulated as a linear programming problem, it is also guaranteed to find the global optimum (for a given nodal discretization).

Limit analysis also has the advantage that it typically only requires two strength parameters for any material modelled, the cohesion (c) and the angle of shearing resistance (ϕ). However more complex yield surfaces may be specified.

7.3 Stress States in Yielding and Non-yielding Zones

In the context of DLO the aim is to find a mechanism that results in collapse under the lowest load. The corollary of this is to find a network of discontinuities where the stress state is on the point of yielding. Solutions can thus be used to correctly represent force distributions in yielding (failing areas). Outside these areas the solver needs only to find a set of forces that does not cause yield, no other conditions being stipulated. If these are examined, then erratic distributions may be observed. This is not an error, but an inherent result of the application of plasticity theory which is only concerned with the collapse state.

Thus LimitState:GEO will generate suitable force distributions that can be used to check the ultimate limit state in a structural element, but it should not be used (or expected) to generate force distributions for determining deflections in structural members that are not yielding or adjacent to yielding material.

7.4 Yield Surface

The most common yield surface utilised in standard textbook limit analysis problems is the linear Mohr-Coulomb model as shown by the solid line in Figure 7.1, and defined by parameters c' and ϕ' for drained analysis. Use of a linear yield surface facilitates the simple derivation of closed form solutions.

In reality uncemented soils do not possess shear strength at zero normal stress (σ'). In these cases the linear yield surface (solid line in Figure 7.1) is an approximate fit to the true yield surface e.g. the dashed line in the figure. This provides a good approximation at higher stress levels, but not at lower stress levels, and can lead to over-prediction of collapse load in certain problems.

A non-zero value of c' is often associated with a yield surface defined in terms of the peak strength of a dense soil. A yield surface defined in terms of the critical state strength typically has a cohesion intercept of $c' = 0$.

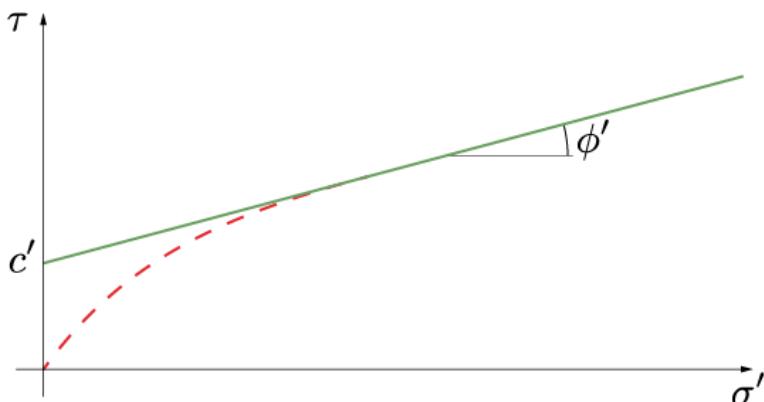


Figure 7.1: Linear Mohr-Coulomb yield surface

Computational limit analysis methods such as DLO as used in LimitState:GEO are not re-

stricted to modelling linear yield surfaces. However for simplicity in LimitState:GEO curved yield surfaces are defined by *combinations* of linear surfaces. For example the Hoek-Brown yield surface (solid line in Figure 7.2) is closely approximated by three linear yield surfaces (dashed lines in the figure). The solver will always choose the lowest point of any set of yield surfaces.

Such combined yield surfaces may be defined in LimitState:GEO using the multi-material facility described in Section 6.1.6.

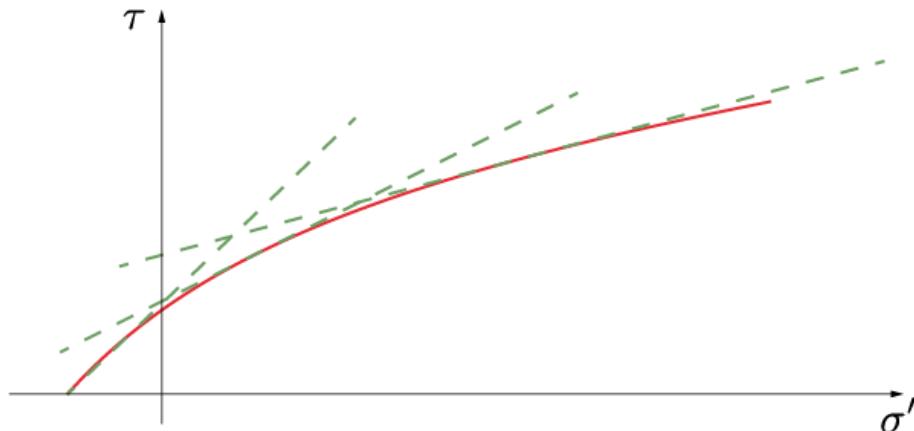


Figure 7.2: Approximating a Hoek-Brown yield surface by three linear Mohr-Coulomb yield surfaces

7.5 Strain Compatibility

The inherent assumption in limit analysis that collapsing bodies mobilize the same strength everywhere at the same time. In reality strains are likely to differ within the problem domain and thus different soil strengths will be mobilized in different locations at collapse. This may be exacerbated where there are two or more materials present in the problem with very different stress-strain characteristics. For example consider the problem of a footing founded on a layer of dense sand, overlying soft clay as depicted in Figure 7.3. In the collapse mechanism it is likely that the strains in each layer will be of broadly similar magnitude. However If the peak strengths of each material are used in the model then these can correspond to very different strain levels as depicted by the circles in Figure 7.4. This will in turn is likely to lead to an over prediction of the collapse load.

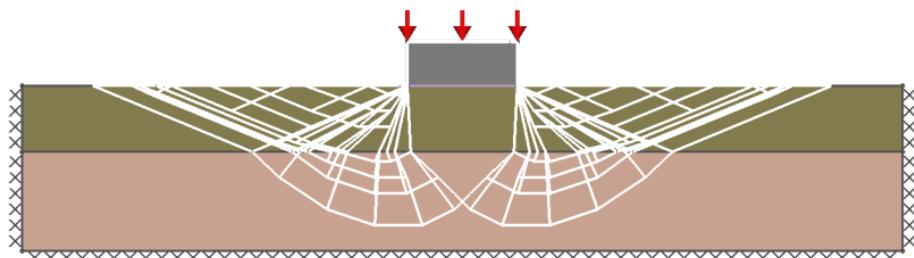


Figure 7.3: Collapse mechanism for a footing founded on a layer of dense sand overlying soft clay.

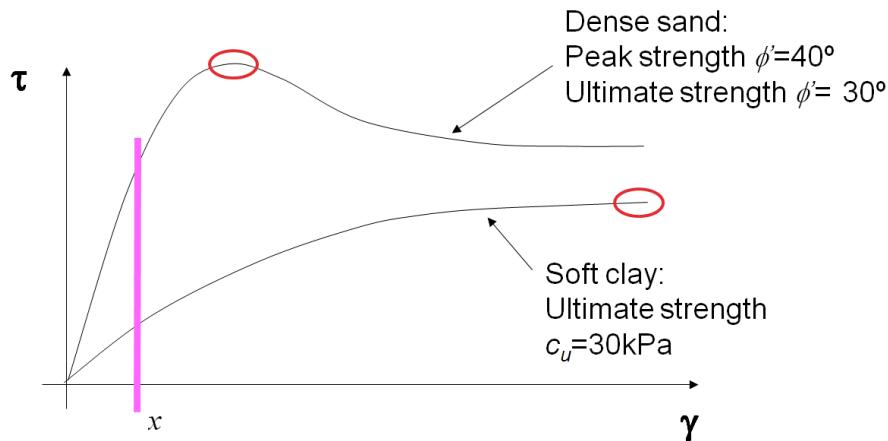


Figure 7.4: Example stress-strain responses for dense sand and soft clay (y -axis representation is not necessarily to scale for each material).

This is an issue that has always required consideration by geotechnical engineers when using any form of limit analysis solution. A conservative recommendation often made is to assume the critical state strength for all materials. At sufficiently high strains most soil materials should reach their critical state (or ultimate) strengths thus rendering the limit analysis assumption valid as long as excessive geometry change has not occurred in the process of mobilising this critical state strength everywhere. It may be reasonable to assume strengths higher than critical state but this requires the judgement of the engineer.

An alternative approach is to use the concept of mobilised strength, where the problem is analysed as a limit analysis problem with the ‘yield’ strength of the material set to the strength mobilised at a specified shear strain e.g. $x\%$ (as indicated by the vertical bar in Figure 7.4). If the ‘collapse’ load is determined on this basis using limit analysis, then it is argued that shear strains everywhere within the problem domain must be lower than $x\%$. The method can in principle be used as an approximate control on deformation. Further information on this approach may be found in BS8002 (BSI 1994). BS8002 recommends that assumed mobilised strengths should always be less than or equal to the critical state strength to avoid the risk of localised strain softening.

For soil-structure interaction problems such as multi-propped sheet pile walls, caution is also required when assessing bending failure of the walls between props. The software will model the ULS state with full mobilisation of the retained soil strength and associated arching. This may give predicted ULS bending moments significantly lower than an SLS calculation would determine.

7.6 Non-associativity and Kinematic Constraints

The upper and lower bound theorems of limit analysis require that the materials modelled obey the associative flow rule. In effect this requires that all shearing resistance is modelled as dilation rather than a combination of dilation and friction as occurs with real soils. This is normally accurate for undrained problems where the angle of shearing resistance is zero, however for drained problems it typically leads to a small overestimate of load capacity. In extreme cases it

can lead to volumetric locking and no collapse.

Experience has shown that for moderately unconstrained problems, the increase in load estimate is minor. However it is not possible to give guidance on its effect on all problems and the user is referred to the literature for specific guidance. It may be noted that limit analysis and therefore an assumption of associative flow is commonly used for typical geotechnical problem types in geotechnical textbooks and design codes. For example the formula for bearing capacity recommended by Eurocode 7 implies use of a limit analysis model. For further information on the effects of non-associativity for specific problem types, reference may be made to the following:

Slopes [Manzari & Nour \(2000\)](#), indicate that non-associative results for cohesive-frictional slope stability problems typically give values 3-10% lower than for the associated flow rule case (pure plasticity model). To put this into context, this corresponds to using an angle of shearing resistance in an associated flow model approximately 3% lower than the actual angle. This is of the order of 1° .

Footings [Loukidis et al. \(2008\)](#) indicate that non-associative results for the N_γ problem (i.e. a rigid rough footing on the surface of a cohesionless soil) typically give values 15-30% lower than for the associated flow rule case (pure plasticity model). To put this into context, this corresponds to using an angle of shearing resistance in an associated flow model approximately 3% lower than the actual angle. This is of the order of 1° .

Part III

Modelling

Chapter 8

Generic Principles

8.1 Model Definition and Solver

8.1.1 Model Definition

Problem geometries are built up using **Geometry objects** (see Section 15.1). The two key geometry objects relevant to model definition are:

Solid This is a 2D polygon defining a body of soil or other material. Its extent is defined by the surrounding **Boundary** objects.

Boundary This is a straight line that defines the edge or boundary of a **Solid**, or an interface between two **Solids**.

Generally the problem will be defined in terms of **Solid** objects. **Boundary** objects are automatically generated around **Solid** objects. Single **Solid** objects should be used for bodies of one material type. A problem such as a simple slope stability problem may thus consist of one solid, while a simple bearing capacity problem might consist of two solids, the footing and the underlying soil.

Boundary objects are used to define interface properties and set boundary conditions. In the example of the footing and the underlying soil, the soil/footing interface properties may be independently defined within the **Boundary** object that is the interface between the **Solid** objects representing the soil and the footing. A **Boundary** can also be used to describe a 1D **Engineered Element** such as a soil nail or sheet pile wall.

8.1.2 Solver Specification

The specification of the LimitState:GEO solver is as follows:

1. The software is designed to generate the optimal layout of slip-lines that make up the critical or failure translational sliding block mechanism for a specified plane strain problem.

2. The slip-lines are restricted to those that connect any two nodes within a predefined grid.
3. Slip-lines are restricted to those that connect nodes within a single **Solid** object, or between a node within a **Solid** object, and a node lying on an adjacent **Boundary** object.
4. The solution is given in the form of an **Adequacy factor**. This is the factor by which specified load, material self weight or body acceleration must be multiplied by to cause collapse.
5. In LimitState:GEO problems involving rotational kinematics are modelled by permitting **Solid** objects to rotate as a single body. If such bodies rotate into a deformable body, then the rotational kinematics are modelled as equivalent translational kinematics at the interface between these bodies.

Solutions are generated using the upper bound theory of plasticity. Plasticity theory is a very common technique utilized in geotechnical engineering. It is assumed that the user is fully familiar with the advantages and limitations of plasticity theory. Discussion of some of the advantages and limitations may be found in Chapter 7.

8.2 Adequacy Factor and Factors of Safety

8.2.1 Introduction

Many different definitions of factors of safety (FoS) are used in geotechnical engineering. Three in common usage are listed below:

1. Factor on load.
2. Factor on material strength.
3. Factor defined as ratio of resisting forces (or moments) to disturbing forces (or moments).

The calculation process used to determine each of these factors for any given problem will in general result in a different failure mechanism, and a different numerical factor. Each FoS must therefore be interpreted according to its definition.

In general any given design is inherently stable and will be well away from its ultimate limit state. Therefore, in order to undertake an ultimate limit state (ULS) analysis it is necessary to drive the system to collapse by some means. This can be done implicitly or explicitly. In many conventional analyses the process is typically implicit. In a general numerical analysis it must be done explicitly.

There are three general ways to drive a system to ULS corresponding to the FoS definitions previously mentioned:

1. Increasing an existing load in the system.
2. Reducing the soil strength.

3. Imposing an additional load in the system.

LimitState:GEO solves problems directly using either Method 1 or 2, by means of the **Adequacy factor**. However, it is also relatively straightforward to find a Factor of Safety appropriate for Method 3.

The **Adequacy factor** is defined as the factor by which specified loads must be increased, or material strengths decreased, in order for the system under consideration to reach a collapse state. There are thus two types of **Adequacy factor** used in the software:

- **Adequacy factor** on load
- **Adequacy factor** on soil strength

In principle both may be defined for any given problem, though typically an **Adequacy factor** on load would be found with the **Adequacy factor** on strength = 1.0, or an **Adequacy factor** on strength found with the **Adequacy factor** on load = 1.0. (Note that these factors are applied over and above any pre-applied partial factors).

By default, 'empty' problems are set to solve for a factor of safety on the applied load(s) (Section 8.2.2) and problems generated using one of the built-in wizards (see Chapter 4) will automatically suggest the type of analysis based on the particular model being generated. However, this can be altered by selecting the desired method from a dropdown box in the **Analysis** toolbar.

In general:

Adequacy factor ≥ 1.0 the problem is safe against collapse

Adequacy factor < 1.0 the problem is unsafe against collapse.

However, note that there is not necessarily a margin of safety in the above definition. A margin of safety may be incorporated by additionally applying partial factors to the input parameters, such as soil strength or applied load (as in Eurocode 7).

Alternatively the problem can be considered in terms of deformations. LimitState:GEO utilises a rigid-plastic model of material behaviour. In this framework, if the **Adequacy factor** is greater than 1.0, there is no movement, if the **Adequacy factor** is less than 1.0, the system displaces without limit.

However in real materials it would be anticipated that there is progressively increasing displacement as the **Adequacy factor** decreases towards 1.0. The principle of mobilised strength is useful in this context. As the **Adequacy factor** decreases towards 1.0, it may be assumed that a soil is mobilising a greater and greater percentage of its full strength until an **Adequacy factor** of 1.0 is attained. Thus in principle a ULS analysis can be used to approximately control displacements in design by using a reduced mobilised strength in the analysis. Use of a reduced strength implies that the soil undergoes a correspondingly reduced strain, thus leading to small overall deformations. This is the principal utilised in e.g. BS8002.

Note that partial factor based design codes such as Eurocode 7 do not explicitly compute a factor of safety, but pre-apply factors to problem parameters. Application of this approach in LimitState:GEO is described in more detail in Section 8.3.

8.2.2 Method 1 - Factor of Safety on Load(s)

Consider the problem in Figure 8.1. The question that is posed here is as follows:

"How much bigger does the load need to be to cause collapse?"

or,

"By what factor, a, does the load need to be increased to cause collapse?"

This factor a is the same as the **Adequacy factor** on load as reported by LimitState:GEO when in **Factor Load(s)** mode.

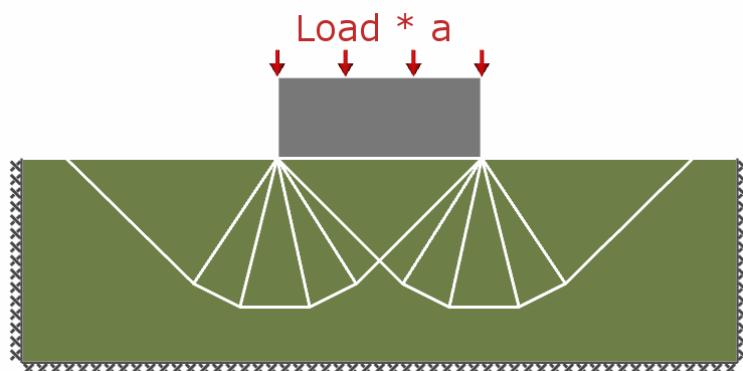


Figure 8.1: Factor of Safety on Load (Method 1)

A **Factor Load(s)** analysis can be conducted by selecting the appropriate option in the **Analysis** tab of any wizard or by choosing it from the drop-down menu in the **Analysis Toolbar**. In this mode the **Adequacy factor** on strength = 1.0.

The user may select which load or load(s) have the **Adequacy factor** on loads applied to them by setting the property **Adequacy** in the **Property Editor** to true. This property is available for **Loads**, **Solid** self weights and **Seismic accelerations**.

Where the **Adequacy factor** is applied to more than one action then it is important to be clear about the interpretation of the result. The presented solution is that for which all the actions that have **Adequacy** applied to them, are multiplied by the **Adequacy factor**. This means that any displayed stress distributions (including tensile/compressive stresses, and bending moments in engineered elements) correspond to these factored actions.

Detailed information about the actions that have been factored and the corresponding factored values can be obtained by clicking on the displayed **Adequacy factor**. This will display the dialog displayed in Figure 8.2.

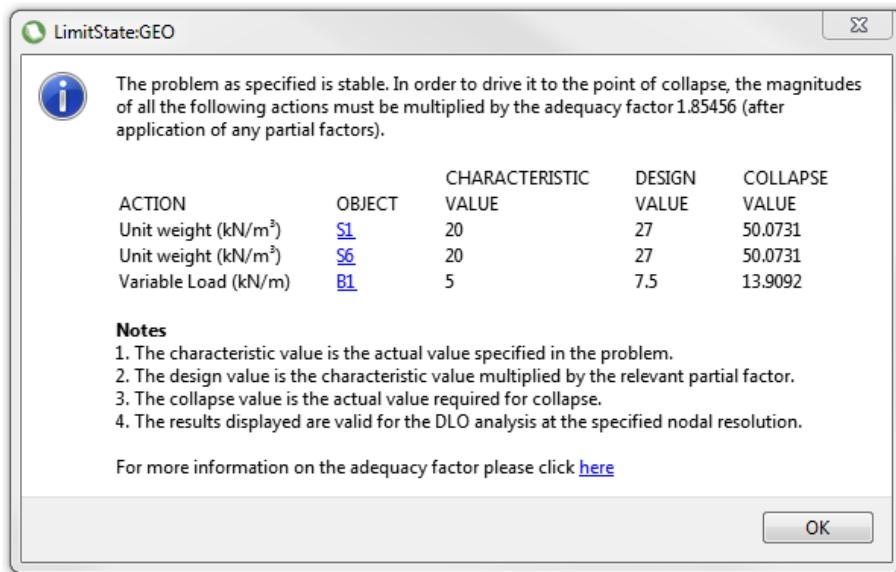


Figure 8.2: Example of detailed information about application of the adequacy factor in **Factor Load(s)** mode

8.2.3 Method 2 - Factor of Safety on Material Strength(s)

Consider the problem in Figure 8.3. The question that is posed here is as follows:

"How much weaker does the soil need to be under the design load to cause collapse?"

or,

"By what factor F does the soil strength need to be reduced to cause collapse?"

This factor F is the **Adequacy factor** on soil strength(s) reported by LimitState:GEO when in **Factor Strength(s)** mode

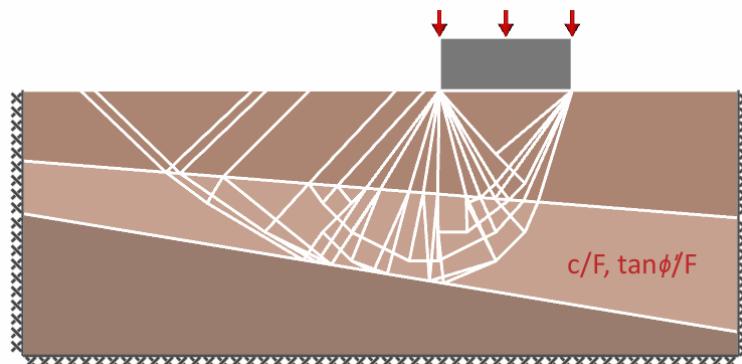


Figure 8.3: Factor of Safety on Material Strength (Method 2)

A **Factor Strength(s)** analysis can be conducted by selecting the appropriate option in the **Analysis** tab of any wizard or by choosing it from the drop-down menu in the **Analysis Toolbar**.

In this analysis, the software will automatically vary F , using an interpolation bisection algorithm, to obtain a solution that results in collapse with the **Adequacy factor** on load = 1.0.

The value of F that produces this is the **Adequacy factor** (or factor of safety) on soil strength. This factor is applied to all Mohr-Coulomb materials in the problem and therefore to values of c' , $\tan\phi'$, and c_u as appropriate. This factor is not applied to Cutoff, Rigid and Engineered Element material properties.

Detailed information about the soil strengths that have been factored and the corresponding factored values can be obtained by clicking on the displayed **Adequacy factor**. This will display the dialog displayed in Figure 8.4.

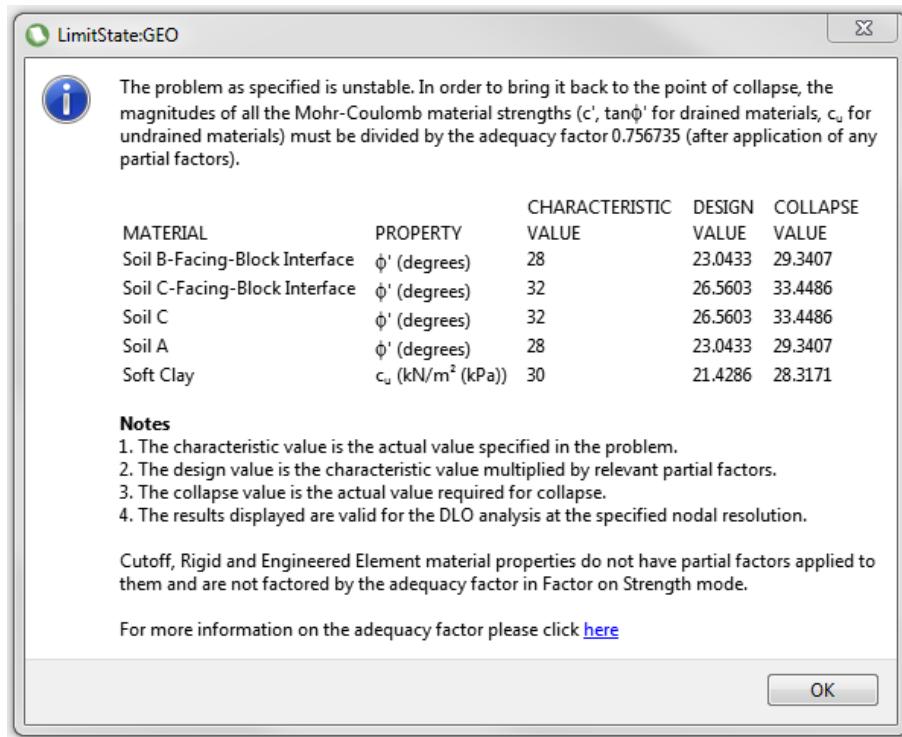


Figure 8.4: Example of detailed information about application of the adequacy factor in **Factor Strength(s)** mode

The presented solution is that for which all the soil strengths are divided by the **Adequacy factor**. This means that any displayed stress distributions (including tensile/compressive stresses, and bending moments in engineered elements) correspond to these factored strengths.

The relationship between F and the the **Adequacy factor** on load can be smooth or highly non-linear. The returned **Adequacy factor** on strength therefore corresponds to either:

1. A solution where the **Adequacy factor** on load is within 0.0001% of the specified target value (normally 1.0).
2. A bracketed solution where values of the **Adequacy factor** on strength are found that generate **Adequacy factor** on load that bracket the target value (normally 1.0). If the higher value is within the specified tolerance of the lower value then the search terminates and the average value is returned as the solution.

The tolerance on the bracketed solution is governed by the **Analysis Options > Factor Strength(s) > Solution Tolerance (%)** (set at **Project** level in the **Property Editor** or via the **Analysis** menu).

Note that for the bracketed solution, it is possible that one or both of the two bracketed results could be a *Locked* or *Unstable* result. In such a case no actual solved result (i.e. mechanism) may be available.

If one of the last bracketed values solved as normal, then this result will be available. However it may not correspond to an **Adequacy factor** on load of exactly 1.0 and this must be borne in

mind when e.g. interpreting internal stresses in the result. To generate an **Adequacy factor** on load closer to 1.0, the solution tolerance should be reduced.

Alternatively if the final result of the iteration process does not produce a solution, then it can be re-run as a **Factor Load(s)** analysis with the identified factor on strength applied via the **Scenario Manager**. For purely frictional problems such as a simple slope, it may additionally be necessary to introduce a small cohesion into the problem.

Advanced Features

By default, when the **Analysis Type** is set to **Factor Strength(s)**, all actions in the system (loads, weights of material and seismic body forces) are set to have **Adequacy** equal to *true*. This approach will automatically apply overall unfavourable conditions to the system in order to allow the correct factor on strength to be identified.

The default settings can be overridden by setting the advanced feature **Automatic Adequacy on Load(s)** to *False* in the **Property Editor** and then assigning **Adequacy** to only the areas of interest (e.g. **Loads** or self weight of **Solids**). Solving again should then result in a solution being obtained. Toggling the setting back to *True* will revert all **Adequacy** settings to *True*.

The target **Adequacy factor** on load(s) may also be changed from 1.0 using the advanced feature **Factor on Load(s)** in the **Property Editor**. For example when using **Adequacy** only on seismic accelerations, it can be useful to set the target **Adequacy factor** on load(s) to 0.0 to find the correct **Adequacy factor** on strength (i.e. when the seismic accelerations are zero).

In problems containing purely frictional materials, a **Factor Strength(s)** analysis can cause slow convergence towards an answer or an *Unknown* solution being returned. This can generally be alleviated by the provision of a small **Artificial Cohesion** value, which improves convergence and has the added effect of ensuring that a failure mechanism is identified. By default the value of this cohesion is set to 0.1kPa, but this can be modified by the user if appropriate. Typically a small value of 0.1kPa will have a minor effect on the value of the **Adequacy factor** identified, but certain problems (e.g. of small dimensions) may be affected more significantly.

8.2.4 Method 3 - Factor of Safety on Ratio of Forces

Consider the problem in Figure 8.5. The question that is posed here is as follows:

"If the soil is failing around the structure, what is the ratio R of resisting forces to disturbing forces?"

In this example, the factor $R = (P + S)/A$ is the factor of safety of interest.

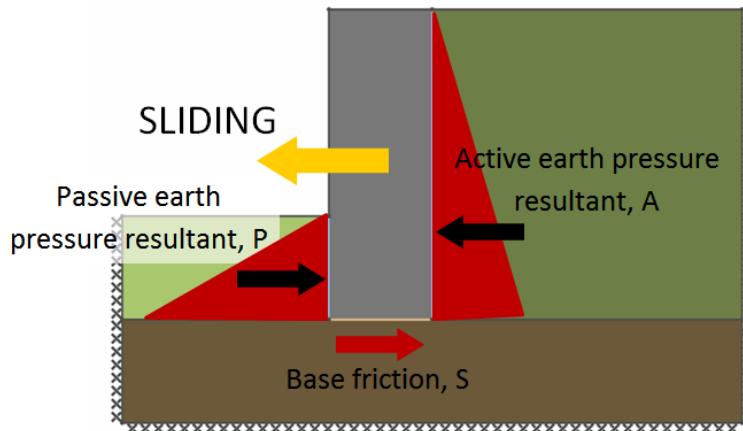


Figure 8.5: Method 3 Factor of Safety

Note that if $R > 1$:

- The passive earth pressure and base friction significantly exceed the active earth pressure.
- The system is therefore out of equilibrium.
- The assumed earth pressures are not possible without some external disturbing agent.

In a numerical analysis, equilibrium is required at all times. Therefore in order to undertake a 'Method 3' type analysis, it is necessary to apply a 'hypothetical' external force H in the direction of assumed failure (as depicted in Figure 8.6) and increase this force until failure occurs. It is then possible to determine the ratio of other resisting to disturbing forces as before (but H itself is ignored in this calculation). For this method it is therefore necessary to pre-determine the mode of failure.

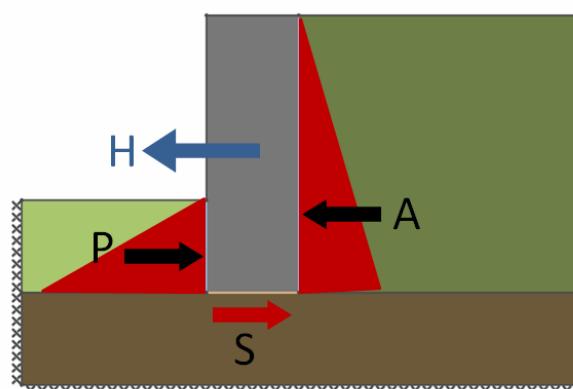


Figure 8.6: Method 3 usage of hypothetical external force to determine Factor of Safety

8.2.5 Application of the Adequacy Factor on Load

8.2.5.1 Typical Usage

The **Adequacy factor** on load may be applied to one or more of the following parameters that result in a force within a problem:

1. an **applied load** (see Section 19.1),
2. a material **self weight** (see Section 19.2),
3. a seismic **acceleration** (see Section 21).

The reference in brackets refers to the part of the manual that details how the **Adequacy factor** is set on the given parameter.

In most problems, the **Adequacy factor** will be applied to a load. For problems such as a slope stability problem where there is no externally applied load, then the **Adequacy factor** may be applied to a material self weight or a body acceleration. This is discussed further in Chapter 10. In such cases, and where the **Adequacy factor** is applied to material self weight, it is recommended that adequacy should be applied to all materials in the problem and not to a single isolated block of soil, as discussed in Section 8.2.3.

For seismic problems, it is often most convenient to apply the adequacy factor to a body acceleration. This is discussed further in Chapter 12.

Where the **Adequacy factor** is applied to more than one parameter, then it is applied equally to each.

Note that when the **Adequacy factor** is applied to a material self weight and this material lies below the water table, then adequacy is also applied to the water pressure. In effect this means that for static groundwater conditions the **Adequacy factor** is applied to the buoyant weight of the soil.

8.2.5.2 Non-standard usage

In some cases it may be desirable to apply the **Adequacy factor** to the self weight of a single body e.g. a buried foundation or pipeline, in order to determine the factor of safety on weight for that object.

However it should be noted that where this is undertaken in a zone where water pressures are also applied the principle outlined earlier applies i.e. adequacy is also applied to the water pressure. This will mean that in the **Report** the free body diagrams for the single body will include factored water pressures while adjacent blocks of soil will not. It will thus appear as if adjacent blocks are *not* in equilibrium with the single body.

In practice, for static groundwater conditions, the results are equivalent to applying the **Adequacy factor** to the buoyant weight of the single body. For non-static groundwater conditions

there is no simple equivalent interpretation. For simplicity and clarity it is therefore recommended that:

1. The **Adequacy factor** is applied to *all* bodies in a problem in which a water table is present (or preferably the **Adequacy factor** is applied to a global body acceleration).
2. If it is desirable to apply the **Adequacy factor** to the self weight of a single body, then where possible, it is applied instead as an **Adequacy factor** on an additional load on that body (for solid bodies that may mean ‘cutting’ a hole in the body to allow application of the load).

Where the above is not possible, then the **Adequacy factor** should be applied to the self weight of the body and care should be applied in the interpretation of the results in cases of non-static groundwater. However it should be noted that in all cases where the **Adequacy factor** equals 1.0, then all quantities require no special interpretation.

8.2.6 Adequacy Factor Sensitivity

As has been previously mentioned, LimitState:GEO provides solutions in terms of **Adequacy factor**. An **Adequacy factor** may be applied to any load or to the self weight of any body of material. The **Adequacy factor** that is returned by LimitState:GEO when it has completed solving is either:

- the factor by which all the specified loads/self weights/seismic accelerations must be multiplied by to cause collapse, or
- the factor by which the material strengths must be divided by to cause collapse.

It is important to note that if there are several actions driving collapse, yet an **Adequacy factor** is applied only to one of them, then the **Adequacy factor** may seem to have a misleadingly high sensitivity to parameter changes.

For example in the Gravity Wall problem in Figure 8.7, both the surface load and weight of soil behind the wall are driving it to collapse. If the **Adequacy factor** is applied only to the surface load, but the load on the wall is dominated by the soil self weight, then large changes in **Adequacy factor** will be required to cause any change in collapse state. Indeed if the wall would collapse due to the soil self weight alone, then no value of **Adequacy factor** could be found to (just) cause collapse and a value of *Unstable* will be returned.

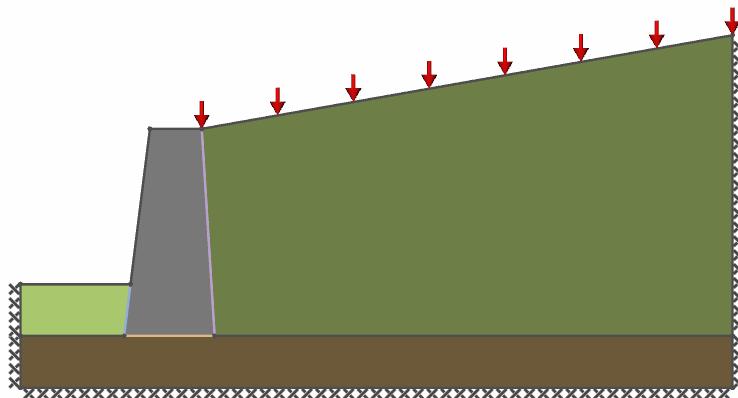


Figure 8.7: Gravity Wall Problem

8.2.7 Adequacy Factor Direction

Occasionally, LimitState:GEO may generate a failure mechanism that is unexpected by the user. This can sometimes relate to how the **Adequacy factor** is used.

When an **Adequacy factor** is applied to a load or self weight or acceleration, then associated with the factor is an **Adequacy Direction** (AD) . This direction is defined as follows:

Load The AD is in the direction of application of the load and relates to the area of application of the load only.

Material Self Weight The AD is directed vertically downwards and relates to the entire zone to which the **Adequacy factor** is applied.

Acceleration The AD is in the direction of the seismic acceleration (horizontal or vertical) taking into account the sign. This AD relates to the entire problem geometry.

The mathematical formulation of DLO utilised in LimitState:GEO requires that the identified critical failure mechanism must result in net positive work being done by the parameter to which the **Adequacy factor** is applied. In simple terms it means that the failure mechanism must result in collapse that involves net movement in the AD.

For example the movement of a load on a rigid foundation at collapse must be in the direction of the AD. Note that the component of movement of the load perpendicular to the AD is unrestricted.

For investigating the collapse of a slope in a single zoned body of homogeneous soil , the **Adequacy factor** is often applied to the soil self weight. In this case the identified collapse mechanism must involve more of the soil moving in the direction of the AD than away, i.e. more must move downwards than upwards.

This principle will normally give rise to intuitively expected solutions and does not normally require further consideration. However consider the issues involved in determining the active and

passive collapse loads for the rigid wall (with smooth base) depicted in Figure 8.8. Active collapse of the wall would result in a required force on the wall less than that for passive collapse. However even though LimitState:GEO finds the smallest **Adequacy factor** that will result in collapse, it will find the *larger* passive value because only this mechanism results in the applied load moving in the direction of its application (the AD) as shown in Figure 8.8. Intuitively this is what would normally be expected.

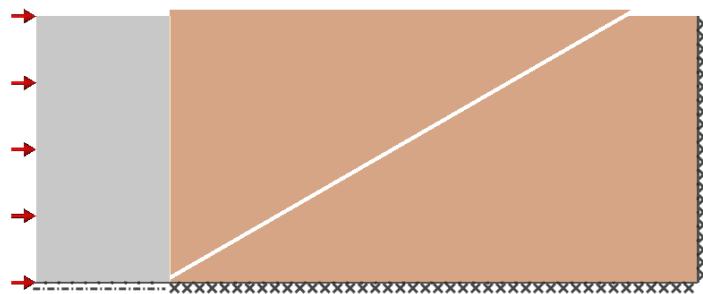


Figure 8.8: Investigating passive failure of a gravity wall. Base of wall is modelled as smooth, utilising the 'symmetry' boundary condition.

The challenge does however remain as to how to determine the active force required for collapse. It is not possible to simply reverse the direction of the force to which the **Adequacy factor** is applied. In this case LimitState:GEO will return 'Unstable' since there is no positive value of the force that would bring the problem just to the point of collapse (only positive values of **Adequacy factor** can be found).

In order to find the active force, it is necessary to additionally apply a passive dead load DH against the wall (where D is the applied stress in kN/m^2 , and H is the height of the wall in m) that exceeds the expected active force (any value will do) as depicted in Figure 8.9. In this figure the green arrows represent the dead load (D) while the red arrows represent the load to which **Adequacy factor** is applied. Let this be a unit stress (1 kN/m^2). The solver will thus find the maximum value of **Adequacy factor** to cause collapse in the AD (i.e. active direction) as shown in Figure 8.9. The required active load may be computed as a force of $(D - AF \times 1) \times H$.

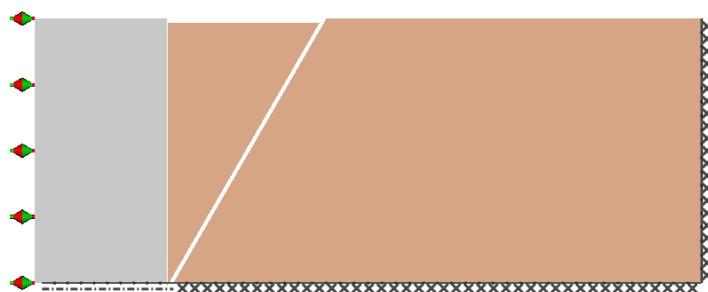


Figure 8.9: Investigating active failure of a gravity wall

A further example illustrating the effect of AD in problems involving seismic actions is given in Section 12.4.

Finally it should be noted that where an **Adequacy factor** is applied to more than one parameter, then the identified collapse mechanism must involve movement in at least one of the specified **Adequacy** directions.

8.3 Use of Partial Factors

8.3.1 Introduction

LimitState:GEO is designed to work closely with the Eurocode 7 approach to Ultimate Limit State design. It has therefore adopted the Eurocode 7 definitions of actions and partial factors, which may be used if required in any analysis. These are sufficiently broad based enough to cover the needs of most other design codes.

In any design code it is important to distinguish whether factors are to be applied before (input factoring) or after (output factoring) the analysis phase. The LimitState:GEO partial factoring system is designed to facilitate input factoring which is typical for material factoring approaches such as Eurocode 7 Design Approach 1 Combination 2 and Design Approach 3, and is frequently adopted for action factoring approaches such as Eurocode 7 Design Approach 1 Combination 1. In general, input factoring makes the best use of the power of numerical modelling (see for example [Smith & Gilbert \(2011a\)](#) and [Smith & Gilbert \(2011b\)](#)). Action and resistance factoring approaches such as Eurocode 7 Design Approach 2 will generally adopt output factoring. Such an approach may be achieved using LimitState:GEO, by specific designation and comparison of actions and resistances acting on a specific body. This at the choice of the user. Once designated, such actions and resistances can be determined from the analysis and factored as appropriate.

For input factoring, partial factors are pre-applied to loads (as multipliers) and/or material properties (as divisors) prior to analysis. Assessment of safety is then undertaken by testing whether in the subsequent analysis, the available resistance to collapse exceeds the actions causing collapse. In LimitState:GEO this is equivalent to checking whether the **Adequacy factor** (applied to any unfavourable load or self weight, or to strength), is greater than 1.0. (Note that the **Adequacy factor** is applied over and above any pre-applied partial factors).

The setting of Partial Factor values is carried out using the **Scenario Manager**. The available factors that may be set are depicted in Figure 8.10. Further details about the Scenario Manager may be found in Chapter 30.

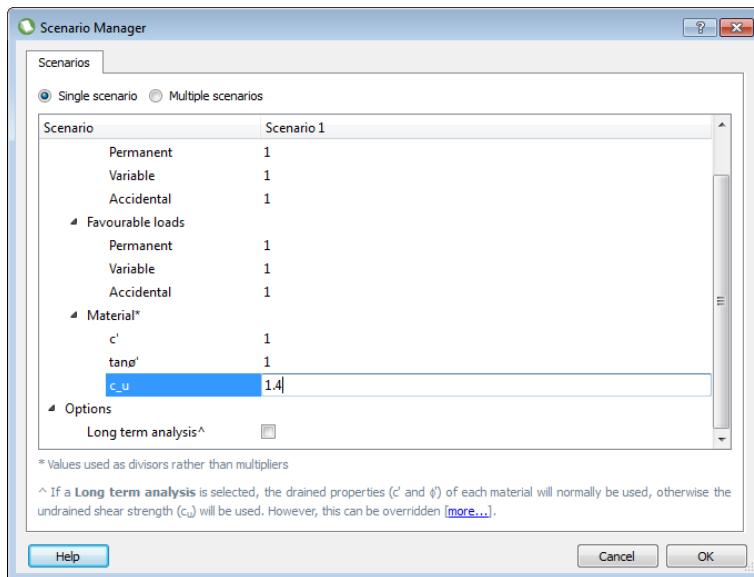


Figure 8.10: Available Partial Factors in the **Scenario Manager**

The general principles implemented in LimitState:GEO are described below. However with respect to Eurocode 7, the following is not to be taken as a definitive guide. The engineer is expected to apply their own understanding of Eurocode 7, especially with regard to some of the subtleties that can arise in certain situations. If there are any inconsistencies between what is described below and the documented Eurocode, then the documented Eurocode should be followed.

8.3.2 Factoring of Actions (Loads)

Eurocode 7 specifies three different types of actions (loads) to which multiplying partial factors can be applied. These are all available within LimitState:GEO:

1. **Permanent**
2. **Variable**
3. **Accidental**

The relevance of each action is the nature of the partial factor to be applied to it, with the corresponding values taken from the **Scenario Manager**. A Variable action will typically have a higher partial factor applied to it in comparison to a Permanent action.

Actions may be loads applied to external boundaries or may arise from the self weight of a block of material. The above settings can thus be applied to both **Boundary** loads and to **Solids**. Self weights are regarded as Permanent actions in LimitState:GEO.

Eurocode 7 also requires that each action is assessed as to its effect on the overall stability calculation. If it contributes to stability then it is Favourable, if it contributes to collapse then it

is Unfavourable. Its **Loading Type** affects the value of partial factor to be applied to it. The following **Loading Types** may be applied to any **Solid** or **Boundary**:

Favourable: Apply the favourable partial factors to any loads on a boundary or to the self weight of the materials within a solid.

Unfavourable: Apply the unfavourable partial factors to any loads on a boundary or to the self weight of the materials within a solid.

Neutral: Do not apply any factors to the loads on this boundary or to the self weight of the materials within a solid. (NB the type of action, permanent, variable or accidental has no relevance in this case.)

By default all **Boundaries** and **Solids** are set to *Neutral* when first created. It is up to the user to explicitly set them to *Favourable* or *Unfavourable* if required.

The purpose of the *Neutral* setting is to:

1. ensure that settings for any new problems that do not require analysis with partial factors, remain unambiguous and unaffected by partial factors.
2. ensure that for any problem that is to be analysed using partial factors (such as when using Eurocode 7), that the user must make explicit decisions about the nature of the actions i.e. change the setting to either *Favourable* or *Unfavourable*.
3. to facilitate modelling of problems where partial factors are to be applied to *effects of actions* rather than to the *source actions* themselves. In these cases *Neutral* might be used for the self weights of certain bodies.

Automatic factoring of *source actions* only is implemented in the current version of LimitState:GEO. To factor *effects of actions* a procedure similar to that described in Section 8.2.4 may be followed manually.

Note that in the Wizards, external loads are preset to *Favourable* or *Unfavourable* as appropriate. The self weight of structural elements such as footings may also be set to *Unfavourable* where they are unambiguous unfavourable actions.

For certain problems it can be a matter of debate as to whether the self weight of a soil body acts favourably, unfavourably or both. Thus in the LimitState:GEO Wizards, soil body self weights are always set to *Neutral* and should be amended by the user as appropriate.

In Eurocode 7, *Neutral* is equivalent to *Favourable* in Design Approach 1, Design Combination 1. In Design Combination 2, the factors on permanent actions are the same for both *Favourable* and *Unfavourable* effects, and therefore the setting is irrelevant.

For certain problems it can be unclear at the start whether a particular load is *Favourable* or *Unfavourable*. LimitState:GEO provides additional assistance in these cases. Following determination of the collapse load LimitState:GEO performs a check on all external actions to determine whether they acted favourably or unfavourably. If these are inconsistent with the original specifications, then the user is alerted to this (see Section 19.4) and may alter the specification and re-solve.

In a very small number of cases it is possible that the amended Favourable / Unfavourable settings may result in a different collapse mechanism and another set of inconsistent Favourable / Unfavourable settings. This is not a inherent problem with LimitState:GEO but simply a consequence of the Partial factor values. As always in these cases it is up to the engineer to apply their own judgement consistent with the principles underpinning the design code.

8.3.3 Factoring of Material Properties

Partial factors may also be applied as divisors on material properties. Different factors are applied to the key parameters controlling collapse: the drained cohesion intercept (c'), the tangent of the angle of shearing resistance ($\tan \phi'$) and the undrained cohesion (c_u). In general self weight (regarded as a material property rather than as contributing to an action) is not factored.

8.4 Solution Accuracy

8.4.1 Introduction

This section discusses solution accuracy within the context of the DLO numerical method itself. For discussion of the accuracy of limit analysis in general, refer to Section 7.

As with any numerical method, solution accuracy is dependent on the resolution of the underlying model. With DLO this relates to the distribution of nodes within **Solid** and **Boundary** objects. The method will provide the most critical sliding block mechanism that can be generated using slip-lines connecting any of the nodes. In many cases a sufficiently accurate solution will be generated for a coarse distribution of nodes. To assess solution accuracy, it is recommended that the nodal resolution be progressively refined, thereby allowing an assessment of the convergence characteristics to be made (towards the ‘exact’ solution).

8.4.2 Benchmarking Results

When comparing LimitState:GEO results with known analytical solutions, it is important to interpret them with regard to the sensitivity of the result to parameter variation.

For example, the bearing capacity problem is notoriously sensitive to small changes in the angle of friction. The predicted bearing capacity of a surface footing on a cohesionless soil of high strength can double with an increase of friction angle of $\approx 3^\circ$.

In these circumstances it is logical to think about the problem in terms of the input values rather than the output values which is the philosophy taken, for example, by Eurocode 7 Design Approach 1.

To illustrate this, consider the example of a simple footing problem, with 1m wide footing, soil parameters: $c'=5$ kPa, $\phi=30$ degrees, $\gamma = 20$ kN/m³, smooth base, and soil domain 0.75m

high by 2.25m wide. The exact benchmark solution for the collapse load is 268 kN. The results obtained from LimitState:GEO using a symmetrical half space model, and the listed nodal densities are as follows:

- Coarse 286 kN (6% variation from 268kN)
- Medium 282 kN (5% variation from 268kN)
- Fine 277 kN (3% variation from 268kN)

However if c' and $\tan \phi$ are decreased by 1% to give $c' = 4.95\text{kPa}$ and $\phi = 29.75^\circ$ then the solution at a fine nodal resolution is the same as the exact benchmark solution. Thus the error in input parameters is 1%. It would be hard to predict c' and ϕ to an accuracy of 1% from site investigation/lab test data.

For the medium nodal density the error is 1.5% : $c'=4.925 \text{ kPa}$, $\phi = 29.63^\circ$. For the coarse nodal density the error is 1.66% : $c'=4.917 \text{ kPa}$, $\phi = 29.50^\circ$.

LimitState:GEO is regularly benchmarked against a known set of limit analysis solutions from the literature. These tests are described in more detail in Appendix A and may be accessed via the internet. Reference to these results can provide useful guidance as to the expected accuracy of the software over a range of problem types.

Other factors that can affect accuracy are described in the following sections.

8.4.3 Interfaces

To maximize computational efficiency, the solver does not model slip-lines that cross the interface (**Boundary** object) between one solid and another. The smoothness of the solution in the vicinity of the boundary is thus dependent on the nodal density on that boundary. LimitState:GEO automatically assigns a higher nodal density on boundaries and the net effect on the solution is usually very small. However for coarse nodal resolutions, the effect can be noticeable. Thus if a **Solid** object is split in half by a new **Boundary** object, then the value of the solution may increase slightly. The user may individually set the nodal density on boundaries (see Section 22.3.3). The software will provide a warning in the **Diagnostics window** if **Boundary** nodal densities are mismatched with adjacent **Solid** nodal densities.

When using **Engineered Elements** it is also necessary to ensure a high nodal density is present on the relevant boundaries to ensure accuracy.

8.4.4 Small Solid Areas

As with all numerical software, numerical tolerance issues can cause the generation of unexpected results. This can occur when the software attempts to compute solutions based on numerically very small problem sizes. To assist users, LimitState:GEO undertakes a pre-solve check (see Section 22.2.1) and issues a warning when the total area of the problem is below 0.25 m^2 . If the problem is below $0.25 \times 10^{-4} \text{ m}^2$ then solve is prevented and an error is displayed in the **Diagnostics window**.

8.4.5 Singularities

The DLO procedure is particularly suited to identifying singularities, i.e. fan zones. However it must be noted that for frictional soils, stress levels can increase exponentially around the fan. This means that solutions with singularities may be particularly sensitive to the number of slip-lines and thus nodal resolution in these areas. To improve the accuracy of a solution, increasing the number of nodes within **Solid** objects which contain dense patterns of slip-lines is likely to be beneficial.

8.4.6 Model Extent

In many cases, the problem domain is not of finite size, but is typically of semi-infinite extent, e.g. a footing on horizontal soil. In this case the user must specify a suitable domain that is large enough not to influence the solution. Since LimitState:GEO generates mechanism based solutions, it is straightforward to determine if the boundaries do or do not influence a solution; if the mechanism intersects or touches any of the boundaries, then it is likely that they have an influence, and may lead to a significant increase in load capacity. If the intention is to model an infinite or semi-infinite body of soil , then the boundaries should be moved outwards until the mechanism does not intersect or touch the given boundary.

8.4.7 Failure Mechanisms Dominated by Rotations

In problems where significant rotational displacements of the soil occur (e.g. if modelling a vane test), the **Edge Rotations** model employed by LimitState:GEO can produce a mechanism that dissipates too much energy. Refer to Section 5.3.3 for further discussion of this issue and approaches that may be employed to minimize its effect.

8.5 Adapting Plane Strain Results to 3D

It is not possible to give specific guidance on adapting plane strain results to three-dimensional cases, since the situation will vary for any given geometry.

Shape factors are generally available for converting plane strain bearing capacity results to 3D geometries. However these are generally only valid for surface footings on homogenous soil bodies . It is not necessarily valid to utilize these factors for other soil geometries e.g. layered soils .

However it can generally be said that in most cases any 3D collapse mechanism will be more unfavourable (will require a higher load to fail) than an equivalent plane strain mechanism (though defining the equivalence of mechanisms requires careful consideration). Thus a plane strain analysis such as is provided by LimitState:GEO will in general be conservative.

However it should be noted that there are exceptions to this principle. This is clearly illustrated by comparing the plane strain and axisymmetric collapse pressures for a footing founded on the surface of a cohesionless soil with angle of shearing resistance ϕ and on a soil of undrained

shear strength c_{u0} at the ground surface and undrained shear strength gradient k . Example results (from limit analysis) for rough and smooth footings are given in Table 8.1 and Table 8.2. It can be seen that the axisymmetric collapse pressure is lower than the plane strain collapse pressure for values of ϕ' less than $\sim 30^\circ$ in the drained case and for values of kB/c_{u0} greater than ~ 2.5 respectively.

	$q/\gamma B$			
	Rough		Smooth	
ϕ' (degrees)	plane strain	axisymmetric	plane strain	axisymmetric
10	0.217	0.161	0.140	0.103
15	0.591	0.466	0.350	0.267
20	1.42	1.21	0.79	0.64
25	3.25	3.04	1.73	1.49
30	7.38	7.76	3.83	3.56
35	17.2	21.0	8.79	9.02
40	42.8	61.9	21.6	25.1
45	117	209	58.8	79.9
50	371	855	186	309

Table 8.1: Comparison of plane strain and axisymmetric collapse pressures (q) for a footing of width or diameter B , founded on soil of unit weight (weight density) γ , and angle of shearing resistance ϕ' .

	q/c_{u0}			
	Rough		Smooth	
kB/c_{u0} (degrees)	plane strain	axisymmetric	plane strain	axisymmetric
0	5.14	6.05	5.14	5.69
0.5	5.97	6.54	5.59	5.98
1	6.61	6.95	5.98	6.25
2	7.60	7.63	6.66	6.72
2.5	8.02	7.93	6.97	6.94
5	9.80	9.23	8.34	7.94
10	12.7	11.4	10.7	9.64

Table 8.2: Comparison of plane strain and axisymmetric collapse pressures (q) for a footing of width or diameter B , founded on soil of undrained shear strength c_{u0} at the ground surface and undrained shear strength gradient k .

Further discussion of the relationship between plane strain and 3D collapse pressures for foot-

ings may be found in [Salgado et al. \(2004\)](#) and [Lyamin et al. \(2007\)](#).

8.6 Troubleshooting

8.6.1 Insoluble Problems

Normally when a model is set up, it is preferable to carry out initial modelling utilising a *Coarse* or *Medium* nodal distribution in order to check that the material models, geometry and boundary conditions have been correctly set up. Any error in these will become apparent in the nature of the solution or will be highlighted by the pre-solve diagnostic checks carried out by the software (see Section 22.2.1). However, certain circumstances can lead to no solution being found. These include the following:

8.6.1.1 Unstable Solutions

Parts of the problem are *Unstable*, and the computed **Adequacy factor** on load can be reduced down to zero (or beyond). Examples of such situations include:

1. Slopes in cohesionless soils steeper than the angle of shearing resistance of the soil. This can include even short sections of over steep slope on a generally shallow but undulating slope. This can also apply if the slope is at a greater angle than the material shearing resistance, but the resistance is decreased due to the use of Partial Factors.
2. Zero or insufficient fixed boundaries to hold the soil in place.
3. Zero strength soil.
4. Free floating or partially supported blocks of soil within the problem. This can sometimes not be immediately apparent if there is a narrow gap between blocks. Zooming in on suspect blocks may assist in identifying such areas. Alternatively select the suspect solid and drag it with the mouse. If it does not drag the adjacent solid with it, then it is not fully connected and there is a gap.

8.6.1.2 Locked Solutions

No solution will be found if the problem is *Locked*. In this case no value of the **Adequacy factor** on load will induce failure. Examples of such situations include:

1. Slopes in frictional or cohesive-frictional soils shallower than the angle of shearing resistance of the soils .
2. Problems where the adequacy factor is applied to both disturbing and restoring forces. For certain problems, failure cannot be induced if each force is increased simultaneously.

3. Problems where boundary conditions are set such that deformation of a body of soil is fully constrained, for example attempting to indent a soil contained within fully fixed boundaries.
4. Problems where frictional soils are modelled with boundary conditions that are highly restricted (but not necessarily fully fixed everywhere). This is due to geometrical locking. In plasticity models of frictional soil, shear deformation is accompanied by dilation. i.e. if there is no room available for the dilation to take place, then no solution will be found.
5. Problems where a partial factor of zero is applied to a load/action to which the **Adequacy factor** is also applied.

8.6.1.3 Unknown Solutions

The solver used in LimitState:GEO is generally stable. Occasionally, however, due to the nature of the way in which it works, it will fail to find a valid solution and an *Unknown* solution will be reported. In such instances it is prudent to modify the problem very slightly and **Solve** again. Aspects to modify include the geometry (e.g. move a vertex by a small distance) and the material properties (e.g. change the unit weight or strength by a small amount).

8.6.2 Troubleshooting Insoluble Problems

1. Consult the **Diagnostics** information to assist with checking that the problem is set up as required. Use of the **Solid Explorer** and the **Boundary Explorer** can also help to rapidly clarify how common properties have been set in the problem.
2. If not immediately obvious, it is recommended that the soil properties in all or suspected zones are changed to increase the influence of cohesion by increasing c_u or c' as appropriate. If a solution is then found, try gradually reducing c_u or c' until no solution is found. The cause of the problem should then be much clearer.
3. If the above does not work for a drained material, try also setting the friction to zero when first increasing the value of c' and then as c' is gradually reduced, simultaneously increase ϕ' back to its original value.
4. An alternative to the above is to reduce the soil self weight, or set it to zero. This should help to identify cases where the soil is collapsing under its own self weight.
5. Be aware that if the problem had arisen due to a small part of the problem geometry being unstable (e.g. a small section of steep slope), then the animated mechanism may not be immediately obvious since it will be small. It may be necessary to zoom in on the screen to identify it.

8.6.3 Problems Giving Solutions That Appear Incorrect

On occasion LimitState:GEO can find a solution, with an accompanying adequacy factor and/or collapse mechanism, that does not lie within the anticipated values. This can occur if the adequacy factor is inadvertently applied to parts of the problem that were not initially intended.

Commonly this occurs with the adequacy factor applied to **Solids**. It is worth checking that the adequacy factor is applied as required by interrogating the settings in the **Solid Explorer**.

In some circumstances the problem may relate to the issue of adequacy direction (see Section 8.2.7).

in long solve times or an *Unknown* solution being returned. This

Chapter 9

Example Problems

LimitState:GEO comes supplied with a range of example files covering a broad range of applications. These present recommended approaches to modelling these problem types. If guidance is being sought on modelling a specific type of problem then following the method used in these files or modifying these files to represent the required problem can be an effective approach.

Additional information on particular modelling issues associated with each example file is given in the **Project Details** box (accessible under the **Tools** menu item).

The example files may be accessed via the LimitState:GEO program menu:

Start > All Programs > LimitState:GEO 3.6.1 > Example Files.

Examples types include problems involving:

- Slope stability
- Foundations on heterogeneous soils
- Embankment stability
- Reinforced earth walls
- Propped excavation
- Quay wall
- Tunnel
- Pipeline

Chapter 10

Slope Stability Problems

10.1 Principles of Slope Stability Limit Analysis

This section outlines the principles applied when performing a slope stability analysis using LimitState:GEO.

In a conventional slope stability analysis (e.g. using the method of slices) a pre-determined slip surface is assumed and the stability of the failing soil mass is evaluated by comparing resisting and disturbing forces/momenta. Usually many trial slip surfaces are investigated and the most critical one identified. This typically requires specification of a search zone and entry and exit points, and can be very sensitive to the shape of slip surfaces used (e.g. circular or non-circular).

In contrast the general purpose limit analysis procedure used by LimitState:GEO does not require the form of the collapse mechanism to be pre-specified. However, the use of a general purpose procedure does mean that an particular approach must be adopted by the software to identify the critical failure mechanism.

10.2 Factor of Safety on Strength

In the majority of cases, the aim of a slope stability analysis will be to determine a factor of safety on the strength of the constituent materials. LimitState:GEO makes this easy for the user by providing the option to undertake a **Factor Strength(s)** analysis (see Section 8.2.3). This essentially provides the user with the facility to ‘set up and go’, without having to concern themselves with any further adjustment to the problem parameters.

In a **Factor Strength(s)** analysis, the software searches for a suitable factor on soil strengths that results an **Adequacy factor** on load of 1.0. Dependant on the types of materials present in the slope, a number of things should be taken into consideration.

10.2.1 Slopes in Cohesionless Soils

In a purely cohesionless soil, geotechnical theory indicates that the slope is:

- stable for any friction angle greater than, or equal to, the slope angle, and
- unstable for any friction angle less than the slope angle.

Collapse is therefore entirely independent of the self weight of the soil (crudely speaking, as the self weight of the material goes up, the normal stresses go up, but frictional resistance also goes up in exact proportion). For the above reasons, in a slope that is shallower than the angle of friction, factoring up the self weight on a frictional soil will not cause collapse and LimitState:GEO will return a **locked** result. Conversely, if the slope is steeper than the angle of friction then it will be found to be **unstable** under its own self weight.

For this reason, in a slope containing cohesionless soil(s) the bisection algorithm can sometimes jump between **locked** and **unstable** solutions as it converges on a solution that provides an **Adequacy factor** on load of 1.0. Mainly this process will be quick and an answer returned in a few iterations. However, on rare occasions this process will be extended and require rather more computation time. In such cases it can be prudent to override the default settings of the **Factor Strength(s)** analysis in order to expedite a solution (see Section 8.2.3).

Note that the changeover from **locked** to **unstable** is sudden in theory and this is mirrored in LimitState:GEO. For example imagine a brick lying on a plank (Figure 10.1). If one edge of the plank is lifted and the plank's angle to the horizontal is gradually increased, the brick will remain stable until the plank angle reaches the angle of friction between the brick and the plank. At this point the brick slides. Beyond this point the situation is inherently unstable. The situation is either stable or unstable, and is entirely unaffected by the weight of the brick i.e. applying the **Adequacy factor** to the self weight of the brick will not drive the problem to failure.

If however there is cohesion and friction between the brick and the plank, then weight will have an effect, but only when the angle of the plank \geq the friction angle. By analogy, in this type of problem LimitState:GEO will give solutions of **locked** for the type of case where the slope angle is less than the friction angle, but will give an adequacy factor for problems with slope angle greater than the friction angle.

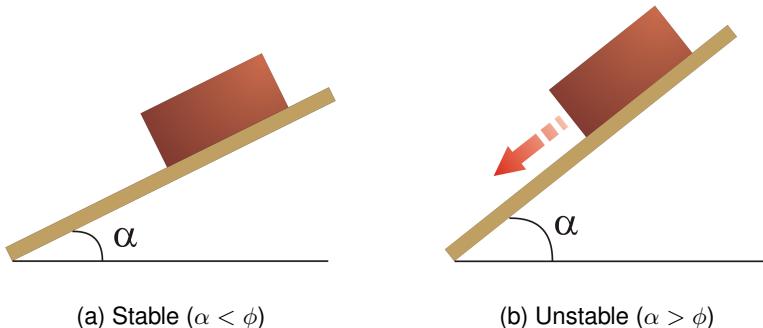


Figure 10.1: Stability of a brick on an inclined plank with brick/plank interface friction angle ϕ .

An alternative method by which to drive a cohesionless slope problem to failure is to use an adequacy factor applied to a horizontal pseudo static acceleration (see Chapter 12).

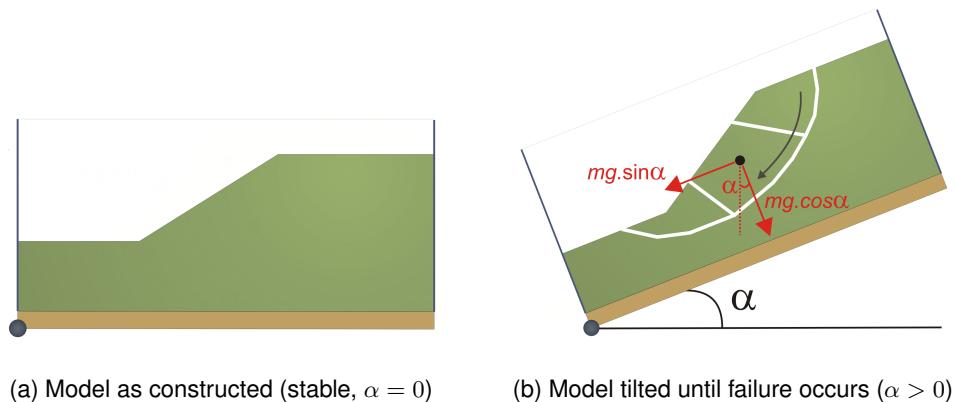
10.2.2 Slopes in Cohesive Soils

In a slope containing purely cohesive soil, a numerical value of **Adequacy factor** will always be returned (i.e. *locked* and *unstable* results should not occur) when undertaking a **Factor Strength(s)** analysis.

10.3 Factor of Safety on Load

It is generally expected that a slope stability problem will be solved using a **Factor Strength(s)** analysis (see Section 8.2.3). However, it is possible to obtain a solution using the alternative **Factor Load(s)** approach.

As an alternative to the tilting plank example described in Section 10.2.1, consider a laboratory model of a slope stability problem with the model contained within a tank, such that the tank may be pivoted about one end (Figure 10.2). In order to establish how close to the point of instability the slope is, the tank can be slowly tilted from the horizontal to a steeper and steeper angle α_{crit} until failure occurs. The larger this angle, the more stable the slope.



(a) Model as constructed (stable, $\alpha = 0$) (b) Model tilted until failure occurs ($\alpha > 0$)

Figure 10.2: Laboratory slope stability model.

Tilting the tank is analogous to modifying the body forces exerted by any mass of soil or structure. Within the frame of reference of the tank, the vertical body force becomes $mg \cos \alpha$ and a horizontal body force is introduced $mg \sin \alpha$, as shown in Figure 10.2, where m is the mass of the body and g the acceleration due to gravity.

In LimitState:GEO, horizontal body forces can be introduced using the **Seismic Actions** facility in the **Property Editor**. By setting the **Horizontal Accel. k_h (g)** to 1.0, **Adequacy (on k_h)** to **True** (and ensuring Adequacy is not set on any other parameter), this requires the software to find the horizontal acceleration required to cause collapse. This will be returned as the **Adequacy factor (AF)**.

This can be converted to an equivalent value of α_{crit} using the expression $\alpha_{crit} = \tan^{-1}(AF)$. Thus the problem is stable if $AF \geq 0.0$.

It is important to note that:

1. Whereas in normal usage stability normally corresponds to $AF \geq 1.0$ (e.g. for a foundation stability problem), in this special case a value of $AF \geq 0.0$ is required.
2. The significance of the order of magnitude of AF will differ from conventional usage. For example, a 24.8° slope of soil of friction angle 30° is just stable with a factor of safety of 1.25 on soil strength. This corresponds to a value of $\alpha_{crit} = 5.2^\circ$ (or $AF = 0.091$). It is thus recommended to conceptually interpret the **Adequacy factor** as an angle $\alpha_{crit} = \tan^{-1}(AF)$.

10.4 Unusual Failure Surfaces for Slopes in Cohesive Soils

In a purely cohesive soil, for certain slope geometries, the failure surface will always touch the edge of the domain (see for example Figure 10.3). While counter-intuitive it is entirely in accordance with slope theory which predicts a failure mechanism of semi-infinite extent for e.g. finite shallow slopes in a semi-infinite domain of purely cohesive soil .

Thus changing the distance to the outer boundary of the problem (whether in the wizard or by dragging a boundary in the viewer) will not succeed in bringing the mechanism fully within the boundaries. However it will be found that the adequacy factor will converge to a fixed value.

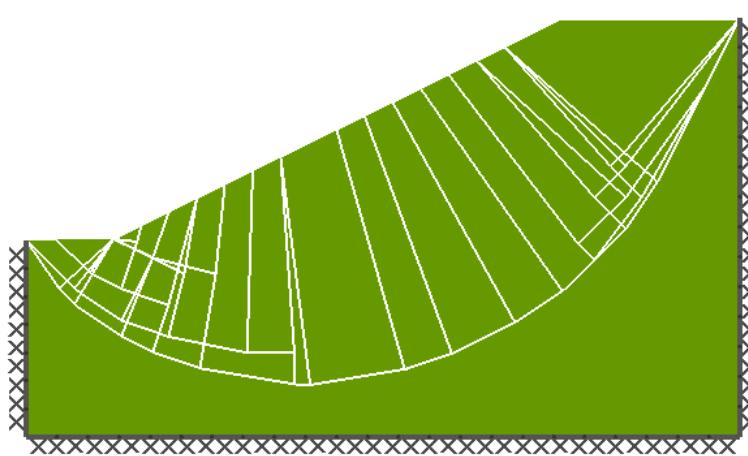


Figure 10.3: Slope failure mechanism in a cohesive soil, touching boundaries at the left and right edges.

Chapter 11

Reinforced Soil

11.1 Introduction

Soil reinforcement is typically modelled using the **Engineered Element** material. This is a material that can be applied to **Boundaries** to provide essentially one dimensional (in section) objects such as soil nails, geotextiles, and sheet pile walls. Specific properties of an **Engineered Element** can be set to allow soil to flow through or past it if required as would be expected of e.g. a soil nail. This would otherwise not be possible in a plane strain program such as LimitState:GEO.

In other modelling approaches, soil reinforcement is often only modelled by its effect such as an imposed point force on a slip surface. This makes it challenging to model complex failure mechanisms and requires the user to pre-judge to some extent the effect of the reinforcement. In LimitState:GEO the software is given full freedom to determine the critical failure mechanism involving soil reinforcement. The parameters required for the soil reinforcement may thus not be familiar to users familiar with other approaches. However these parameters are typically required when estimating e.g. an imposed point force on a slip surface.

The parameters defining the behaviour of an **Engineered element** are:

Pullout factor, T_c The contribution of the material cohesion to the pullout resistance of the element.

Pullout factor, T_q Used to determine the contribution of the overburden to the pullout resistance of the element.

Lateral factor, N_c The contribution of the material cohesion to the lateral resistance of the element.

Lateral factor, N_q Used to determine the contribution of the overburden to the lateral resistance of the element.

Plastic moment, M_p The plastic moment of resistance of the element. (If rigid joints are being modelled then this parameter should be set to 1E+30).

Rupture strength, R The maximum tensile force that can be carried by the material.

Compression strength, C The maximum compressive force that can be carried by the material.

Advanced > Subdivide at nodes? Specifies whether failure of the element between end vertices is permitted at nodes (e.g. plastic hinge formation and/or tensile failure). This should be set to *False* for compatibility with early versions of LimitState:GEO. The default and recommended value is *True*.

The theoretical background to these parameters is given in Section 6.1.5. Guidance on defining **Engineered element** geometry using LimitState:GEO is given in Section 17.9.2.

11.2 Soil Reinforcement with Specified Interface Materials

As outlined in Section 17.5 it is often appropriate, in problems of soil/structure interaction to assign properties to the interface between a solid body and the surrounding soil that are some function of the properties of the soil. Similarly, where soil reinforcement is specified in a model, it can be appropriate to specify interface properties that are distinct on either side of the element itself. This is anticipated to be useful primarily in cases where ‘flow through’ does not occur (e.g. for a sheet pile wall or a geotextile), as the materials applied at the interfaces will be applied across the full width of the model (and will insert a potentially unintentional ‘wound’ through the model if used in conjunction with e.g. a soil nail).

When creating a new **Engineered Element** material, the user specifies only those properties that relate to the element itself (see Chapter 17). This material can then be applied to 1D boundaries in the usual manner. However, when associated with a **Boundary Object**, the viewer will display three lines, as shown in Figure 11.1:

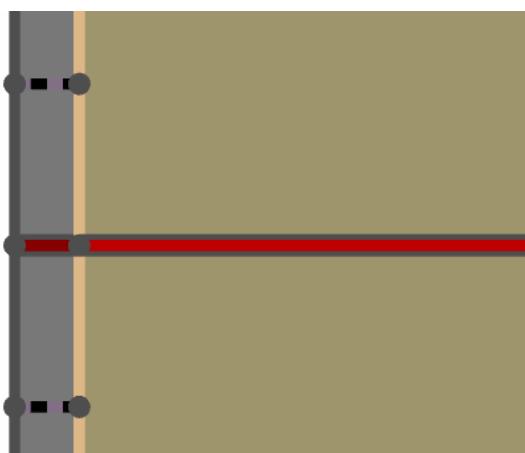


Figure 11.1: Section of a reinforced soil wall problem, showing an Engineered Element material (red) on a 1D boundary. Note the dark borders to the element, which signify that edge materials have not been applied.

The inner line depicts the **Engineered Element** itself while the darker outer lines represent the boundary interfaces on either side of the element. These interfaces can be assigned one or more materials so that the interaction of the reinforcement with its surroundings along the

interfaces can be closely modelled. For example, see the sheet pile wall depicted in Figure 11.2, where the **Engineered Element** (Sheet Pile) is bordered by Frictionless and No-Tension interface materials.

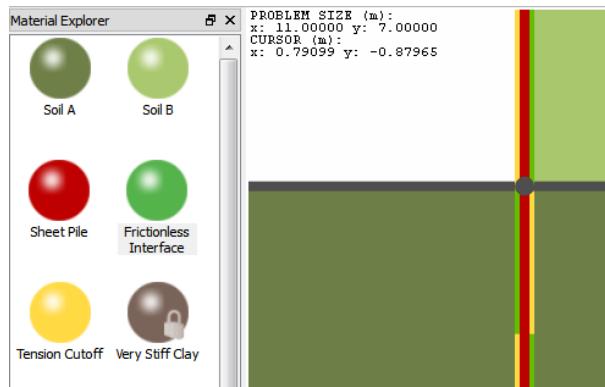


Figure 11.2: Section of a sheet pile wall problem, showing an Engineered Element material (red) on a 1D boundary with frictionless (green) and no-tension (yellow) materials applied to the two outer faces.

11.3 Soil Reinforcement - Predefined Types

11.3.1 Predefined types

When defining a new **Engineered Element**, LimitState:GEO provides five predefined parameter sets, representing different common soil reinforcement types. Dependant upon usage, each parameter set contains fixed and editable properties taken from those listed in Section 11.1.

- Soil Nail (Rigid)
- Soil Nail (Can Yield)
- Soil Nail (Flexible)
- Sheet Pile Wall (Rigid)
- Sheet Pile Wall (Can Yield)
- Geotextile
- Other

11.3.2 Soil Nail - general parameters

For a Soil Nail, the pullout properties T_c , T_q , N_c , and N_q (which are user-defined) are ideally determined by field tests. However estimates using theory may be made. For example if a nail of diameter B is embedded in a cohesive soil of undrained shear strength c_u then plasticity

theory could be used to estimate the ‘Pullout Factor’ T_c as $\pi B c_u$ and the ‘Lateral Factor’ N_c as $\sim 9c_u B$ based on laterally loaded pile theory for widely spaced piles. Both values might then be factored. For an undrained cohesive soil the pullout and lateral factors T_q and N_q are independent of depth and therefore would be set to zero.

Certain current alternative analysis methods do not take account of the lateral resistance of soil nails and so N_c and N_q could be set to zero when comparing with these analyses. However in this situation, LimitState:GEO may occasionally identify a failure mode where the nails simply ‘float’ out of the ground because they have no lateral resistance. It is therefore recommended that a small value of N_c is set, of the order of 1 kN/m².

When modelling a soil nail, no materials should be assigned to the boundary interfaces.

11.3.3 Soil Nail (Rigid)

A rigid soil nail has its plastic moment (M_p) set to 1E+30 kNm/m (effectively infinite) to ensure that the element behaves as a rigid body and cannot form any plastic hinges. The rupture and compression strengths are also set to be effectively infinite (1E+30 kN/m).

11.3.4 Soil Nail (Flexible)

This type of soil nail behaves in the same way as the rigid soil nail except that the plastic moment capacity is set to zero so that the nail may hinge freely at locations along the nail. These locations can either be user-defined **Vertices** or, if the **Subdivide at nodes?** material property is set to *true*, **Nodes** along the length of the element. (By default, LimitState:GEO will set **Subdivide at nodes?** to *true* when creating any material that is designated as being able to yield. This can be modified in the **Property Editor**).

11.3.5 Soil Nail (Can Yield)

This type of soil nail behaves in the same way as the flexible soil nail except that finite values can be set for the axial (rupture and compression) strengths.

11.3.6 Sheet Pile Wall - general parameters

To represent a rigid Sheet Pile Wall the values of T_c , N_c and M_p are all set to 1E+30 i.e. effectively infinity. The values of T_q and N_q are set to zero.

Setting the **Pullout Factor** T_c and **Lateral Factor** N_c to infinity locks the Engineered Element to its boundary interfaces and essentially prevents the flow of soil through the wall. The shear interaction of the Sheet Pile Wall with the adjacent soil is thus controlled by the boundary interfaces as would be expected. If these are not assigned any Materials then the wall/soil interface strength is modelled the same as the adjacent soil.

11.3.7 Sheet Pile Wall (Rigid)

Setting the **Plastic moment** M_p to infinity ensures that the element behaves as a rigid body and cannot form plastic hinges along its length.

Setting the **Rupture strength** R and the **Compression Strength** C to infinity prevents axial failure of the element.

11.3.8 Sheet Pile Wall (Can Yield)

This type of sheet pile wall behaves as the rigid sheet pile wall except that a finite value of plastic moment can be set so that the sheet pile may form plastic hinges at locations along the length. These locations can either be user-defined **Vertices** or, if the **Subdivide at nodes?** material property is set to *True*, **Nodes** along the length of the element. (By default, LimitState:GEO will set **Subdivide at nodes?** to *True* when creating any material that is designated as being able to yield. This can be modified in the **Property Editor**)

The plastic moment per metre width (into the diagram) can be estimated from a knowledge of the wall thickness and the yield strength of the wall material.

11.3.9 Geotextile

To represent a geotextile the values of T_c , N_c are all set to 1E+30 i.e. effectively infinity. The values of T_q , N_q , M_p and C are set to zero.

Setting the **Pullout Factor** T_c and **Lateral Factor** N_c to infinity locks the Engineered Element to its boundary interfaces and essentially prevents the flow of soil through the wall. The shear interaction of the Geotextile with the adjacent soil is thus controlled by the boundary interfaces as would be expected. If these are not assigned any Materials then the geotextile/soil interface strength is modelled the same as the adjacent soil.

For the geotextile, a zero value of plastic moment is set so that the geotextile may freely form 'plastic hinges' at locations along the length. The **Rupture strength** R is set to the tensile strength of the geotextile so that the geotextile may freely form 'plastic hinges' at the same locations along the length. These locations can either be user-defined **Vertices** or, if the **Subdivide at nodes?** material property is set to *True*, **Nodes** along the length of the element. (By default, LimitState:GEO will set **Subdivide at nodes?** to *True* when creating any material that is designated as being able to yield. This can be modified in the **Property Editor**)

11.3.10 Other

This 'Application' type is used for user defined reinforcement properties which do not fit the pre-defined values of the other types. All parameters are freely editable. By default, LimitState:GEO will set **Subdivide at nodes?** to *True* when creating a material of this type.

11.4 Modelling soil nails

Conventionally only the tensile reinforcement action of soil nails is modelled. This is specified using the pullout capacity of the nails and is ideally determined from field tests. The capacity can be split into the constant component T_C and a component that depends on the prevailing vertical effective stress T_q . These values are per metre length of nail. The value for one nail should then be multiplied by the number of nails per metre width (perpendicular to the plane of modelling) to give the value entered in the software. No boundary interface materials should be set for a soil nail.

Conventionally the lateral resistance of the nail is neglected. To do this the parameters N_c and N_q should be set to zero. However in practice it is preferable to set N_c to a small nominal value e.g. 1 kN/m². This avoids the nail 'floating' out of the ground in the model.

If required, tensile failure of a nail can be represented using the Rupture Strength R . Bending failure can also be specified by specifying a plastic moment M_p , but may only be relevant if non-zero lateral factors N_c and N_q are used. Note that if both bending and tensile failure are anticipated then combined bending/tensile action needs to be accounted for and lower strengths may need to be specified. Again the values calculated for one nail should be multiplied by the number of nails per metre width.

11.5 Modelling flexible geosynthetic reinforcement

11.5.1 General principles

This type of reinforcement is not provided as a predefined type. However it is possible to model it using the **Engineered Element** generic model. To determine if this is the case it is necessary that the engineer compare the theoretical model that they wish to use for the geotextile with that provided by LimitState:GEO.

The recommended parameters to be used are the same as for a sheet pile wall, but with the **Plastic moment** M_p set to zero and **Subdivide at nodes?** set to *True* to allow the modelling of many possible 'hinge' locations on the element to allow full flexibility. The boundary interfaces would normally be assigned material properties that are a percentage of the adjacent soil material strengths. This percentage is commonly termed the α factor for the geotextile.

Note that it is not recommended to use Engineered Elements with infinite lateral resistance to model fully enclosed volumes of frictional soil e.g. a sand bag. Due to the dilatational representation of friction in plasticity theory (see Section 7.6), such enclosed volumes will not be able to deform and will behave like rigid blocks.

11.5.2 Geotextiles - Method 1

The recommended approach to modelling a geotextile using an **Engineered element**, is to set parameters as follows:

Pullout factor, $T_c = \infty$ This locks the geotextile to its boundaries. In practice the value ∞ should be represented in the software by the value 1E30.

Pullout factor, T_q This can be set to any value since pullout behaviour is determined by T_c .

Lateral factor, $N_c = \infty$ This prevents the flow of soil through the geotextile. In practice the value ∞ should be represented in the software by the value 1E30.

Lateral factor, N_q This can be set to any value since lateral behaviour is determined by N_c .

Plastic moment, $M_p = 0$ This allows fully flexible movement at any node.

Rupture strength, R This should be set to the tensile strength of the geotextile.

Compression strength, $C = 0$ It is assumed that the geotextile has zero compressive strength.

Advanced > Subdivide at nodes? This should be set to *True* to allow the modelling of many possible 'hinge' locations on the element and/or rupture failure.

The **Engineered element** representing a geotextile should normally be used in conjunction with **Boundary** materials (see Section 11.2) on each side of the geotextile to model the geotextile/soil interface friction. These boundary interfaces would normally be assigned material properties that are a percentage of the adjacent soil material strengths. This percentage is commonly termed the α factor for the geotextile. Use of a **Derived Mohr-Coulomb Material** (see Section 17.5) facilitates straightforward modelling of the α factor.

Note that this geotextile model assumes the principles of limit analysis, i.e. rigid-plastic behaviour and ductility. Where it is required to model complex deformation, it may be necessary to employ a high nodal density along the element to fully capture the behaviour.

11.5.3 Geotextiles - Method 2

An alternative approach to modelling a geotextile may be achieved by directly using the Pullout Factor and by omitting boundary interface materials. This has the following differences from Method 1:

- In circumstances where the geotextile and, for example, the underlying soil remain static and the overlying layer slides along the geotextile, the interface skin friction modelled will be the full soil strength.
- The pullout strength is calculated based on the indicative vertical stress pre-computed before the solution stage as described in Section 6.1.5. Since it is very likely that the vertical effective stress along the element is modified by the overall failure mechanism, this may not be as precise as Method 1, particularly if a **Factor Load(s)** analysis is undertaken with the **Adequacy factor** applied to items that load the geotextile. However it is likely to be closer to conventional hand calculations that utilise indicative vertical stresses.

To use this approach, set parameters as follows:

Pullout factor, T_c See below.

Pullout factor, T_q Factors T_c and T_q should be set such that $T = T_c + T_q \sigma'_v$ gives the pullout resistance per unit length per unit width,

Lateral factor, $N_c = \infty$ This prevents the flow of soil through the geotextile. In practice the value ∞ should be represented in the software by the value 1E30.

Lateral factor, N_q This can be set to any value since lateral behaviour is determined by N_c .

Plastic moment, $M_p = 0$ This allows fully flexible movement at any node.

Rupture strength, R This should be set to the tensile strength of the geotextile.

Compression strength, $C = 0$ It is assumed that the geotextile has zero compressive strength.

Advanced > Subdivide at nodes? This should be set to *True* to allow the modelling of many possible ‘hinge’ locations on the element and/or rupture failure.

Boundary interface materials are not required.

Note that this geotextile model assumes the principles of limit analysis, i.e. rigid-plastic behaviour and ductility.

11.5.4 Geotextiles - Method 3

An alternative that may give a mechanically equivalent model is to represent the geotextile as a fully rigid **Engineered Element** and with zero lateral resistance. LimitState:GEO will then allow soil to flow ‘through’ the element. This can be considered equivalent (for small displacements) to letting the geotextile freely deform with the soil, while still providing transmission of tensile forces. However the resulting deformations seen in an animation may not look intuitively correct and caution should be exercised to ensure that the scenario modelled is not kinematically restricted by the rigid representation of the geotextile.

To use this approach, set parameters as follows:

Pullout factor, $T_c = \infty$ This locks the geotextile to its boundaries. In practice the value ∞ should be represented in the software by the value 1E30.

Pullout factor, T_q This can be set to any value since pullout behaviour is determined by T_c .

Lateral factor, $N_c = 0$ This allows the ‘flow’ of soil through the geotextile.

Lateral factor, $N_q = 0$ This allows the ‘flow’ of soil through the geotextile.

Plastic moment, $M_p = \infty$ This models the geotextile as fully rigid.

Rupture strength, R This should be set to the tensile strength of the geotextile.

Compression strength, $C = 0$ It is assumed that the geotextile has zero compressive strength.

Advanced > Subdivide at nodes? This should be set to *True* to allow the modelling of many rupture failure along the element.

11.6 Anchor tendons

In certain circumstances it is necessary to model an element that can transmit tension and/or compression, but not interact with the surrounding soil. This can be achieved using an **Engineered Element** and setting the values of T_c , T_q , N_c and N_q to zero. With these settings any materials assigned to the boundary interfaces will have no effect, and the element can freely displace through the soil. The bending, rupture and compressive resistance of the element can be controlled by the parameters M_p , R and C . To allow modelling of a flexible anchor tendon, the moment of resistance M_p can be set to zero.

Note that an engineered element representing a flexible anchor tendon (with $M_p = 0$) should not be directly connected to an engineered element representing a retaining wall or anchor plate. If this occurs then the bending strength modelled at the junction will be the lowest of the engineered elements meeting at that junction i.e. zero. This has the effect of introducing a frictionless hinge into the wall or anchor plate and thus the possibility of an unintended failure mode.

The correct approach to linking a low bending strength anchor tendon to a wall element is illustrated in Figure 11.3 where a short stub of wall material is modelled to which the anchor tendon attaches. This leaves the main wall bending strength unaffected.

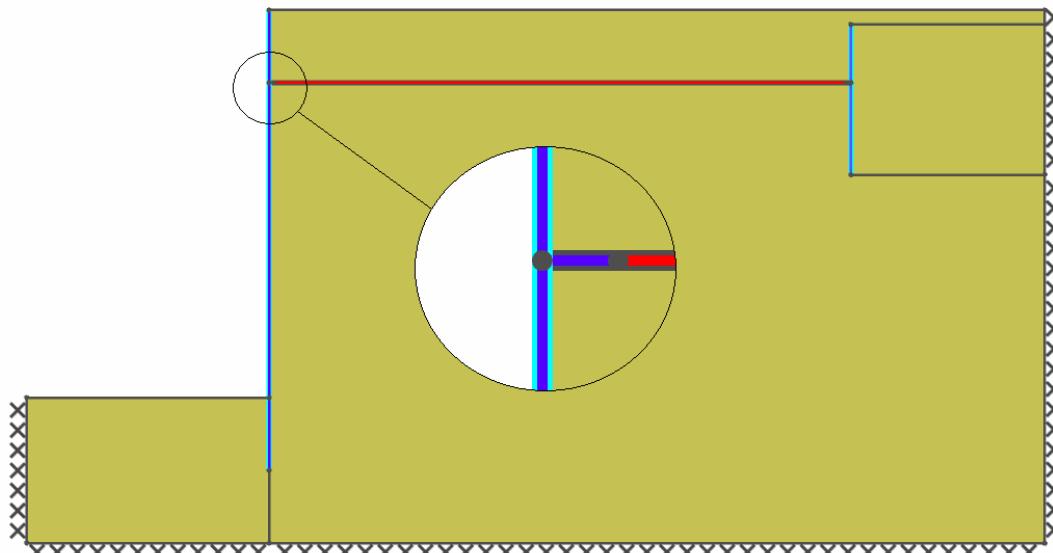


Figure 11.3: Connection of a flexible anchor tendon to a retaining wall modelled using Engineered Elements.

11.7 Example 1 - Simple Reinforcement Pullout

The simple pullout test modelled in Figure 11.4 illustrates the functioning of the parameter T_c . The input file is available in the **Example Files** folder.

The reinforcement (coloured red in the diagram) is embedded in both a weightless rigid block (height 1m) on the left and a body of clay of undrained strength $c_u = 30 \text{ kN/m}^2$ and of length $L = 2\text{m}$ on the right (with zero tensile strength specified on the interface between block and soil). It is assumed that it has a pullout resistance of 10 kN/m^2 in the soil.

The aim of the test is to determine what force is required to displace the rigid block to the left, thus pulling the reinforcement out of the soil.

To ensure the reinforcement remains embedded in the rigid block but can pull out of the soil it is necessary to define two **Engineered Element** materials. The first is given a high value of T_c (e.g. $1E+30 \text{ kN/m}^2$) and is assigned to the portion of the reinforcement in the rigid block (dark red in the diagram). The second is given a value of T_c of 10 kN/m^2 and is assigned to the portion of the reinforcement in the soil block (light red in the diagram). Both materials are given high values of plastic moment of resistance (e.g. $1E+30 \text{ kNm/m}$) and tensile rupture strength (e.g. $1E+30 \text{ kN/m}$).

A unit stress with Adequacy factor (red arrows) is applied to the left hand edge of the rigid block. On solving the block displaces to the left and the light red portion of the reinforcement pulls through the soil. An **Adequacy factor** of 20 is returned, indicating that a force of 20 kN/m is required to pull out the 2m of reinforcement, with $T_c = 10 \text{ kN/m}^2$. Displaying the tensile force diagram confirms that the maximum tensile force experienced in the soil reinforcement is 20 kN/m .

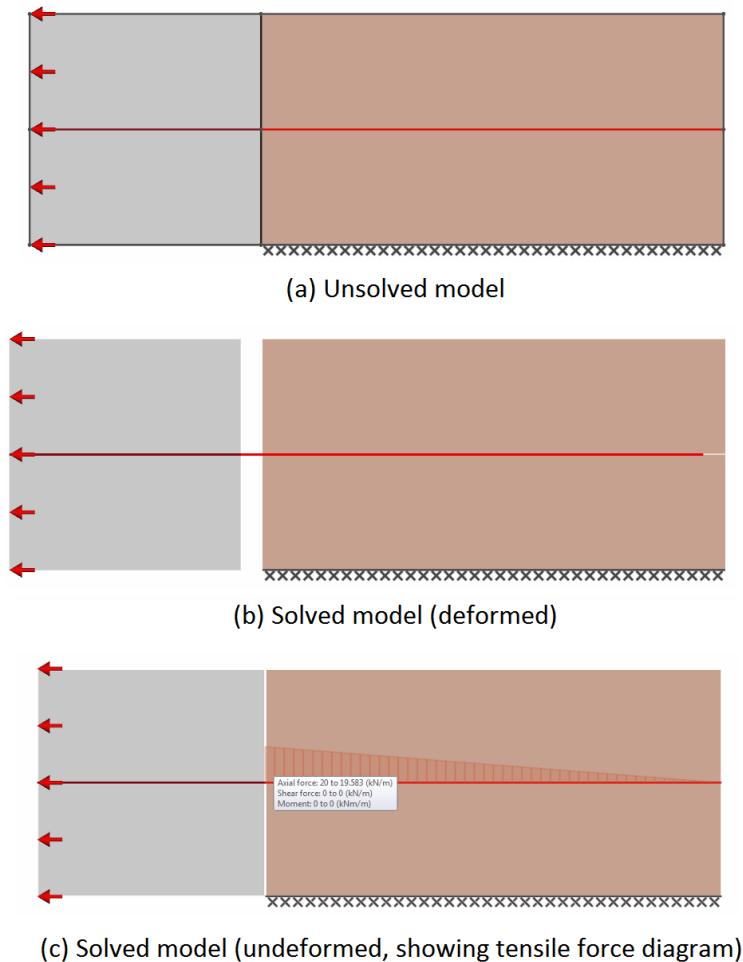


Figure 11.4: Simple reinforcement pullout test.

11.8 Example 2 - Simple Reinforcement Tensile Rupture

The simple test modelled in Figure 11.5 illustrates the functioning of the tensile **Rupture strength** parameter. The input file is available in the **Example Files** folder.

The problem is identical to that of the pullout test (Section 11.7) except that the reinforcement embedded in the body of soil (light red) has a tensile capacity of 40 kN/m and a T_c value of $1E+30$ kN/m². This value will 'lock' the Engineered Element to its boundary interfaces. The force required for the Engineered Element to pullout along its boundary interfaces is equal to $2 \times c_u \times L = 2 \times 30 \times 2 = 60$ kN/m which exceeds the tensile capacity.

The aim of the test is to show that the force required to displace the rigid block to the left and cause the reinforcement to rupture in tension is equal to the rupture strength of the nail. On solving the block displaces to the left and the light red portion of the reinforcement *does not* pull through the soil. Instead the nail is seen to lengthen. An **Adequacy factor** of 40 is returned, indicating that a force of 40 kN/m was required to rupture the reinforcement.

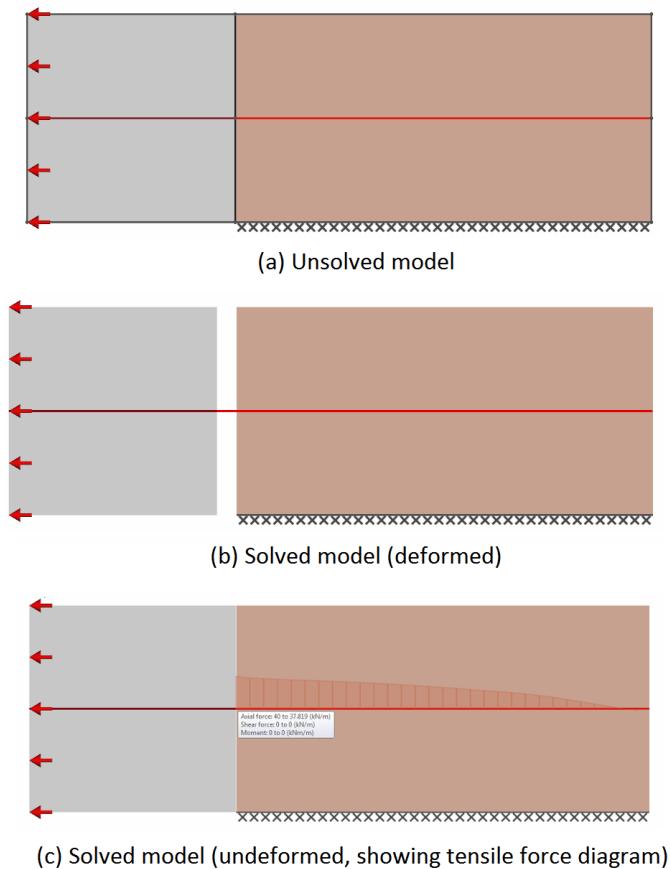


Figure 11.5: Simple reinforcement tensile rupture test.

11.9 Example 3 - Simple Reinforcement Lateral Displacement

The simple lateral displacement test modelled in Figure 11.6 illustrates the functioning of the parameter N_c . The input file is available in the **Example Files** folder.

The reinforcement, for example a soil nail, (coloured red in the diagram) is embedded in a body of soil of width 3m and attached to two weightless rigid blocks (width 0.5m) either side of the soil body (with frictionless interface between the soil and the blocks). It is assumed that it has a lateral displacement resistance of $N_c=25 \text{ kN/m}^2$ in the soil.

The aim of the test is to determine what force is required to displace the rigid blocks upwards, thus displacing the reinforcement through the soil. To ensure that the reinforcement remains fixed to the rigid blocks but can displace through the soil it is necessary to define two **Engineered Element** materials. The first is given a high value of N_c (e.g. $1\text{E}+30 \text{ kN/m}^2$) and is assigned to the portion of the reinforcement embedded within the rigid blocks (dark red in the diagram). The second is given a value of N_c of 25 kN/m^2 and is assigned to the portion of the reinforcement in the soil block (light red in the diagram).

A unit stress with Adequacy factor (red arrows) is applied to the base of each rigid block (i.e. over a total width of 1m). On solving the blocks displace upwards and the light red portion of

the reinforcement diaplaces through the soil. An Adequacy factor of 75 is returned, indicating that a force of $25 \times 3 = 75$ kN/m is required to displace the 3m of reinforcement upwards, with $N_c = 25$ kN/m².

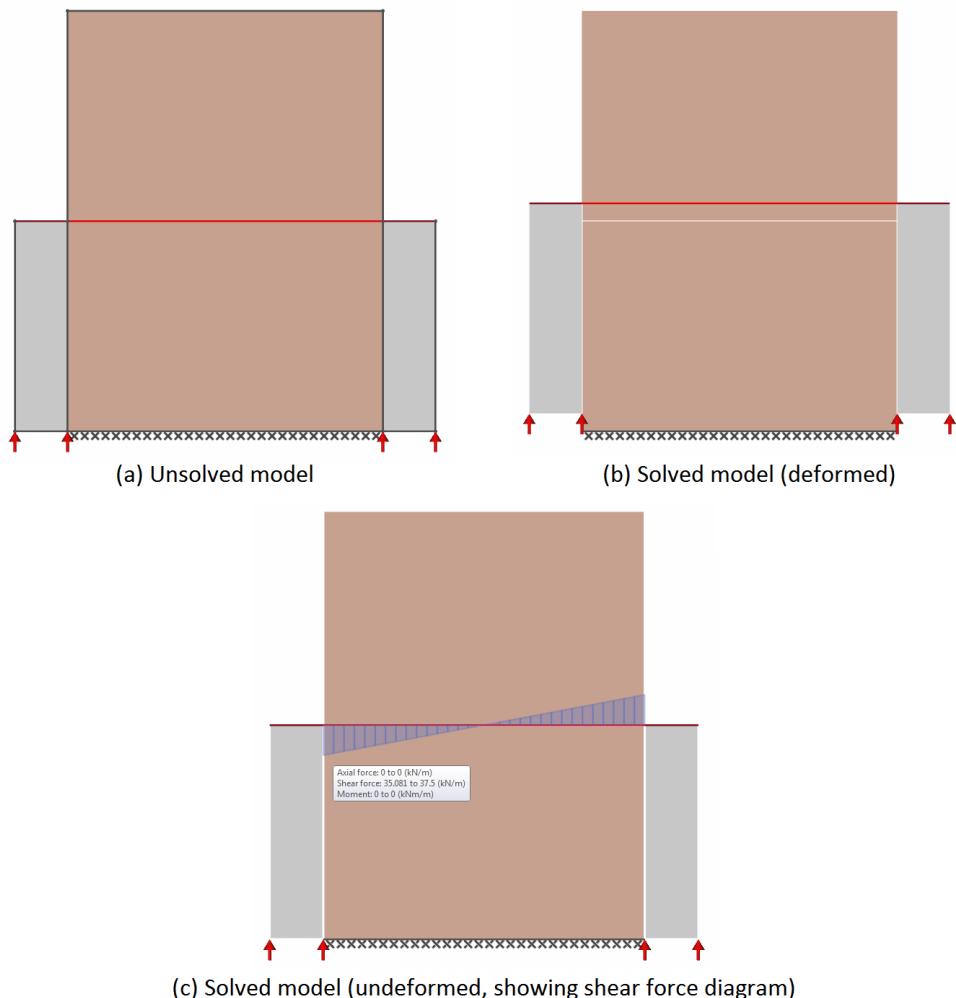


Figure 11.6: Simple reinforcement lateral displacement test.

11.10 Example 4 - Simple Reinforcement Bending

The simple bending test modelled in Figure 11.7 illustrates how the parameter M_p functions. The input file is available in the **Example Files** folder.

A reinforcement frame surrounds a body of soil of width 1m and height 1m. On top of the frame is attached a weightless rigid block (height 0.5m) to which a lateral pressure of 2 kN/m is applied (resulting in a 1kN/m force). It is assumed that the frame has a plastic moment of resistance of 10 kN/m/m.

The aim of the test is to determine what force is required to displace the rigid block and cause bending failure in the frame.

The **Engineered Element** material defining the frame (red in the diagram) is set with high values of T_c and N_c (e.g. 1E+30 kN/m²) and an M_p value of 10 kNm/m. All of the soil properties are set to zero except the undrained cohesion (c_u) which is assigned a value of 10 kN/m².

The Adequacy factor is applied to the lateral load acting on the rigid block (red arrows). A coarse nodal discretization is set (target number of nodes = 250).

On solving, the block displaces to the right and reinforcement frame displaces to form a trapezoidal shape, with bending occurring at the corners. An **Adequacy factor** of 50.63 is returned, indicating that a stress of 50.63 kN/m/m is required to shear the soil and cause sufficient plastic hinges in the frame to form a failure mechanism. The analytical solution for this problem can be easily determined using a simple work equation:

$$1\lambda\theta = 4M_p\theta + 1c_u\theta \quad (11.1)$$

hence:

$$\lambda = 4M_p + c_u = 40 + 10 = 50 \quad (11.2)$$

where λ is equal to the factor on load required to cause failure and θ is equal to the rotation at the yielding positions. Therefore, using a coarse nodal discretization, the solution has an error of approximately 1.2%. Note that the higher the number of target nodes set for the problem, the closer the value will be to the analytical solution.

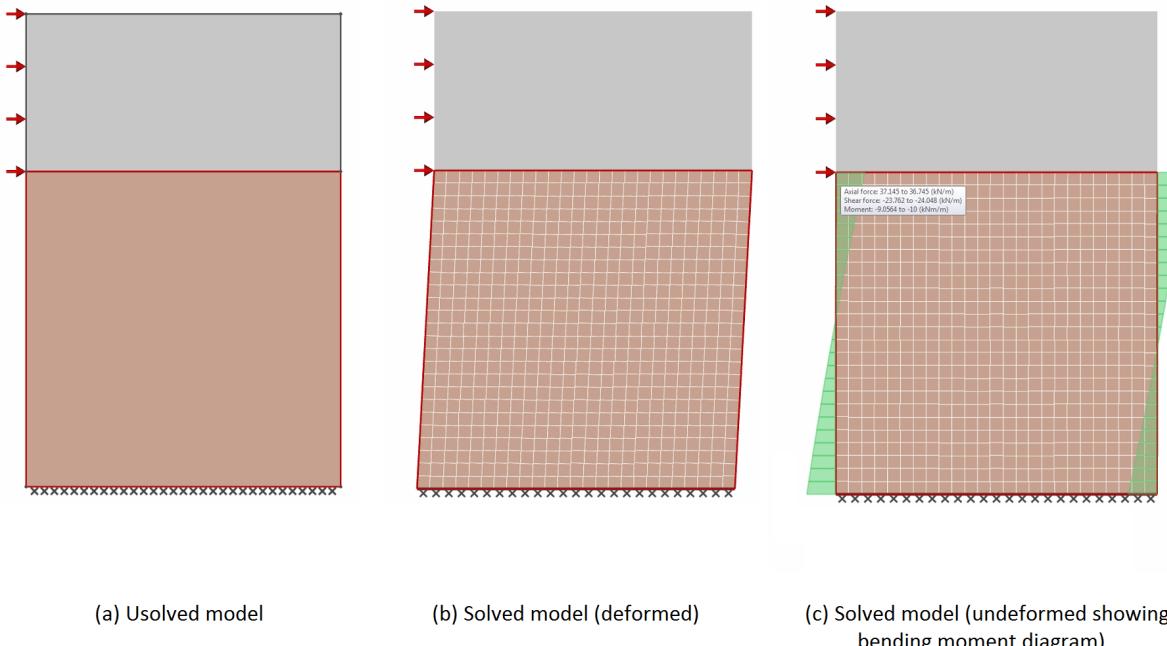


Figure 11.7: Simple reinforcement bending test.

Chapter 12

Seismic Problems

12.1 Modelling of Seismic Loading Using DLO

In LimitState:GEO horizontal and/or vertical pseudo static acceleration may be applied to any problem. These may be applied with or without **Adequacy** applied to them. Often it may be required to check a design against seismic loading and the question may be posed as follows:

'What value of horizontal acceleration is required to cause collapse?'

This may be straightforwardly done by applying adequacy to the horizontal acceleration (details of how to enter seismic parameters via the user interface in LimitState:GEO are given in Section 21).

In general it is recommended that **Adequacy** be applied to horizontal acceleration only or to both horizontal and vertical acceleration. Use of **Adequacy** on vertical acceleration only should be done with caution as it may result in correct but unexpected results as described in Section 12.4)

12.2 Example 1 - Modelling a Simple Mechanics Problem

Consider the sliding block problem in Figure 12.1. The rigid block is of dimensions $1\text{m} \times 1\text{m}$ and of unit weight 1 kN/m^3 . It rests on a rigid platform separated by a frictional interface with $\tan \phi = 0.5$. A fixed unit stress (green arrows) of 1 kN/m^3 is applied to the left hand face of the block. The question is:

'What horizontal acceleration k_h applied right to left will cause failure?'

To determine this using LimitState:GEO the geometry and parameters are entered as described. An example file of this problem (SlidingBlockSeismic.geo) is included in the LimitState:GEO 'example files' directory (see Appendix C for further information on accessing example files).

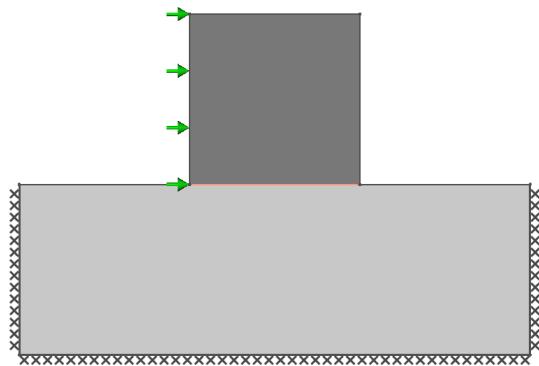


Figure 12.1: Simple sliding block example under horizontal seismic loading.

To find the required horizontal acceleration to cause failure, set **Horizontal Accel. k_h (g)** to 1.0, **Adequacy (on k_h)** to *True* and solve. LimitState:GEO will return an **Adequacy factor** of 1.5 and show a sliding failure. This means that the horizontal acceleration must be 1.5g to cause failure. (Note that the **Horizontal Accel. k_h (g)** could have been set to e.g. 0.3, in which case an **Adequacy factor** of 5 would have been returned, still giving a required horizontal acceleration of 1.5g to cause failure). This can be checked by simple mechanics:

Resolving horizontally,

$$k_h W = P + W \tan \phi \quad (12.1)$$

where W is the weight of the block and P is the applied fixed force. For the given units this gives:

$$k_h \times 1.0 = 1.0 + 1.0 \times \tan \phi = 1.5 \quad (12.2)$$

12.3 Example 2 - Checking the Mononobe-Okabe Equation

Consider the simple retaining wall problem in Figure 12.2. The wall is modelled as a weightless rigid block of height 1m resting on a smooth base. It retains soil of angle of shearing resistance 30° , and the soil/wall interface angle of shearing resistance is also set to 30° . The seismic active earth pressure acting on the wall due to a horizontal acceleration of 0.1g is required.

To determine this using LimitState:GEO the geometry is entered as described. An example file of this problem (SeismicRetainingWall1.geo) is included in the LimitState:GEO ‘example files’ directory (see Appendix C for further information on accessing example files).

To find the required active pressure, set **Horizontal Accel. k_h (g)** to 0.1 and the **Adequacy (on k_h)** to *False*. To determine an active earth pressure it is necessary to set one of the normal loads (e.g. the **Permanent** load) on the left hand face of the wall to 1.0 kN/m^2 without Adequacy and another (e.g. the **Variable** load) on the left hand face of the wall to -1.0 kN/m^2

with Adequacy (see Section 8.2.7 for an explanation of why this is required to obtain active earth pressures). On solving with a medium resolution, LimitState:GEO will return an Adequacy factor of 0.8377 and show a compound wedge failure mechanism.

This means that the active earth pressure is the sum of the applied loads i.e. $1.0 - 0.8377 = 0.1623 \text{ kN/m}$. This is slightly higher than the value of 0.1611 kN/m given by Mononobe-Okabe equation for this problem. The reason for the discrepancy is that the Mononobe-Okabe equation is based on the assumption of a single wedge failure mechanism whereas in fact for a rough wall the true mechanism differs slightly from this.

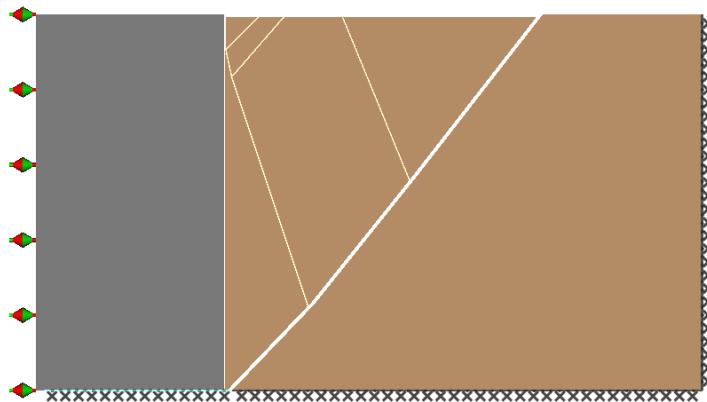


Figure 12.2: Simple retaining wall example under horizontal seismic loading (1).

It is possible to pose this problem in an alternative form asking:

'What horizontal acceleration is required to give a resultant active earth pressure on the wall of 0.1611 kN/m^2 '?

This is modelled by the example file SeismicRetainingWall2.geo as depicted in Figure 12.3. In this case the **Horizontal Accel. k_h (g)** is set to 1.0 and the **Adequacy (on k_h)** to **True**. One of the normal loads (e.g. the **Permanent** load) on the left hand face of the wall is set to 0.1611 kN/m^2 without **Adequacy**.

On solving with a medium resolution, LimitState:GEO will return an **Adequacy factor** of 0.09665 and again show a compound wedge failure mechanism. This implies that the wall will fail at a horizontal acceleration of 0.09665. This is close to the expected horizontal acceleration of 0.1, but again LimitState:GEO has found a slightly more optimal failure mechanism.

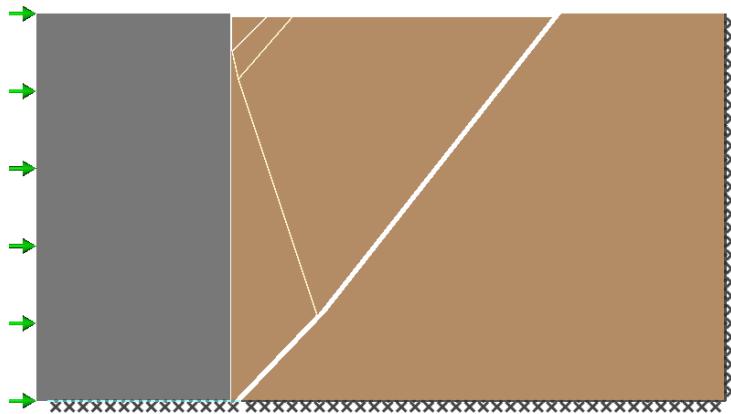


Figure 12.3: Simple retaining wall example under horizontal seismic loading (2).

12.4 Example 3 - Correct use of Adequacy Direction

Consider again the sliding block problem in Figure 12.1 with the same parameters as used in Example 1. In this case the question posed is:

'What value of horizontal acceleration k_h applied right to left in combination with a vertical acceleration $k_v = 0.5k_h$ applied downwards will cause failure?'

To find the required solution, set **Horizontal Accel. k_h (g)** to 1.0, **Adequacy (on k_h)** to True , **Vertical Accel. k_v (g)** to 0.5, **Adequacy (on k_v)** to True and solve. LimitState:GEO will return an **Adequacy factor** of 2.0 and show a sliding failure similar to Example 1.

This can again be checked by simple mechanics:

$$k_h W = P + (1 + k_v)W \tan \phi \quad (12.3)$$

where W is the weight of the block and P is the applied fixed force. For the given units this gives:

$$k_h \times 1.0 = 1.0 + (1 + 0.5k_h) \times 1.0 \times \tan \phi \quad (12.4)$$

and

$$0.75k_h = 1.5 \quad (12.5)$$

or $k_h = 2$.

To facilitate simple comparison of several solutions, it is helpful to model the scenario such that the Adequacy factor returned is a value of 1.0. To do this set **Horizontal Accel.** k_h (g) to 2.0, and **Vertical Accel.** k_v (g) to 1.0.

If now the acceleration on which **Adequacy** is applied is changed then the results in Table 12.1 are obtained. The anomalous result in the final row relates to the issue of Adequacy Direction (see Section 8.2.7). In the first two cases the failure mechanism involves sliding in the direction of the applied horizontal acceleration and, due to dilation, a small upward movement of the block, *against* the direction of the applied vertical acceleration. In the third case, the solver is restricted to finding a failure mode in the specified direction of the applied vertical acceleration only. The sliding mode is thus not possible and since both blocks are rigid, a bearing type failure cannot be found and a result of *locked* is returned.

Adequacy on k_h	Adequacy on k_v	Result (Adequacy factor)
Yes	No	1.0
Yes	Yes	1.0
No	Yes	*locked*

Table 12.1: Result of applying Adequacy to different parameters

Part IV

User Guide

Chapter 13

Introduction to the User Guide

This part of the manual contains the primary program reference guide for LimitState:GEO. The following topics are described:

- The graphical user interface,
- Setting up and solving a problem
 - specifying the problem geometry,
 - setting the analysis mode (long or short term, translational or rotational),
 - setting material properties,
 - modelling soil reinforcement/Engineered Elements,
 - setting boundary conditions,
 - setting loads,
 - modelling water pressures
 - modelling seismic accelerations,
 - setting adequacy factor,
 - solving,
 - viewing and animating the solution,
 - interrogating the solution,
- Generating a report ,
- Solving for multiple scenarios

The sequence listed under ‘setting up and solving a problem’ is the recommended order to follow when defining and solving a problem.

Chapter 14

The Graphical Interface

14.1 Introduction

The LimitState:GEO graphical interface is designed to give the user maximum flexibility over defining the problem and setting problem parameters while providing new users a clear framework for setting up problems. The default LimitState:GEO screen is divided into a number of areas as shown in Figure 14.1.

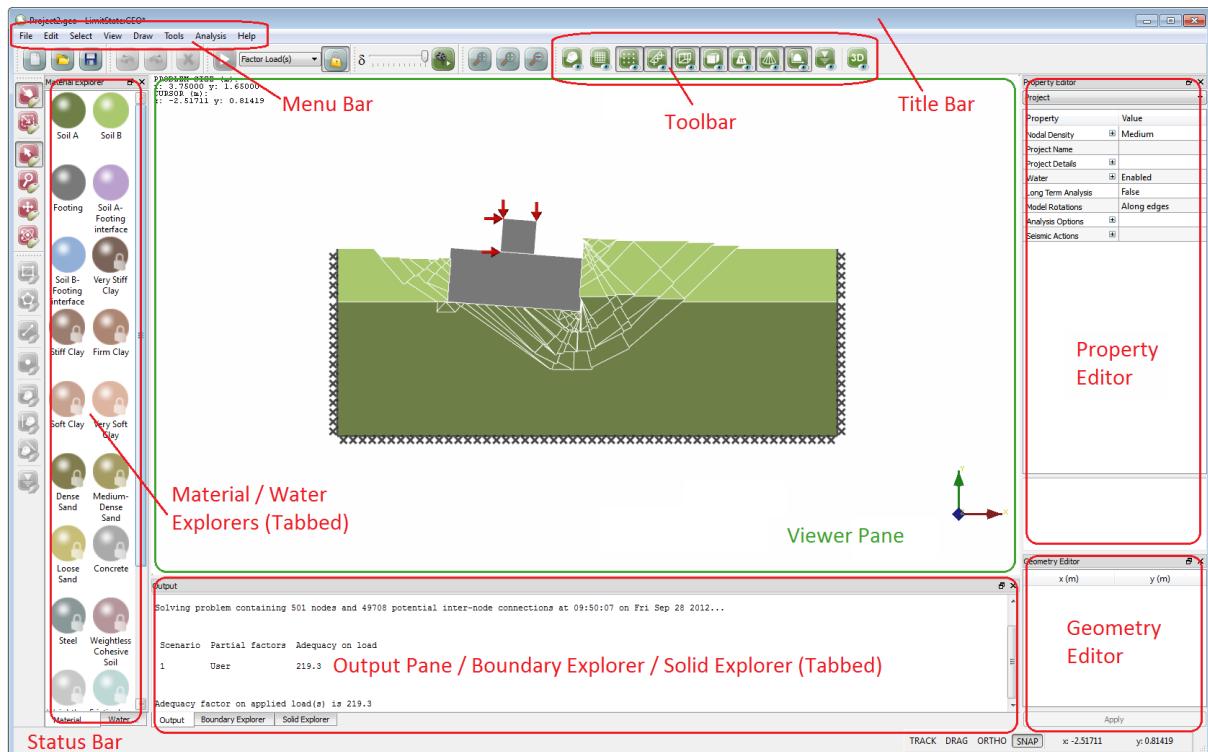


Figure 14.1: Areas of the Graphical User Interface utilized in LimitState:GEO

The areas shown by default are as follows:

- Title bar,
- Menu bar
- Top toolbar
- Left hand toolbar
- **Materials Explorer / Water Regime Explorer** (tabbed),
- **Viewer** pane,
- **Property Editor**,
- **Geometry Editor**,
- **Output pane / Boundary Explorer / Solid Explorer** (tabbed)
- Status bar.

A brief overview of each area is given in the following sections. A fuller description is given in later Chapters. A full list of toolbar and menu items may be found in Appendix B.

Additional items not shown by default are the:

- **Vertex Explorer**
- **Calculator**

14.2 Title Bar

The Title bar (Figure 14.2) displays the name of the software and the name of the file currently being edited. At the right hand side are the **Minimize**, **Restore Down** and **Close** buttons, which may be accessed by left-clicking the relevant icons. These functions may also be accessed via the Title bar context menu, by right clicking anywhere along the bar.



Figure 14.2: The LimitState:GEO Title Bar

14.3 Menu Bar

Menu items (Figure 14.3) may be expanded by left-clicking on the relevant heading. This will bring up a list of functions available for the chosen menu entry. More about the options available in each menu is available in Appendix B.2.

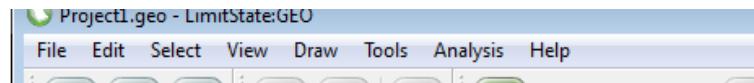


Figure 14.3: The LimitState:GEO Menu bar

14.4 Toolbars

The toolbars (e.g. Figure 14.4) allow direct access to the main functionality of the software. The function relating to each toolbar button may be activated by left-clicking on the relevant icon. Right clicking on any part of a toolbar, brings up the explorer and toolbar selection context menu, which allows toggling the display of interface entities, as depicted in Figure 14.5.



Figure 14.4: The top Toolbars in LimitState:GEO

Tooltips describing to each button are available by hovering the mouse over the icon for a short period. Further description of the toolbars may be found in Section B.3.

The default and optional toolbars are outlined in Appendix B.3.

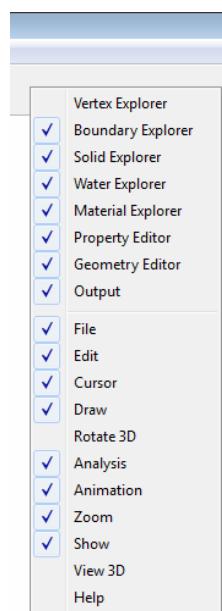


Figure 14.5: Explorer and toolbar selector

14.5 Material Explorer

This pane (Figure 14.6) is, by default, available on the left side of the screen as a tab along with the **Water Explorer**. It displays the materials available to the user. These are divided into two sets: built-in (read only) materials, which are identified by a padlock, and user defined materials, which are freely editable. The user defined materials are displayed at the top of the explorer.

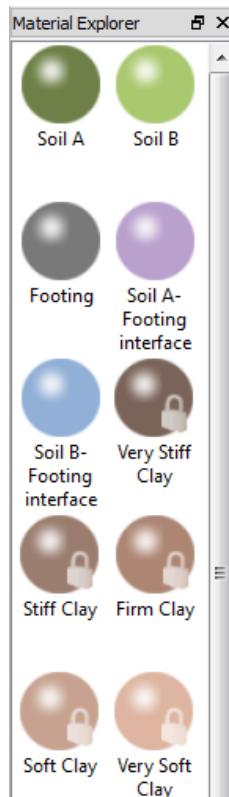


Figure 14.6: The LimitState:GEO Material Explorer

The following actions are possible:

Select Materials may be selected by single clicking the icon with the left mouse button. When selected, material properties appear in the **Property Editor** (Section 14.8), where they can be edited if allowed.

Drag & Drop Materials may be dragged from the explorer and dropped onto objects in the **Viewer** pane by clicking on them with the left hand mouse button and then, with this button held down, dragging them with the mouse to the required location. The object to which the material will be assigned will turn magenta. For further information see Section 17.2.

Material Context Menu Right-clicking on any material brings up a material specific context menu. From here, the following options are available:

- *Delete user defined material* - removes a user-defined material from the **Explorer**

- *Duplicate material* - creates an editable copy of the selected material
- *New material...* - opens a dialog to create a new user-defined material
- *Export materials...* - exports the current materials to CSV
- *Import materials...* - imports pre-defined materials from CSV

14.6 Water Regime Explorer

This pane (Figure 14.7) is, by default, available on the left side of the screen as a tab along with the **Material Explorer**. It displays the water regimes available to the user. These are divided into two sets: built-in (read only) water regimes, which are identified by a padlock, and user defined regimes, which are freely editable. The user defined regimes are displayed at the top of the explorer.

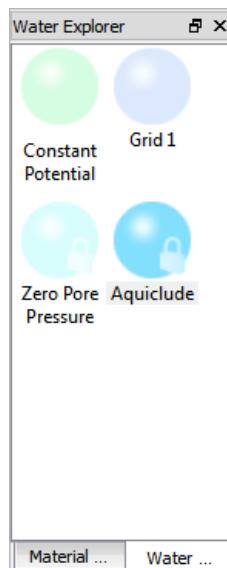


Figure 14.7: The LimitState:GEO Water Regime Explorer

The following actions are possible:

Select Water regimes may be selected by single clicking the icon with the left mouse button.

When selected, material properties appear in the **Property Editor** (Section 14.8), where they can be edited if allowed.

Drag & Drop Water regimes may be dragged from the explorer and dropped onto objects in the **Viewer** pane by clicking on them with the left hand mouse button and then, with this button held down, dragging them with the mouse to the required location. The object to which the regime will be assigned will turn magenta. For further information see Section 20.5.

Water Regime Context Menu Right-clicking on any water regime brings up a water specific context menu. From here, the following options are available:

- *Delete user defined water regime* - removes a user-defined water regime from the **Explorer**
- *Duplicate water regime* - creates an editable copy of the selected water regime
- *New water regime...* - opens a dialog to create a new user-defined water regime
- *Export water regimes...* - exports the current water regimes to CSV
- *Import water regimes...* - imports pre-defined water regimes from CSV

14.7 Viewer Pane

This pane (Figure 14.8) displays the current problem geometry. It provides access to user editable geometry objects.

Object properties may be freely edited using the mouse or keyboard or both depending on their nature. A full description may be found in Chapter 15. Specific geometry objects may be selected by left-clicking with the mouse. The properties of the objects are then displayed in the **Property Editor** (Section 14.8).

The geometry of the model may be freely edited in the viewer by selecting and modifying existing objects or adding new ones.

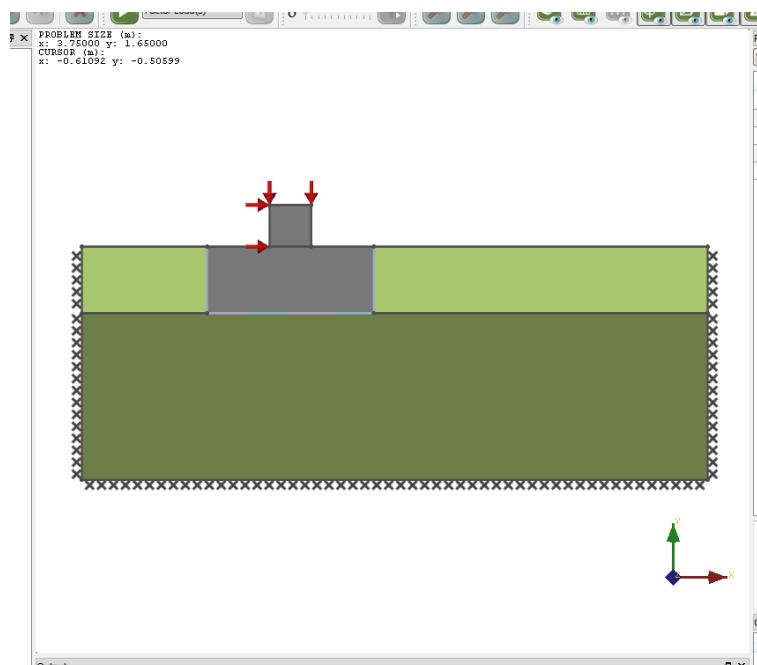


Figure 14.8: The LimitState:GEO Viewer Pane

To scroll the view, use the arrow keys on the keyboard or use the mouse to pan (see also Section 3.8). Similarly, the + and - keys can be used to zoom in and out, while pressing * will zoom to the extents of the model geometry (see also Section 3.7).

14.8 Property Editor

The **Property Editor** provides core access to problem parameters in a direct and intuitive way. In general the properties of any material, water regime or geometry object may be displayed simply by selecting it in the an **Explorer** or the **Viewer** pane. In addition, global project level parameters may be displayed at any time by left-clicking on an empty part of the **Viewer** pane with the mouse. The **Property Editor** is shown in Figure 14.9 and is generally located at the right of the user interface.

Single clicking on any item in the **Property** column of the **Property Editor** gives an expanded explanation of the parameter in the window at the base of the **Property Editor**. In turn this expanded explanation may contain one or more hyperlinks to the manual to take the user to a more comprehensive source of information. An ‘expand’ symbol (+ or arrow) next to an item in the **Property Editor** indicates that there are additional sub-parameters relating to that item, that may be viewed. Click on the expand symbol to access these. Left-clicking on a value in the **Property Editor** allows you to modify it by typing or selecting your choice (unless it is read only). For specific parameters a clickable button may also appear which gives access to a further dialog to provide additional functionality.

The calculator (see Section 14.13) may be used in any numeric data entry cell.

Right clicking on items that have been selected, brings up a context menu relevant to that item.

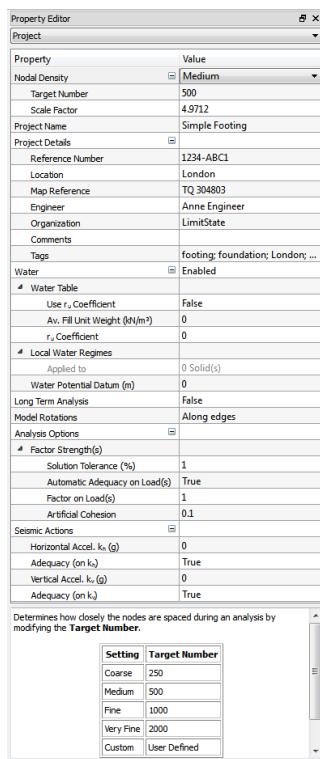


Figure 14.9: Project properties displayed in the Property Editor

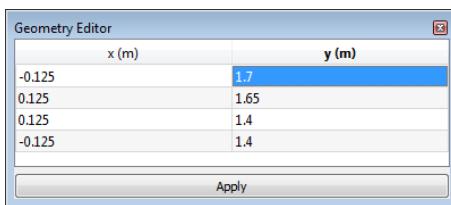
14.9 Geometry Editor

The **Geometry Editor** (Figure 14.10) provides direct access to the co-ordinates of any geometry object. When a geometry object is selected, for example by left-clicking on it in the **Viewer Pane**, the **Geometry Editor** displays the (x, y) coordinate of each vertex associated with that object.

Any value in the **Geometry Editor** may be altered by:

1. Clicking on the value
2. Typing a new value
3. Pressing Enter
4. When all new parameters have been entered to the user's satisfaction, clicking the **Apply** button.

The arrow or tab keys may also be used to navigate through the coordinates.



A screenshot of a Windows-style dialog box titled "Geometry Editor". It contains a table with two columns: "x (m)" and "y (m)". The table has five rows of data. The fifth row, which contains the values "-0.125" and "1.4", is highlighted with a blue selection bar. At the bottom of the dialog is a single "Apply" button.

x (m)	y (m)
-0.125	1.7
0.125	1.65
0.125	1.4
-0.125	1.4

Figure 14.10: The LimitState:GEO Geometry Editor

Any or all values may be selected and copied and pasted to another application. Conversely a table of coordinates may be copied and pasted from another application to the geometry table. To copy, select any rectangular block of data (i.e. single cell, column, row or block of cells) and then right-click with the mouse to display the context menu. Then select **Copy**.

To paste data, copy one or more cells from another application and **Paste** to a cell in the geometry editor. The data will be pasted relative to this cell. Any data extending beyond the bounds of the geometry editor table will be ignored.

14.10 Output Pane

This pane (Figure 14.11) is, by default, available at the bottom of the interface as a tab along with the **Boundary Explorer** and **Solid Explorer**. It is used to display solution information (including the **Adequacy factor**) and messages to the user. Any portion of text may be copied and pasted to another window/text box or other application.

To select text, perform the following steps:

- left-click on the text and drag with the mouse to select a specific block of text,
- double click on a word to select it,
- triple click on a line to select it.

To copy, press CTRL-C or right click and select ***Copy*** from the context menu.

Right clicking anywhere on the window, brings up the context menu.

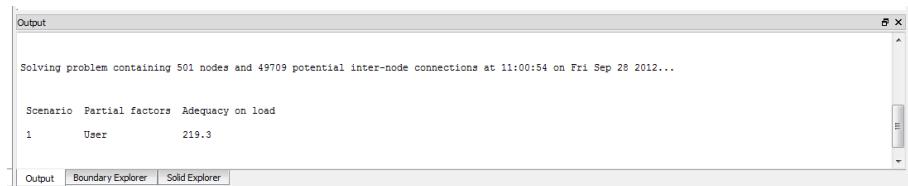


Figure 14.11: The LimitState:GEO Output Pane

The amount of information displayed in the **Output Pane** may also be increased to include the iteration data by ticking the checkbox ***Display iteration information in the output window***. This is available in the **Solve** tab of the **Preferences Dialog** (see Chapter 28).

14.11 Status Bar

The **Status Bar** (Figure 14.12) is used to provide:

- Short term progress messages to the user (at the left side).
- Drawing settings (described further in Section 14.12, Section 15.8, and Section 15.9):
 - **TRACK** - Zooms the viewer to highlight objects selected in **Explorers**
 - **DRAG** - Allows on-screen objects to be moved using the mouse.
 - **ORTHO** - Restricts on-screen movement of objects to the Cartesian directions
 - **SNAP** - Snaps objects to points of interest in the model (e.g. **Vertices**)
- Display of the (x, y) coordinates of the current mouse position, when the mouse pointer is within the **Viewer** pane.

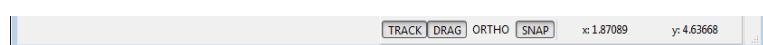


Figure 14.12: The LimitState:GEO Status Bar

14.12 Vertex, Boundary and Solid Explorers

14.12.1 Introduction

The explorers provide lists of all the geometry objects present in the current problem according to type. the **Boundary** and **Solid** explorers also display specific properties of the **Geometry objects** which may be adjusted here in the same way as in the **Property Editor**.

Clicking on any item in the explorer will cause it to be selected and highlighted in the **Viewer** pane. In addition if the **TRACK** setting is active (see Section 14.11) then the **Viewer** pane will zoom and highlight the selected geometry object.

14.12.2 Vertex Explorer

The **Vertex Explorer** (Figure 14.13) lists the **Vertices** present in the problem. If the **TRACK** setting is active (see Section 14.11), clicking on a row in the explorer will select that vertex and cause the viewer to zoom in on it. Additionally, the **Geometry Editor** will display the coordinates of the selected object.

The **Vertex Explorer** is not displayed by default, but may be activated using the **View > Explorers > Vertex Explorer** menu function.

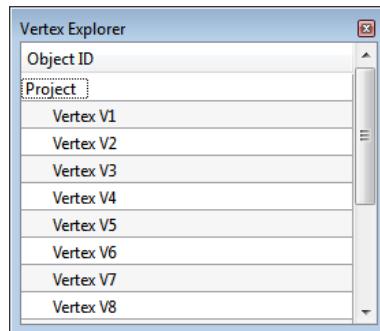


Figure 14.13: The Vertex Explorer

14.12.3 Boundary Explorer

The **Boundary Explorer** (see Figure 14.14) is displayed by default as a tab alongside the **Output Pane** and **Solid Explorer**. It lists all the **Boundaries** present in the problem and can be used to quickly determine or modify the following properties:

Support Type *Free, Fixed or Symmetry* (see Chapter 18).

Loading *Permanent, Accidental or Variable* (see Chapter 19).

Loading Type *Favourable, Unfavourable or Neutral* (see Section 19.4).

Adequacy *True or False* (see Section 8.2).

If the **TRACK** setting is active (see Section 14.11), clicking on a row in the explorer will select that boundary and cause the viewer to zoom in on it.

Boundary Explorer				
Object ID	Support Type	Loading	Loading Type	Adequacy
Boundary B49 Free	-	-	Neutral	-
Boundary B50 Free	-	-	Neutral	-
Boundary B51 Free	-	-	Neutral	-
Boundary B52 Free	-	-	Neutral	-
Boundary B53 Free	-	-	Neutral	-
Boundary B54 Free	Variable	Unfavourable	true	
Boundary B55 Free	-	Neutral	-	
Boundary B56 Fixed	-	Neutral	-	
Boundary B57 Free	-	Neutral	-	
Boundary B58 Free	-	Neutral	-	
Boundary B59 Fixed	-	Neutral	-	
Boundary B60 Fixed	-	Neutral	-	
Boundary B61 Fixed	-	Neutral	-	

Figure 14.14: The LimitState:GEO Boundary Explorer

14.12.4 Solid Explorer

The **Solid** explorer (see Figure 14.15) is displayed by default as a tab alongside the **Output Pane** and . It lists the **Solids** present in the problem and can be used to quickly determine or modify the following properties

Adequacy *True or False* (see Section 8.2).

Loading Type *Favourable, Unfavourable or Neutral* (see Section 19.4).

Water Regime Lists the type of **Water Regime** applied to the zone, if any (see Section 6.2.3).

Clicking on a row in the explorer will select that solid and cause the viewer to zoom in on it.

Solid Explorer			
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 14.15: The LimitState:GEO Solid Explorer

14.13 Calculator

The calculator is used to facilitate entry of data into the **Property Editor** and the Wizards in the following circumstances:

- When the parameter needs computing from several source parameters.

- When the parameter needs to be converted from different units.

If the mouse is clicked on any wizard or **Property Editor** numeric data entry cell, the calculator button  appears in the right hand end of the cell. Click on this to display the calculator. The calculator (see Figure 14.16) has all the normal functionality of a standard calculator plus the ability to convert units. The units conversions are displayed according to the parameter type in the cell. The value appearing in the results window of the calculator automatically appears in the data entry cell. There is no need to copy and paste to the cell in question.

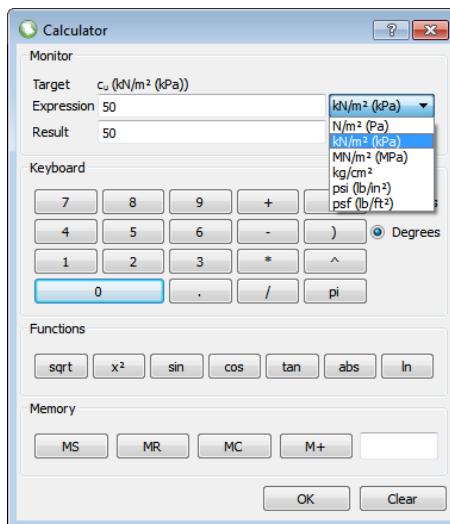


Figure 14.16: The calculator window

Chapter 15

Specifying the Problem Geometry

15.1 Terminology

Problem geometries are built up using **Geometry objects**. There are three types of geometry object:

Solid This is a 2D polygon defining a body of soil or other material. Its extent is defined by the surrounding **Boundary** objects.

Boundary This is a straight line that links 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.

Vertex This is a point 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.

In LimitState:GEO all geometry is defined in terms of straight lines. Curves may be represented by a series of short straight line segments. **Boundary** and **Solid** objects can both be assigned material properties.

In general most geometry will be built up using the draw **Rectangle** and **Polygon** functions. If a polygon is added adjacent to an existing polygon, LimitState:GEO automatically ensures that each polygon shares the interface **Boundary object**.

15.2 Setting the Units

The LimitState:GEO interface can display either Metric (kN, m) or Imperial (lb, ft) units.

To set the units used, open the **Preferences...** dialog in the **Tools** menu. Go to the **Units** tab and select the units that you wish to use (see Figure 15.1). Changes will take effect when LimitState:GEO is next restarted.

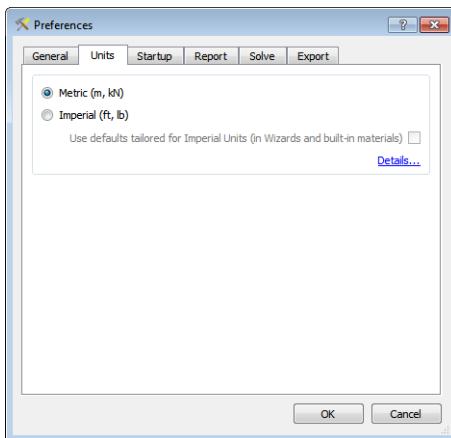


Figure 15.1: Changing units in LimitState:GEO

The default units used in each setting are given in Table 15.1.

Measurement	Metric	Imperial
Length	m	ft
Area	m^2	ft^2
Volume	m^3	ft^3
Force	kN	lb
Weight	kN	lb
Force per unit width	kN/m	lb/ft
Unit weight (weight density)	kN/m^3	lb/ft^3 (pcf)
Pressure	kN/m^2 (kPa)	lb/ft^2 (psf)
Angle	degrees	degrees
Bending moment	kNm	$lb.ft$ (pf)
Cohesive strength	kN/m^2 (kPa)	lb/ft^2 (psf)

Table 15.1: Default units used in Metric and Imperial modes

Imperial conversions of Metric measurements can result in values with many significant figures, which can be difficult to work with. The option to use ‘tailored’ default Imperial values in the Wizards and built-in materials is therefore also provided. By selecting this option, the pre-defined values for problem geometries and material properties are rounded to an appropriate number of significant figures.

15.3 Starting with an Empty Project

While problem geometries may be rapidly built for common scenarios using the Wizards (see Section 4), here it is assumed that the user is starting with an empty project.

To start with an empty project, select **Create a new project** in the **Welcome to LimitState:GEO** dialog and click **OK**. The **New Project** dialog (see Figure 4.1) will then start. Select the **Empty Project** and click **OK** to display the the **Empty Project** dialog shown in Figure 15.2.

15.3.1 Project Details

The first tab on the **Empty Project** dialog (Figure 15.2) is for the input of project data. None of the fields are mandatory, but a default name of **New Project** will be displayed and used unless changed by the user. Once all the necessary fields have been filled in, clicking **Next** will advance on to the next stage of problem definition.

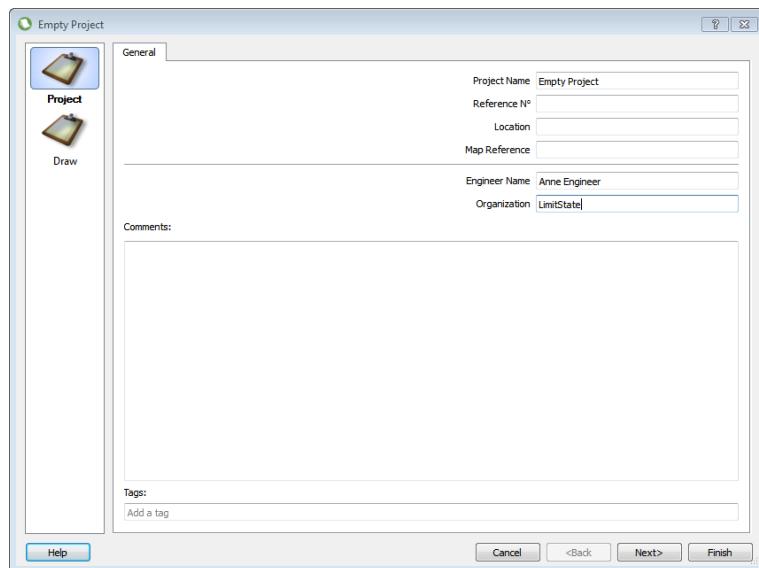


Figure 15.2: The **Empty Project** dialog (**Project Details** tab)

15.3.2 Draw Settings

The second tab in the **Empty Project** dialog displays the options for setting up the drawing environment. It is concerned with establishing a suitable grid geometry and snap settings to enable a model to be produced in a quick and efficient manner. The grid is a useful aid to modelling and, when set up correctly in conjunction with the snaps, can save the user much time and effort.

15.3.2.1 Grid Settings

To display the grid from the start of modelling, simply ensure that the **Grid** check box is ticked in the **Draw Settings** tab (this is done by default). There are then several variables that may be altered in order to tailor the size, spacing and behaviour of the grid:

Auto-calculate extents Ticking this check box causes LimitState:GEO to automatically adjust the extents of the grid during modelling so that it is always slightly bigger than the objects

being drawn. When enabling this feature in the wizard, the initial size of the grid is determined by the values entered for the **Minimum** and **Maximum** project extents and **Major gridline spacing**.

Minimum Sets the initial minimum x and y coordinates of the grid (note that this value will be overridden if **Auto-calculate extents** is enabled).

Maximum Sets the initial maximum x and y coordinates of the grid (note that this value will be overridden if **Auto-calculate extents** is enabled).

Major gridline spacing Sets the initial distances (in metres) between the major x and y gridlines.

Minor gridlines Determines the initial number of minor intervals between the major gridlines.

15.3.2.2 Snap Settings

Object snap in LimitState:GEO is very similar to that found in many CAD and drawing packages - it causes the cursor to become attracted to certain features of the model, such as points or intersections between lines.

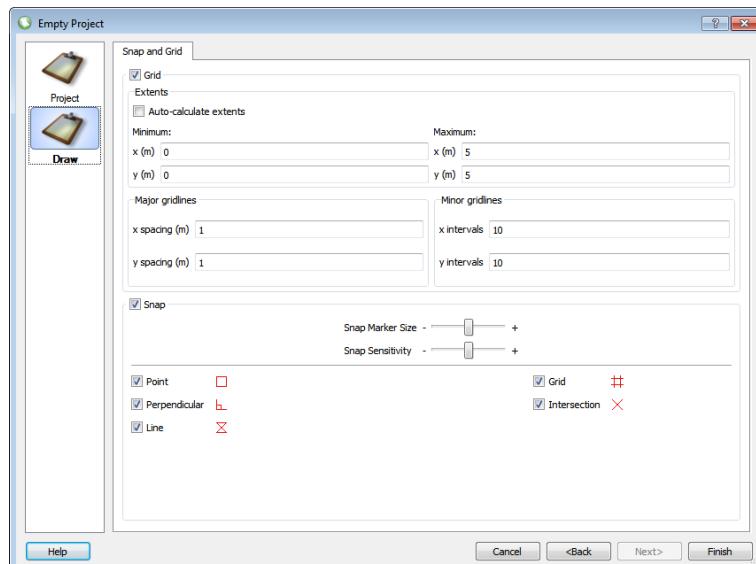


Figure 15.3: The **Empty Project** dialog (**Draw Settings** tab)

To use the snap functionality from the start of modelling, simply ensure that the **Snap** check box is ticked in the **Draw Settings** tab (see Figure 15.3). This is done by default. You can then fine-tune the size of the snap markers and select the type of snaps that you wish to use:

Snap Marker Size Use the slider to make the snap marker larger (+) or smaller (-).

Snap Sensitivity This determines how close the cursor needs to be to a particular feature in order to snap to it. Use the slider to make the distance smaller (+) or larger (-).

Point Causes the cursor to snap to **Points / Vertices**.

Perpendicular When drawing a line or polygon, this will cause the cursor to snap to a point on an existing line that forms a right angle between that and the line being drawn.

Line Causes the cursor to snap to *Line Objects*.

Grid Causes the cursor to snap to points of intersection between the gridlines.

Intersection Causes the cursor to snap to points of intersection between the gridlines and the *Line Objects* of the model.

Clicking **Finish** will end the setup process and display the modelling environment. The grid can subsequently be adjusted and toggled on or off at any time during modelling via the **Settings** (see Section 15.8) dialog of the **Draw** menu and the  icon respectively. Similarly, the snap functionality can be quickly toggled by clicking the **SNAP** button on the status bar.

15.4 Starting with a DXF Imported Geometry

The user may often find it useful to import a problem geometry that has already been pre-defined using a CAD package. This can be accomplished in LimitState:GEO by using the **Import... > Geometry...** function, found under the **File** menu as shown in Figure 15.4

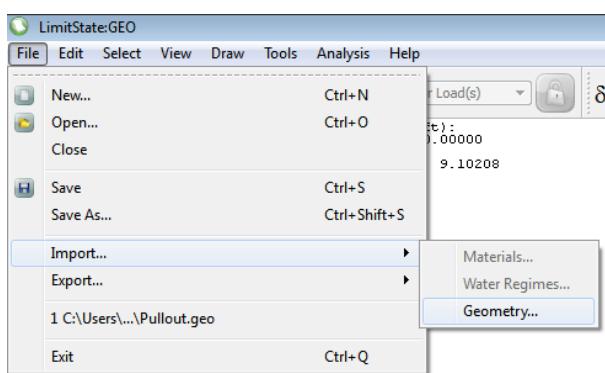


Figure 15.4: Importing geometry (DXF) via the **File** menu

Clicking this will bring up a dialog that allows you to browse for a pre-saved DXF file.

Once a DXF file has been selected, the DXF Import dialog will appear (Figure 15.5). This offers the user several options:

1. **TRANSFORMING THE GEOMETRY** Sometimes the model within a DXF file will be centered far away from the origin. If LimitState:GEO detects that this is the case, the option to translate the model (and potentially also scale at the same time) is presented to the user. First the bounding box is determined. The bottom-left corner will then be moved according to the settings provided in the 'Translate' fields. Finally, if a scale factor is also specified, the whole model will be resized using (0, 0) as the reference point.
2. **HOW TO INTERPRET CLOSED LOOPS**

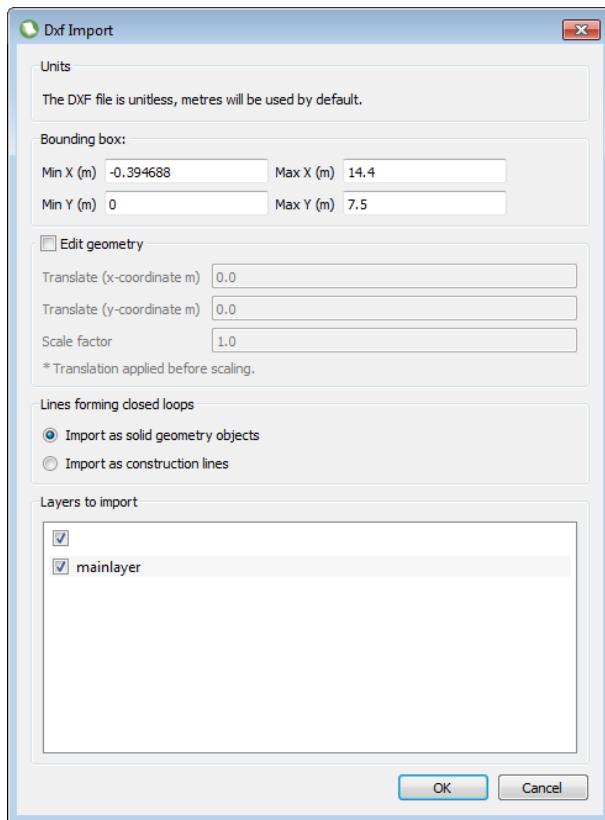


Figure 15.5: The **DXF Import Dialog**

LimitState:GEO deals with solid bodies and the interfaces between them; therefore, by default, the DXF import function recognizes lines that form a closed loop and interprets them as being a solid object (Figure 15.6). Lines that do not form a closed loop will be imported as construction lines so that the user is able to still use them when defining the model geometry. Alternatively, there is the option to interpret all lines within the DXF file as being construction lines. If this option is selected, LimitState:GEO solids and boundaries will not be generated and the user must draw them. In all cases, imported DXF construction lines appear as solid grey lines in order to easily differentiate them from those defined within the program.

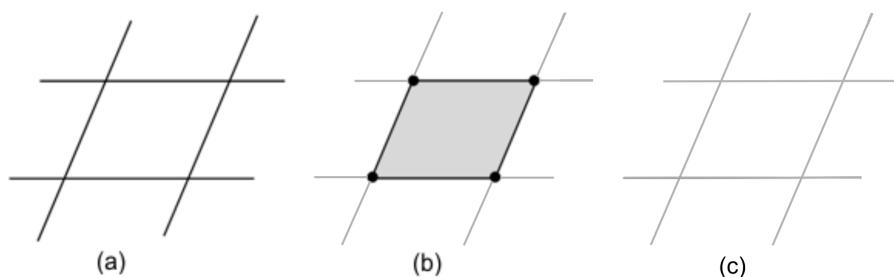


Figure 15.6: Solid object identification using DXF import: (a) Original DXF file contents, (b) Solid object, boundaries, construction lines and vertices as imported with closed loops interpreted as solid objects and (c) construction lines as imported with closed loops interpreted as construction lines.

3. THE CHOICE OF LAYERS TO IMPORT

DXF files can sometimes be composed of several layers and the user may not always wish to import all of them into their LimitState:GEO model. All layers present in a file are listed in the DXF Import dialog; to prevent one or more from being imported, simply un-check the box next to the name of the appropriate layer (N.B. where a layer has not been given a name, ‘Unnamed Layer’ will appear in the import dialog).

15.4.1 Permitted DXF Commands

The following commands can be used to create a DXF file that is readable by LimitState:GEO (other commands, including those used to generate 3D objects, will not be recognized):

- Point
- LWPolyline
- Line
- Polyline
- XLine
- Spline
- Vertex

Closed loops of lines forming ‘islands’ within other closed loops will be recognized and imported without difficulty. However, additional lines will be automatically generated to connect these ‘islands’ to the surrounding geometry (to allow LimitState:GEO to analyse the problem properly).

15.4.2 Tips for Modifying a DXF in an External CAD Package

Whilst working on a DXF file for import into LimitState:GEO it is useful to bear in mind the following points:

ENSURE LOOPS ARE PROPERLY CLOSED In most CAD packages it is possible to work to a very fine precision. This can sometimes lead to situations where lines that appear to intersect do not actually meet in reality. Whilst not necessarily causing a problem in the CAD model, if two lines of a loop do not intersect when importing a DXF into LimitState:GEO, a solid object will not be formed. Care should therefore be taken to ensure that, where objects are expected to be converted to solids, these are properly drawn in the CAD model before exporting to DXF (working with a grid or object snap turned on can help with this).

REMOVE UNNECESSARY OBJECTS Only those types of drawing object listed in Section 15.4.1 will be recognized and imported into LimitState:GEO. If a CAD file is being worked on specifically for import into LimitState:GEO then text, hatching and other similar objects should be removed before the DXF file is generated. Failure to do this may lead

to unwanted objects being created or to a loss of program efficiency as LimitState:GEO processes the superfluous data.

KEEP THE DRAWING AS SIMPLE AS POSSIBLE When importing from a DXF file, every intersection of two or more lines will be converted to a LimitState:GEO vertex. These vertices are, in turn, regarded as nodes along the boundaries of solid elements at the time of analysis. As LimitState:GEO normally works with a fixed number of nodes, excessive detail (leading to large numbers of boundary nodes) will reduce the number of nodes inside solid elements and potentially reduce the accuracy of the solution obtained. Wherever possible, the user is advised to appropriately simplify the ground profile of any problem before importing and try to achieve a suitable balance of boundary and internal nodes in the LimitState:GEO model. This advice is particularly relevant when working with DXF files containing multiple layers.

WORK AT THE CORRECT SCALE LimitState:GEO is set to work in either metres or feet (dependant upon the choice of units in the **Preferences** menu). It should therefore be ensured that all imported DXF files are drawn to an appropriate scale.

BE AWARE THAT LAYERS ARE FLATTENED Since LimitState:GEO works with planar models, when multi-layered DXF files are imported, all layers are first flattened. Any overlapping lines that were distinct in the original file will no longer be separate, which may result in undesired problem geometries. It is recommended that such DXF files are simplified to a single layer representing the problem the user wishes to model.

WORK IN THE XY PLANE Care should be taken to ensure that the correct viewpoint is being used when preparing a DXF file for import. LimitState:GEO works in the *xy* plane, which corresponds to the ‘Plan’ or ‘Top’ 3D viewpoint in most CAD packages. Objects that lie predominantly in the *xz* or *yz* planes may be imported in a way that is not desirable. It is recommended that DXF files containing 3D components are simplified to a single (2D) layer representing the problem the user wishes to model.

15.4.3 Permitted DXF Types

DXF files can be produced in either ASCII or binary format. LimitState:GEO will only recognize those files that have ASCII encoding. It is therefore recommended that the user checks the type of any DXF file before attempting an import.

15.4.4 Following DXF Import

Following a successful DXF import, it will be necessary to apply the requisite material properties, loading and support conditions to the model before an analysis can be attempted.

15.4.5 Export to DXF

The **Export... > Geometry...** command (also found in the **File** menu) will save the geometry of any LimitState:GEO project to a suitably formatted DXF file. If the project is in a solved state, it is possible also to save the entities generated during the solution process, such as nodes and slip-lines, to different layers in the DXF (see also Section 25).

15.5 Construction Lines

In addition to the Grid, construction lines can provide a useful tool when setting up a problem geometry. These lines appear as dotted grey lines in the viewer pane and possess the same snap functionality as the standard geometry objects but, like the grid, construction lines are not counted as part of the problem during analysis.

Construction lines can be dragged and deleted, just like standard drawing objects. In addition, to clear all of the construction lines at the same time, simply choose ***Clear all construction lines*** from the ***Construction lines*** section of the ***Draw*** menu or right-click in the viewer to bring up the viewer context menu and select ***Clear all construction lines*** from there.

There are three types of construction line available: Horizontal, Vertical and Custom (angled):

15.5.1 Horizontal Construction Lines

To draw a horizontal construction line, click the  button in the toolbar. When hovering over the **Viewing Pane**, the cursor will be accompanied by a solid, horizontal black line that denotes the current position of the construction line; to place this in an approximate position, simply use the cursor position to choose an appropriate y coordinate and left-click the mouse. For a more accurate positioning, right-click the mouse and choose the ***Set y coordinate*** option. A dialog will then appear that allows a precise y coordinate to be entered. The option to continue placing construction lines will remain until the user presses the ***ESC*** button on the keyboard or right-clicks the mouse and chooses the ***Cancel*** option.

15.5.2 Vertical Construction Lines

To draw a vertical construction line, click the  button in the toolbar. The procedures for adding vertical construction lines are the same as for adding horizontal construction lines except that the x coordinate is now set instead of the y.

15.5.3 Custom Construction Lines

To draw a custom (angled) construction line, click the  button in the toolbar. The next left click of the mouse will then set the first point of the construction line - this will be taken as the origin of rotation. Alternately, right-click the mouse and choose ***Add point*** to set the origin more precisely.

A solid black line will now appear that rotates around the origin point with any movement of the cursor. To set the angle by specifying a second point, simply left-click the mouse in the **Viewing Pane** or right-click and choose ***Add point*** to set the destination more precisely. Should it be necessary to enter an angle of rotation instead of a second point, right-click the mouse and choose the ***Set angle*** option. A dialog will then appear that allows an angle (in degrees) to be entered. This is measured around the origin point, from the horizontal axis in an anti-clockwise direction.

15.6 Drawing Functions

15.6.1 Rectangle

Click on the toolbar button , or from the menu select **Draw, Rectangle**. Left click with the mouse anywhere on the geometry window and then with the mouse button held down, drag a rectangle. This function is used to draw horizontal soil layers and structural elements.

15.6.2 Polygon

Click on the toolbar button , or from the menu, select **Draw, Polygon**. Left click with the mouse anywhere on the geometry window to draw the first vertex of the polygon. Then move the mouse to the position of the next vertex and left click to draw this. Continue until all vertices of the polygon have been drawn. To complete the polygon, right-click to display the context menu and select **Finish**. This will connect the last vertex drawn to the first vertex (there is no need to click on the first vertex again).

To exit from drawing the polygon and clear the currently drawn vertices, press ESC.

When drawing a **Polygon**, it is permissible to utilize existing **Vertex objects** as part of the polygon. LimitState:GEO can be set to snap to existing geometry objects to facilitate this (see Section 15.8). This is the normal way in which polygons are added adjacent to existing polygons.

15.6.3 Line

Lines (**Boundary objects**) are not allowed to exist in isolation in LimitState:GEO and are used to split existing **Solid objects**. To draw a line, click on the toolbar button , or from the menu, select **Draw, Line**. Now left-click on any existing **Line** or **Vertex** and then using the mouse connect it to another **Line** or **Vertex** by clicking on it. The **Line** so drawn must pass across an existing **Solid object**. If this is not the case then a warning message is displayed. If the line is legitimate then a message appears in the output pane stating that the **Solid object** has been split.

When a **Line** is drawn from a point on one **Line** to a point on another across an existing **Solid object**, additional **Vertices** will automatically be created where required.

15.6.4 Vertex

Vertex objects are not allowed to exist in isolation in LimitState:GEO but may be used to split existing **Boundaries**. To add a **Vertex**, click on the toolbar button , or from the menu, select **Draw, Vertex**. and left-click on any existing **Boundary**. To specify multiple vertices

at different locations in the same action, hold down the CTRL button while left-clicking at the desired locations.

If the vertex is legitimate then a message appears in the output pane stating that the boundary has been split.

15.7 Selecting Objects

15.7.1 Single Click Selection

Single selection mode may be activated by using the toolbar buttons ‘Select’  and ‘Click’ . In this mode any object may be selected by clicking on it with the mouse. As soon as it is selected it will be highlighted and its properties will appear in the **Property Editor**. If it is a **Geometry Object**, then its coordinates will also appear in the **Geometry Editor**.

15.7.2 Rectangle Selection

Rectangle selection mode may be activated by activating the toolbar buttons ‘Select’  and ‘Rectangle’ . In this mode any object or set of objects may be selected by dragging a rectangle around them with the mouse. The rectangle must fully enclose the required items. As soon as they are selected they will be highlighted and common properties shared by all the selected items will appear in the **Property Editor**. To select only specific types of objects, use the **Selection Filter** dialog (Figure 15.7) from the main menu (Select - Filters...).

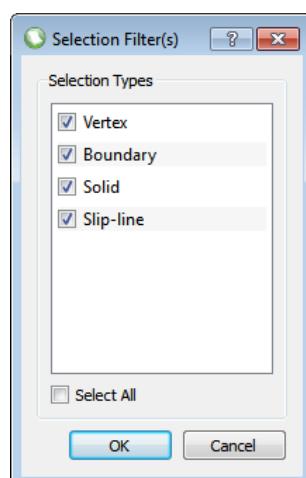


Figure 15.7: The Selection Filter dialog

15.7.3 Multiple Selection

Once a selection has been made using ‘Rectangle’ or ‘Click’, additional objects may be added to the selection by holding the **CTRL** key down while making additional selections using either ‘Rectangle’ or ‘Click’. Already selected objects may also be removed from the selection by re-selecting them using either ‘Rectangle’ or ‘Click’.

15.8 Snapping to Other Objects

It is possible to snap to most other geometrical objects on the screen. Extensive control is provided via the Snap tab in the Preferences dialog accessible via the **Draw > Settings** menu as shown in Figure 15.8.

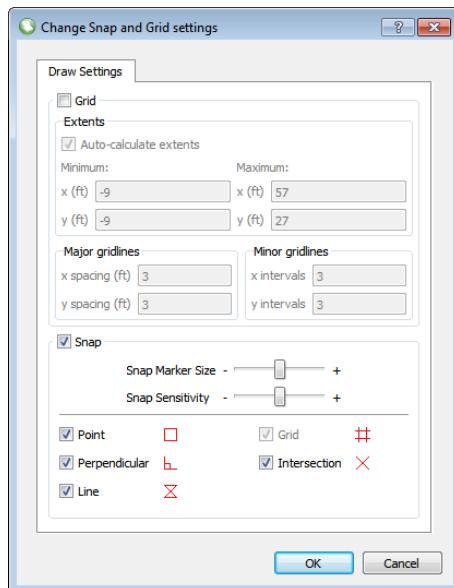


Figure 15.8: The settings dialog in the **Draw** menu

Global control over snap is also provided by the **SNAP** button on the **Status bar**.

15.9 Modifying the Geometry

15.9.1 Using the Mouse

Any geometry object (**Solid**, **Boundary**, **Vertex**) may be selected and then dragged using the mouse (hold down the left button while moving the mouse). This provides a quick and flexible way to modify the geometry. If the modification should distort the geometry into an invalid form (e.g. turn a polygon inside out) then a warning message is displayed and the original geometry is restored.

The following actions are possible:

- Dragging a **Solid object** will drag the solid and its adjacent **Boundary objects** and **Vertex objects**.
- Dragging a **Boundary object** will drag the boundary and its end point **Vertex objects**.
- Dragging a **Vertex object** will drag the vertex and the ends of the adjacent **Boundary objects**.

Dragging is enabled by clicking on the **DRAG** button on the **Status bar**. Dragging may be constrained to orthogonal directions only by clicking the **ORTHO** button on the **Status bar** or by pressing **F8**.

To **Delete**, the following methods are recommended for the listed object types:

Vertex and Boundary drag the Vertex at one end point of the Boundary onto the Vertex at the other end point of the Boundary. The Vertex objects will be merged and the Boundary object deleted.

Vertex Select the Vertex and click Delete . The Vertex and one of the adjacent Boundary objects will be deleted. It is not defined which of the two Boundary Objects will be deleted. If this would result in an ambiguous or illegal geometry this action is not permitted. To ensure a specific Boundary Object is deleted, use the method for deleting a Vertex and Boundary.

Boundary Select the Boundary and click Delete . The outcome for this action depends on the location of the Boundary:

- If the Boundary object is an internal interface between two Solids, then the Boundary object is deleted and the Solids are merged.
- If the Boundary object is an external Boundary, it will be deleted together with its adjacent Vertex objects and one of the adjacent Boundary objects. The Vertex objects at the other end points of these deleted Boundary objects will be linked by the remaining Boundary. It is not defined which of the two Boundary Objects will be deleted. If this action would result in an ambiguous or illegal geometry this action is not permitted.

Solid Select the Solid and click Delete . The Solid is then deleted together with all adjacent Boundary and Vertex objects that are not shared with other Solids.

It is usually preferable to delete objects by dragging and merging so that the properties of the remaining amended objects are retained.

15.9.2 Connecting Geometry Objects

To connect a **Solid object** to another **Solid object**, it is best to work with the relevant vertices. Ensuring that Snap to Point is activated (see Section 15.8) will make certain that the objects share common vertices rather than simply having vertices in very close proximity to each other.

For the example in Figure 15.9, two initially unconnected square solid zones are to be modified so that, after the process is completed, they share a common boundary:

- a** The first vertex is selected. It is then clicked and the left mouse button held down.
- b_i** The selected vertex is dragged over the target vertex (a red rectangle signifies that the two will be merged upon release the left mouse button). The left mouse button is then released.
- b_{ii} and c** The above steps are repeated for the second vertex.

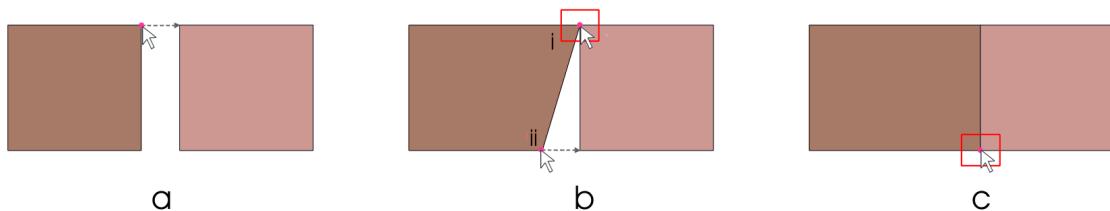


Figure 15.9: Connecting two solids by merging their vertices

15.9.3 Overlapping Geometry Objects

To overlap one **Solid object** with another, drag the former over the latter. This process will then split the original two **Solid objects** into three new **Solid objects**. If a **Solid object** is dragged over more than one other **Solid object** then a multiple split will be carried out.

For the example in Figure 15.10, two initially unconnected square solid zones are to be modified so that, after the process is completed, they overlap one another:

- a** The first solid is selected. It is then clicked and the left mouse button held down.
- b** The selected solid is dragged over the target solid and the left mouse button is then released.
- c** A new zone is created by the overlapping areas of the two solids. Should the two original solids have different associated materials (as in this case), the new zone will contain a combination of all the materials and will be displayed with a 'checkerboard' pattern (for more information on assigning materials to zones see Section 17.2).

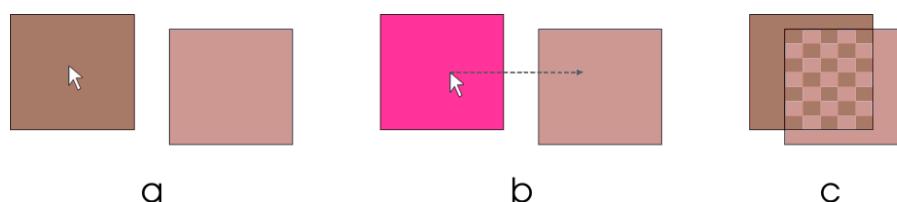


Figure 15.10: Overlapping two solids using drag and drop: a) selection of the first solid; b) dragging the solid and c) dropping the solid and creation of a new zone.

15.9.4 Changing the End Vertex of a Boundary

There are several ways in which the end vertex of a line can be changed. However, in situations where the geometry is perhaps rather complex, the following method may be used (Figure 15.11):

- a Using the **Line** function (see Section 15.8) draw a new boundary line in the desired position.
- b Use the mouse pointer to select the original boundary line then press the **Delete** button to remove it.
- c If the zone created in step 2 now contains a mixture of materials (c), you can alter this by setting the materials as described in Section 17.2.
- d If desired / appropriate, the original vertex may now be deleted.

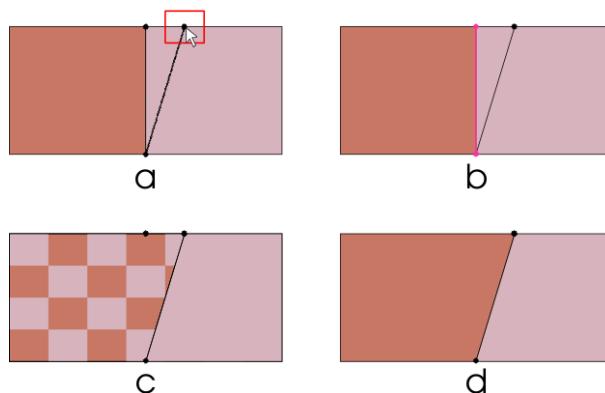


Figure 15.11: Moving a Boundary

Easier methods in the above case would have been to edit the coordinates of the vertex in the **Geometry Editor** or drag and drop the original endpoint of the line using the mouse (possibly with ORTHO enabled). However, these shortcut approaches are not always suitable in practice, for example where the vertex also defined the geometry of another **Solid** which was to be left unchanged.

15.9.5 Creating One Object Inside Another

It is often useful to create a zone that lies entirely within another solid (for example to model a material lens or large void). To achieve this, simply draw the new zone in the required position using the rectangle () or polygon () draw functions. Additional boundary lines will be created in order to link the new zone to the rest of the model and the new zone will automatically be assigned the material of the surrounding solid. To create a lens of material from the new zone, change the material as described in Section 17.2. To create a void, select the new zone with the mouse and press the **Delete** button (Figure 15.12).

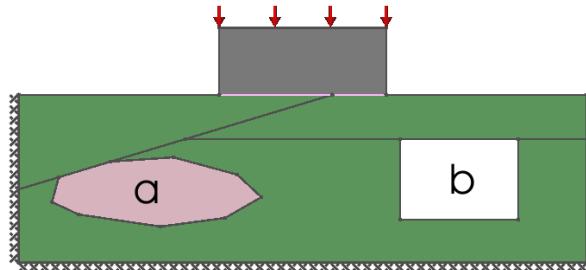


Figure 15.12: Creating Zones Within a Solid. a) Lens and b) Void

15.9.6 Using the Geometry Editor

Instead of utilising the mouse, the coordinates of points making up any geometry object may be directly edited within the Geometry Editor (see Section 14.9). Select the required object and its coordinates will appear in the Geometry Editor. When all changes have been made, click the **Apply** button in the Editor to commit the changes.

15.10 Undo/Redo

Many specific modifications may be undone utilising the Undo button or the menu **Edit**, **Undo**. Any undone actions may be redone using the Redo button or the menu **Edit**, **Redo**. Redo may only be carried out if no intervening modifications to the geometry have been made. If **Undo** or **Redo** are not available then the toolbar buttons and menu items are disabled (appear in grey). Hovering the mouse over the Redo or Undo button will display a tooltip describing the last action.

15.11 Editing Geometry Object Properties

In addition to displaying the coordinates of a **Geometry Object** in the **Geometry Editor**, selecting any object displays its properties in the **Property Editor** and these may be edited. The properties available for a Vertex, Boundary and Solid are depicted in Figure 15.13, Figure 15.14 and Figure 15.15.

Property Editor	
Vertex	
Property	Value
Object ID	Vertex V3
x (m)	5
y (m)	7

Figure 15.13: Property Editor display for Vertex

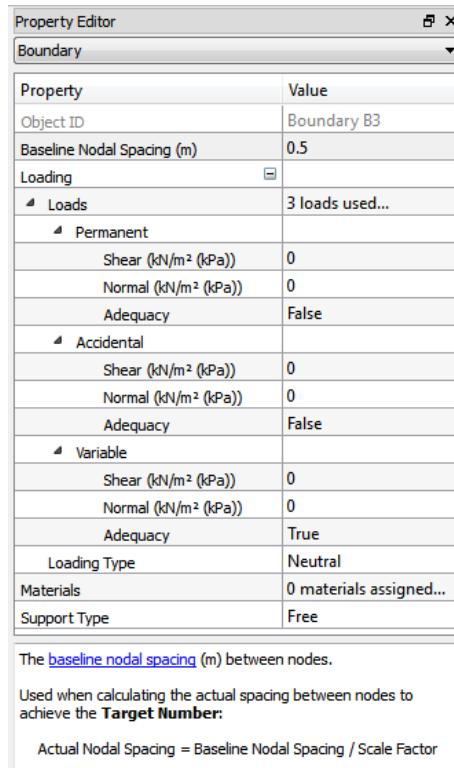


Figure 15.14: Property Editor display for Boundaries

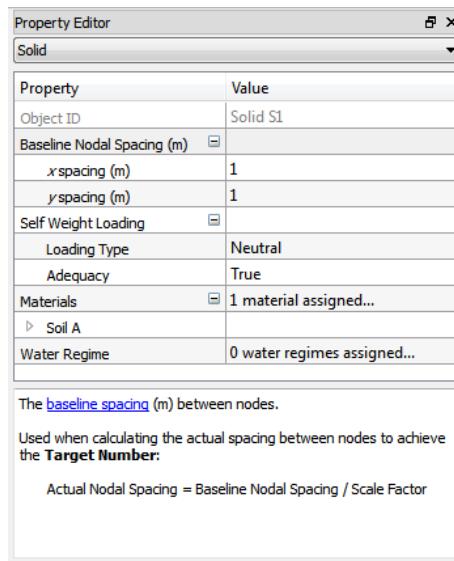


Figure 15.15: Property Editor display for Solids

Chapter 16

Setting the Analysis Mode

16.1 Short and Long Term Stability

In geotechnical analysis, it is a common requirement to check both the short term and long term (undrained and drained) stability of a problem.

LimitState:GEO provides the facility to rapidly switch between short and long term analysis mode via the **Property Editor**. To change the mode, click anywhere on an empty part of the **Viewer pane** to bring up the Project properties in the **Property Editor**. Then select the **Long Term Analysis** entry and choose *True* or *False* from the drop down selection box.

Specifying the mode causes LimitState:GEO to select either **drained** or **undrained** parameters for each material in the problem according to Table 16.1.

Example material	Drainage behaviour	Short term mode	Long term mode
Sand / gravel	always drained	drained	drained
Clay	drained/undrained	undrained	drained
Solid rock	always undrained	undrained	undrained

Table 16.1: Selection of drained/undrained parameters for long and short term analysis modes

The **Drainage behaviour** is a parameter defined for each material and described further in Section 17.4.2. The behaviour of drained materials is affected by the prevalent pore water pressure which in turn is determined by the location of the **Water table**, r_u value (if present), or an assigned **Water regime** as described in Chapter 20 .

16.2 Translational and Rotational Solution Mode

The current implementation of DLO in LimitState:GEO generates solutions based on translational mechanisms. In order to model mechanisms involving rotation of structural elements such as cantilever retaining walls or eccentrically loaded footings, LimitState:GEO can allow rotations to be modelled along **Boundaries** as an approximation to modelling rotational and translational failure everywhere. Further information on the theory underlying this process may be found in Section 5.3.3

To change the mode, click anywhere on an empty part of the **Viewer pane** to bring up the Project properties in the **Property Editor**. Then select the **Model Rotations** entry and choose *False* or *Along Edges* from the drop down selection box.

If this value is reset to *False* then a purely translational solution will be found. With *Along Edges* set, LimitState:GEO allows rotations to be modelled along **Boundaries**. This is the recommended default value.

16.3 Factor Load(s) vs Factor Strength(s)

16.3.1 Changing Analysis Type

LimitState:GEO is set up to conduct analyses that either calculate an **Adequacy factor** for loads within the problem (**Factor Load(s)**) or material strengths (**Factor Strength(s)**). Control over the type of analysis to be conducted is provided in a number of places:

Analysis Toolbar This is generally located at the top of the window and also includes the **Solve** and **Animate** buttons. In the drop-down menu, select the type of analysis to be undertaken before solving.

Wizard Analysis Tab The final tab in each wizard allows the user to specify the analysis type and **Nodal resolution**. In the drop-down menu, select the type of analysis to be undertaken before solving.

For more information on the differences between the **Factor Load(s)** and **Factor Strength(s)** analysis types, see Section 8.2.

16.3.2 Factor Strength(s) Settings

A **Factor Strength(s)** analysis has a number of associated settings. These are accessible in the **Property Editor** at Project level, under **Analysis Options > Factor Strength(s)** and from the **Analysis** menu under **Options...** (Figure 16.1):

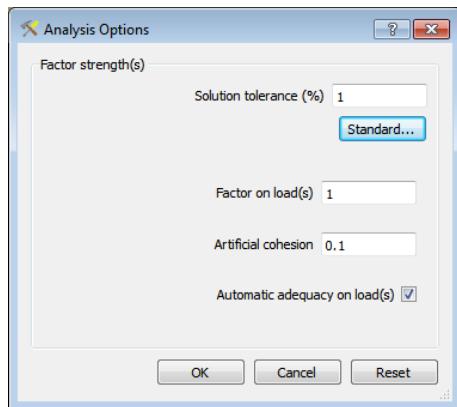


Figure 16.1: The Analysis Options dialog (accessed from the Analysis menu)

Solution Tolerance This is the relative percentage tolerance on the factor of strength solution when in **Factor Strength(s)** solution mode (default value is 1.0). For example, set the value to 0.5 to terminate the analysis when the solution is bracketed such that the upper solution is within 0.5% of the lower solution.

Automatic Adequacy on Load(s) When *True* true, the software will use the automatic global 'adequacy on load' settings to search for a factor on strength. When set to *False*, the software will use the existing user-defined 'adequacy on load' settings, so **Adequacy** should be assigned to only the **Loads** or self weight of **Solids** that are of interest.

Factor on Load(s) Defines the target **Adequacy factor** on load(s). This should generally be set to 1.0 but, for example when using **Adequacy** only on seismic accelerations, it can be useful to set the target **Adequacy factor** on load(s) to 0.0 to find the correct **Adequacy factor** on strength (i.e. when the seismic accelerations are zero).

Artificial Cohesion Adds a (generally small) cohesion to purely frictional soils when undertaking a **Factor Strength(s)** analysis. This has the effect of improving convergence towards a solution ensuring that a failure mechanism is identified. By default the value of this cohesion is set to 0.1kPa.

For more information on the settings used in a **Factor Strength(s)** analysis, see Section 8.2.3.

Chapter 17

Setting Material Properties

17.1 Material Types

LimitState:GEO provides a selection of built in materials primarily based on the Mohr-Coulomb failure envelope. This encompasses the majority of soil types likely to be encountered and also allows many structural materials to be reasonably represented. Additionally, a *Frictionless* material, with zero weight and shear strength, is provided to allow the modelling of smooth interfaces. Later on it will be seen that more sophisticated material models can be generated using the 'multi-material' technique. These materials may be assigned to **Solids** and also interfaces i.e. **Boundary Objects**.

In addition LimitState:GEO provides a special **Engineered Element** material which may only be assigned to **Boundary Objects**.

LimitState:GEO comes with a number of predefined materials. These may be viewed in the **Materials Explorer** as shown in Figure 17.1.

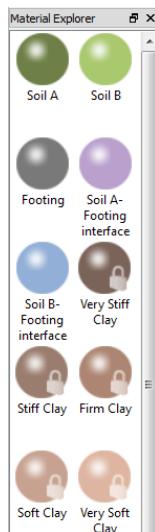


Figure 17.1: The Materials Explorer

The materials provided have typical properties according to their type and are provided to allow an easy introduction to LimitState:GEO and may be used as templates for user defined materials. The properties are not intended to correspond to those in any specific soil description standard (these could be defined by the user if required). It is *not* anticipated that these materials will be used for any ‘real’ design calculations. Note that the set of predefined materials provided is determined by the chosen system of units to be used in the software. If the software has been set to work in Imperial units, then a separate set of predefined ‘Imperial’ materials are provided. The reason for this is simply convenience in terms of providing materials with typical properties with rounded values. Thus a *Soft Clay [Imperial]* will not have quite the same properties as a *Soft Clay*.

A material may be modelled as a:

1. Standard Mohr-Coulomb material with directly defined cohesion (c) and friction (ϕ),
2. Derived Mohr-Coulomb material whose cohesion (c) and friction (ϕ) properties are defined with reference to another standard Mohr-Coulomb material
3. Mohr-Coulomb material for which the cohesion varies linearly with depth.
4. Mohr-Coulomb material for which the cohesion varies spatially.
5. Cutoff material (tension and/or compression),
6. Rigid material.
7. Engineered Element.

Click on any material in the **Materials Explorer** to bring its properties up in the **Property Editor**. Values may be interrogated and edited as described in Section 14.8.

17.1.1 Converting Yield Stress to Cohesion

It is evident that, in LimitState:GEO, material strengths are generally represented in terms of cohesion (c). However, users may be more familiar with the representation of material strength in terms of limiting tensile and / or compressive yield stresses. To convert a yield stress (σ_y) to a suitable cohesion, the Von Mises yield equation for shear can be employed, i.e.:

$$\sigma_y = c\sqrt{3} \quad \therefore \quad c = \frac{\sigma_y}{\sqrt{3}} \quad (17.1)$$

Alternatively, users may wish to use the (slightly more conservative) Tresca yield equation:

$$\sigma_y = 2c \quad \therefore \quad c = \frac{\sigma_y}{2} \quad (17.2)$$

Ultimately, however, the choice of a suitable conversion factor is left up to the user.

17.2 Assigning a Material to a Solid or a Boundary

Materials may be assigned to a **Solid object** or **Boundary Object** by the following methods:

Drag and drop Select a material in the Materials explorer and drag it onto the required Geometry Object (which will turn pink when hovered over).

The ‘Change’ dialog Select the required Geometry Object. In the Property Editor click on the right hand box in the row ‘Materials’ to display the ‘Change’ button (see Figure 17.2). Click this to display the ‘Edit Object Material(s)’ dialog (see Figure 17.3). Check or uncheck the relevant boxes to include or remove materials from the object.

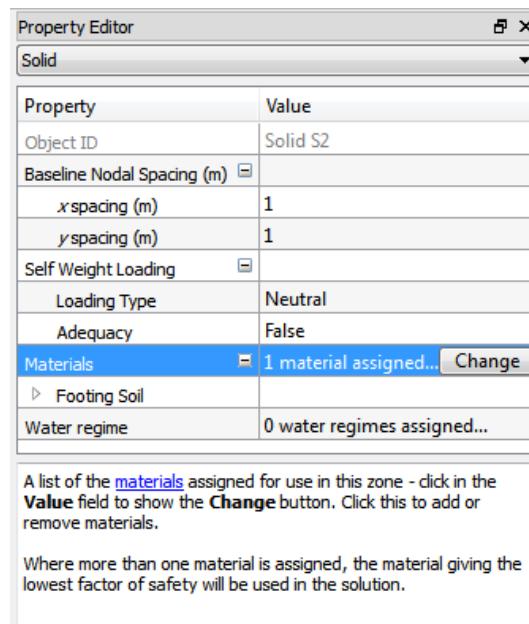


Figure 17.2: Displaying the ‘Change’ material button in the Property Editor

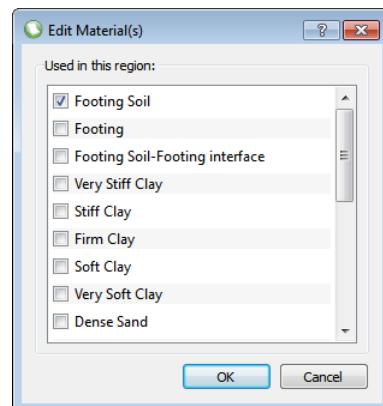


Figure 17.3: Edit Object Material(s) dialog box

If a material is assigned to an object which already has one or more materials assigned to it,

a warning message appears asking if the material is to replace the existing one, or be added to the materials in the object. For further information on the use of multi-materials see Section 6.1.6.

17.3 Post-Solve Diagrams

The stresses acting on **Solid** objects and **Boundaries** with assigned **Engineered Element** materials can be queried and / or displayed once a solution has been obtained (see Figure 23.3 and also Section 23.4).

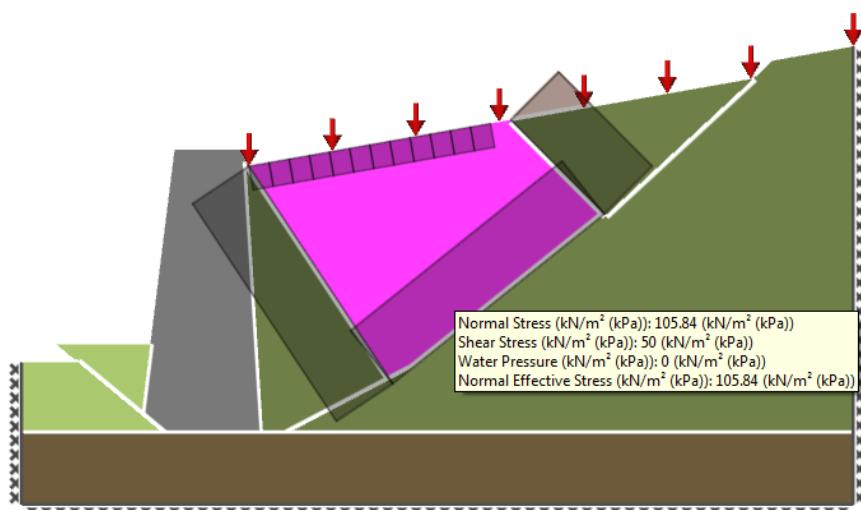


Figure 17.4: Shear stress diagrams for a solid object selected after solve

In some instances, it can be useful to display a particular stress diagram by default after solve whilst in others the user may wish to display diagrams only after clicking on the desired object with the mouse. The behaviour of LimitState:GEO materials can be tailored to show the diagrams of most use to the user by selecting the appropriate options in the **Post-Solve Display** dialog which is accessed through the **Property Editor** in the following manner:

Select the desired material in the **Material Explorer**.

Click in the **Value** field to show the **Change** button. Click this to open the **Post-Solve Display** dialog (Figure 17.5 for **Solids** and Figure 17.6 for **Boundaries**).

Select *After Click* to cause a zone containing the chosen material to display the chosen diagram type after being selecting in the viewer. Select *After Solve* to cause a zone containing the chosen material to display the chosen diagram type after the analysis is completed. Note that, in order to prevent the viewer from becoming crowded with multiple overlapping diagrams, the option to display post-solve diagrams *After Solve* is unavailable for **Solids**.

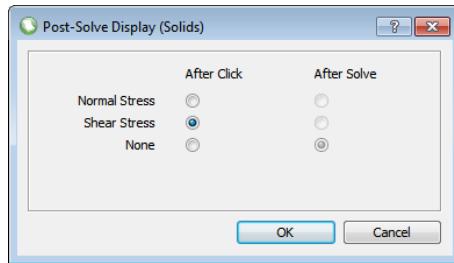


Figure 17.5: The **Material - Post-Solve Display** dialog for **Solids**

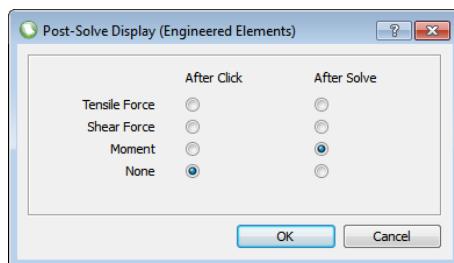


Figure 17.6: The **Material - Post-Solve Display** dialog for **Engineered Elements**

For **Solids**, the options are to display the *Normal Stress* or *Shear Stress* on each edge. For **Boundaries** containing an **Engineered Element** material, the options are to display the *Tensile Force*, *Shear Force* or *Moment* along the object. Alternatively, select *None* to display no diagram after solve.

In the **Viewer Pane**, the animation slider can be altered freely and the diagrams will move accordingly. To switch between diagrams of different types, right-click on an object and select the new diagram type from the available selection (for more information see Section 23.4).

17.4 Standard Mohr-Coulomb Material

17.4.1 Setting Soil/Material Strength Properties

For a standard Mohr-Coulomb material the properties depicted in Figure 17.7 and listed below may be edited:

Property Editor	
Material	
Property	Value
ID	Soil A
Type	Mohr-Coulomb
Color	
c' (kN/m ² (kPa))	0
ϕ' (degrees)	30
c_u (kN/m ² (kPa))	50
Unit Weight (kN/m ³)	20
Sat. Unit Weight (kN/m ³)	20
Drainage Behaviour	Always undrained
Advanced	
c_u Gradient (kN/m ² (kPa)/m)	0
Datum c_u Elevation (m)	0
Override Strength	False
Post-Solve Display	Click: Shear

Figure 17.7: Mohr Coulomb material properties dialog box

ID A unique identifier given to all objects in the project.

Type The underlying material type.

Color The color as displayed in the viewer, click in the **Value** field to show the **Change** button. Click this to modify the color.

Drained Cohesion Intercept (c') Drained material strength property - see Section 6.1.2.

Drained Angle of Shearing resistance (ϕ') Drained material strength property - see Section 6.1.2.

Undrained shear Strength (c_u) Untrained material strength property - see Section 6.1.2.

Unit Weight Unit weight (weight density) of material located above the water table and in a zone where no water regime is applied). In general this is synonymous with dry unit weight. However if a saturated material lies above the water table, then its saturated unit weight should be entered here.

Saturated Unit Weight Unit weight (weight density) of material located below the water table or within a zone containing a water regime. In general this is synonymous with saturated unit weight. However if a saturated material lies above the water table, then its saturated unit weight should also be entered under **Unit Weight**.

Drainage Behaviour Determines the behaviour of the material in long and short term analysis modes. 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.

Advanced Access advanced material property specifiers:

c_u Gradient The change in c_u value per m increase in depth below the datum. This parameter is used for a material with linear variation of undrained strength with depth. If the functionality of this parameter is not required, it should be reset to its default value of 0.0.

Datum c_u Elevation The elevation (y coordinate) at which c_u is equal to its datum value.

The datum c_u value is that entered in the **Property Editor** as c_u . This parameter is used for a material with linear variation of undrained strength with depth.

Override Strength Override the strength properties assigned for use in the selected zone using a spatially varying grid function multiplier (see Section 17.4.4).

Post-Solve Display Specify the diagrams to display following a solve. Click in the **Value** field to show the **Change** button. Click this to open the **Post-Solve Display (Solids)** dialog (Section 17.3). Select *After Click* to display the chosen diagram type after selecting a solid or *After Solve* to display the chosen diagram type immediately after solving.

It can be seen that it is possible to enter drained and undrained properties for each material. The **Long Term Analysis** project setting (Chapter 16) allows the user to switch between long and short term (drained and undrained) analyses without having to redefine all the material properties in the problem.

For clay soils, typically undrained shear strength c_u would be entered for the undrained cohesion. The effective stress properties of the clay, c' and ϕ' , are entered for the drained cohesion and drained angle of shearing resistance.

17.4.2 Setting Short Term/Long Term Behaviour

For each material it is necessary to define how it behaves when subjected to short or long term loading. Table 16.1 indicates the scenarios that might arise. It must be emphasized that this table is indicative and the user is free to choose the behaviour as they see fit. In the context of unfissured rock, the behaviour would be undrained in that its deformation up to the point of failure is unaffected by water pressure and thus undrained. At failure the rock may fracture and water, if present, would affect the analysis. How this would be modelled depends on the specific scenario.

Note that water pressures only affect soil behaviour where soils have a non-zero angle of shearing resistance. Thus an undrained material with zero ϕ_u behaves in the same way irrespective of the pore pressures.

The **Drainage behaviour** setting is provided for clarity and to avoid unnecessary pore pressure computations for undrained materials.

17.4.3 Mohr-Coulomb Material with Linear Variation of Strength with Depth

Frequently it is found that the undrained shear strength of clay soils varies with depth and that in many cases this variation can be approximated as a linear function of depth.

It is necessary to enter the datum level for where the baseline undrained strength is found, and in addition the strength gradient ($\text{kN/m}^2/\text{m}$). These values are found under the **Advanced** setting in the **Property Editor** as shown in Figure 17.7. Partial factors are applied to both the baseline strength and the strength gradient to give the correct factored strength at any depth. The baseline undrained strength is the value of c_u used for constant strength bodies of soil.

If the baseline undrained strength and the strength gradient are defined such that the average undrained shear strength on any given slip-line is less than zero, then a zero value of average undrained shear strength is assigned to that slip-line.

For step changes in strength properties, it is necessary to define separate zones with differing materials.

17.4.3.1 Zero and negative strengths within a solid

It is possible to apply materials with linearly varying undrained shear strength to any **Solid** zone. Depending on the location of the zone, the combination of initial material strength, datum level and gradient may mean that the **Solid** could possess a zero or negative strength at certain points, which will result in an *unstable* solution being obtained upon solve. This is demonstrated graphically in Figure 17.8. Here a solid measuring 3m x 5m and centered at (1.5, 2.5) includes a material where the datum is set at $y = 3\text{m}$ and the c_u gradient is 20 kPa per metre. The initial undrained shear strength of the material is 20 kPa.

Below the datum level, the material strength *increases* from 20 kPa to a value of 80 kPa at the base. However, above the datum, the material strength *reduces* at 20 kPa per metre. Hence, at a height of 4m the material has zero strength and at the top of the solid ($y = 5\text{m}$) the strength is -20 kPa. A zero or negative material strength within a solid will cause it to become unstable.

The pre-solve diagnostic function (Section 22.2.1) will warn the user when such a case is present in a model.

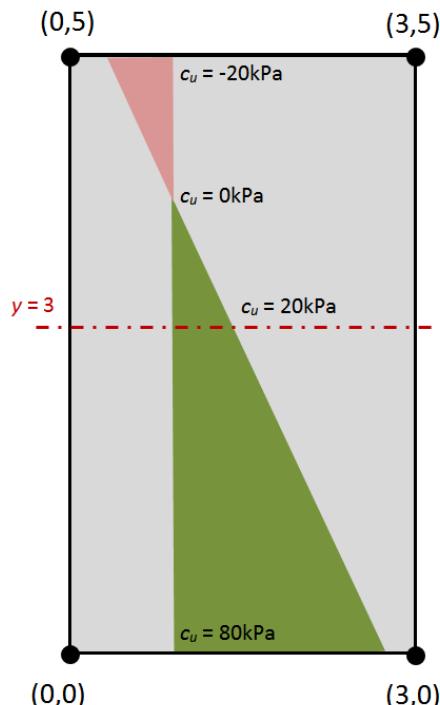


Figure 17.8: Negative undrained shear strength in a solid containing a material with linearly varying strength with depth.

17.4.4 Mohr-Coulomb Material with Spatial Variation of Strength

*This section describes a feature that is available as an add-in for LimitState:GEO.
Please contact [LimitState](#) for purchasing details.*

Sometimes it may be desirable to model the cohesive strength properties of clay soils as varying spatially with depth and width. This can be achieved by selecting to **Override Strength** for a particular material, where the cohesion parameters c' and c_u are multiplied by the spatially varying factor defined in a grid. The drained angle of shearing resistance is unaffected. Any factors applied to c' and c_u will therefore automatically be applied to the overridden strength. The c_u *Gradient* parameter has no effect on the values calculated.

With a material selected, click in the **Value** field of the **Property Editor** to show the **Change** button. Click this to define the override grid parameters using the Grid Interpolation dialog (Figure 17.9).

- First, set the **Origin** of the grid. This is the coordinate corresponding to the top-left point of the grid and does not have to be located within any particular zone (in this way you can specify a single grid for the entire problem rather than having to repeat on a per-zone basis).
- Second, set the desired **Spacing** of the grid points. For the spacing in the vertical direction, positive values relate to upwards spacing and negative downwards. Therefore it is convenient to specify a negative value for **dy**.
- Third, set the **Number of divisions** of the grid in the x and y directions using the scroll boxes. The table will grow such that the appropriate number of fields are present.
- Last, populate the table with the desired multiplier values. This can be done manually or by copying and pasting from a spreadsheet program such as Excel. To paste copied data press **CTRL + V**. Note that the size of the copied data must match the size of the table.

When the Grid Interpolation dialog is displayed then a contour/gradient plot of the grid factors will be automatically displayed in the **Viewer Pane** at the correct coordinates. An example is shown in Figure 17.10. Note that if the grid does not fully encompass a zone to which the material is applied, then zero strength multipliers will be modelled in the areas not defined by the grid. By default the contour plot will update dynamically as values in the grid are adjusted. However this process can be disabled by unchecking the 'Update graphical view on value change' tickbox. This can be faster if many values are being altered in one go.

The Grid Interpolation may be disabled by unchecking the **Override strength?** checkbox in the **Grid Interpolation** dialog. The values entered are retained so that re-enabling the grid will use the most recently entered values. Click **Reset** to set the grid to the default values. More about the grid interpolation method used is given in Appendix E.

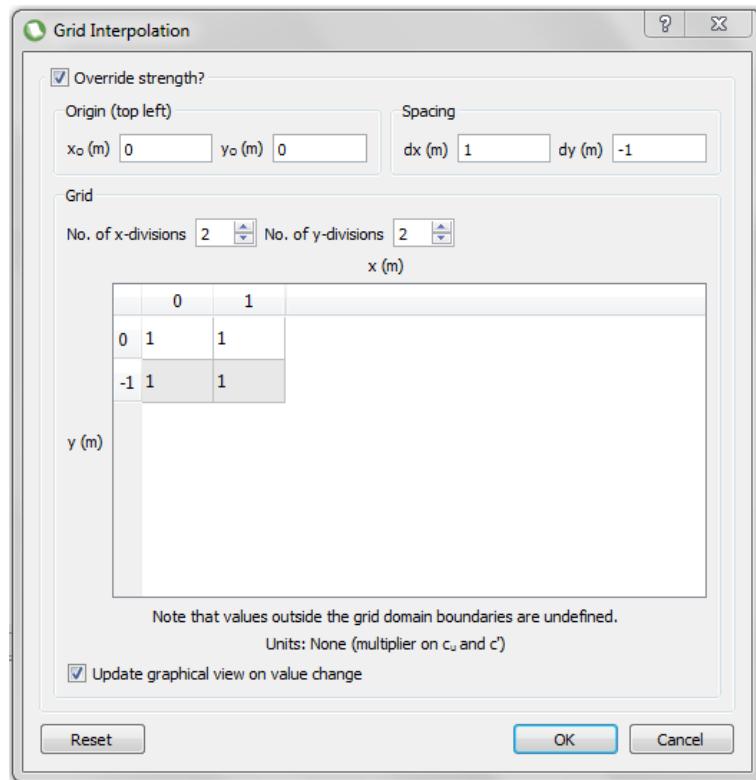


Figure 17.9: The **Grid Interpolation** dialog for defining spatially varying Mohr-Coulomb material strengths

```
PROBLEM SIZE (m):
x: 20.00000 y: 9.80000
CURSOR (m):
x: 1.44133 y: 8.79286
```

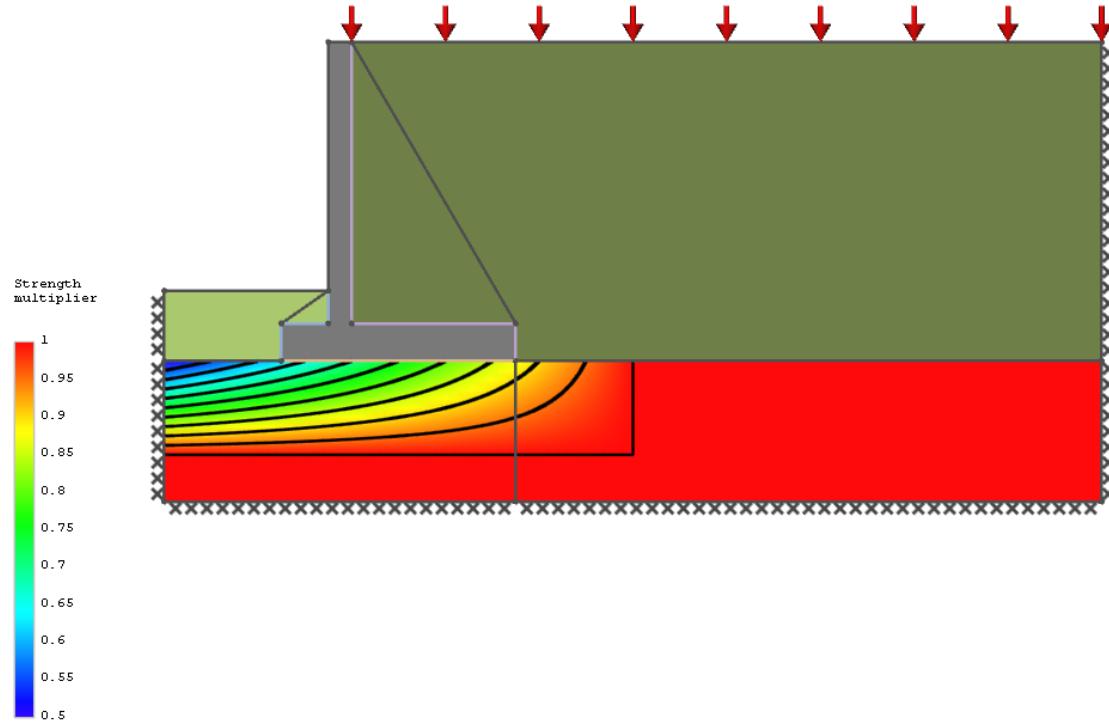


Figure 17.10: Contour display for a material using a **Grid Interpolation**. Note that the extent of the grid can exactly fit a zone as shown here or extend beyond the zone. If the grid does not fully cover the zone then the undefined areas will be modelled with a zero strength multiplier.

17.5 Derived Mohr-Coulomb Material

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 multiple of the adjacent soil strengths. This may be achieved using a **Derived** material type and entering values for cohesion c and shear resistance ($\tan \phi$) multipliers in the **Property Editor** as depicted in Figure 17.11. The other fields allow you to modify the properties of the **Parent Material**, **Drainage Behaviour** and **Post-Solve Display** behaviour of the material as discussed for standard Mohr-Coulomb materials in Section 17.4.1.

Property Editor	
Material	
Property	Value
ID	Soil A-Footing Interface
Type	Mohr-Coulomb (Derived)
Color	
Parent Material	
ID	Soil A
Type	Mohr-Coulomb
Color	
c' (kN/m ² (kPa))	0
ϕ' (degrees)	30
c _u (kN/m ² (kPa))	50
Unit Weight (kN/m ³)	20
Sat. Unit Weight (kN/m)	20
Drainage Behaviour	Always undrained
▷ Advanced	
Post-Solve Display	Click: Shear
Multiplier on c', c _u	0.5
Multiplier on tan ϕ'	0.5
Drainage Behaviour	Always undrained
Post-Solve Display	Click: Shear

The [parent material](#) that this material is derived from.

Figure 17.11: A Derived Material shown in the **Property Editor**

Note that in many design codes the interface friction property is often set to be a function of $\tan \phi_{crit}$, the critical state angle of shearing resistance. If this is not the actual ϕ value entered e.g. a peak value is entered, then it will be necessary either to make an additional modification to the multiplier or to use a ‘standard’ material instead of a ‘derived’ material type. The main advantage of the Derived material is that if the soil properties are altered, then the interface properties are automatically adjusted.

17.6 Cutoff Material

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, tensile stresses may arise. To model such cases, it is possible to specify a tension cutoff. The property defined is the normal stress at which tensile failure occurs and may be specified in the **Property Editor** as depicted in Figure 17.12.

A tension cutoff material is typically used on its own at the interface between a solid body e.g. a concrete block or rock and another solid or material with high cohesion. This allows the blocks to separate or undergo rocking displacement. The tension cutoff material is also often used in combination with other materials. For further information on this topic refer to the use of multi-material zones in Section 6.1.6.

Property Editor	
Material	
Property	Value
ID	No Tension
Type	Cutoff
Color	
Limiting Stress - Comp. (kN/m ² (kPa))	1e+30
Limiting Stress - Tens. (kN/m ² (kPa))	0
Unit Weight (kN/m ³)	0
Sat. Unit Weight (kN/m ³)	0
Post-Solve Display	Click: Shear

Figure 17.12: Cutoff material Properties Editor

Additionally a compressive cutoff can be specified e.g. for materials with a finite crushing strength (see Section 6.1.3).

17.7 Rigid

This material type may only be assigned to **Solids**, but not **Boundaries**. If it is the only material present in a **Solid**, it prevents the solver from assigning any nodes to that solid. Thus no deformation can take place within that solid. Where it is known that the solid will not deform (e.g. for a retaining wall or footing), use of a **Rigid** material ensures efficient use of nodes in the overall problem.

Additionally nodes will not be assigned to **Boundaries** that lie between two **Rigid Solids** or between a **Rigid Solid** and an external boundary unless the **Boundary** also has a material assigned to it.

The user is able to assign dry and saturated **Unit Weight** values to a **Rigid** material.

17.8 Setting Soil/Material Unit Weight (Weight Density)

Unit weights corresponding to material above and below the water table, or within a zone that includes a water regime (including Aquiclude), may be entered for each material.

Whether the **Unit Weight** or **Saturated Unit Weight** is used in an analysis depends on a number of factors:

The corresponding **Property Editor** entries are listed in Table 17.1. Note that the term ‘unit weight’ is used in LimitState:GEO and is synonymous with ‘weight density’.

For drained materials such as sands, different values of **Unit Weight** and **Sat. Unit Weight** will normally be specified. The **Unit Weight** will typically correspond to the dry unit weight.

For materials such as rock, concrete or steel, represented as an undrained material, the unit

Water table	Water regime	Label in Property Editor
Above	No regime applied	Unit Weight
Below	No regime applied	Saturated Unit Weight
Above	Water regime applied	Saturated Unit Weight
Below	Water regime applied	Saturated Unit Weight

Table 17.1: Unit weight (weight density) parameters used according to zone location

weight is normally unaffected by the location of the water table and the **Unit Weight** and **Sat. Unit Weight** values should be set to be the same.

Similarly, clay soils are often saturated above the water table due to capillary action and thus the **Unit Weight** and **Sat. Unit Weight** values would normally be set to be the same.

Unit weights are always defined as resulting in **Permanent** loads or actions. They may be set to act in a *Favourable*, *Unfavourable* or *Neutral* way within a problem. It is also possible to find an **Adequacy factor** in terms of a unit weight. These parameters may be set in the **Property Editor** values for the **Solid** containing the material, under the entry **Self Weight Loading**.

Click on the  to display the entries **Loading Type** and **Adequacy**. Further information on modelling with the Loading Type settings may be found in Section 8.3.

17.9 Engineered Element

17.9.1 Introduction

In LimitState:GEO **Engineered Elements** such as soil nails are modelled as a special material that may be assigned to a **Boundary** object. The theory behind their implementation in the software is given in Section 6.1.5 and guidance on their use in modelling in Section 11.

For the **Engineered element** type, the following properties (also depicted in Figure 17.13) may be edited.

Pullout factor, T_c The contribution of the material cohesion to the pullout resistance of the element.

Pullout factor, T_q Used to determine the contribution of the overburden to the pullout resistance of the element.

Lateral factor, N_c The contribution of the material cohesion to the lateral resistance of the element.

Lateral factor, N_q Used to determine the contribution of the overburden to the pullout resistance of the element.

Plastic moment, M_p The plastic moment of resistance of the element. (If rigid joints are being modelled then this parameter should be set to 1E+30).

Rupture strength, R The maximum tensile force that can be carried by the material.

Compression strength, C The maximum compressive force that can be carried by the material.

Advanced > Subdivide at nodes? Specifies whether failure of the element between end vertices is permitted (e.g. plastic hinge formation and/or tensile failure). This should be set to *False* for compatibility with earlier versions of LimitState:GEO.

Post-Solve Display The diagrams that are displayed by default following solve.

Engineered element pullout and lateral resistance properties are given in kN per metre length of element per metre width in the problem and may be specified absolutely and/or as a linear function of the nominal effective stress acting at the element midpoint (see Section 6.1.5).

The bending resistance is given in kNm per metre width in the problem. The plastic moment can be set so that the element may form plastic hinges at locations along their length. These locations can either be at user-defined **Vertices** or, if the **Subdivide at nodes?** material property is set to *True*, at **Nodes** along the length of the element. (By default, LimitState:GEO will set **Subdivide at nodes?** to *True* when creating any material that is designated as being able to yield.) Note that partial factors (if defined) are not applied to the properties of any Engineered element.

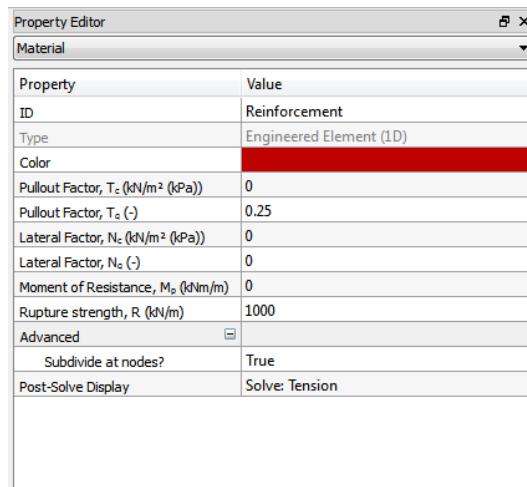


Figure 17.13: Engineered element material properties

Note that when an **Engineered Element** is assigned to a **Boundary** object, additional boundary interfaces are created that can also determine the overall behaviour of the element in a soil body. Further guidance on this is given in Section 11.

Engineered elements may represent items such as soil nails or reinforcing strips around which surrounding materials may flow, or items such as sheet piles walls through which soil does not flow. Individual elements may be connected to each other. With appropriate choice of parameters, an **Engineered elements** material can be used to represent:

1. Soil nails.

2. Soil reinforcement, such as within a reinforced earth wall (see Figure 17.14).
3. Connecting rods.
4. Soil anchor (the tendon and anchor may be modelled by two connected elements with differing properties)
5. A pile such as within a non-contiguous pile wall used to stabilise a slope.
6. A sheet pile wall.
7. A geotextile.

However it is necessary to be take account of the constraints of the **Engineered element** model when representing these types. When creating a new **Engineered element** material LimitState:GEO provides pre-programmed and suggested values for common types of soil reinforcement. This feature is described further in Section 11.

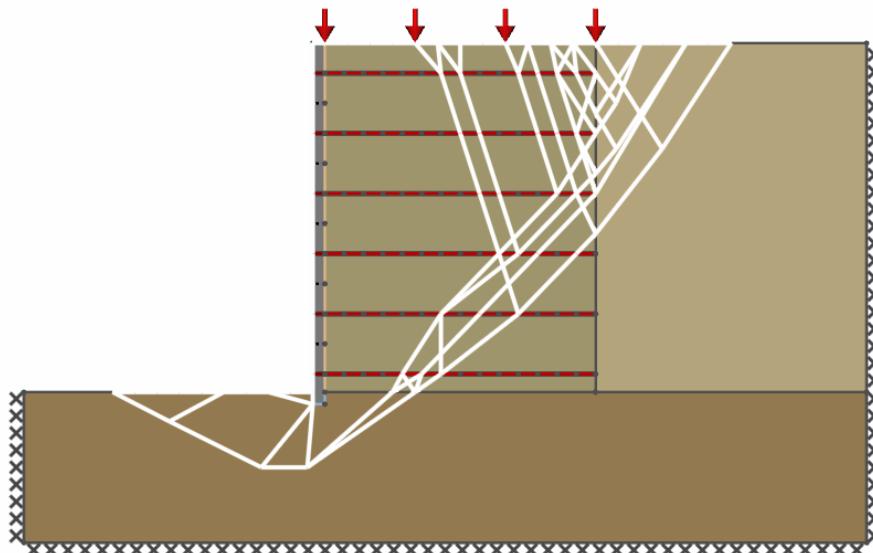


Figure 17.14: Example of reinforced earth wall modelled using the **Engineered element** material

17.9.2 Defining Engineered Element Geometry

A simple example of an **Engineered element** modelled within a soil and an adjacent wall element is depicted in Figure 17.15.

To model an engineered element within the wall solid, it is first necessary to draw a boundary element, such as AB, and then drag and drop an **Engineered element** material onto this boundary.

To continue the engineered element into the soil, draw the boundary element BD and then draw the vertex C onto this boundary. Then drag and drop an **Engineered element** material onto the boundary BC. The engineered element now runs from A through B to C.

Appropriate pullout and lateral resistances then need to be assigned to the engineered elements. Since these properties are likely to be a function of the material in which the **Engineered element** is embedded, it will usually be necessary to specify more than one **Engineered element** material - each with the appropriate properties - and assign these to appropriate segments of the element. For example, the pullout and lateral factors for the element embedded in the wall will typically be much higher than for the element in the soil.

For elements to connect to each other, it is not necessary for the same material to be used in each element, as long as they are both **Engineered element** materials.

Note that it is necessary for the boundary element CD to be present to satisfy the geometrical constraints of LimitState:GEO. For purely visual purposes it can be simply 'hidden' by assigning the same soil type to it as the surrounding soil. This has no effect on the analysis.

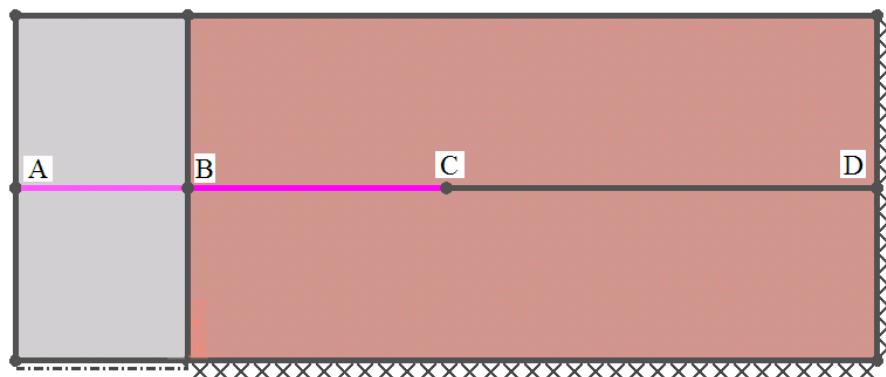


Figure 17.15: Example of **Engineered element** modelled within a soil and anchored within a wall solid

Also note that when **Engineered Elements** do connect, they are modelled with a joint that may rotate with plastic moment M_p . There is no restriction as to the initial angle of the joint. This means that several connected elements are modelled as an object that may pull through the soil as an undeformed rigid object if M_p is set to a high value, or that may bend if M_p is set to a lower value.

Also note that **Engineered Elements** do not connect to boundaries. For example if the **Engineered Element** in Figure 17.15 extends from point B to D, it is not rigidly connected to the right hand fixed boundary. If it is required to fix the element to the boundary, then this can be achieved in several ways. A simple approach is to create a new fully rigid **Engineered Element** with all parameters set to 1E30 (this can be quickly generated by selecting the *Sheet Pile Wall (rigid)* Application in the Create New Material dialog). This should then be assigned to the relevant fixed boundary as shown in Figure 17.16.

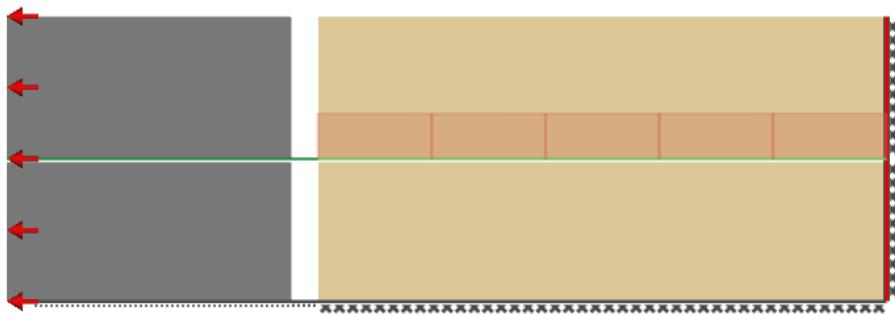


Figure 17.16: Example of **Engineered element** modelled within a soil and connected to a fixed boundary using an intermediate rigid **Engineered element** (coloured red). The animated failure shown above is tensile failure of the reinforcement.

17.9.3 Post Solve Information

17.9.3.1 Force and Moment Diagrams

The forces acting on engineered element objects can be queried and / or displayed in the **Viewer Pane** once a solution has been obtained (see Section 23.4). In some instances, it can be useful to display a particular force (or moment) diagram by default after solve whilst in others the user may wish to display diagrams only after clicking on the desired element type with the mouse. The behaviour of LimitState:GEO materials can be tailored to show the diagrams of most use to the user by selecting the appropriate options in the **Post-Solve Display (Engineered Elements)** dialog (Figure 17.17) which is accessed through the **Property Editor** in the following manner:

Select the desired material in the **Material Explorer**. Click in the **Value** field to show the **Change** button. Click this to open the **Post-Solve Display (Engineered Elements)** dialog. Select *After Click* to display the chosen diagram type after selecting a solid or *After Solve* to display the chosen diagram type immediately after solving.

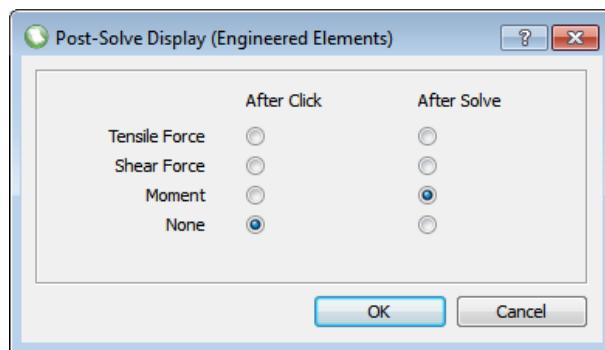


Figure 17.17: The **Material - Post-Solve Display** dialog for Engineered elements

17.9.3.2 Property Editor Data

Additional information relating to all **Engineered Elements** of a specific material type may be obtained via the **Property Editor** and the **Viewer Pane** after solving (but before Unlocking). To obtain this data it is necessary to switch off any force or moment diagram display. If the mechanism has been animated, it is also recommended to revert to an unanimated display by dragging the animation slider bar back to the left.

To obtain the information in the **Property Editor** it is necessary to select one of the required **Engineered Elements** in the **Viewer Pane**. The properties depicted in Figure 17.18 will then be displayed for that particular element (note that depending on other display settings, more elements than the one selected may be highlighted). This permits checking of the stresses assumed by the software in the computation of the lateral and pullout resistances for the **Engineered Elements**. Note that these are not displayed if the corresponding coefficients are zero.

Property Editor	
Engineered Element	
Property	Value
ID	Engineered Element 1
σ_v (kN/m ² (kPa))	32
u (kN/m ² (kPa))	0
σ'_v (kN/m ² (kPa))	32
Pullout Factor, T_c (kN/m ² (kPa))	0
Pullout Factor, T_a (-)	0.25
Lateral Factor, N_c (kN/m ² (kPa))	0
Lateral Factor, N_a (-)	0
Working Pullout Resistance, T (kN/m ² (kPa))	8
Working Lateral Resistance, N (kN/m ² (kPa))	0
Rupture Strength, R (kN/m)	1000
Compression Strength, C (kN/m)	1000
Moment of Resistance, M_p (kNm/m)	0

Figure 17.18: Post solve **Property Editor** information available for an **Engineered Element**

ID The identifier (name) of the **Engineered Element**.

Nominal Total Vertical Stress, σ_v This is based on the weight of material and any surface load vertically above the midpoint of the engineered element, and the water pressure at the Element midpoint. Note that this value is computed assuming all partial factors and the adequacy factor equal 1.0.

Nominal Pore Water Pressure, u This is based on the water pressure at the midpoint of the **Engineered Element**.

Nominal Effective Vertical Stress, σ'_v This is based on the weight of material and any surface load vertically above the midpoint of the engineered element. Note that this value is computed assuming all partial factors and the adequacy factor equal 1.0.

Pullout Factor, T_c See Section 17.9.1.

Pullout Factor, T_a See Section 17.9.1.

Lateral Factor, T_c See Section 17.9.1.

Lateral Factor, T_c See Section 17.9.1.

Working Pullout Resistance, T This is the working value of pullout resistance based on the nominal effective vertical stress. It is calculated from $T = T_c + T_q\sigma'_v$.

Working Lateral Resistance, N This is the working value of lateral resistance based on the nominal effective vertical stress. It is calculated from $N = N_c + N_q\sigma'_v$.

Rupture Strength, R See Section 17.9.1.

Compression strength, C See Section 17.9.1.

Moment of Resistance, M_p See Section 17.9.1.

17.10 Creating and Deleting User Defined Materials

17.10.1 Materials Explorer Context Menu

LimitState:GEO pre-defined materials are not editable (though those additional materials provided via wizards are editable).

To access material specific functions, with the mouse over a material in the Materials Explorer, right click to bring up the context menu depicted in Figure 17.19.

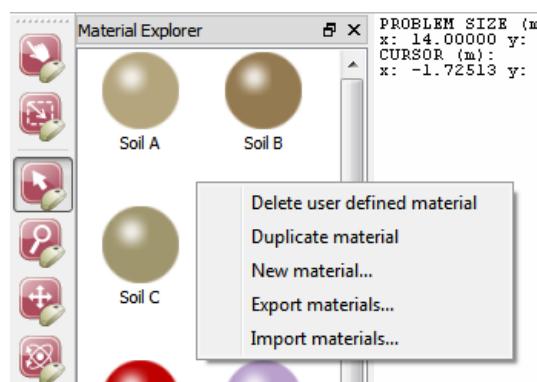


Figure 17.19: Materials Explorer context menu

17.10.2 Creating a New Material

Select **New material...** in the context menu (or from the main menu, select Tools - Create New Material...) to display the **Create New Material** dialog as shown in Figure 17.20.

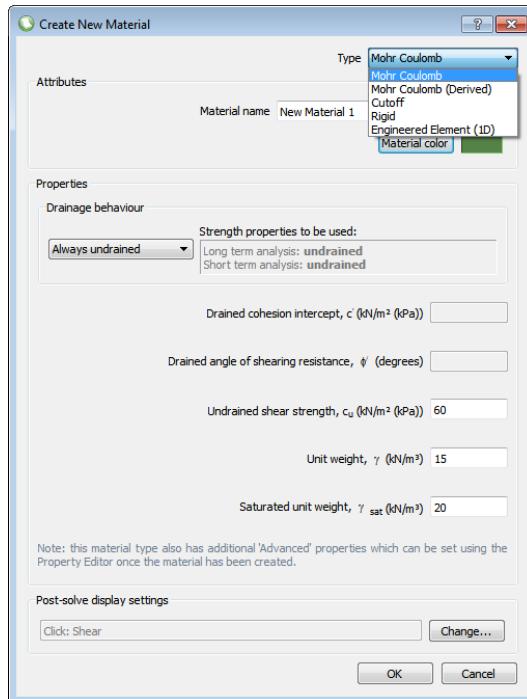


Figure 17.20: Create New Materials Dialog

Enter the required parameters and click **OK**. Note that only the core material parameters are requested in this dialog. Other parameters may be set subsequently using the **Property Editor**.

To set the type of diagram that is displayed for the new material after solve, click the **Change...** button next to the **Post-Solve Display Settings** field. This will open a dialog where the default post-solve diagrams (e.g. Normal Stresses, Shear Stresses etc.) can be modified.

When *Engineered Element (1D)* is entered in the **Type** drop down box, it is possible to select a specific ‘Application’ as shown in the **Create New Material** dialog in Figure 17.21

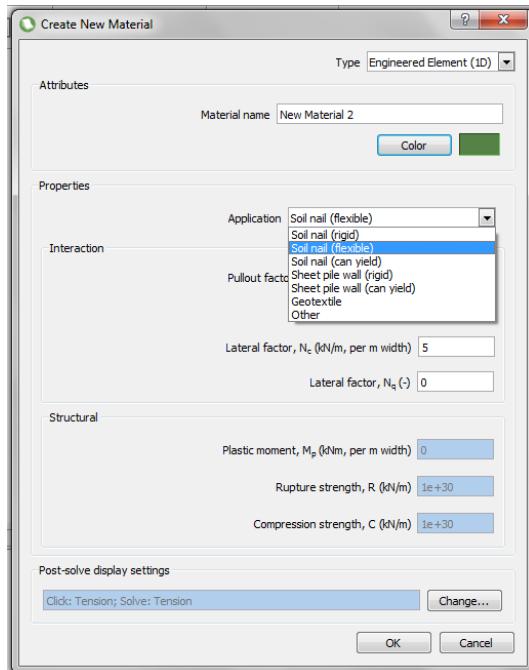


Figure 17.21: Create New Materials Dialog (Engineered Element)

The different applications are primarily concerned with the ability of soil to flow around the element. Thus the *Nail* type applications have low cohesive **Lateral Factor** values (N_c) while the *Sheet Pile* materials have effectively infinite values for the same parameter. Further information regarding the **Engineered Element** material type is available in Section 6.1.5 and Chapter 11.

17.10.3 Creating a Duplicate Material

To duplicate a material, select the appropriate icon by left-clicking with the mouse in the **Material Explorer**. Now right-click to bring up the context menu and select **Duplicate material**. A new material will be created with the name 'Copy of X' (where 'X' is the parent material). All the properties of the new material will be the same as those of the parent. The user can then modify the properties as required.

17.10.4 Deleting a Material

To delete a material, select the appropriate icon by left-clicking with the mouse in the **Material Explorer**. To delete it from the project, select **Delete material** in the context menu. {rovided that it is not used in any **Solid** or **Boundary** object within the project, this action will delete the selected material. If the material *is* being used, a suitable warning message will be displayed.

17.11 Exporting and Importing Materials

The materials in the **Materials Explorer** can be exported to a .CSV (comma separated values) file for import into other projects.

To export materials, right click the mouse in the **Materials Explorer** and select **Export materials...** in the context menu. You will be prompted for a name under which to save the exported file. Alternatively, the same functionality can be accessed from the menus by selecting **File > Export... > Materials...**.

The CSV file format allows the data to be easily edited within a text editor or spreadsheet if required.

To import materials into a project, right click the mouse in the **Materials Explorer** and select **Import materials...** in the context menu. You will be prompted to search for and select the desired CSV file. After doing this, the saved materials will appear within the **Materials Explorer**. Alternatively, the same functionality can be accessed from the menus by selecting **File > Import... > Materials...**.

17.12 Previewing Strength Distributions

The distribution of the cohesive strength component of the Mohr Coulomb materials within the model may be previewed using the **Preview Strength ▶** menu item on the **Analysis** menu. Either **Short term** or **Long term** values of cohesion may be specified. Either c_u or c' will then be displayed according to the material settings (see Section 16.1). The variation of the relevant strength across the model is then displayed graphically in the **Viewer Pane** as shown in Figure 17.22. This allows variations of non-uniform materials, either due to a linear variation of undrained strength with depth or due to a grid multiplier to be displayed.

```

PROBLEM SIZE (m):
x: 20.00000 y: 9.80000
CURSOR (m):
x: 11.61990 y: -0.05918
Short Term, Strength cu, kN/m2 (kPa): 50

```

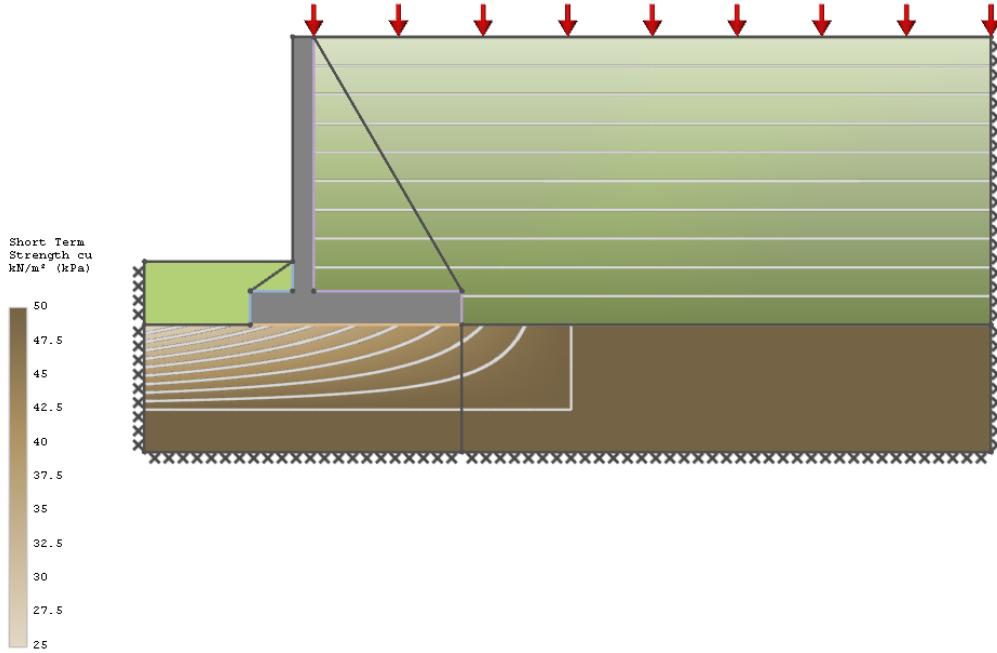


Figure 17.22: Preview of strength distribution across a model.

If the cursor is positioned over any part of the model, then the value of the relevant strength will be displayed in the top left hand corner of the **Viewer Pane**, below the cursor co-ordinates. The legend will display the relevant range for that material. If any given material is used in more than one zone within the model, then the legend range will be displayed across the range of all used instances of the material in the model.

If more than one material is allocated to any **Solid**, then the following process is followed in order to determine what is displayed:

- if a mix of uniform and non-uniform materials are used with only one non-uniform material present, then its properties are displayed graphically;
- if more than one non-uniform material is used then a warning message is displayed and the relevant **Solid** will be highlighted in purple;
- if more than one uniform material is used then the last one applied to the **Solid** is displayed graphically and the previous ones ignored. It should be clear from the values and colour shown which material is being displayed.

If an interpolated grid is used to define a material strength that does not fully cover a Solid to which it is assigned then a warning message is also displayed and the relevant **Solid(s)** are highlighted in purple.

Note that it is necessary to click **Unlock** afterwards to edit the problem parameters.

Chapter 18

Setting Boundary Conditions

In LimitState:GEO the following boundary conditions may be set on any external Boundary (Figure 18.1):

Free The boundary is free to displace in any direction.

Symmetry This provides a symmetry condition. It is implemented as a fixed boundary with a smooth interface. Thus it may also be used to represent this condition if desired.

Fixed Only displacements parallel to this boundary are permitted. The shear conditions along this boundary will be determined by the materials in the adjacent solid and/or any materials specified on the boundary. Sliding will always take place in the weakest material present.

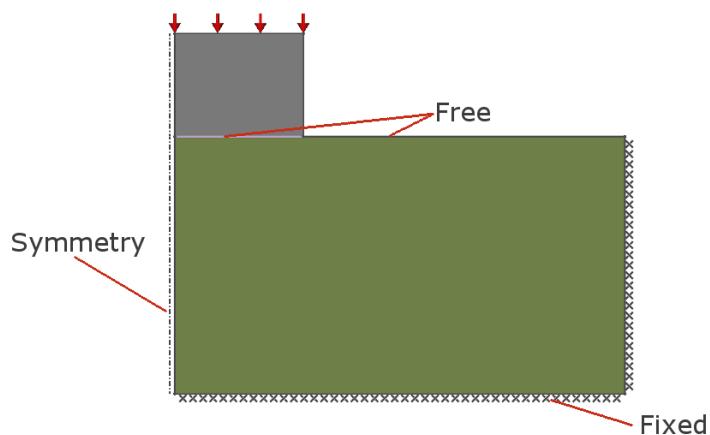


Figure 18.1: Boundary conditions in LimitState:GEO

All internal **Boundary Objects** i.e. boundaries that are at the interface of two solids may only have a boundary condition of **Free**.

Select a boundary object by left-clicking with the mouse. In the **Property Editor**, click **Support Type** to expand the data as shown in Figure 18.2. To change the support condition, simply choose an alternative option to the current one.

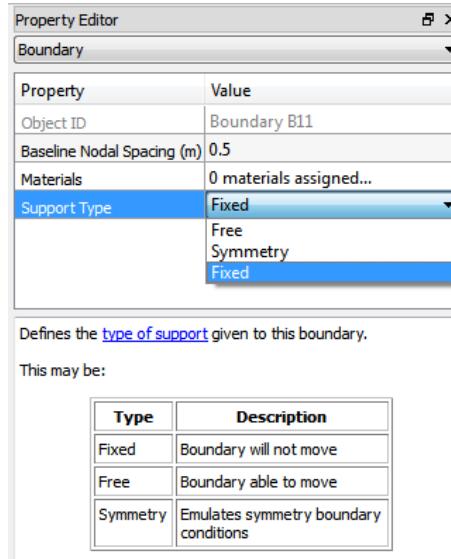


Figure 18.2: Setting boundary conditions

Chapter 19

Setting Applied Loads

19.1 Specifying Boundary Loads

In LimitState:GEO loading may be specified on any external **Boundary Object**. To set the values, select the required **Boundary Object**, and then using the **Property Editor**, click **loads** to expand the loads data as depicted in Figure 19.1.

Property Editor	
Boundary	
Property	Value
Object ID	Boundary B6
Baseline Nodal Spacing (m)	0.5
Loading	[...]
↳ Loads	2/3 loads used (Variable, Accidental)...
↳ Permanent	
Shear (kN/m ² (kPa))	0
Normal (kN/m ² (kPa))	0
Adequacy	False
↳ Variable	
Shear (kN/m ² (kPa))	-0.868241
Normal (kN/m ² (kPa))	4.92404
Adequacy	True
↳ Accidental	
Shear (kN/m ² (kPa))	0
Normal (kN/m ² (kPa))	1
Adequacy	False
Loading Type	Unfavourable
Materials	0 materials assigned...
Support Type	Free

Figure 19.1: Loads, displayed in the **Property Editor** for a selected **Boundary**

In LimitState:GEO, loads may be specified as **Permanent**, **Variable**, and **Accidental**. The purpose of these labels is purely to permit correct application of Partial Factors (see **Scenario Manager**, Chapter 30) to obtain a design load. No additional meaning is given to these labels. Each load may have an **Adequacy** factor applied to it and may be designated as *Favourable*, *Unfavourable* or *Neutral* (see Section 19.4).

On the **Viewer** pane, loads are displayed as a series of coloured arrows:

1. Loads with Adequacy = *false* : green

2. Loads with Adequacy = *true* : red

19.2 Specifying Self Weight Loading

In LimitState:GEO loads also inherently arise from the self weight of **Solid Objects**. Each self weight load may have an **Adequacy** factor applied to it and may be designated as *Favourable*, *Unfavourable* or *Neutral* (see Section 19.4). To set these values select the required **Solid Object**, and modify the values in the **Loading Type** field of the **Property Editor** as depicted in Figure 19.2.

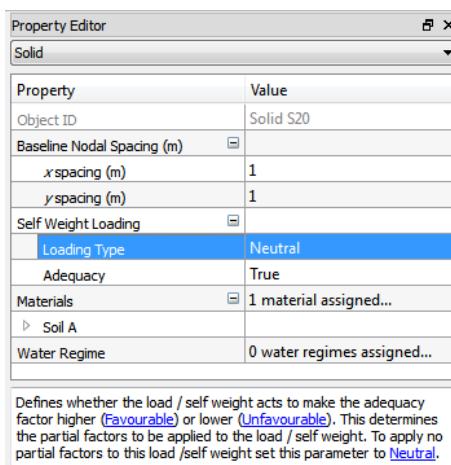


Figure 19.2: Modifying the self-weight loading type of a **Solid** object

Note that when the **Adequacy** factor is applied to a material self weight and this material lies below the water table, then adequacy is also applied to the water pressure. In effect this means that for static groundwater conditions the **Adequacy** factor is applied to the buoyant weight of the soil. Further discussion of the implications of this may be found in Section 8.2.5.1.

19.3 Adequacy Factors

In order that the solver can find a solution that causes collapse, it is necessary to apply an **Adequacy factor** to one or more of the loads or self weights. Consider a simple example of a vertically loaded footing with a permanent load of 1000kN and suppose that the soil resistance is 2000kN. To find a solution an adequacy factor must be applied to the permanent load. The solver will then find the factor that causes collapse, which in this case is 2. An adequacy factor > 1 means that the geotechnical design is safe against collapse (within the scope of the analysis technique), an adequacy factor < 1 means that the geotechnical design is unsafe.

19.4 Favourable/Unfavourable Setting

In design codes such as Eurocode 7, different partial factors are applied to loads dependent on whether they are favourable or unfavourable (i.e. help resist collapse or help cause collapse respectively). LimitState:GEO provides a facility to specify whether loads applied to a boundary are favourable or unfavourable. This specification only affects the choice of partial factor to be applied. The setting is applied to all loads (**Permanent**, **Variable**, and **Accidental**) applied to that boundary and may be specified under the **Loading Type** entry in the **Property Editor**. In addition to *Favourable* and *Unfavourable*, a *Neutral* setting may also be specified. With this latter setting no factors are applied to the load. Further information on modelling with these settings may be found in Section 8.3.

For certain problem scenarios, especially those involving several soil/structure interactions, it may not be obvious whether a given load is favourable or unfavourable from the start.

LimitState:GEO assists the user in this case by carrying out a post analysis diagnostic test, checking whether a given load dissipated or added energy into the problem. In most scenarios, input of energy is indicative of an unfavourable load, dissipation of energy is indicative of a favourable load. After each analysis a message is displayed in the console indicating on which **Boundary objects** the favourable/unfavourable settings did not match the sign of the energy dissipation in the actual solution.

The displayed **Boundary objects** are hyperlinked. If a hyperlink is clicked the boundary is selected and its properties displayed in the Property Editor, where they can be edited if desired.

Note that this diagnostic assessment of favourable/unfavourable is carried out using the combined design load after partial factors have been applied. If the favourable and unfavourable partial factors are identical then no diagnostic assessment is output. Also note that it is the responsibility of the user to decide the final unfavourable/favourable setting. The output from LimitState:GEO is only provided as guidance.

Chapter 20

Setting Water Pressures

20.1 Introduction

Water pressures may be specified in several ways:

- By drawing a water table.
- By specifying an r_u value.
- By assigning a water regime to a **Solid**.

Water pressures are determined according to the theory described in Section 6.2.2.

In LimitState:GEO, pressures defined by a **Water Regime** override any that may be defined by a **Water Table**. The water pressure at a free surface is always defined by the **Water Table** (and will thus be zero if the water table is below the free surface, or is not defined).

The distinctive difference between a **Water Table** and a **Water Regime** is that if a water table crosses a **Solid**, it automatically splits that **Solid** so the part of the **Solid** above the water table uses the **Unit weight** of any materials contained within it and is modelled with zero pore water pressure, while the part below the water table uses the **Sat. Unit weight** and is modelled with linear variation of pore water pressure with depth.

In contrast a **Water Regime** uses the **Sat. Unit weight** and the specified water pressure variation for the whole **Solid**. Thus **Water Regimes** are normally used in confined bodies of soil.

20.2 Enabling and Disabling Water Pressures

The setting **Water** in the project **Property Editor** is used to define whether water pressures are taken into consideration during an analysis.

To turn on water pressures, change the setting to *True* and to disable them, set it to *False*.

When set to *False*, all materials are assumed to lie above the water table and are treated as dry (i.e. zero pore pressure conditions for drained materials). The unit weights of all materials used is the specified **Unit Weight**. Any water table is then displayed in a faint grey.

20.3 Drawing a Water Table

To draw a water table, click on the water table toolbar button , and then with the mouse click on a series of points across the geometry to define the water table. To finish, right click the mouse to bring up the context menu and select **Finish**. The water table will then be displayed as shown in Figure 20.1. It is displayed in light blue if the **Water** setting in the project **Property Editor** is set to true. If not enabled, it is displayed in faint grey.

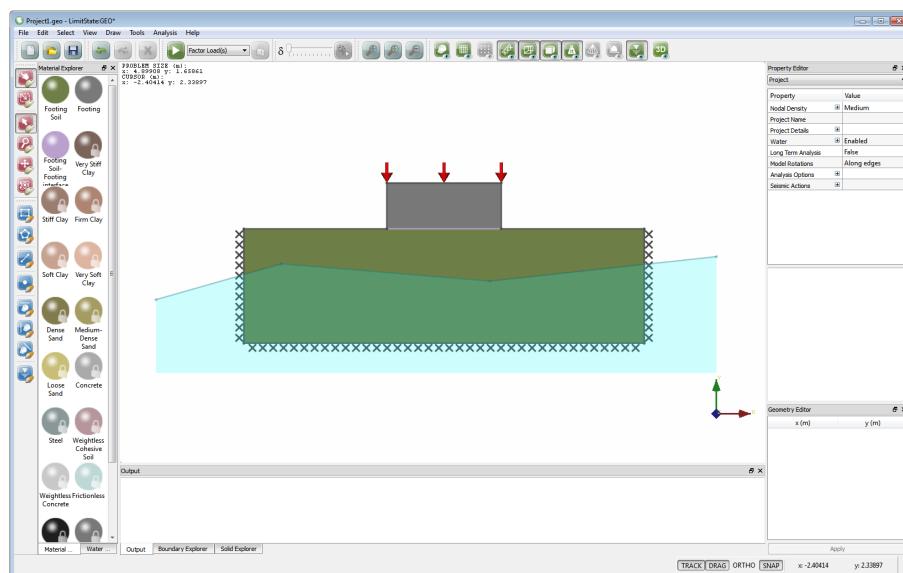


Figure 20.1: Example water table added to a simple footing problem

To select the water table, click on any part of the water table. The entire water table or individual vertices may be selected and moved using the mouse. When selected, the water table coordinates also appear in the **Geometry Editor** where they may be edited. Vertices for the water table must be specified with monotonically increasing x-coordinates.

To clear the water table, right click anywhere in the **Viewer Pane** to display the context menu and select **Clear Water Table**.

Note

In versions prior to LimitState:GEO 3.0, if a water table caused a solid to be split by a new boundary, nodes on the new boundary were counted towards the overall specified Target Number of nodes. In Version 3.0 nodes on the new boundary are not counted towards the specified Target Number. This means that solution accuracy should be broadly consistent regardless of the location of the water table, or whether it is disabled. In some cases, this will mean a larger overall number of nodes is used in a solution. In general it would be expected that files solved in Version 3.0 should therefore return slightly lower (more accurate) results than the identical file

solved in previous versions, though with a slightly longer solution time. An information message will be issued by the software if a version 1.0 or 2.0 file is solved with a water table. To avoid seeing the message again, save the file (which will save it as a Version 3.0 file) and re-open it.

20.4 Use of an r_u Value

An r_u value may be utilised in place of a water table. In this case the water pressure u at any point is given by

$$u = r_u \gamma h \quad (20.1)$$

where γh is the overburden pressure which may be represented by defining h as the vertical depth of the point in question below the ground surface.

To give additional flexibility in LimitState:GEO, the nominal ground surface, when using an r_u value, is defined using the water table. For conventional use, the user will draw the water table along the soil surface.

To enable the use of r_u , click the next to the **Water** entry in the **Property Editor**. Three new values will appear as shown in Figure 20.2. Set **Use r_u Coefficient** to *True*.

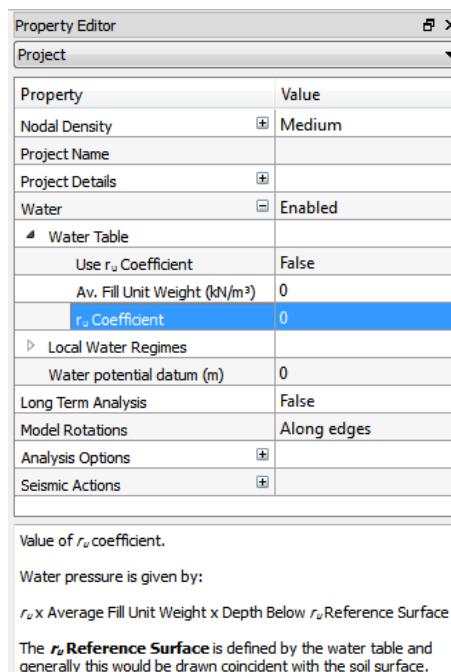


Figure 20.2: r_u parameters displayed in the **Property Editor**

The **Average Fill Unit Weight** is the value of γ to be used in equation 20.1, and the required r_u value is entered adjacent to r_u **Coefficient**.

20.5 Localized Water Pressures (Regimes)

A localized water pressure may be applied to any individual block of material (**Solid**) by defining and assigning a **Water Regime**.

Water regimes work in a similar manner to **Materials** in that they can be user-defined according to type and are assigned using an **Explorer** or the **Property Editor**.

LimitState:GEO comes with a number of predefined water regimes. These cannot be edited and are displayed with a padlock on the icon. They may be viewed in the **Water Explorer** as shown in Figure 20.3. By default, display of the **Water Explorer** is achieved by clicking the tab at the base of the **Material Explorer**, although it may be dragged to other areas of the screen.

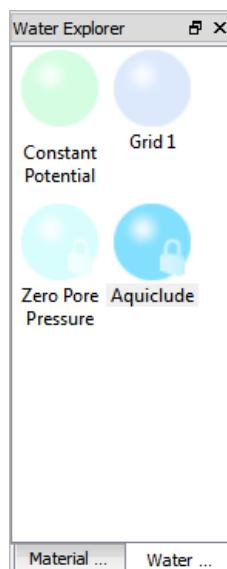


Figure 20.3: The Water Explorer

Water regimes may be assigned using the following two methods:

Drag and drop Select a regime in the **Water Explorer** and drag it on to the required object.

The ‘Change’ dialog Select the required object. In the **Property Editor**, click on the right hand box in the row **Water Regime** to display the ‘Change’ button (see Figure 20.4). Click this to display the **Edit Water Regime(s)** dialog. Check or uncheck the relevant boxes to include or remove materials from the object.

When a regime is applied to a zone, its colour will change to include that of the selected regime.

Unlike with materials, an object can only contain a single water regime. Therefore, if the user attempts to assign a second regime, a warning will be displayed.

Any solid to which a water regime has been applied (including Aquiclude) will use the defined saturated unit weights (**Sat. Unit Weight**) of any materials within that solid.

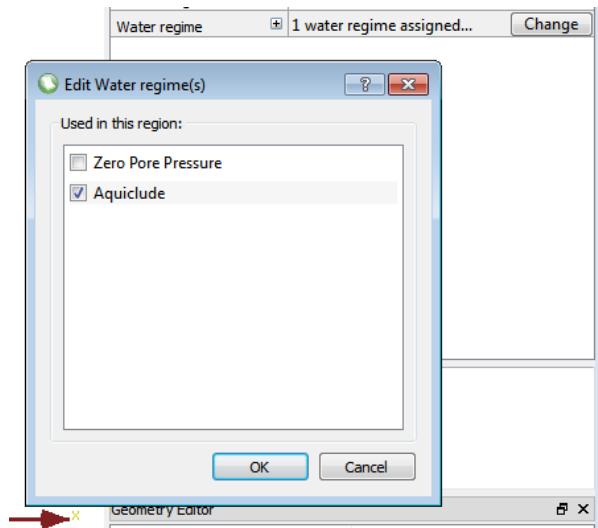


Figure 20.4: Assigning a water regime using the Property Editor

20.5.1 Defining a New Water Regime

By default, two water regimes are provided in LimitState:GEO:

Zero Pore Pressure A Constant Potential

Aquiclude This models zero water pressures in the solid, but is intended to be used for impermeable materials that are internally unaffected by water pressure.

Additional water regimes may be specified by right-clicking the mouse in the **Water Explorer** to bring up the context menu, then selecting **New water regime...**. This will cause the **Create New Water Regime** dialog to appear (e.g. Figure 20.6).

To create a new regime, first select the type from the **Type** dropdown.. Each new regime can be assigned a **Name** and **Color** as well as properties particular to the type chosen:

20.5.1.1 Constant Potential

In a **Constant Potential** regime, the affected zone is subject to water pressures determined from a specified water potential. The datum level against which this potential is measured is global for the entire project (i.e it is the same for all zones containing a potential) and is therefore set outside the **Create New Water Regime** dialog in the **Property Editor** Figure 20.5.

Two properties are required when defining a **Constant Potential** regime:

Water potential The assumed height of water above the global datum level.

Factor on unit weight of water A multiplier on the unit weight of the water in the affected zone(s).

Using this regime type it is possible to define areas containing negative pressures (e.g. to represent capillary rise).

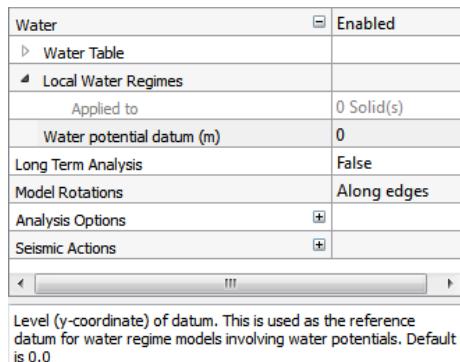


Figure 20.5: Definition of a the **Water potential datum** level in the **Property Editor**

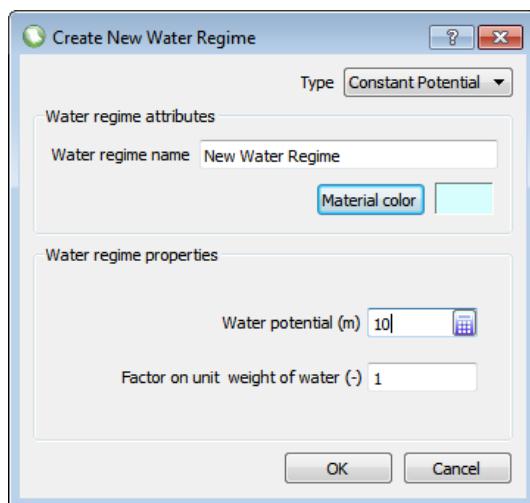


Figure 20.6: Definition of a new **Constant Potential** water regime

20.5.1.2 Constant Pressure

In a **Constant Pressure** regime, the affected zone is subject to a specified constant pressure (Figure 20.7):

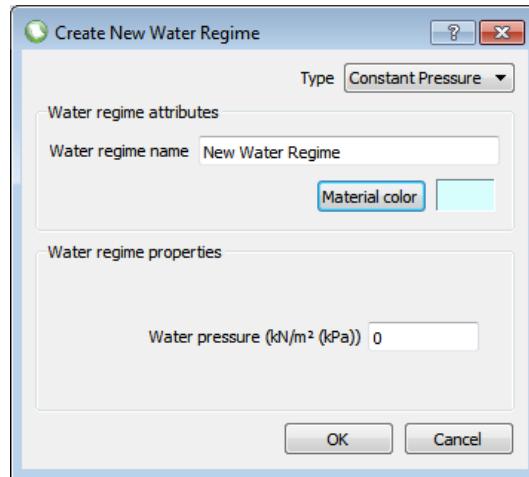


Figure 20.7: Definition of a new **Constant Pressure** water regime

This regime type might typically be used to assign areas of zero pressure or constant suction.

20.5.1.3 Interpolated Grid

In an **Interpolated Grid** regime, the affected zone is subject to spatially varying water pressures defined by a series of user-specified points and associated pressure values. (Figure 20.8):

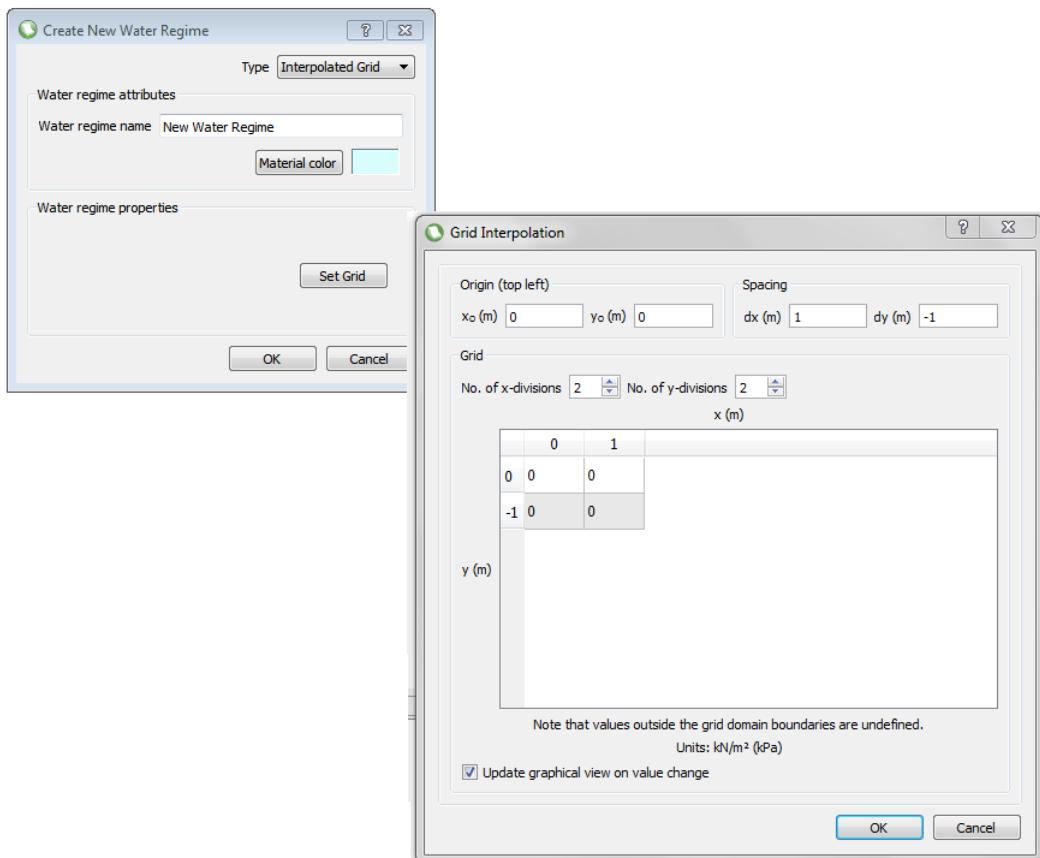


Figure 20.8: Definition of a new ***Interpolated Grid*** water regime

After setting the **Name** and **Color**, a grid of pressures must be defined. To do this, click the 'Set Grid' button to bring up the **Grid Interpolation** dialog (Figure 20.8):

- First, set the **Origin** of the grid. This is the coordinate corresponding to the top-left point of the grid and does not have to be located within any particular zone (in this way you can specify a single grid for the entire problem rather than having to repeat on a per-zone basis).
- Second, set the desired **Spacing** of the grid points. For the spacing in the vertical direction, positive values relate to upwards spacing and negative downwards. Therefore it is usual to specify a negative value for **dy**.
- Third, set the **Number of divisions** of the grid in the x and y directions using the scroll boxes. The table will grow such that the appropriate number of fields are present.
- Last, populate the table with the desired pressure values. This can be done manually or by copying and pasting from a spreadsheet program such as Excel or from the grid itself. To paste copied data press **CTRL + V**. To copy data press **CTRL + C**.

Intermediate pressures, between the defined points, are calculated using a bilinear interpolation algorithm (see Appendix E).

When the Grid Interpolation dialog is displayed then a contour/gradient plot of the grid factors will be automatically displayed in the **Viewer Pane** at the correct coordinates. An example is shown in Figure 20.9. Note that if the grid does not fully encompass a zone to which the material is applied, then zero strength multipliers will be modelled in the areas not defined by the grid. By default the contour plot will update dynamically as values in the grid are adjusted. However this process can be disabled by unchecking the 'Update graphical view on value change' tickbox. This is faster if many values are being altered in one go.

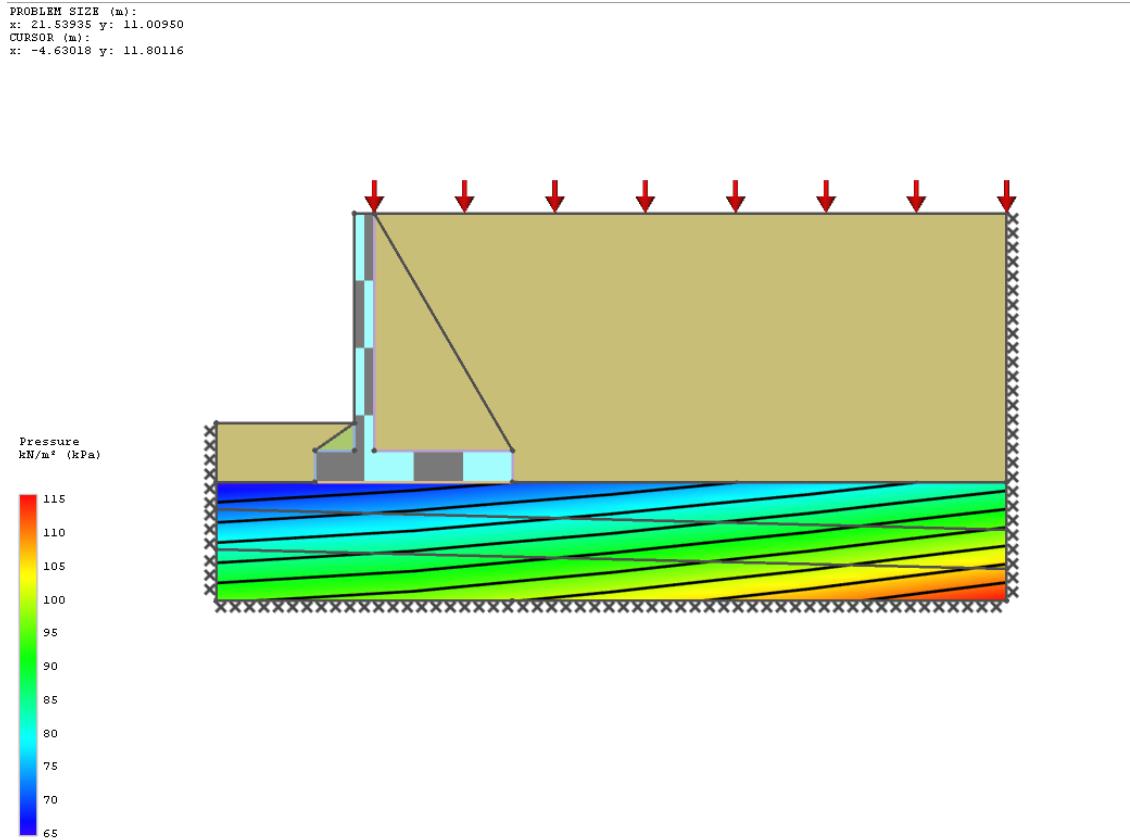


Figure 20.9: Contour display for a material **Water Regime** using a **Grid Interpolation**. Note that the extent of the grid can exactly fit a zone or extend across several zones as shown here. If the grid does not fully cover the zone then the undefined areas will be modelled with a zero water pressure.

20.5.1.4 Aquiclude

In an **Aquiclude** the affected **Solid** zone (**Aquiclude**s can not be applied to **Boundary** objects) is internally subject to zero water pressure and is treated as being 'Dry'. There are no additional properties to set for this particular type.

Assigning an aquiclude regime to a zone prevents LimitState:GEO from checking the continuity of pressures at the boundaries of that zone but allows the water pressure in adjacent zones to act on the body represented by that zone. It should generally only be used for undrained or impermeable objects such as mass concrete.

For more information on **Aquiclude**s see Section 6.2.3.

20.5.2 Creating a Duplicate Water Regime

To duplicate a **Water Regime**, hover over the icon of the target regime with the mouse and right-click to bring up the context menu. Select **Duplicate water regime** to create a copy of the selected regime. Click the icon for this new regime to bring up its properties in the **Property Editor**, where they can then be edited.

20.5.3 Deleting a Water Regime

To delete a **Water Regime**, hover over the icon of the target regime with the mouse and right-click to bring up the context menu. Select **Delete user defined water regime** to remove the selected regime from the project. Note that a warning message will be shown if the regime is currently assigned to an object

20.5.4 Exporting and Importing Water Regimes

To export or import the regimes listed in the **Water Explorer** in a .csv (comma separated values) file, select **Export water regimes...** or **Import water regimes...** in the context menu.

20.5.5 Viewing Water Regime Affected Objects

To quickly gain an overview of the solids that are affected by water regimes, click on the tab for the **Solid Explorer** (by default this is located at the bottom of the screen along with the **Output Pane**). Columns in the selected explorer will list the applied regimes. Clicking on any row will cause the object to be highlighted in the **Viewer Pane** and, if **TRACK** is also selected, the object will be zoomed in to.

20.6 Previewing Modelled Water Pressures

The distribution of water pressures specified in the problem may be previewed using the **Pre-view Water Pressure** menu item on the **Analysis** menu. The variation of water pressure across the model is then displayed graphically in the **Viewer Pane** as shown in Figure 20.10.

```

PROBLEM SIZE (m):
x: 21.53935 y: 11.00950
CURSOR (m):
x: 14.93112 y: 3.99862
Pressure, kN/m2 (kPa): 22.5879

```

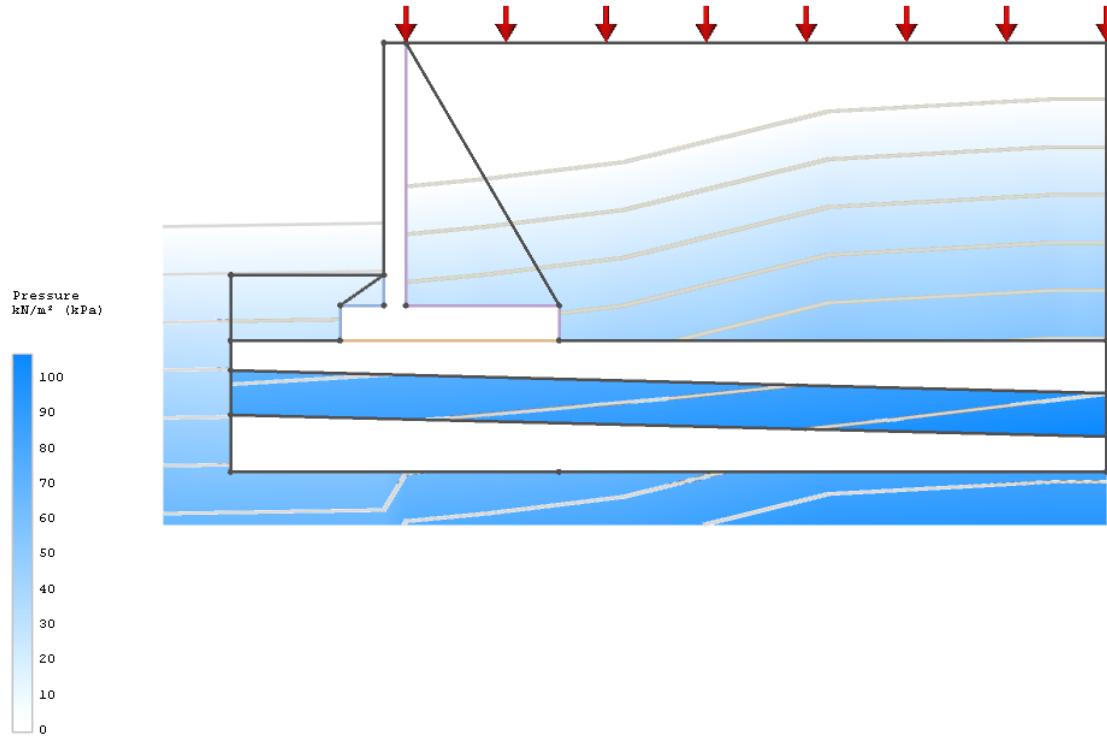


Figure 20.10: Preview of water pressure distribution across a model.

If the cursor is positioned over any part of the model, then the value of the water pressure will be displayed in the top left hand corner of the **Viewer Pane**, below the cursor co-ordinates.

If there is a mismatch in any water pressures across any boundary, then a warning text is displayed and any affected boundaries are displayed in red. A mismatch in water pressures can lead to inaccuracy in the solution provided (see Section 6.2.1).

If an interpolated grid is used to define water pressure that does not fully cover a Solid to which it is assigned then a warning message is displayed and the relevant Solid(s) are highlighted in purple.

Note that it is necessary to click **Unlock** afterwards to edit the problem parameters.

Chapter 21

Setting Seismic Parameters

LimitState:GEO is able to apply horizontal and/or vertical pseudo static accelerations to any problem. These may be applied as fixed accelerations or it is possible to apply an adequacy factor to one or both of these parameters. Further information on modelling with these parameters may be found in Section 12.

To set seismic accelerations, click the  next to the **Seismic Actions** entry in the project level **Property Editor**. Four new values will appear as described below and shown in Figure 21.1.

Horizontal Accel. k_h (g) Horizontal base acceleration. 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. Note: when in 'Factor strength(s)' mode this value may be overridden.

Vertical Accel. k_v (g) Vertical base acceleration. Specify a positive value for a 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. Note: when in 'Factor strength(s)' mode this value may be overridden.

To set a horizontal acceleration of e.g. $0.3g$ (i.e. $0.3 \times 9.81 \text{ m/s}^2$), set **Horizontal Accel. k_h** to 0.3. To apply this as a fixed acceleration set **Adequacy (on k_h)** to **False**. Positive values are modelled as resulting in a horizontal right to left acceleration on the problem. (This corresponds to an equivalent base acceleration in the opposite i.e. positive x-direction).

Frequently the user may wish to ask the question:

'At what horizontal acceleration will the modelled soil construction collapse?'

In order to get LimitState:GEO to answer this question set **Horizontal Accel. k_h (g)** to 1.0, and **Adequacy (on k_h)** to **True**, while ensuring that **Adequacy** is not set on any other parameter

Property Editor	
Project	
Property	Value
Nodal Density	Medium
Project Name	
Project Details	
Water	Enabled
Water Table	
Local Water Regimes	
Water potential datum (m)	0
Long Term Analysis	False
Model Rotations	Along edges
Analysis Options	
Factor type	Factor Load(s)
Seismic Actions	-
Horizontal Accel. k_h (g)	0
Adequacy (on k_h)	True
Vertical Accel. k_v (g)	0
Adequacy (on k_v)	True

Define whether [seismic](#) ground accelerations are present.

Figure 21.1: Seismic parameters displayed in the **Property Editor**

e.g. a load. On solving, LimitState:GEO will return an **Adequacy factor** equal to the required horizontal acceleration. This procedure will also work with a fixed vertical acceleration.

An exactly equivalent procedure is applicable to the vertical acceleration. Positive values are modelled as resulting in an additional vertical downward acceleration on the problem. (This corresponds to an equivalent base acceleration in the opposite i.e. positive y-direction).

Additionally, if it is required to answer the question:

'At what horizontal acceleration will the modelled soil construction collapse such that the vertical acceleration equals 0.2 times the horizontal acceleration?'

then set **Horizontal Accel. k_h (g)** to 1.0, **Adequacy (on k_h)** to *True*, **Vertical Accel. k_v (g)** to 0.2, and **Adequacy (on k_v)** to *True*. On solving, LimitState:GEO will return an **Adequacy factor** equal to the required horizontal acceleration.

Chapter 22

Analysis

22.1 Overview

To perform an analysis, on the **Analysis** menu click **Solve**. Alternatively this command can be accessed via the  button on the toolbar; the keyboard shortcut for the command is **F5**.

The analysis procedure first allocates nodes to all zones and generates a list of potential slip-lines.

An initial analysis is then undertaken and the solver iterates towards the optimal solution by progressively adding slip-lines at critical positions (see Chapter 5). Stages in the solution are displayed on the **Status bar**.

By default, the output displayed in the **Output** pane from an analysis is restricted to the critical load case and failure adequacy factor only. To enable the display of *all* iteration data, go to the **Preferences** dialog in the Tools menu, select the **Solve** tab and check the option to **Display iteration information in output window**.

22.2 The Solver

A solver is required to find the critical collapse load factor and associated collapse mechanism. The internal forces in the structure must satisfy all specified yield constraints; these are set up for a particular problem by LimitState:GEO.

The solver used by LimitState:GEO is [Mosek](#), a powerful interior point linear programming solver. In LimitState:GEO, Mosek is called as a subroutine and, to maximize efficiency, the problem data is passed *via* memory.

22.2.1 Pre-Solve Checks and the Diagnostics Tool

During model generation and prior to solving, LimitState:GEO runs a number of checks to ensure the integrity and rationality of the model.

Some checks are carried out continuously and result in singular warning or error messages being displayed to the user. These pertain to issues such as invalid licenses and geometries and are provided in Table 22.1.

Check	Warning
Check if a license was held then lost	The connection to the network license server or hardware dongle was lost. You need to restart the application to be able to solve again.
Check if license does not have permission to solve for certain features	Cannot solve as the following feature(s) in this project are not supported by this <LICENSE_TYPE> license.'
Check for two coincident vertices	Two coincident points encountered
Check for a zero length line	Zero length boundary encountered
Check for two coincident boundaries	Two coincident boundaries encountered
Check for two intersecting lines without a vertex at the crossover point	Vertex missing at boundary object intersection.
Check for an 'invalid' polygon	Illegal solid geometry.
Check for a zero area polygon	Zero area solid.
Check for a problem where the overall area is too small	Total area of problem is too small, which may lead to numerical tolerance problems - for more details, see here.
Check for solids overlapping without vertices on their coincident points	Solids are overlapping (see output window for details.)

Table 22.1: Continuous checks carried out by LimitState:GEO and their associated warnings.

Other messages are collated and shown at solve-time, or when the user opens the **Diagnostics tool** dialog (as shown in Figure 22.1) by going to the **Analysis** menu and selecting **Diagnostics...** These messages are categorized according to severity as follows:

Information Useful reminders about the problem set-up that require no user action prior to solve.

Warning Messages regarding the problem set-up that *may* require user action prior to solve, but which do not prevent solving.

Error Messages regarding the problem set-up that *must* be resolved by the user prior to solve.

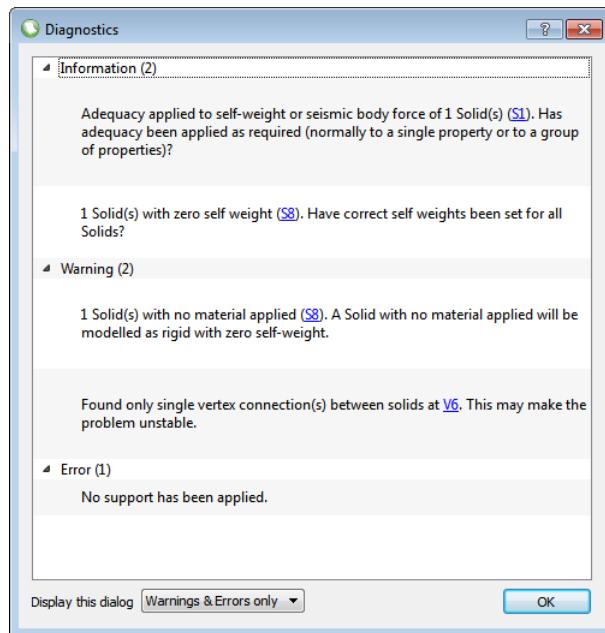


Figure 22.1: The LimitState:GEO Diagnostics Tool

The **Display this dialog** drop-down menu at the base of the **Diagnostics** tool allows the user to specify the circumstances under which the dialog is displayed:

Always Displays the dialog before every **Solve**, irrespective of whether there are any warnings or errors to report.

Warnings & Errors only Displays the dialog before **Solve** only when there are warnings or errors to report.

Errors only Displays the dialog before **Solve** only when there are errors to report.

If the setting is changed to *Errors only* then any warnings encountered will instead be highlighted in the **Output Pane** by a hyperlink. Clicking the hyperlink will open the **Diagnostics** dialog with the warnings present.

A list of the messages provided by the **Diagnostics** tool and their interpretation is provided in Table 22.2.

Severity	Check	Warning
Information	Check which objects have Adequacy applied to them (total number is 20 or fewer).	Adequacy applied to: - Self weight of #NUM Solid(s) (LIST) - Seismic body forces in the (x/y) direction(s) - #NUM Boundary load(s) (LIST) Has adequacy been applied as required (normally to one or more Boundary load(s) or Solid(s))?
Information	Check that no solids have zero unit weight.	#NUM Solid(s) with zero self weight (LIST). Have correct self weights been set for all Solids?

Continued on next page...

Information	Check to see whether a Water Table is using an r_u coefficient in pore pressure calculations.	A water table is active and is set to use an r_u coefficient in pore pressure calculations (rather than hydrostatic pressures). Average fill unit weight = VALUE, r_u coefficient = VALUE.
Information	List the different loading types applied to Boundary and Solid zones.	The following 'Loading Types' have been specified on Boundary loads: Unfavourable: LIST Neutral: LIST Favourable: LIST The following 'Loading Types' have been specified on Solid self weights: Unfavourable: LIST Neutral: LIST Favourable: LIST Note that the Boundary and Solid Explorers can also be used to quickly identify and edit the above settings.
Information	Check for the inclusion of purely frictional materials in a Factor Strength(s) analysis	A Factor Strength(s) analysis has been specified. Detected material(s) with zero cohesion that will be used in one or more scenarios. 'Artificial cohesion' will be added to these materials to improve convergence. This may have a small effect on the solution obtained.
Warning	Check which objects have Adequacy applied to them (total number is above 20).	Adequacy applied to self weight or seismic body force of #NUM Solid(s) (LIST) and to #NUM Boundary load(s) (LIST). Has adequacy been applied as required (normally to a single property or group of properties)?
Information	Check to see whether non-unit partial factor sets have been used and to prompt user to set loading type accordingly.	Non unit load factors have been specified in one or more of the scenarios to be modelled. Have all load types been specified as Unfavourable, Favourable or Neutral as required? Note that Neutral loads are not factored in any situation.
Information	Check to identify Engineered Element materials with zero pullout interaction.	Engineered Element material(s) LIST have been specified with Pullout Factors Tc and Tq both set to zero. This will lead to no pullout (longitudinal) interaction of the engineered element with the soil and therefore no conventional reinforcement action by these elements.
Information	Check to identify Engineered Element materials with zero lateral interaction.	Engineered Element material(s) LIST have been specified with Pullout Factors Nc and Nq both set to zero. This will lead to no lateral interaction of the engineered element with the soil and therefore no conventional reinforcement action by these elements.
Information	Check that rotational failure mechanisms can be identified.	Model rotations has been set to 'False'. This means that rotational failure mechanisms will not be identified. Is this intentional?
Warning	Check that the problem size is sufficient to avoid numerical tolerance issues.	Defective geometry encountered - the total area of problem is small, which may lead to numerical tolerance issues.
Warning	Check for a single Vertex joining two or more Solids .	Found only single vertex connection(s) between solids at VERTEX. This may make the problem unstable.
Warning	Check for loading applied to an internal Boundary .	Loads should only be applied to free boundaries; loads on other types of boundary will now be removed.
Warning	Check for interfaces between Solids of different strength materials where no material is set.	Found materials with different strengths separated by interface without material, at LIST.
Warning	Check for zero strength material within a Solid .	Found Mohr Coulomb material with zero strength within the following Solid(s): (LIST) It is strongly recommended that all materials used in Solids have non-zero drained and/or undrained strength properties.
Warning	Check that the water table (if present) does not include any vertical sections.	The Water Table has been defined with one or more vertical (or near vertical) faces. The water pressure on the vertical faces is defined as zero. This may give unintended results where, for example, water pressure is required to act on a concrete dam wall and the water table has been drawn along its face.

Continued on next page...

Warning	Check that adjacent Water Regimes in the problem do not contain mismatched properties across their common boundaries.	One or more Water Regimes have mismatched water pressures along the following zone boundaries: LIST This will lead to unpredictable results.
Warning	Check for Solids without any set material type.	#NUM Solid(s) with no material applied (LIST). A Solid with no material applied will be modelled as rigid with zero self-weight.
Warning	Check that Boundarie do not have significantly higher nodal spacing than the solids to either side	One or more zone boundaries have significantly higher nodal spacings than their adjacent solids: LIST. This can lead to poor results.
Warning	Check that Engineered Element objects are able to fail or articulate at intermediate positions between end vertices.	Subdivide at nodes has been set as 'False' for LIST. This means that the chosen post-solve diagram(s) will be misleading (unless a sufficient number of intermediate Vertices have been manually added).
Warning	Check to determine whether an Engineered Element exists in a Factor Strength(s) analysis.	Only material strengths, cohesion and friction are factored in factor strength mode. Engineered element parameters (pullout strength, rupture strength, bending moment) are unaffected in this mode.
Warning	Check to determine whether cutoff material exists in a Factor Strength(s) analysis.	Warning: Only material strengths, cohesion and friction are factored in factor strength mode. Cutoff tensile and compressive strengths are unaffected in this mode.
Warning	Check to identify Engineered Element materials with zero rupture and compressive strengths.	The Engineered Element material(s) LIST have been specified with rupture and compression strengths both set to zero. This is likely to lead to an unstable solution.
Warning	Check to identify any linked (joined) series of Engineered Elements that has been specified to have no interaction with the surrounding zone.	The linked series of Engineered Elements consisting of LIST have all been specified with pullout factors Tc and Tq and lateral factors Nc and Nq all set to zero. This will lead to no interaction of the linked series of engineered elements with the soil.
Warning	Check to identify any straight line series of Engineered Elements that has been specified to have no lateral interaction with the surrounding zone.	The straight line linked series of Engineered Elements consisting of LIST have all been specified with lateral factors Nc and Nq both set to zero. This will lead to no lateral interaction of the engineered element with the soil. This may lead to unusual and unrealistic failure mechanisms involving large lateral displacements of the engineered elements.
Warning	Check to identify Cutoff materials with zero rupture and compressive strengths.	The Cutoff material(s) LIST have been specified with zero compressive and tensile limiting stresses. This is likely to lead to an unstable solution.
Warning	Check to see if an interface material has been applied to the free surface of an Engineered Element.	An Engineered Element has been detected on a free boundary with different interface properties applied to each side. In the current version of the software all interface materials will be applied to the side in contact with the Solid geometry object.
Warning	Check to see if the problem contains materials with a linearly varying strength with depth and that these do not create a zero or negative strength within any Solid objects that they are applied to (thus promoting an *unstable* solution).	Found material(s) that have linear variation of undrained strength with depth giving a zero or negative strength within the following Solid(s): LIST
Warning	Check to see that a sensible value of artificial cohesion has been specified for Factor Strength(s) analyses	An Artificial Cohesion has been specified for use in Factor Strength(s) analyses. The value (VALUE) is greater than the default value (VALUE) - are you sure that this is intended?
Warning	Check to see that any derived materials applied to boundaries are adjacent to zones containing the parent material	Found a boundary that contains a derived material, but is not adjacent to a solid containing the parent material: LIST
Warning	Check to see that low strength Engineered Elements are not creating potential hinges in high strength Engineered Elements at the point where they join.	One or more of the connected Engineered Elements consisting of LIST have been specified with a low or zero bending strength (Mp) but are linked to other engineered elements of high bending strength. This will effectively introduce a low strength hinge into the set of high strength engineered elements which may be unintended.
Error	Check that the problem size is greater than $0.25 \times 10^{-4} \text{ m}^2$.	Defective geometry encountered - the total area of problem is too small.
Error	Check that at least one Solid has an applied Material .	No materials have been applied, drag and drop a material onto a geometry object.

Continued on next page...

Error	Check that at least one Boundary is acting as a Support .	No support has been applied.
Error	Check that a geometry has been defined.	No geometry objects have been defined.
Error	Check that Adequacy has been defined as <i>True</i> for at least one load or weight.	An adequacy factor needs to be applied to either a load or to a solid object with non-zero self weight.

Table 22.2: Checks carried out by the LimitState:GEO Diagnostics Tool.

22.3 Analysis Settings

22.3.1 Overview

A number of the top-level project **Property Editor** parameters (as shown in Figure 22.2) provide key control over the type and accuracy of the solution to be obtained.

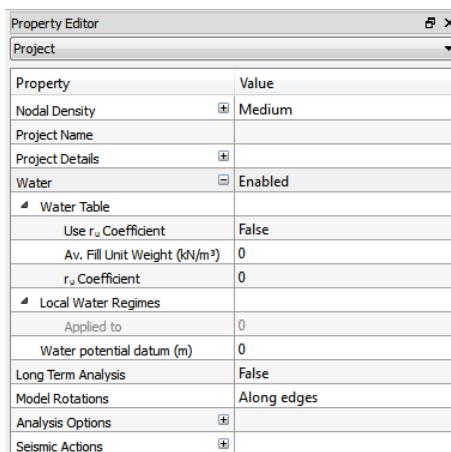


Figure 22.2: Project properties

Nodal Density This setting controls the nodal resolution and accuracy of the solution obtained (Section 22.3.2).

Water This setting controls whether water pressures are enabled or disabled during the analysis (Section 20.2) as well as providing control over the global **Water potential datum** and **Water table** r_u settings.

Long Term Analysis This setting controls whether the problem solves as either a long term (typically drained) or short term (typically undrained) problem as described in Chapter 16. If multiple scenarios (see Chapter 30) are defined with differing individual long term/short term settings, then this parameter will display *Varies*. Changing this setting to *True* or *False* will change the long term/short term setting for all the scenarios.

Analysis Options The settings in this section control the parameters of a **Factor Strength(s)** analysis (Section 8.2.3).

Model Rotations This setting controls whether the analysis is carried out as a translational analysis or with rotations permitted at **Solid** edges (Section 8.1.2).

Seismic Actions This setting controls whether pseudo static horizontal and or vertical accelerations are applied to the problem (Chapter 21) . One or both accelerations may have an **Adequacy factor** applied to them.

22.3.2 Setting and Previewing the Nodes

Solution accuracy is determined by the number of nodes utilised in the solution as described in Section 5. The number of nodes to be used may be set using the **Nodal Density** setting in the Project level parameters accessible in the **Property Editor**. The settings are *Coarse* (250 nodes), *Medium* (500), *Fine* (1000) and *Very Fine* (2000). It is recommended that initial scoping calculations are carried out in *Coarse* or *Medium* mode which typically generates solutions within a few seconds. For determination of the solution for final accuracy, the setting should be changed to *Fine*, *Very Fine* or higher using the *Custom* setting described below.

At any time, it is possible to view the arrangement of nodes prior to solve to check that these are as required. To do this, first select the **Preview Nodes** menu item on the **Analysis** menu. It will then be possible to toggle on and off the view of nodes using the **Show Nodes** button . Note that it is necessary to click **Unlock** afterwards to edit the problem parameters.

For advanced use, the **Nodal Density** may be set to *Custom* . The user has two options on how to control the Nodal density:

Target Number Set this to the total number of nodes that the software will attempt to use when solving the problem. The software will normally be able to meet this target within a few percent. Altering this value will cause the **Scale Factor** on the **Baseline Nodal Spacing** to change, thus altering the Actual Nodal Spacing:

$$\text{Actual Nodal Spacing} = \text{Baseline Nodal Spacing} / \text{Scale Factor}$$

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. Modifying the **Target Number** will automatically cause the **Scale Factor** to change.

Scale Factor A global factor used to scale the **Baseline Nodal Spacing** to achieve (or alter) the **Target Number** of nodes.

$$\text{Actual Nodal Spacing} = \text{Baseline Nodal Spacing} / \text{Scale Factor}$$

i.e. doubling the **Scale Factor** causes the number of nodes used along boundaries to double and the number of nodes used in solids to quadruple.

Note: Modifying the **Scale Factor** will automatically cause the **Target Number** to change.

Setting the **Baseline Nodal Spacing** is described in Section 22.3.3.

22.3.3 Setting Nodal Distribution within Geometry Objects

The baseline nodal spacing within **Geometry objects** may be set by selecting the required object and modifying the following parameter(s) in the **Property Editor**:

Boundary object Nodal spacing: this is the baseline spacing along the Boundary object.

Solid object Linear nodal spacing: Expand to view the x-spacing and y-spacing. These specify the baseline spacing on a rectangular grid within the solid object. The software will position the grid within the solid appropriately.

If it is expected that failure will not occur in certain geometry objects (e.g. concrete elements) then setting a large nodal spacing in these objects will ensure computational resources are not wasted.

22.3.4 Optimizing Nodal Layout

For classes of problem where the failure mechanisms are likely to involve some degree of soil body rotation (e.g. gravity retaining walls) the **Analysis tab** of the wizards includes the option to **Optimize [the nodal layout] for problem type**. By checking this box, extra nodes will be allocated to the **Boundaries** where rotations are likely to occur, thus improving the solution obtained. However, if significant rotations are anticipated then it is better to adopt the procedure for dealing with rotational failure mechanisms described in Section 5.3.3.2.

22.4 Solving

To solve a problem, select the solve icon  or press F5.

LimitState:GEO is set up to conduct analyses that either calculate an **Adequacy factor** for loads (**Factor Load(s)**) or material strengths (**Factor Strength(s)**). To switch between analysis types, select the required option from the drop-down menu in the **Analysis** toolbar (alternatively, the final tab in each wizard allows the user to specify the analysis type).

Advanced settings for a **Factor Strength(s)** analysis are available in the **Property Editor**.

For more information on the differences between the **Factor Load(s)** and **Factor Strength(s)** analysis types, see Sections 8.2 and 16.3.

22.5 Analysis Results

22.5.1 Collapse Adequacy Factor Found

Following an analysis the critical collapse **Adequacy factor** will be displayed in the **Output** pane.

The **Adequacy factor** is the factor by which specified loads must be increased, or material strengths decreased, in order for the system under consideration to reach a collapse state. Depending upon the type of analysis selected prior to solve (chosen using the dropdown menu

in the **Analysis Toolbar** or the **Analysis Tab** in a wizard) there are two types of **Adequacy factor** that can be reported:

- **Adequacy factor** on load
- **Adequacy factor** on soil strength

When assessing a design using a design code analysis with partial factors applied to loads, a value of this factor of greater than or equal to 1.0 indicates that the design is safe. A value less than 1.0 indicates that it is unsafe.

22.5.2 No Solution Found

22.5.2.1 ‘Locked’ Result

It is possible that the applied load can be increased without limit. In this case the problem can be described as being geometrically ‘locked’. This result can occur in the following scenarios:

1. Where a highly constrained body of soil is modelled with a large angle of shearing resistance (or using a *Rigid* material).
2. In a problem involving a sloped frictional material, when the adequacy factor is applied to the self weight; here it will often be found that the self weight can be increased without affecting stability. (As an analogy consider a book resting on a slightly sloping table; if the book is initially in stable equilibrium then simply making the book progressively heavier will never lead to the book sliding off the table.) Further discussion of these issues may be found in Section 8.6.1.
3. Where the partial factor that is applied to the load or loads that also have the **Adequacy factor** applied to them is set to zero. Partial factors are set in the **Scenario Manager**, see Section 30.

22.5.2.2 ‘Unstable’ Result

It is possible that no solution can be found because material in the problem is ‘unstable’ under its own self weight and/or other applied forces. i.e. no viable equilibrium state can be identified. This result will typically occur if a body of soil is loaded, but has no viable means of resisting the load. For example, the ‘unstable’ result will occur in a slope problem involving a frictional material when the slope angle is greater than the angle of friction. (As an analogy consider a book resting on a steeply sloping table; if the book is not initially in stable equilibrium then increasing or reducing the weight of the book will not affect this, and the book will slide off the table.) Further discussion of these issues may be found in Section 8.6.1.

22.5.2.3 ‘Terminated’ Result

If the user aborts the analysis (see below for details on how to do this) then a ‘terminated’ status message will be reported. Note also that the software will internally abort an analysis if the number of iterations required is excessive (though this should very rarely happen).

22.5.2.4 ‘Unknown’ Result

The solver used by the software is very robust but can occasionally encounter numerical difficulties, leading to failure to find a solution of definite status (i.e. it will fail to find a value for the ‘adequacy factor’ or the ‘locked’, ‘unstable’ or ‘terminated’ results). If this occurs then an ‘unknown’ status will be reported.

In such cases, it is often beneficial to try to solve the problem again, initially in completely unchanged form, and subsequently after very minor changes have been made to one or more of the following:

- nodal density
- material properties
- loading regime

However, in the rare instance that a solution is still not found, please contact LimitState support for assistance.

22.5.2.5 ‘Always Locked’ Result

In a Factor Strength(s) analysis, a problem that continually returns a ‘locked’ solution will not fail for any value of factor on material strength. For example, this may occur if attempting to determine the **Adequacy factor** on the strength of a rigid material.

22.5.2.6 ‘Always Unstable’ Result

In a Factor Strength(s) analysis, a problem that continually returns an ‘unstable’ solution will fail for all values of factor on material strength. For example, this may occur if attempting to determine the **Adequacy factor** on the strength of a material zone lying on a sloped, frictionless interface.

22.5.2.7 ‘Slow Progress’ Result

The solver is encountering difficulties in converging to an optimum solution. The solution returned in the current iteration may or may not be of acceptable quality. Therefore the result is

displayed in square brackets e.g. '[1.3334]', and a subsequent iteration is undertaken in order to attempt to fix the problem and return a good solution.

For the **Factor Load(s)** mode, if more than two "Slow Progress" messages are returned by the solver then the solve is terminated.

For the **Factor Strength(s)** mode the solver will attempt to iterate to a correct converged solution.

22.5.2.8 ‘Out of Space’ Result

The solver has exhausted all the available memory without achieving a converged solution. In such cases, it may be necessary to reduce the complexity of the problem or, more generally, increase the amount of RAM in the PC or switch to an operating system that provides a higher limit on the amount of memory that a process can utilize.

Note that, to use LimitState:GEO in 64-bit mode¹, you will require a special ‘add-in’ license - please contact **sales@limitstate.com** for more information.

22.5.3 Aborting an Analysis

After the Mosek solver is started, LimitState:GEO then waits for a solution to be found. To abort this process, the user should click on the red stop button  on the toolbar or press the **Esc** key to abort the analysis and return control to the user.

22.5.4 Lock and Unlock

After a solution is found, the slip-line mechanism is displayed in the viewer window, and the solution is ‘locked’. A ‘locked’ solution prevents any modification to the problem geometry or properties, but allows access to the properties of individual slip-lines, pressure distributions on individual blocks and animation of the mechanism. It is possible to animate the mechanism (see Section 23.2), click on individual slip-lines to determine normal and shear forces or stresses on the line (see Section 23.3), and display pressure diagrams for individual blocks (see Section 23.4).

To unlock the problem to allow editing of the problem parameters, click the unlock button  or select **Unlock** on the **Analysis** menu.

¹By default, LimitState:GEO will run as a 32bit process on 64bit operating systems

Chapter 23

Post-Analysis Functions

23.1 Adequacy factor

Further information concerning the parameters to which the **Adequacy factor** has been applied may be obtained by clicking on the displayed **Adequacy factor**.

Further details concerning the data displayed may be found in Section 8.2.2 for **Factor Load(s)** mode and in Section 8.2.3 for **Factor Strength(s)** mode.

23.2 Animation

LimitState:GEO provides a useful facility to animate any failure mechanism to facilitate interpretation. This is achieved by magnifying the instantaneous displacements of each block of material at failure. Note that though the solution is strictly only valid for infinitesimal displacements, large displacements are displayed to assist visualization of the collapse mode.

To animate the mechanism following solve, use one of two options:

- Use the slider bar 
- Click on the animate button 

The animation may also be saved as an .avi or .gif file or as a series of still images (Chapter 25). The preferences controlling the size and content of the file may be accessed via the **Tools > Preferences** menu under the **Export** tab (see Section 25.4).

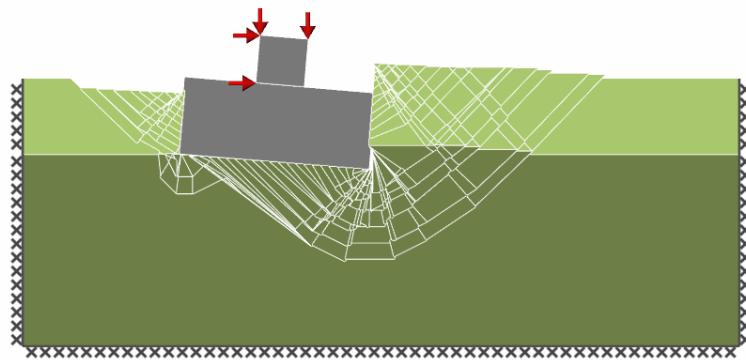


Figure 23.1: An animated LimitState:GEO failure mechanism

23.3 Slip-lines

Following Solve and display of the solution, individual slip-lines may be interrogated by selecting them with the mouse for the following information:

- Normal displacement
- Stress displacement
- Normal force (per unit width into the diagram)
- Shear force (per unit width into the diagram)
- Normal stress
- Shear stress

This information is displayed in the **Property Editor** for any selected slip-line (as depicted in Figure 23.2). Note that any given slip-line may be split into contiguous small segments. Forces and stresses are displayed as total (not effective) values i.e. inclusive of water forces (or pressures).

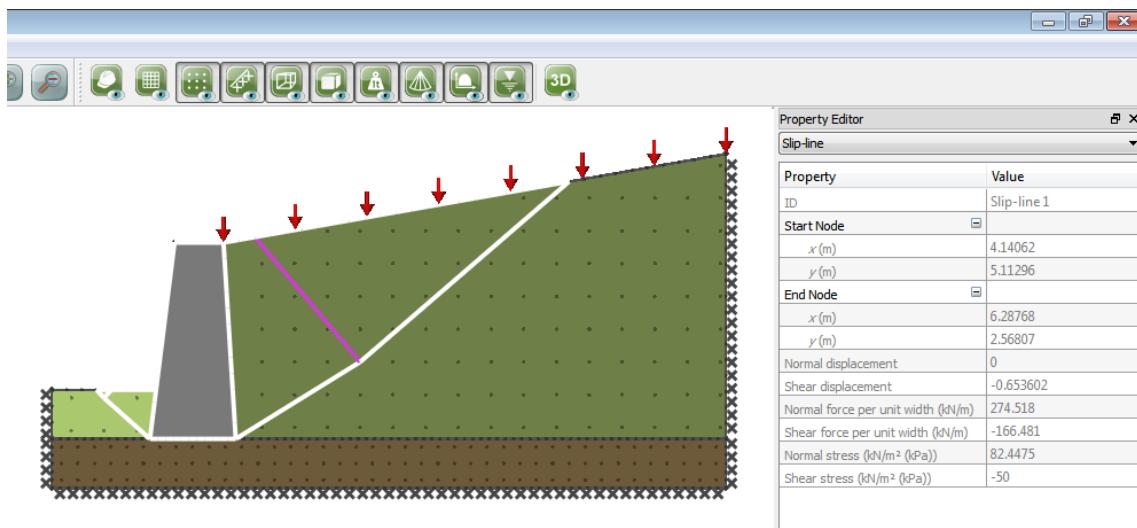


Figure 23.2: Interrogation of slip-line properties

23.4 Pressure Distributions around Blocks

Following Solve and display of the solution, in addition to interrogation of individual slip-lines, it is possible to display the pressure distributions around rigid **Blocks** of any material (i.e. blocks delineated by slip-lines). This is achieved by left-clicking with the mouse on the chosen object.

The distribution is displayed as a bar plot with each bar indicating the magnitude of the average stress on the relevant part of the selected object in the collapse mechanism as shown in Figure 23.3 . The bar width is defined by the nodal spacing on that slip-line. Specific values of average stress and water pressure corresponding to each bar may be viewed by hovering the mouse over the relevant part. Note that compressive values are displayed as negative.

Also note that in some cases apparently erratic stress distributions may be displayed by the solver. This is not an error but results from the nature of the solution method. For further discussion of this issue refer to Section 7.3.

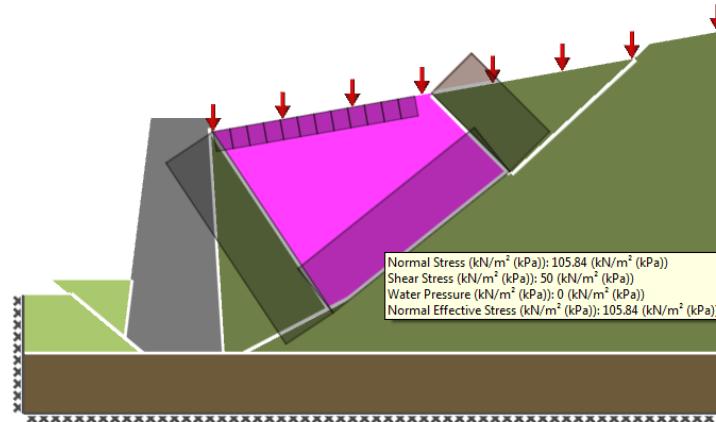


Figure 23.3: Display of post-solve pressure distribution diagrams for a selected **Block** object

Control over the type of diagram that is displayed is provided in several ways:

23.4.1 Post-Solve Display Dialog

The **Post-Solve Display** dialog (Figure 23.4) is accessible from a number of locations:

New Material Dialog When creating a new material, the option to set the default diagram display types is presented at the bottom. Clicking the **Change...** button will display the dialog.

Property Editor After selecting a material in the **Material Explorer**, a **Post-Solve Display** field is presented. Clicking the **Change...** button will display the dialog. This is available for both user-defined and pre-existing materials.

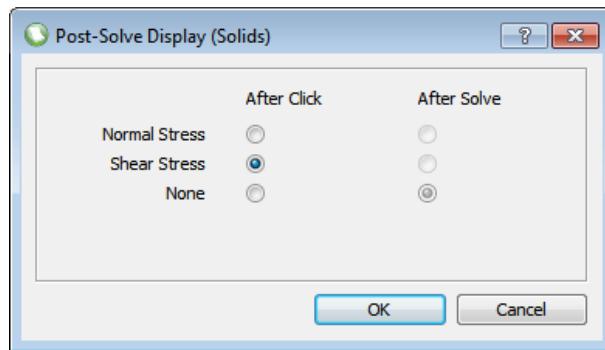


Figure 23.4: The **Post-Solve Display** dialog

Select *After Click* to display the chosen diagram type after selecting an object or *After Solve* to display the chosen diagram type for all objects containing that material immediately after solving. Note that the *After Solve* option applies only to **Engineered Element** material types (see Section 23.5).

23.4.2 Block Context Menu

The type of post-solve diagram being displayed can also be changed using the context menu. To bring up the menu, solve the problem then hover the cursor over the desired **Block** and right click the mouse. The menu will appear, with options to display the *Normal Stress* or *Shear Stress* on each edge (Figure 23.5):

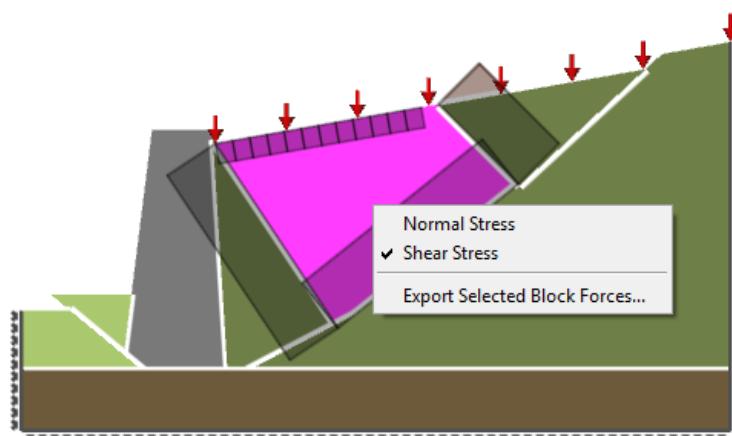


Figure 23.5: The **Blocks** post-solve display context menu

The **Export Selected Block Forces...** option in this menu also allows the force distribution around the selected block to be exported as a .csv file. Note that, to avoid the coordinates being displaced in the output, this should be done while the solution is in a non-animated state.

The values exported are:

- *Face*
- *Angle (degrees clockwise from x-axis)*
- *Start Point x (m)*
- *Start Point y (m)*
- *End Point x (m)*
- *End Point y (m)*
- *Mid Point x (m)*
- *Mid Point y (m)*
- *Length (m)*
- *Normal Stress (kN/m² (kPa))*
- *Shear Stress (kN/m² (kPa))*
- *Normal Force (kN/m)*
- *Shear Force (kN/m)*
- *Horizontal Equilibrium Term (kN/m)*
- *Vertical Equilibrium Term (kN/m)*
- *Water Pressure (kN/m² (kPa))*
- *Normal Effective Stress (kN/m² (kPa))*
- *Water Force (kN/m)*
- *Normal Effective Force (kN/m)*

The (average) stresses, pressures, and forces are those act on the at the **Block** between the Start and End Points. The force values are equal to the equivalent (average) stress multiplied by the length of the line segment. Mid point values are provided to facilitate simple graph plotting if required.

23.5 Force Distributions along Engineered Elements

Following Solve and display of the solution, it is also possible to display the axial force, shear force and moment distributions along a boundary containing an **Engineered Element**. This is achieved by left-clicking with the mouse on the chosen object.

The distribution is displayed as a bar plot with each bar indicating the linearised variation of the magnitude of the force or moment between nodes on the selected object as shown in Figure 23.6. The bar width is defined by the nodal spacing on that boundary. Specific values of the force/moment variation on each bar may be viewed by hovering the mouse over the relevant part.

The sign convention adopted is as follows:

Axial force Tension positive

Shear force Clockwise positive

Moment Hogging positive (for a horizontal element)

The graphical displays are plotted so that positive values are plotted above the element for a horizontal element and to the right of an element for a vertical element (i.e. following standard graph plotting convention). For inclined elements, the sign convention adopted is depends on which axis direction the line is nearest to. If not clear the signs can be confirmed by using the tool-tip function.

Valid moment values are calculated except when **Model Rotations** value is set to *False* or when the **Moment of Resistance** M_p value is set to $1E30$ i.e. a rigid element is to be modelled. In these cases no plot is produced and the tool-tip returns a value 'Unavailable'.

Also note that in some cases apparently erratic force distributions may be displayed by the solver. This is not an error but results from the nature of the solution method. For further discussion of this issue refer to Section 7.3. Thus the values displayed are unique only in yielding areas.

In non-yielding areas values may therefore be higher than required to prevent collapse. Hence if adopting the values of e.g. the Moment for design it is recommended to specify these explicitly as properties of the Engineered Element and re-run the analysis to confirm the effect of the value selected.

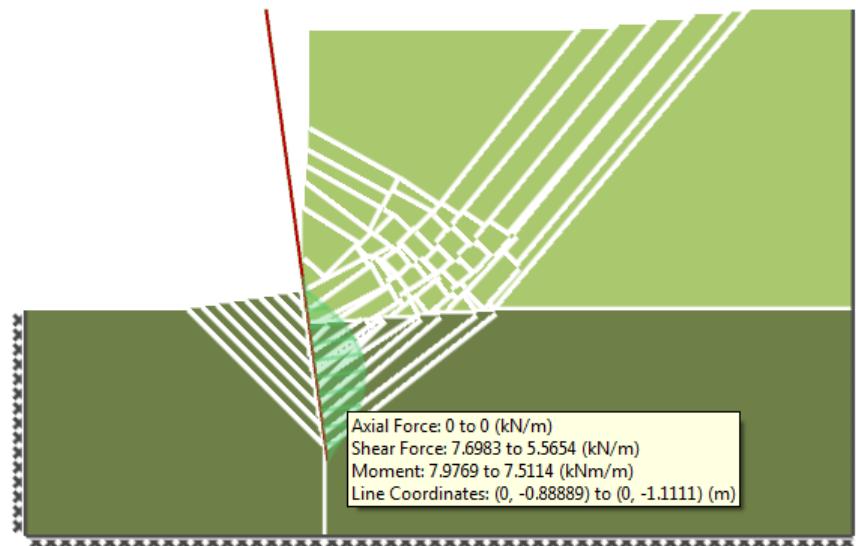


Figure 23.6: Display of post-solve pressure distribution diagrams for a selected **Engineered Element**

Control over the type of diagram that is displayed is provided in several ways:

23.5.1 Post-Solve Display Dialog

The **Post-Solve Display** dialog (Figure 23.7) is accessible from a number of locations:

New Material Dialog When creating a new material, the option to set the default diagram display types is presented at the bottom. Clicking the **Change...** button will display the dialog.

Property Editor After selecting a material in the **Material Explorer**, a **Post-Solve Display** field is presented. Clicking the **Change...** button will display the dialog. This is available for both user-defined and pre-existing materials.

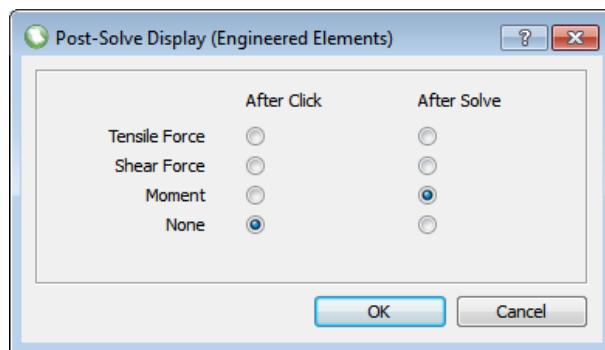


Figure 23.7: The **Post-Solve Display** dialog for Engineered Elements

Select *After Click* to display the chosen diagram type after selecting an object or *After Solve* to display the chosen diagram type for all objects containing that material immediately after solving. Note that the *After Solve* option applies only to **Engineered Element** material types.

23.5.2 Engineered Element Context Menu

The type of post-solve diagram being displayed can also be changed using the context menu. To bring up the menu, solve the problem then hover the cursor over the desired **Engineered Element** and right click the mouse. The menu will appear, with options to display the *Tensile Force*, *Shear Force* or *Moment* per unit length at each node along the object (Figure 23.8):

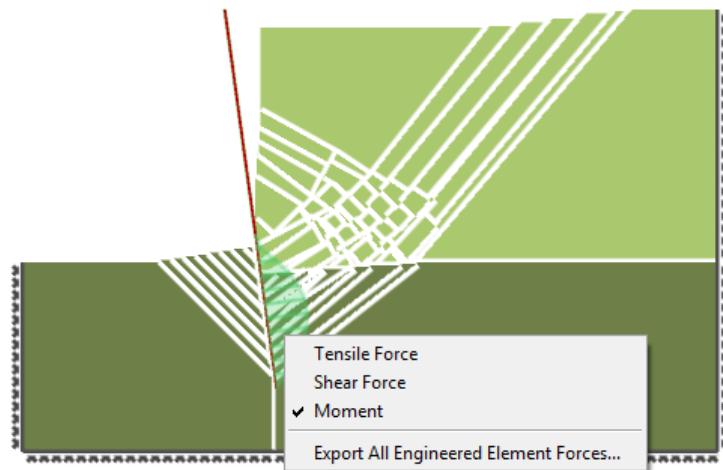


Figure 23.8: The **Engineered Elements** post-solve display context menu

The **Export All Engineered Element Forces...** option in this menu also allows the force/moment distribution of the selected object to be exported as a .csv file. Note that, to avoid the coordinates being displaced in the output, this should be done while the solution is in a non-animated state.

The values exported are:

- *Point x (m)*
- *Point y (m)*
- *Engineered Element ID*
- *Axial Force (kN/m)*
- *Shear Force (kN/m)*
- *Moment (kNm/m)*

The forces and moments are those which act in the element at coordinates (x, y) .

Chapter 24

Report Output

24.1 Viewing the Report Output

Following an analysis a customizable report can easily be generated. LimitState:GEO can output a report in Microsoft Rich Text (.rtf) format which can then be edited if required by the user using a word processor. It is also possible to save the report in HTML format, e.g. for company intranet usage. To generate the report, on the **Analysis** menu click **Report...**: The following **Report options** dialog (Figure 24.1) allows full control over what is displayed in the Report.

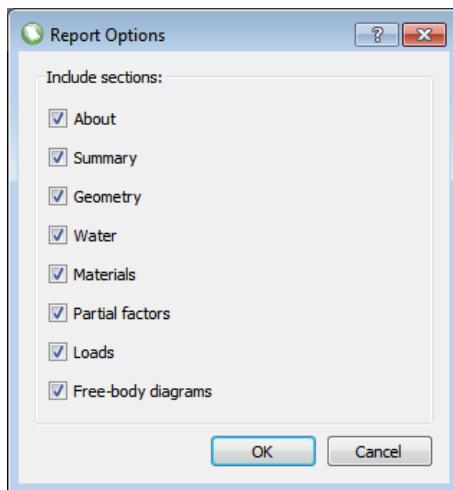


Figure 24.1: LimitState:GEO **Report options** dialog

About Include an introductory section containing information about LimitState:GEO, the DLO analysis method and the Adequacy Factor.

Summary Include an overview of the project details, main analysis settings and an image of the critical collapse mechanism.

Geometry Include information about the geometry of the problem (boundaries and solids) including the constituent vertices, support type, nodal spacing and applied materials / water regimes.

Water Include information about the geometry of any defined water table or regime.

Materials Include property data for the materials defined in the project.

Partial factors Include information on all the partial factor sets defined in the project.

Loads Include information on loads applied in the project, whether on Boundaries or the self-weight of a Solid object.

Free-body diagrams Include free body diagrams for the solid objects delineated by slip lines in the analysed problem.

An example first of the report output is shown in Figure 24.2.

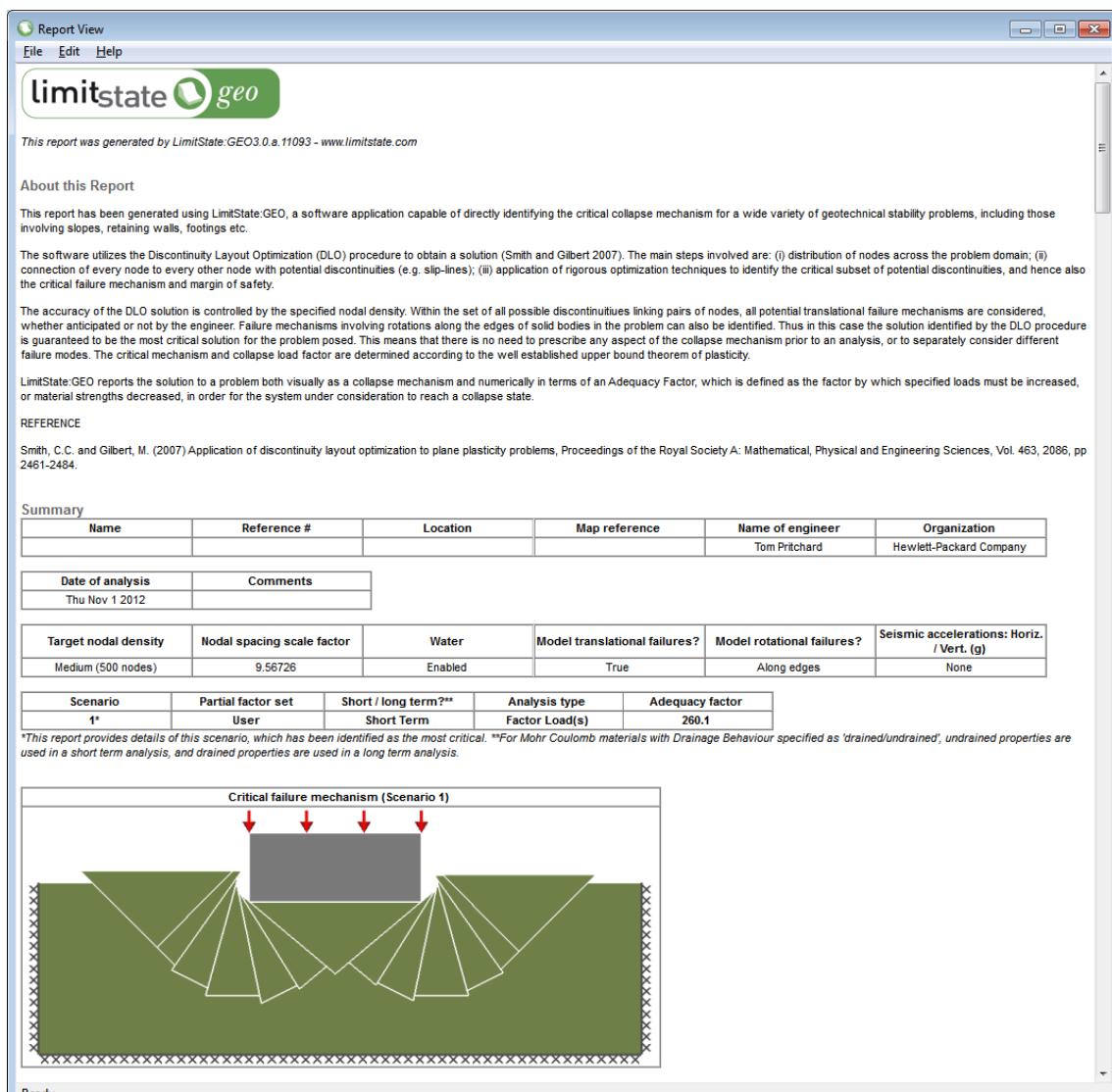


Figure 24.2: Example of LimitState:GEO report output

24.2 Saving the Report Output

The report output from LimitState:GEO can be saved to one of the following formats:

PDF Portable Document Format (.pdf)

RTF Rich Text Format (.rtf) - editable in most word processing software

HTML Hypertext Markup Language (.html)

To save the report, select **File > Save As...** in the report dialog.

24.3 Customizing the Header or Footer

Custom header and footer images can be included in the report by selecting the **Tools** menu, clicking **Preferences** and selecting the **Report** tab as shown in Figure 24.3.



Figure 24.3: LimitState:GEO report preferences

Chapter 25

Exporting Graphical Output

25.1 Overview

LimitState:GEO provides a number of options for exporting graphical output of the problem and the identified collapse mechanism. These may all be accessed from the main **File/Export...** menu as follows:

Geometry... This allows the geometry of the problem to can be saved as a Drawing Exchange Format (DXF) file for use in an external CAD package or import into a new LimitState:GEO project. Alternatively the geometry can be saved as an Encapsulated PostScript (EPS) or Asymptote (ASY) file.

Image... This allows the image displayed in the **Viewer** pane to be saved as either a Portable Networks Graphics (PNG), Joint Photographic Experts Group (JPG), Tagged Image File Format (TIFF), Encapsulated PostScript (EPS) or Postscript (PS) file.

Animation... This allows an animation of the solved problem to be saved in either Audio Video Interleave (AVI) or Graphics Interchange Format (GIF) file formats.

25.2 Geometry Export

The original (unsolved) problem geometry may be saved as a DXF file for subsequent modification in a CAD package or for re-importing to LimitState:GEO. This process is further described in Section 15.4.5.

The mechanism generated following solve may also be exported, along with the original geometry, as a line only vector graphics image (rather than a full colour image) in either DXF - AutoCAD (.dxf), EPS - Encapsulated Postscript (.eps) or ASY - Asymptote (.asy) formats. Example output is shown in Figure 25.1 and may be contrasted with the **Image** output as shown in Figure 25.2. Where allowable, the different entities (nodes, geometry, failure mechanism etc. will be output to separate layers).

The latter Asymptote format is a script based graphical programming language. The output

file provides a number of options for customising the image which can then be converted by Asymptote to a wide range of graphical output formats (e.g. EPS). The Asymptote file format can be viewed in any text editor and includes comments that may be consulted for further information.

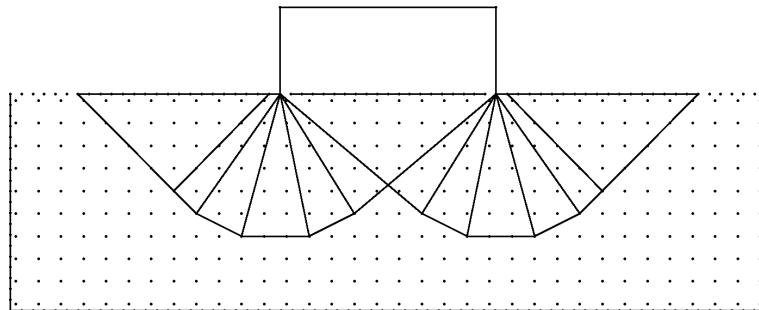


Figure 25.1: Example of line only vector graphics output for a Simple Footing problem

25.3 Image Export

This function allows the image displayed in the **Viewer** pane (including displaced solutions) to be saved in one of the following formats:

PNG Portable Network Graphics raster image (.png)

JPG Joint Photographic Experts Group raster image (.jpg)

TIFF Tagged Image File Format (.tiff)

EPS Encapsulated Postscript vector image (.eps)

PS Postscript vector image (.ps)

An example raster image (.png) is shown in Figure 25.2.

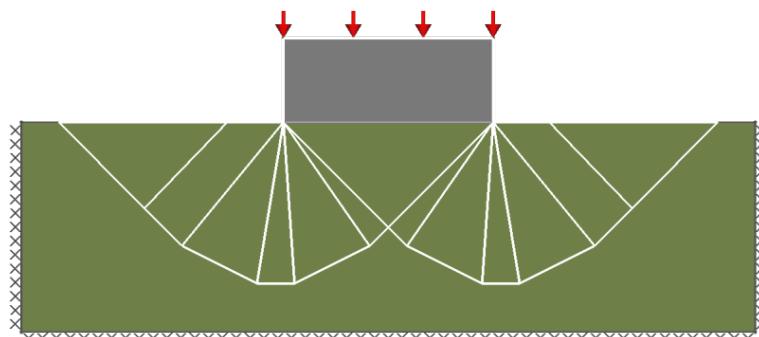


Figure 25.2: Example of image graphics output for a Simple Footing problem

For raster images, the image size width in pixels may be controlled using the **Export** tab in the **Preferences** dialog (accessed from the **Tools** menu) (see Figure 25.3).

25.4 Animation Export

An animation of the failure mechanism (as seen in the **Viewer** pane) can be exported to either:

AVI Audio Video Interleave (.avi)

GIF Graphics Interchange Format (.gif)

Both formats are readily incorporated into most modern Presentation packages.

To export an animation, go to the **File** menu and select **Export... >Animation...**. Here you can select the format of the file and save location.

In addition to the slip line mechanism, other visual aspects of the solution can be included in the exported animation. These are controlled via the **Export** tab of the **Preferences** dialog in the **Tools** menu (see Figure 25.3).

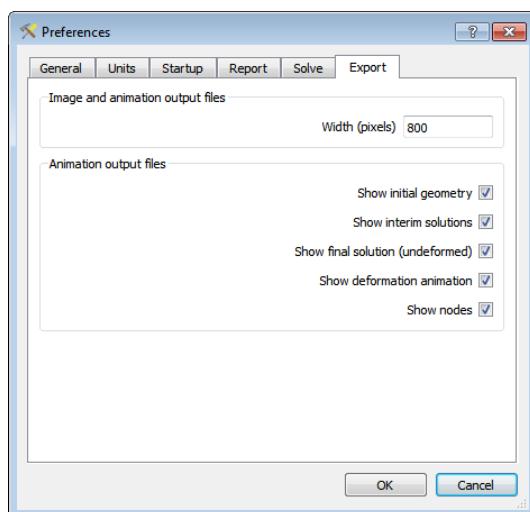


Figure 25.3: Export settings for animation output

Width (pixels) Controls the width of the image / animation (to a maximum width of 1200 px).

Show initial geometry Insert frames showing the starting geometry of the model (pre-analysis).

Show interim solutions Insert frames showing the failure mechanisms associated with each iteration of the analysis.

Show final solution (undeformed) Insert frames showing the critical failure mechanism without deformation.

Show deformation animation Insert frames showing each step of the deformation animation.

Show nodes Insert frames outlining the distribution of nodes in the model.

An example animation (.avi) is shown in Figure 25.4.

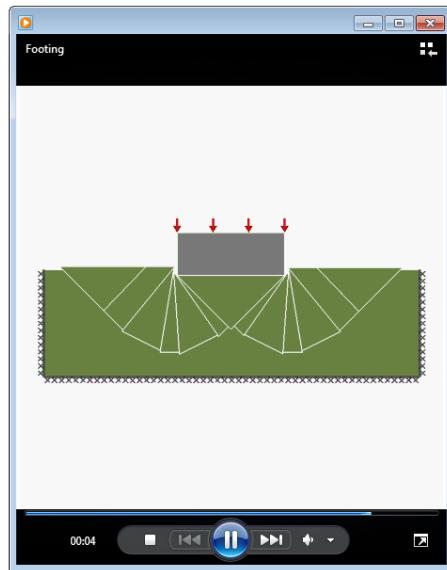


Figure 25.4: Animated failure mechanism from LimitState:GEO

25.5 Animations and Multiple Scenario Problems

Where a problem involves more than one scenario, the critical case is selected by default after solving and the image or animation relating to this will be exported. However, in some situations it may be desirable to export data relating to a non-critical case. To do this, simply select the tab at the bottom of the **Viewer** pane that relates to the case in question before exporting.

Chapter 26

Opening and Saving Projects

26.1 Opening and Saving Projects

The standard load and save functions are available:

Save This saves the file in the default directory using the current name. If a name has not been specified by the user then a default name: ‘Projectn’ is used where *n* is set to 1 unless this name already exists in which case it is set to 1 more than the largest existing value. The Save function may also be accessed using the  toolbar button.

For users running Windows 7 or above, a thumbnail image of the model and a number of descriptive tags can also be saved with the project. These help to identify individual files without the need to open them. More information is provided in Sections 2.1.1 and 4.2.2

Save as... The user is prompted for a filename and may also change the directory if required.

Open... The user is prompted to select a file from the default directory. This directory may be changed if required. The Open function may also be accessed using the  toolbar button.

26.2 Auto-recovery Files

LimitState:GEO saves auto-recovery files every five minutes, and also when certain actions are performed by the user. If the software does not close down properly, for example due to a power outage or a crash in LimitState:GEO, an auto-recovery file will be available. On next startup, LimitState:GEO will inform the user that a recovery file is available, and ask whether to load it. If the user does not recover the file, it will be deleted.

While generating auto-recovery files, a message appears in the status bar and the user is temporarily blocked from making changes to the current project.

Chapter 27

Thumbnails, Tags and Searching

27.1 Thumbnails

LimitState:GEO will take a snapshot of the model geometry when saving the file and display this instead of the icon when navigating folders (note that this functionality may need to be enabled - see Section 2.1.1.1 for instructions on how to do this)¹. Figure 27.1 shows an example of a folder containing three LimitState:GEO files:

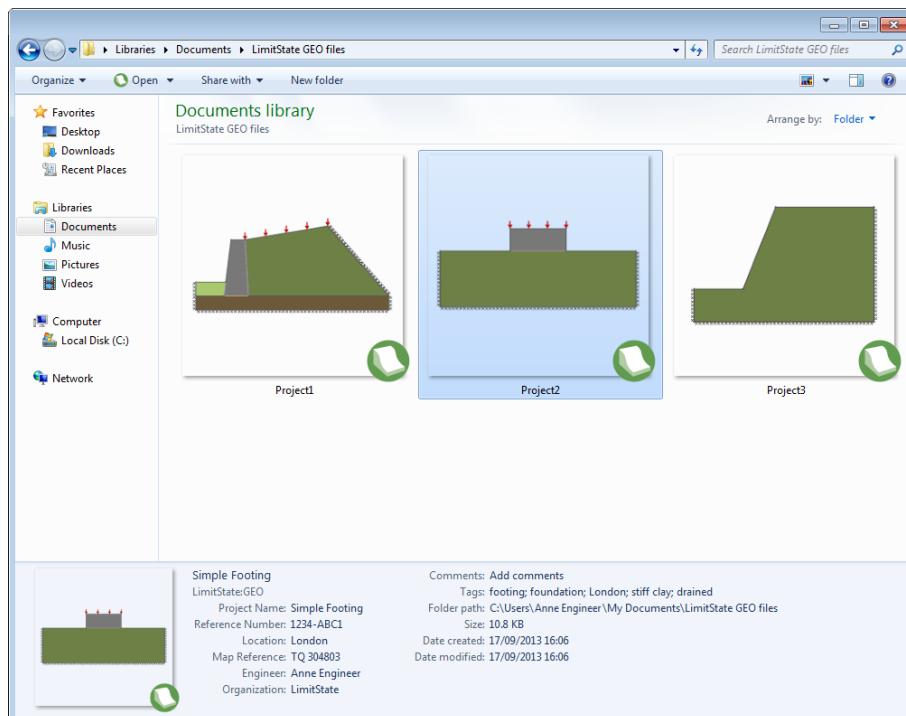


Figure 27.1: Thumbnails and tags are saved with the LimitState:GEO project file.

¹Windows Vista and newer operating systems only

27.2 Tags and Searching

27.2.1 Tags

Tags (or keywords) allow the user to categorize a file and search for it on their Windows system² at a later date without necessarily having to open it and inspect the properties.

Tags can be edited within LimitState:GEO either within the **Project Details** dialog or within the **Property Editor** at **Project** level (Figure 27.2).

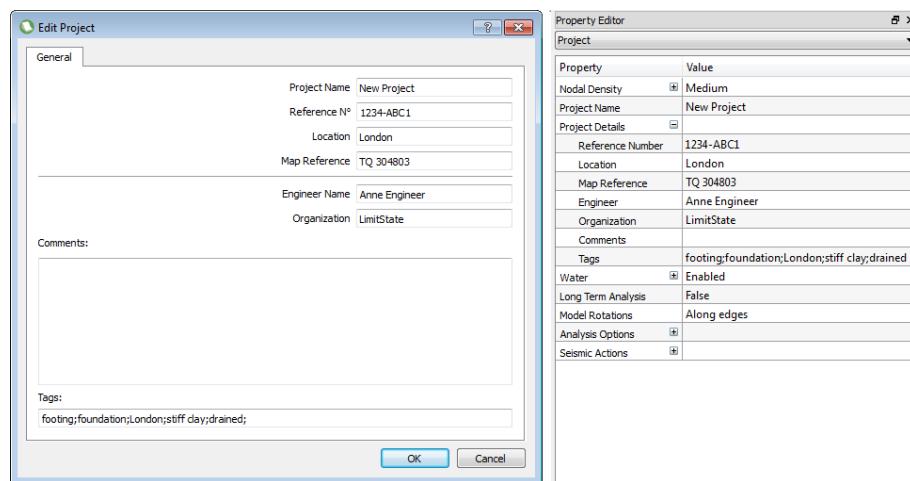


Figure 27.2: Tags specified within the LimitState:GEO **Project Details** dialog and **Property Editor**

Each tag in the **Tags** field should be separated by a semi-colon (automatically generated). For example, in Figure 4.2 the tags "footing; foundation; London; stiff clay; drained" are specified. These are saved with the file and the individual words will register in a search of the system.

The tags for a project can also be edited outside LimitState:GEO by either:

- Selecting a file and clicking in the "Tags" field at the base of an Explorer window (selecting to **Save** once editing is complete), or
- Right-clicking the project file, selecting **Properties** and clicking the **Tags** field on the **Details** Tab.

27.2.2 Searching

To search for a LimitState:GEO file by name, simply type some or all of the name into the Windows search box (ensuring you are in an appropriate directory). Remember to suffix the ".geo" file extension.

²Windows 7 or above

To search for a tag within a LimitState:GEO file, append the "*tags:*" prefix to any term in the search box (e.g. search for "*tags:drained*").

To search for a file with specific project details (e.g. Location, Engineer or Organization) it may be necessary to first configure your system to allow file content to be searched. Details on how to do this are given in Section 2.1.1.2. Once this has been done, simply enter the appropriate terms in the search box.

Chapter 28

Preferences

The **Preferences** dialog, available from the **Tools** menu, allows a number of LimitState:GEO parameters to be set and saved for each time it is re-started.

28.1 General

The **General Preferences** tab (Figure 28.1) provides the ability to modify the language and look of the program. It also allows the user to define how the software checks for updates.

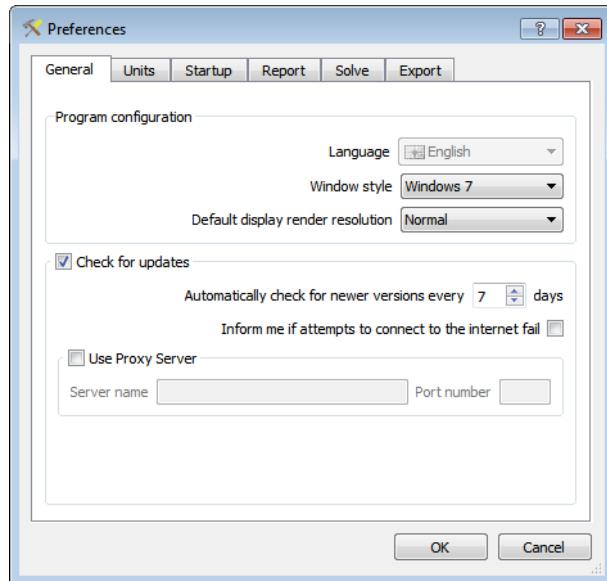


Figure 28.1: The **General** preferences tab

Language Change the language used by the software.

Window style Change the appearance of the software interface.

Default display render resolution Set the quality of the on-screen rendering.

Check for updates Uncheck this to prevent the software from periodically checking for newer versions.

Automatically check for newer versions every X days Use the up and down arrows to set the length of time between checks.

Inform me if attempts to connect to the internet fail Check this to be alerted if update checks are not successful.

Use Proxy Server Check this option to use a proxy server as the gateway for update checks (server name and port number required).

28.2 Units

The **Units** tab (Figure 28.2) provides the ability to toggle between *Metric* and *Imperial* units systems. If *Imperial* units are selected, the user has a further option to use default values that have been rounded appropriately for ease of use.

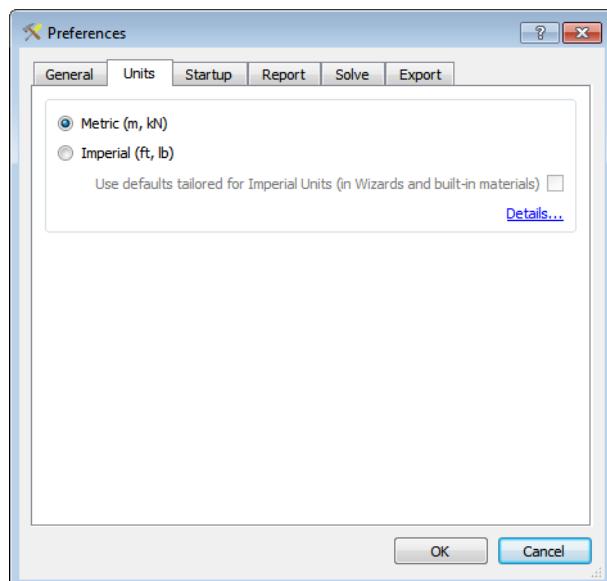


Figure 28.2: The **Units** preferences tab

28.3 Startup

The **Startup Preferences** tab (Figure 28.3) provides the ability to modify the features presented to the user each time the program is started.

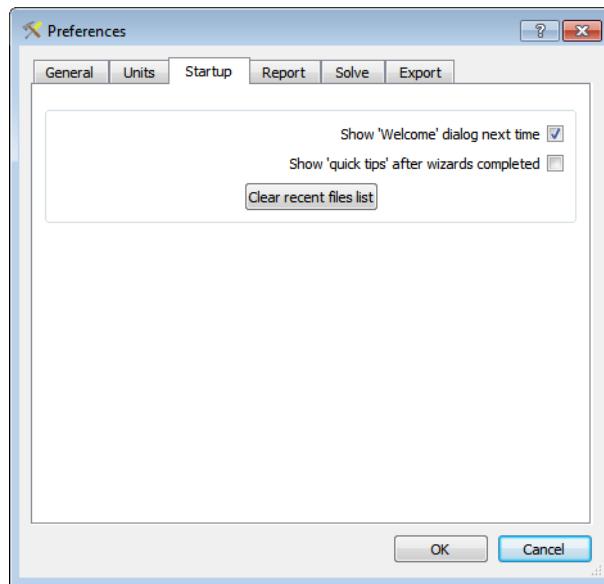


Figure 28.3: The **Startup** preferences tab

Show 'Welcome' dialog next time Display the welcome screen at startup. If this is deselected, the software will start without offering a choice of Wizard to the user.

Show 'quick tips' after wizards completed Check this to be provided with a dialog of useful information after completing a Wizard.

Clear recent files list Click this button to clear the recent files list (found under the **File** menu).

28.4 Report

The **Report Preferences** tab (Figure 28.4) allows the user to set the default header and footer images used in the report.

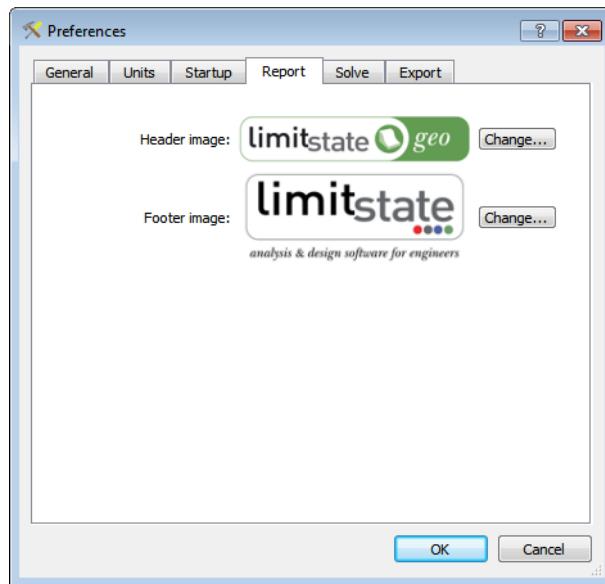


Figure 28.4: The **Report** preferences tab

Header image Click the **Change...** button to select a new PNG image to use at the head of the report.

Footer image Click the **Change...** button to select a new PNG image to use at the foot of the report

28.5 Solve

The **Solve Preferences** tab (Figure 28.5) allows the user to set when the pre-solve **Diagnistics** dialog is displayed and alter the solution information that is presented in the **Output Pane**.

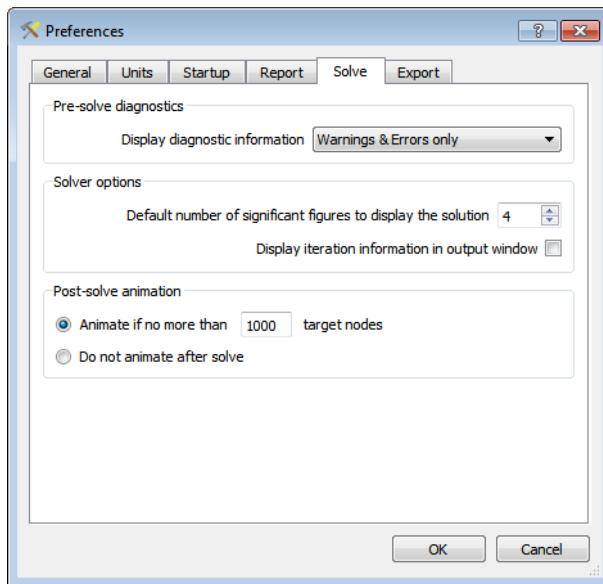


Figure 28.5: The **Solve** preferences tab

Display diagnostic information Use the drop-down list to set whether the **Diagnostic** dialog is displayed at Solve time a) *Always*, b) when there are *Warnings or Errors* or c) when there are *Errors Only*.

Default number of significant figures to display the solution Change this to alter the number of significant figures to which the **Adequacy factor** is reported in the **Output Pane** and **Report**.

Display iteration information in output window Check this box to add extra columns to the solution table in the **Output Pane**. Columns added include:

- **Iteration** - the iteration number.
- **Variables** - the number of variables in the linear programming matrix at this iteration.
- **Constraints** - the number of constraints in the linear programming matrix at this iteration.
- **Violation %** - the maximum percentage violation of the slip-line adding criterion encountered at this iteration.
- **Vars added** - the number of variables added to the linear programming matrix on this iteration.
- **Cons added** - the number of constraints added to the linear programming matrix on this iteration.
- **Time(secs)** - the time taken to solve the problem at this iteration.

Animate if no more than X target nodes If the target number of nodes for the project is fewer than the value of X, the solution will automatically animate on completion of the solve step.

Do not animate after solve Do not animate on completion of the solve step, irrespective of the complexity of the problem.

28.6 Export

The **Export Preferences** tab (Figure 28.6) allows the user to set the default contents and size of images and animations when exported to file.

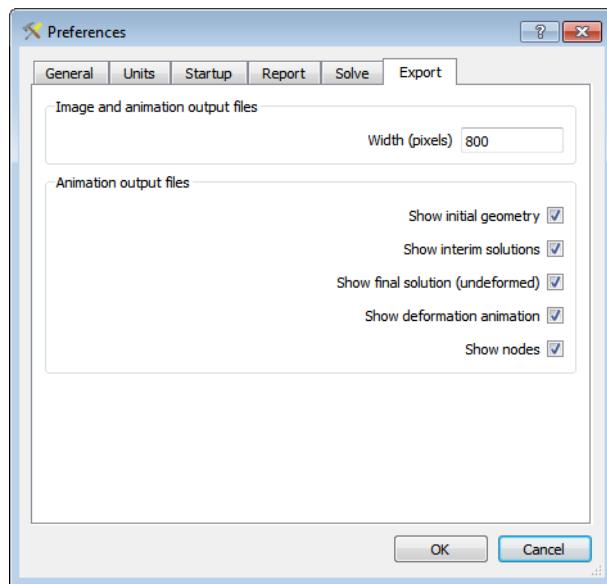


Figure 28.6: The **Export** preferences tab

Width (pixels) Controls the width of the image / animation (to a maximum width of 1200 px).

Show initial geometry Insert frames showing the starting geometry of the model (pre-analysis).

Show interim solutions Insert frames showing the failure mechanisms associated with each iteration of the analysis.

Show final solution (undeformed) Insert frames showing the critical failure mechanism without deformation.

Show deformation animation Insert frames showing each step of the deformation animation.

Show nodes Insert frames outlining the distribution of nodes in the model.

Chapter 29

License Information Dialog

The **License Information** dialog (Figure 29.1) is accessed from the **Help** menu and provides information on the license currently being used as well as the ability to swap to using a different license (e.g. as the result of an upgrade or renewal).

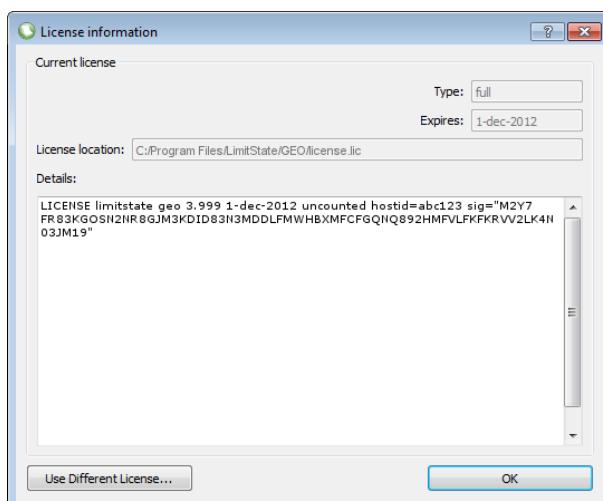


Figure 29.1: The License Information dialog

Type Describes the type of license being used (*full*, *trial*, etc.). This determines the level of functionality available.

Expires The date on which the license will cease to work.

License location The path to the license file being used.

Details The contents of the license file. This can be used to determine, amongst other things, the product, the highest version number that can be licensed, the hostid (computer identifier) and the license expiry date.

Use Different License... Click this button to disassociate from the currently used license and cause LimitState:GEO to request a new one. This is generally used when upgrading or replacing a license file and will require the software to restart. It is advised that all work is saved before using this.

Chapter 30

Scenario Manager and Partial Factors

30.1 Introduction

The **Scenario Manager** (depicted in Figure 30.1) is a powerful facility that permits several scenarios to be solved as a single set. This is a common requirement in geotechnical engineering, where for example undrained and drained cases may need to be checked, or where several design combinations of partial factors may need to be checked for the same geometry as is required by codes such as Eurocode 7. It is also used to specify partial factors for solution of single problems.

It should be noted that LimitState:GEO uses a similar convention to European codes of practice in that factors on load are treated as multipliers and factors on material properties are treated as divisors. In other geographical areas, this convention may be different and the user is advised to check before entering values into the software.

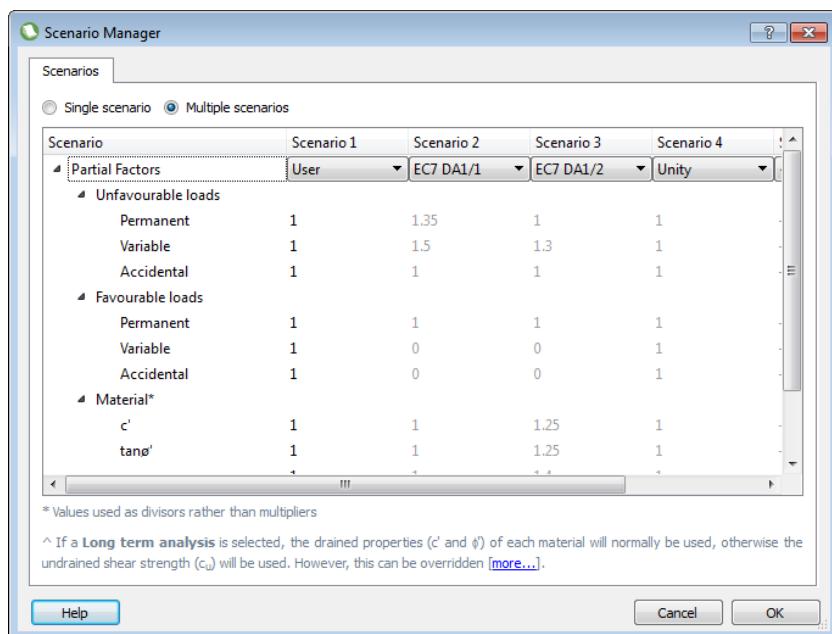


Figure 30.1: The Scenario manager

30.2 Specification of Partial Factors for a Single Scenario

Ensure that the **Single scenario** radio button is selected. One scenario **Scenario 1** is available. The **User** Partial Factor set is listed by default. Its default settings are all unity. To change any factor, double click on the entry (as shown in Figure 30.2), and change its value.

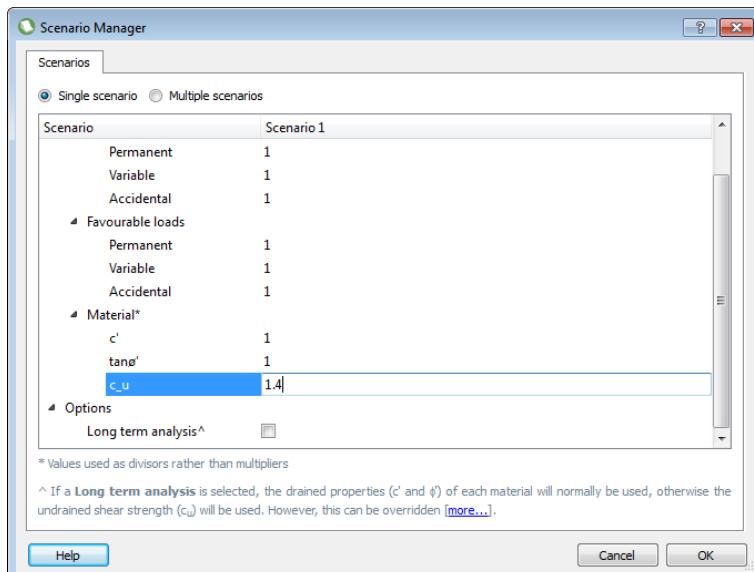


Figure 30.2: Editing Partial Factors in the Scenario manager

These Partial Factors will then be applied to the current problem. Loads may be individually defined as **Permanent**, **Variable** or **Accidental** and favourable / unfavourable / neutral (see Chapter 19). Material self weight is defined as **Permanent** but may be specified as favourable / unfavourable / neutral (see Section 17.8). Note that no partial factors are applied to the basic soil nail material type.

In addition to specifying individual Partial Factors, the user may select from a pre-defined set of partial factors by using the drop down box (as shown in Figure 30.3) or create a new list.

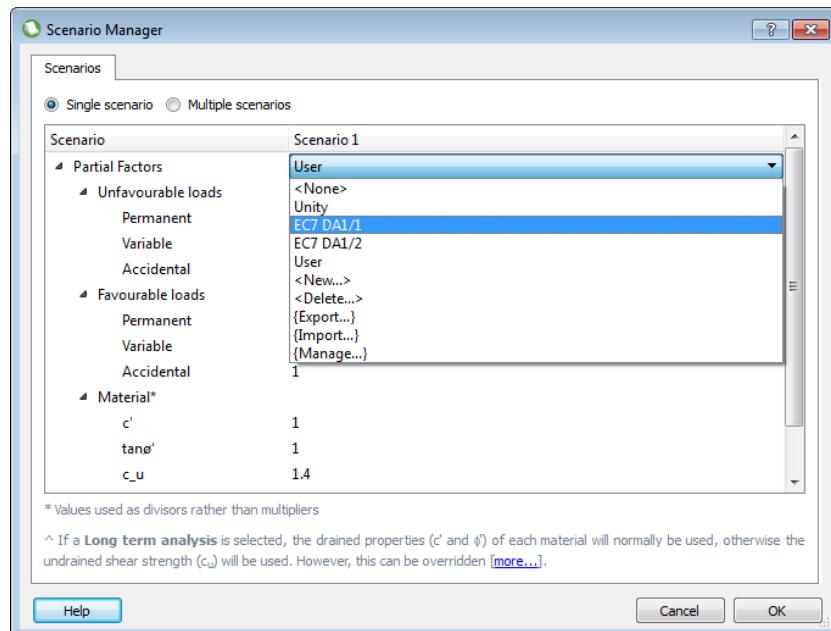


Figure 30.3: Partial Factors list in the Scenario manager

30.3 Specification of Long and Short Term Analysis Mode

Under the options heading (as shown in Figure 30.2). It is possible to set the scenario to solve as either a long term (typically drained) or short term (typically undrained) problem as described in Chapter 16. This setting may also be set globally via the project properties using the **Property Editor** as described in Section 22.3.

30.4 Specifying Multiple Scenarios

Ensure that the **Multiple scenario** radio button is selected. On first use, two scenarios **Scenario 1** and **Scenario 2** are listed. The **User** Partial Factor set is listed by default for **Scenario 1** and **<None>** is listed for **Scenario 2**. This may be changed to an actual Partial Factor set using the drop down box. If this is done, a new empty scenario (**Scenario 3**) is displayed. Further scenarios may be defined in this way.

To remove any scenario from the set of scenarios, select **<Delete>** from the drop down box. An error message will be displayed if this action would reduce the number of scenarios to zero.

30.5 Using the Partial Factor Set Manager

The partial factor set manager allows the user to control the partial factor sets that are saved with the file and listed in the drop down menu in the **Scenario Manager** itself.

Select {**Manage...**} from the list to open the **Manage User Defined Partial Factor Sets** dialog (Figure 30.4).

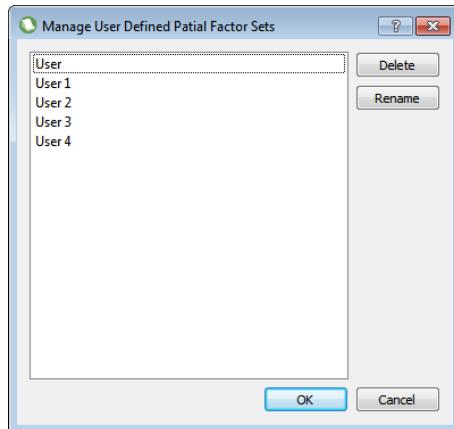


Figure 30.4: The LimitState:GEO Manage User Defined Partial Factor Sets dialog

Two options are then available:

Delete Select a partial factor set by clicking it in the list with the left mouse button (multiple sets can be chosen by holding down CTRL whilst selecting). Clicking the **Delete** button will remove the selected sets from the list. Sets that are being used in the project will not be deleted.

Rename Select a partial factor set by clicking it in the list with the left mouse button. Click the **Rename** button to bring up the **Rename Partial Factor Set** dialog, where the name of the set can be changed.

30.6 Exporting and Importing Scenarios

All Partial factor sets listed in the drop down box may be exported to a .csv file by selecting {**Export...**}.

To import previously exported sets of Partial Factors, select {**Import...**}. Partial Factor sets with the same name are not overwritten.

All Partial Factors are saved with the main file.

30.7 Solving with Multiple Scenarios

When multiple scenarios are defined, clicking **Solve** solves each scenario in turn and displays the failure mechanism in a separate tab in the **Viewer pane** (see Figure 30.5). Numeric solutions are also displayed in the **Output pane** and the critical case is highlighted. The other solutions may be viewed by clicking on the respective tabs.

Overall solution time is typically faster than if each scenario were solved individually since solution mechanisms are typically fairly similar for common Partial Factor sets.

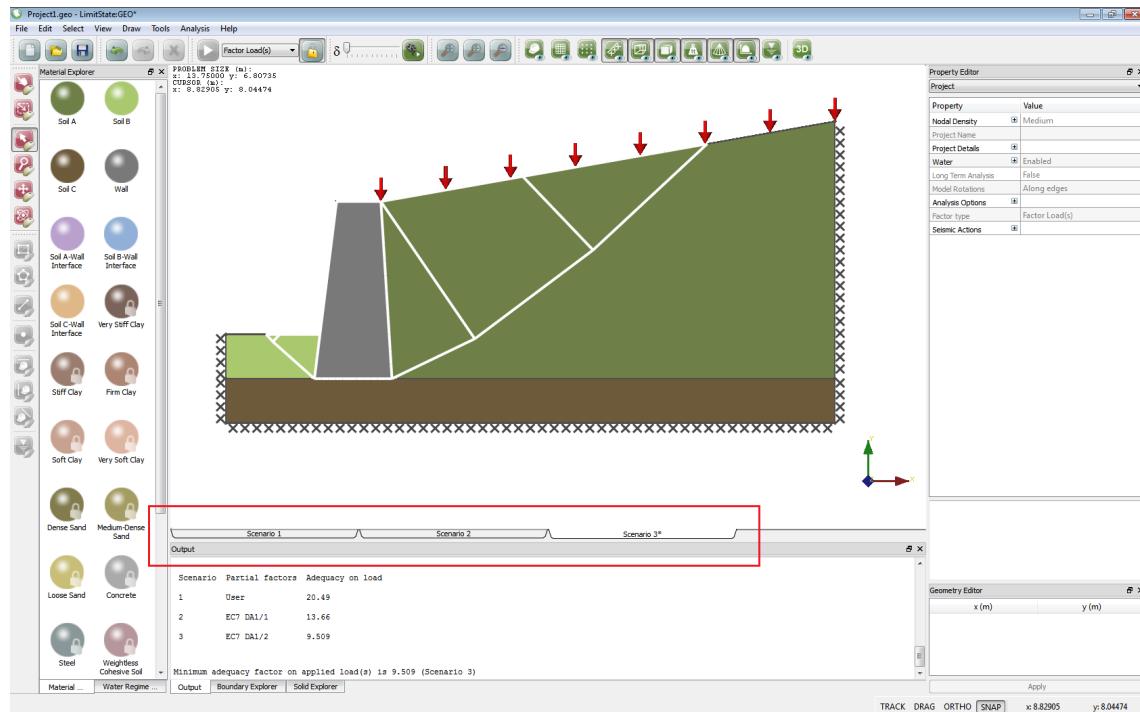


Figure 30.5: Presentation of results in tabs when multiple scenarios are specified in the **Scenario Manager**

Part V

Appendices

Appendix A

Verification

A.1 Verification Tests

A list of tests carried out using LimitState:GEO to verify its accuracy against known limit analysis solutions may be found at the following web page: <http://limitstate.com/geo/verification>.

These tests are run every time the software code is updated, and cover a broad range of problem types such as footings, retaining walls, slopes, and anchors.

Results are presented in terms of deviation from benchmark collapse load and also in terms of change in input parameters required for the LimitState:GEO solution to match the benchmark solution. This latter approach matches a design approach that considers uncertainty in material parameters. It is particularly relevant where the collapse load is a highly non-linear function of the soil strength. For example a change in ϕ' of 5% can result in a change of footing bearing capacity by 25% for simple footing problems on cohesionless soil.

The majority of these tests are carried out using the coarse nodal density. In general, even when using this coarse density, accuracy in most tests varies from exact to $\sim 5\%$ in terms of the input parameters. Increased accuracy is straightforwardly achieved by increasing the nodal density.

An animated image and further details of each test including the source of the benchmark can be obtained by clicking on a hyperlink on the web page. The input file used to generate the test result may also be downloaded.

A.2 Academic Papers

The following papers may be consulted for comparisons of DLO solutions with known solutions for the listed topics:

[Smith & Gilbert \(2007a\)](#) Footings on frictional soil, footings on confined layers.

[Smith & Gilbert \(2007b\)](#) Footings on layered soils, modelling of water pressures in DLO, effect of water table on bearing capacity of footings.

[Smith & Gilbert \(2008\)](#) Slope stability analysis, footings on slopes.

[Lee et al. \(2008\)](#) Pipeline bearing capacity.

[Merifield & Smith \(2010\)](#) Multi-plate strip anchors.

[Clarke et al. \(2010\)](#) Sheet pile walls.

Other papers that discuss the DLO procedure are listed below:

[Gilbert et al. \(2009\)](#) Plastic Limit Analysis using Discontinuity Layout Optimization (DLO).

[Babiker et al. \(2010\)](#) Application of discontinuity layout optimization to problems involving non-associative friction.

[Gilbert et al. \(2010\)](#) Application of discontinuity layout optimization to geotechnical limit analysis problems.

[Smith & Gilbert \(2010a\)](#) Advances in Computational Limit State Analysis and Design.

[Smith & Gilbert \(2010b\)](#) Ultimate Limit State design to Eurocode 7 using numerical methods.

Appendix B

Menu and Toolbar Reference

B.1 General

B.1.1 Scrollbars

Vertical and horizontal scrollbars allow the display area to be shifted in the vertical and horizontal sense respectively.

B.1.2 Current Mouse Position

The coordinates of the mouse are shown in the bottom right hand corner of the screen on the **Status Bar** and also in the top left hand corner of the **Viewer Pane**.

B.1.3 Scrolling Wheels

Many mice are now equipped with a third button that is often used for scrolling whilst browsing the internet. LimitState:GEO makes use of this additional functionality by allowing the user to pan and zoom the display:

- To **pan**, simply press and hold the third button whilst in the display window. Moving the mouse will now pan the image around the screen.
- To **zoom**, simply roll the wheel up to zoom out and down to zoom in.

The pan and zoom functions are also accessible via the menu system described in Section B.3.1 and via keyboard shortcuts (see Section 3.7 and Section 3.8).

B.2 Menus

B.2.1 File Menu

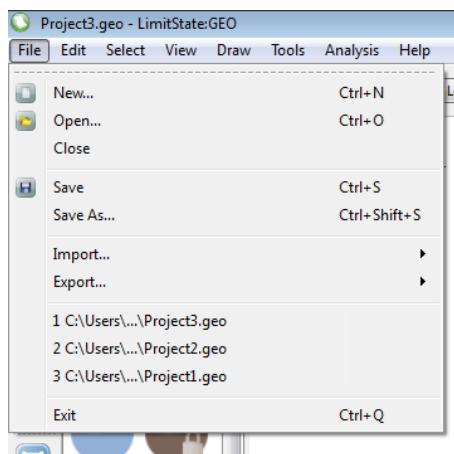


Figure B.1: The LimitState:GEO File menu

Function	Shortcut	Description
New	Ctrl+N	Create a new project.
Open	Ctrl+O	Open an existing project.
Close		Close the current project.
Save	Ctrl+S	Save the current project.
Save As	Ctrl+Shift+S	Save the current project under a specified name.
Import...		Import predefined <i>Materials</i> (Section 17.11) or problem <i>Geometry</i> (Section 15.4)
Export...		Export problem <i>Materials</i> (Section 17.11) <i>Geometry</i> (Section 25.2) or an <i>Image</i> (Section 25.3) or <i>Animation</i> (Section 25.4) of the model.
Open recent file		Open one of the 5 most recently accessed files.
Exit	Ctrl+Q	Exit LimitState:GEO.

Table B.1: File menu functions

B.2.2 Edit Menu

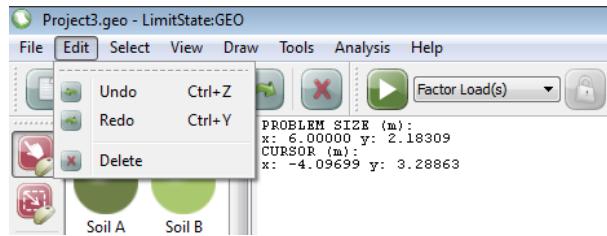


Figure B.2: The LimitState:GEO Edit menu

Function	Shortcut	Description
Undo	Ctrl+Z	Step back to the point immediately before the last action was taken.
Redo	Ctrl+Y	Redo a previously undone action.
Delete		Delete the selected object(s).

Table B.2: Edit menu functions

B.2.3 Select Menu

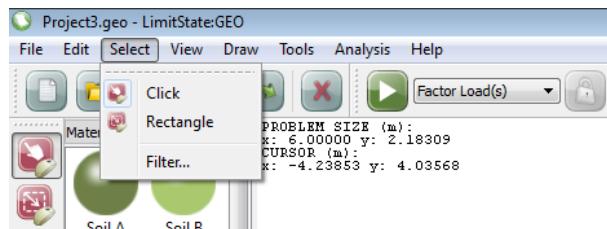


Figure B.3: The LimitState:GEO Select menu

Function	Shortcut	Description
Click		Select a single object (Section 15.7.1).
Rectangle		Select objects within a rectangular area (Section 15.7.2).
Filter...		Brings up the Selection Filter dialog (Section 15.7.2), allowing the user to select only <i>Vertices</i> , <i>Boundaries</i> , <i>Solids</i> , <i>Slip-lines</i> or a combination of these.

Table B.3: Select menu functions

B.2.4 View Menu

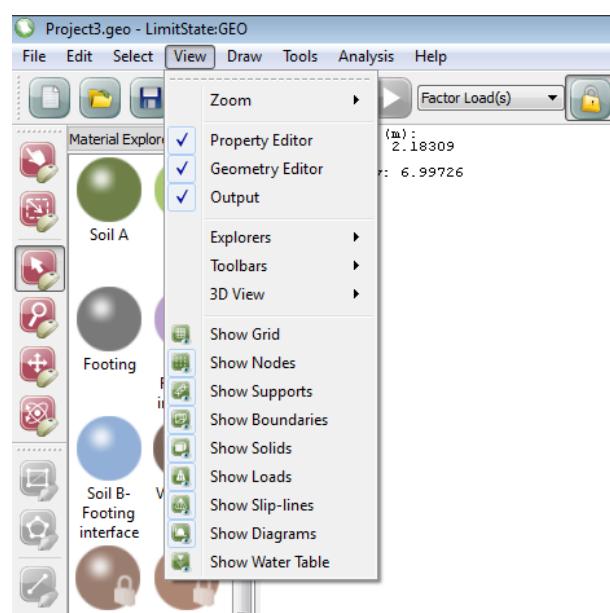


Figure B.4: The LimitState:GEO View menu

Function	Shortcut	Description
Zoom ►		Access the zoom functions (Section 3.7).
Property Editor		Toggle the display of the Property Editor (Section 3.10 / Section 14.8).
Geometry Editor		Toggle the display of the Geometry Editor (Section 14.9).
Output		Toggle the display of the Output pane (Section 14.10).
Explorers ►		Toggle the display of the available Explorers (Section 14.12).
Toolbars ►		Toggle the display of the available Toolbars (Appendix B.3).
3D View ►		Choose from a variety of pre-defined 3D camera viewpoints or toggle the display of the global axis icon.
Show Grid		Toggle the display of the Grid (Section 15.3).
Show Nodes		Toggle the display of Nodes (Solve or Preview Nodes must be executed before any effect is seen).
Show Supports		Toggle the display of Supports (i.e. Boundary Conditions) (Chapter 18).
Show Boundaries		Toggle the display of Boundaries and Vertices (Section 15.1).
Show Solids		Toggle the display of Solids (Section 15.1).
Show Slip-lines		Toggle the display of Slip-lines (Section 23.3. Note Solve must be executed before any effect is seen).
Show Diagrams		Toggle the display of post-analysis pressure and force diagrams (Section 23.4).
Show Water Table		Toggle the display of the Water Table (Section 20.3).

Table B.4: View menu functions

B.2.5 Draw Menu

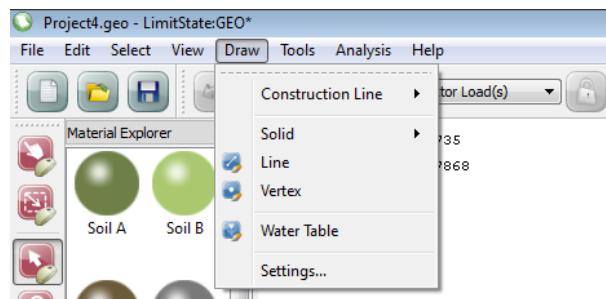


Figure B.5: The LimitState:GEO Draw menu

Function	Shortcut	Description
Construction Line ►		Display the Construction Line sub-menu (Section 15.5). From here you can <i>Draw</i> new construction lines or <i>Clear</i> the current set.
Solid ►		Display the Solid drawing sub-menu (Section 15.6). From here you can create new <i>Rectangle</i> or <i>Polygon</i> solids.
Boundary		Draw a Boundary line linking two existing Vertices or Boundaries (Section 15.6.3).
Vertex		Add a Vertex to an existing Boundary line (Section 15.6.4).
Water Table		Draw a Water Table (Section 20.3).
Settings...		Open the Change Snap and Grid Settings dialog (Section 15.3.2.1).

Table B.5: Draw menu functions

B.2.6 Tools Menu

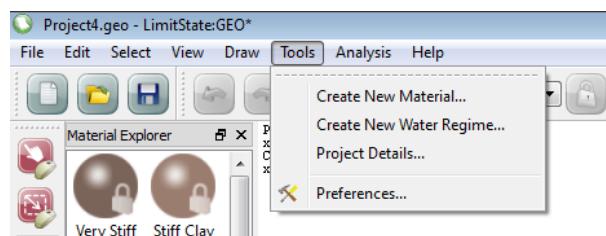


Figure B.6: The LimitState:GEO Tools menu

Function	Shortcut	Description
Create New Material...		Open the Create New Material dialog (Section 17.10).
Create New Water Regime...		Open the Create New Water Regime dialog (Section 20.5).
Project Details...		Open the Project Details dialog (Section 15.3.1).
Preferences...		Open the Preferences dialog, where project default settings can be specified (Section 28.1).

Table B.6: Tools menu functions

B.2.7 Analysis Menu

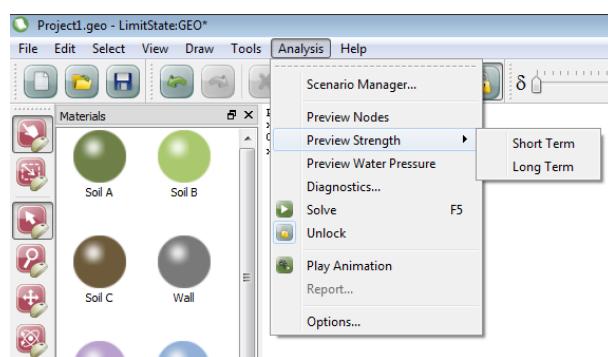


Figure B.7: The LimitState:GEO Analysis menu

Function	Shortcut	Description
Scenario Manager		Open the Scenario Manager dialog (Section 30).
Preview Nodes		Preview nodes at the current nodal density setting without solving.
Preview Strength ►		Preview the distribution of cohesive strength (c_u or c') in the model (Section 17.12).
Preview Water Pressure		Preview the distribution of water pressures in the model. (Section 20.6).
Diagnostics...		Open the Diagnostics dialog (Section 22.2.1).
Solve	F5	Analyse the current problem (Section 22.3).
Unlock		Unlock the project for editing.
Play Animation		Animate the critical failure mechanism post solve (Section 23.2).
Report...		Open the analysis Report dialog (Chapter 24).
Options...		Open the Analysis Options dialog (Section 8.2.3).

Table B.7: Analysis menu functions

B.2.8 Help Menu

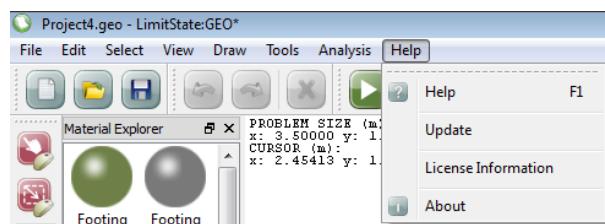


Figure B.8: The LimitState:GEO Help menu

Function	Shortcut	Description
Help	F1	Open the Help dialog (Section 1.5).
Update		Check for updates to the software.
License Information...		Open the License Information dialog, which displays details of the license being used and allows the user to change to an alternative license (Chapter 29).
About		Display LimitState:GEO version details.

Table B.8: Help menu functions

B.3 Toolbars

B.3.1 Default Toolbars

By default, the following toolbars are displayed when LimitState:GEO is opened:

Toolbar	Functions
File	New / Open / Save (Appendix B.2.1).
Edit	Undo / Redo / Delete (Appendix B.2.2).
Cursor	Click / Rectangle / Select (Section 15.7). Zoom / Pan / Rotate 3D (Section 3.7 / Section 3.8).
Draw	Rectangle / Polygon / Line / Vertex (Section 15.6). Horizontal Construction Line / Vertical Construction Line / Custom Construction Line (Section 15.5). Water Table (Section 20.3).
Analysis	Solve / Unlock (Chapter 22). Analysis Type (Section 8.2).
Animation	Magnify Displacements / Play Animation (Section 23.2).
Zoom	Zoom All / Zoom In / Zoom Out (Section 3.7).
Show	Show Construction Lines / Show Grid (Section 15.5). Show Nodes / Show Supports / Show Boundaries (Chapter 18 / Section 15.1). Show Solids (Section 15.1) / Show Loads (Chapter 19) / Show Slip-lines (Section 23.3). Show Water Table (Section 20.3) / View 3D.

Table B.9: Default LimitState:GEO toolbars

B.3.2 Optional Toolbars

To access some of the less commonly utilized features of LimitState:GEO, it may be necessary to open a separate toolbar. To do this, click **View** and select **Toolbars**. The option to open the following is now available:

Toolbar	Functions
Rotate 3D	Rotate the problem in 3D or about different axes: Rotate 3D / Rotate About x / Rotate About y / Rotate About z
View 3D	View the problem from different angles: Top / Bottom / Left / Right / Front / Back
Help	Access: Help (Section 1.5) / About

Table B.10: Optional LimitState:GEO toolbars

B.4 Context Menus

Depending upon the position of the cursor, right clicking the mouse within the LimitState:GEO environment will bring up one of several context menus:

B.4.1 Viewer Pane Context Menu

Right-clicking within the **Viewer Pane** will bring up the following context menu:

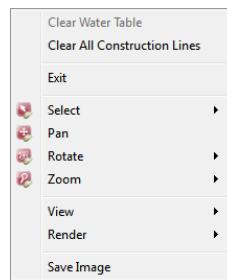


Figure B.9: Viewer Pane context menu

From here, you may many of the display-related functions of the toolbars may be easily accessed, as well as several other independent functions:

Option	Functions
Clear Water Table	Removes any user-defined water table from the model (Section 20.3). Note that this does not remove water regimes from zones or change whether water is <i>Enabled</i> in the Property Editor .
Clear All Construction Lines	Removes all construction lines from the model (Section 15.5).
Exit	Exits the context menu.
Select	Switch between <i>Click</i> and <i>Rectangle</i> object selection (Section 15.7).
Pan	Pan the model in the Viewer Pane (Section 3.8).
Rotate	Rotate the view - either freely or about Cartesian axes (Appendix B.3.2).
Zoom	Activate Zoom mode or access the Zoom functions (Section 3.7).
View	Select a pre-defined 3D viewpoint, toggle 3D mode or toggle the Global Axis markers.
Render	Switch the quality of the rendering between <i>Normal</i> , <i>Low</i> and <i>High</i> resolution and <i>Rendered</i> or <i>Wireframe</i> modes. <i>High Render</i> resolution provides a crisper view but may slow the editing process on older or less powerful machines.
Save Image	Save the current view in raster or vector format (Section 25.3).

Table B.11: Viewer Pane context menu

B.4.2 Toolbar / Property Editor Context Menu

Right-clicking within any toolbar or **Property Editor** will display the menu shown in Figure B.10:

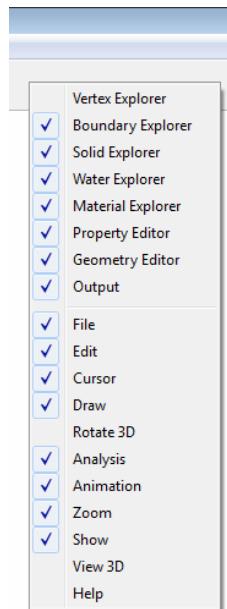


Figure B.10: Toolbar / Property Editor context menu

From here, you can toggle the display of the **Property Editor** (Section 14.8), **Output Pane** (Section 14.10), **Explorers** (Section 14.12) and **Toolbars** (Section 14.4).

B.4.3 Geometry Object Explorer Context Menu

Right-clicking within any geometry object explorer will display the menu shown in Figure B.11:

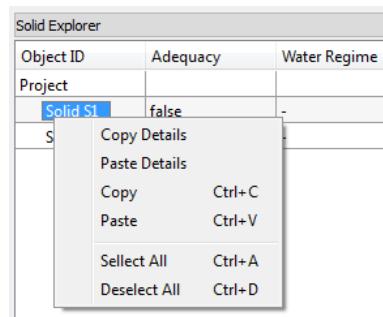


Figure B.11: Object Explorer context menu

From here, the functions described in Table B.12 are available:

Option	Shortcut	Functions
Copy Details		Copy the settings (i.e. row data) for the selected object.
Paste Details		Paste row data for the selected object.
Copy	CTRL+C	Copy data from the selected cells.
Paste	CTRL+V	Paste data to the selected cells (if a single cell is selected it will be treated as the top left cell and the data pasted accordingly).
Select All	CTRL+A	Select the entire table.
Deselect All	CTRL+D	Deselect the entire table

Table B.12: Geometry object Explorer context menu functions

Appendix C

Accessing Example Files

To open the specified example file, locate it in:

[ROOT]/example files/

where **[ROOT]** is the installation path of LimitState:GEO, usually:

C:/Program Files/LimitState/GEO3.6.1 /

Alternatively, open the Example Files html page from the LimitState:GEO program menu and open the file from there:

Start > All Programs > LimitState:GEO 3.6.1 > Example Files.

Appendix D

Derivation of Theory

D.1 Work Done by Cohesion by Rotation of a Log Spiral

The work done W on a 180° arc of a log spiral overcoming cohesion c for a relative body rotation of ω is as follows:

$$W = \int_0^\pi c.ds.(r\omega \cos \phi), \quad (\text{D.1})$$

where ds is the arc length. Thus

$$W = c\omega \cos \phi \int_0^\pi r.ds. \quad (\text{D.2})$$

Now

$$\cos \phi = \frac{r.d\psi}{ds}, \quad (\text{D.3})$$

where ψ is measured clockwise along the arc from its starting point.

Hence

$$W = c\omega \int_0^\pi r^2.d\psi = c\omega \int_0^\pi (r_0 e^{\psi \tan \phi})^2.d\psi \quad (\text{D.4})$$

$$W = r_0^2 c\omega \int_0^\pi e^{2\psi \tan \phi}.d\psi = r_0^2 c\omega \left[\frac{e^{2\psi \tan \phi}}{2 \tan \phi} \right]_0^\pi \quad (\text{D.5})$$

$$W = \frac{r_0^2 c\omega}{2 \tan \phi} \left(e^{2\pi \tan \phi} - 1 \right) \quad (\text{D.6})$$

$$W = \frac{c\omega}{2 \tan \phi} (r_1^2 - r_0^2) = \frac{c\omega}{2 \tan \phi} (r_1 - r_0) l = \frac{c\omega u l^2}{\tan \phi} \quad (\text{D.7})$$

Appendix E

Interpolated Grid

E.1 Grid Square Interpolation

LimitState:GEO has the option to use bilinear grid interpolation to determine strength (Section 17.4.4) and water pressure (Section 20.5.1.3) values throughout a zone.

The interpolation is carried out using grid squares or rectangles on a 2D Cartesian mesh with values defined at corner points. For a grid square with corners (x_1, y_1) – (x_2, y_1) – (x_2, y_2) – (x_1, y_2) (Figure E.1), suppose it is required to determine the value of a function $f(x, y)$, given that the values of f are known at the four corners.

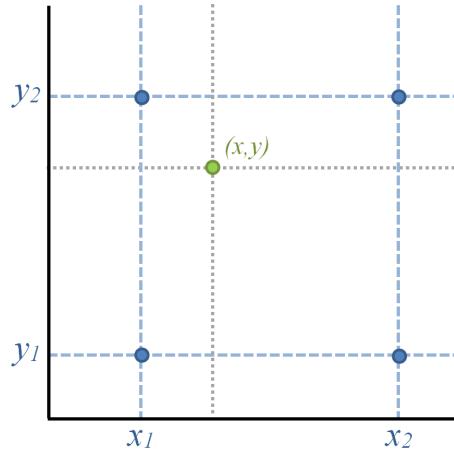


Figure E.1: Bilinear interpolation on a grid to determine the value of $f(x, y)$

Firstly an interpolation is carried out in the x-direction:

$$\begin{aligned} f(x, y_1) &\approx \frac{x_2 - x}{x_2 - x_1} f(x_1, y_1) + \frac{x - x_1}{x_2 - x_1} f(x_2, y_1), \\ f(x, y_2) &\approx \frac{x_2 - x}{x_2 - x_1} f(x_1, y_2) + \frac{x - x_1}{x_2 - x_1} f(x_2, y_2) \end{aligned} \quad (\text{E.1})$$

Secondly, an interpolation is carried out in the y-direction:

$$f(x, y) \approx \frac{y_2 - y}{y_2 - y_1} f(x, y_1) + \frac{y - y_1}{y_2 - y_1} f(x, y_2) \quad (\text{E.2})$$

Substituting the original x-direction interpolation terms (E.1) into E.2 provides the required estimate of $f(x, y)$:

$$\begin{aligned} f(x, y) &\approx \frac{f(x_1, y_1)}{(x_2 - x_1)(y_2 - y_1)} (x_2 - x)(y_2 - y) \\ &+ \frac{f(x_2, y_1)}{(x_2 - x_1)(y_2 - y_1)} (x - x_1)(y_2 - y) \\ &+ \frac{f(x_1, y_2)}{(x_2 - x_1)(y_2 - y_1)} (x_2 - x)(y - y_1) \\ &+ \frac{f(x_2, y_2)}{(x_2 - x_1)(y_2 - y_1)} (x - x_1)(y - y_1) \end{aligned} \quad (\text{E.3})$$

The same result is obtained if the interpolation is carried out in the y and then x-direction. Assuming the following:

$$\hat{x} = \frac{x - x_1}{x_2 - x_1} \quad (\text{E.4})$$

$$\hat{y} = \frac{y - y_1}{y_2 - y_1} \quad (\text{E.5})$$

E.3 can be re-written so that E.4 and E.5 each lie between 0 and 1:

$$f = f(x_1, y_1)(1 - \hat{x})(1 - \hat{y}) + f(x_2, y_1)\hat{x}(1 - \hat{y}) + f(x_1, y_2)(1 - \hat{x})\hat{y} + f(x_2, y_2)\hat{x}\hat{y} \quad (\text{E.6})$$

which can be restated as:

$$f = C_1 + C_2(x - x_1) + C_3(y - y_1) + C_4(x - x_1)(y - y_1) \quad (\text{E.7})$$

where

$$C_1 = f(x_1, y_1) \quad (\text{E.8})$$

$$C_2 = \frac{f(x_2, y_1) - f(x_1, y_1)}{x_2 - x_1} \quad (\text{E.9})$$

$$C_3 = \frac{f(x_1, y_2) - f(x_1, y_1)}{y_2 - y_1} \quad (\text{E.10})$$

$$C_4 = \frac{f(x_1, y_1) - f(x_2, y_1) - f(x_1, y_2) + f(x_2, y_2)}{(x_2 - x_1)(y_2 - y_1)} \quad (\text{E.11})$$

.

Appendix F

Command-Line Interface

F.1 Introduction

The LimitState:GEO command-line interface add-in allows users to run the software and / or change problem parameters from a command prompt or batch file. This is especially useful when conducting parametric studies, generating interaction diagrams or when needing to run a sequence of analyses on a PC unattended (e.g. if models are particularly large).

Parametric studies With the command-line interface, users can modify a wide range of parameters in a single LimitState:GEO model without the need to manually edit a file or group of files. This allows parametric studies to be conducted quickly and with minimum user intervention. Full reports and solution files are automatically generated, as well as individual input files corresponding to each parameter change, so the results can be queried in detail.

Unattended analyses When a large number of analyses need to be run, and the operator can't be present to inspect and/or save the results, a simple batch file can be created to run LimitState:GEO command-line instructions, and to undertake a sequence of unattended analyses.

F.2 License Requirements

In order to run LimitState:GEO from the command-line, the user must first have access to a license that enables the use of this mode. To check that the type of license held allows command-line usage, go to **Help > License information** and see that the text of the license includes:

```
options="command_line"
```

Note that other options, relating to other add-in modules, may also be enabled.

F.3 File Format

LimitState:GEO project files (.geo) are saved in compressed Extensible Markup Language format (XML - [see here](#)). As such, they can be uncompressed (using a tool installed with LimitState:GEO) and the contents inspected in order to determine the descriptors used to assign properties to different materials, geometry objects etc. Figure F.1 shows an example section of XML from a LimitState:GEO file, describing a material (Soil A):

```
<geo_PhysicalDefinition material="44" isReadOnly="false">
<lo_PhysicalDefinition>
<eng_PhysicalDefinition isUsed="true" color="110, 128, 71, 255">
<clf_Object k="43" P="1" C="Soil A"/>
</eng_PhysicalDefinition>
</lo_PhysicalDefinition>
<geo_MaterialMohrCoulomb drainageBehaviour="Always undrained" drainedCohesion="0" drainedPhi="30" tanPhiMultiplier="1"
undrainedCohesion="25" undrainedPhi="0" datumDepthForCohesion="0" cohesionMultiplier="1" cohesionGradient="0">
<geo_Material physicalDefinition="43" saturatedUnitWeight="20">
<ult_Material diagramAfterClick="2" uW="20">
<clf_Object k="44" C="Soil A"/>
</ult_Material>
</geo_Material>
</geo_MaterialMohrCoulomb>
</geo_PhysicalDefinition>
```

Figure F.1: Definition of a LimitState:GEO material in uncompressed XML.

It can be seen that the properties of ‘Soil A’ are set using simple descriptors such as ***undrained-Cohesion*** or ***saturatedUnitWeight***. These can be used in the command-line interface when modifying the properties they are associated with. Short versions of the most commonly-used of these descriptors (e.g. ***cu*** or ***satuw***) are also available - these are outlined in Section F.9.

F.3.1 Uncompressing Files

To uncompress a LimitState:GEO file, select and right click with the mouse. The option to **Convert to XML** will be displayed in the context menu. Select this and the file will be converted and a copy saved in the same location with the suffix ‘UnCompressed’ e.g. ‘testFile.geo’ will be uncompresssed and saved as ‘testFileUnCompressed.geo’. The file can then be read into any text editing software (e.g. MS Notepad).

Alternatively, files can be uncompresssed from the command line (see Section F.9.1).

Note that:

1. LimitState:GEO files cannot be uncompresssed using standard compression software.
2. A license is not required in order to access the convert functionality.

F.4 Input

Running LimitState:GEO from the command-line requires a valid LimitState:GEO (.geo) input file.

F.5 Units

LimitState:GEO files store all data in metric units (metres, kilonewtons etc.), irrespective of the unit system used in GUI mode. Therefore, any modifications made to the input must be carried out using metric units.

F.6 Output

The output from running a LimitState:GEO analysis from the command-line includes on-screen information and a number of files which can be enabled or disabled as required.

F.6.0.1 On-Screen Output

Typical on-screen output from the command-line is shown in Figure F.2. Note that verbose output will be given if this is set in GUI mode, otherwise, brief solve information will be displayed.

```

C:\Windows\system32\cmd.exe
Opened file test.geo at 09:53:18 on Wed Dec 19 2012
Diagnostic information messages: 2
Solving problem containing 511 nodes and 44341 potential inter-node connections at 09:53:19 on Wed Dec 19 2012...
Scen. P.F. Iterat. Var. Con. A.L. Viol% Var add Con add Time
1 Unity 1 10612 5999 1.139 42.4798 173? 529 0.234
1 Unity 2 12349 6578 1.101 15.9552 1371 457 0.579
1 Unity 3 13720 7835 1.094 6.25836 120 40 0.532
1 Unity 4 13840 7075 1.094 1.53884 21 ? 0.546
1 Unity 5 13861 7082 1.094 0 0 0 0.546
1 Unity 6 fltr 13861 7082 1.094 0 435 307 0.592
Adequacy factor on applied load(s) is 1.094
Press enter to continue...

```

Figure F.2: Typical LimitState:GEO command-line output (verbose).

Diagnostic messages Informs the user how many diagnostic ‘information’ or ‘warning’ messages are encountered before solve. If an ‘error’ is encountered then the file will not solve. If the file is being solved as part of a batch command, the next command in the list will be executed [shown by default].

Scen. The scenario number [shown by default].

P.F. The partial factor set being used [shown by default].

Iterat. The iteration number. Numbers with ‘fltr’ after them signify that the iteration includes a step to filter crossover points between slip-lines and add a new node at this position.

Var. The number of variables in the LP matrix at this iteration.

Con. The number of constraints in the LP matrix at this iteration.

A.L. The calculated adequacy factor on load(s) [shown by default for a factor load(s) analysis].

A.S. The calculated adequacy factor on strength(s) [shown by default for a factor strength(s) analysis].

Viol% The maximum percentage violation of the slip-line adding condition at this iteration.

Var add The number of variables added to the LP matrix at this iteration.

Con add The number of constraints added to the LP matrix at this iteration.

Time The time taken (in seconds) to solve this iteration.

F.6.0.2 File Output

By default, if running the file ‘test.geo’ without any modifications to the model properties, the output will be:

test.txt A text file containing the solution output from the analysis. All data that would normally appear in the Output window when in GUI mode will be written to this file. In the event of a crash, all data up to that point will be saved. Note, however, that the data on the file is overwritten on each Solve.

test.rtf An editable report corresponding to the analysis (the content included in the report can also be specified by the user).

If running ‘test.geo’ *with* modifications to properties, a LimitState:GEO file containing the modifications will also be output by default:

test_<SUFFIX>.geo A LimitState:GEO file including any modifications made.

<SUFFIX> will be replaced by automatically generated text describing to any modifications made *or* suffix text specified by the user as a command-line argument.

Additionally, but not by default, a comma separated solution file may also be output. This will aggregate the relevant analysis output (date and time, solution, input file name, modifications) from one or more analyses if specified:

Note that:

- The .geo, .txt and .rtf output files will be saved in the same directory as the original input file, *not* the directory from which LimitState:GEO is called. Write permissions are required in this output directory, otherwise output will not be saved.
- All paths containing a space should be enclosed in quotation marks when entered into the command-line interface.
- Once an analysis has been initiated, the user is free to use the command-line for other tasks. The analysis will continue to run and output will be sent to the screen and the solution files saved as normal.

F.7 Running

To run LimitState:GEO from the command-line, first open a command prompt.

All calls to run the software from the command line must be made from within the ‘bin’ folder in the LimitState:GEO installation directory (if the full path is to be omitted) or include the relative or full path to the executable from the current directory. Similarly, filenames must be preceded by the path if not present in the calling directory.

Note that LimitState:GEO is not added to the ‘Path’ in the Environment Variables of the machine it is installed on. Adding it will allow command line calls to be made without the requirement to include the full path to the executable file in the call.

F.8 Help

To access the command-line interface help documentation, open a command-line window, navigate to the directory containing the LimitState:GEO executable and enter:

```
> geo -help
```

F.9 Syntax

The syntax for running LimitState:GEO via the command-line takes the following form:

```
> <APPLICATION> [OPTIONS] <INPUT_FILE>
```

Where:

< APPLICATION > is the path to, and name of, the LimitState:GEO executable file.

[OPTIONS] are optional parameters that influence the nature and output of the analysis.

[INPUT_FILE] is the name of the LimitState:GEO input file.

A typical command-line call might read:

```
> geo -property:undrainedCohesion:44=30 -execute -suffix:"_my_test" test.geo
```

or, using short commands:

```
> geo -p:cu:44=30 -x -sf:"_my_test" test.geo
```

Entering either of these commands would execute test.geo with the undrained cohesion (c_u) of a material set to 30kPa. The output files will be saved with the suffix “_my_test”. Object keys are discussed further in Section F.9.2.

F.9.1 Options Syntax

After specifying the application name, the user has a number of options to choose from. These are primarily concerned with the properties of the file during analysis and the output generated after it is analysed. Each option has a long and a short A full list of available options is given in Table G.1:

Option	Short	Default?	Description
-open	-o	*	Opens the specified file.
-property	-p		Modify a property of the problem.
-execute	-x		Solve the problem.
-save	-s	* (if file modified)	Save the (modified) input file.
-no_save	-ns		DO NOT save the input file.
-report	-r	* (if file modified)	Save the report output (.rtf) from the analysis.
-no_report	-nr		DO NOT save the report output from the analysis.
-log	-l	* (if file modified)	Save an output file (.txt) containing the analysis data.
-no_log	-nl		DO NOT save an output file containing the analysis data.
-sol_file	-sl		Save a solution file (.csv) containing the most important details from one or more analyses.
-suffix	-sf		Add user-defined suffix text to the files output from the analysis.
-version	-v		Displays the version number of the copy of LimitState:GEO that is being used.
-uncomp	-u		Uncompress the named LimitState:GEO file.
-comp	-c		Compress the named LimitState:GEO file.

Table F.1: Command-line options.

F.9.1.1 The Solution File

The **-sol_file** (-sf) option specifies that a comma-separated value (.csv) solution file containing the most relevant analysis data is to be created (or added to if it already exists). The data included in the file is:

Date and time The date and time that the analysis was started

Adequacy factor The calculated adequacy factor for the solved problem

File name The name of the solved file

Modifications A list of the modifications made to the file

The solution file will, by default, be saved in the directory where the LimitState:GEO executable lies. This allows data from multiple input files to be collated in an easily identified location (especially useful if running a batch file). The user can specify an alternative location for the solution

file if required (see below). Write permissions are required in the save directory, otherwise output will not be saved.

Saving a solution file requires a file name to be specified, the syntax is as follows:

```
-sol_file:<SOLUTION_FILE_NAME>
```

Where <SOLUTION_FILE_NAME> is the path and name of the solution file (including the .csv extension). For example, to execute a file named ‘test.geo’ and log the analysis in "C:\Users\AnneEngineer\My Documents\geo_log.csv" enter:

```
> geo -x -sl:"C:\Users\AnneEngineer\My Documents\geo_log.csv" test.geo
```

If the solution file does not already exist, it will be created containing a header line describing the contents of the file.

F.9.2 Object Keys

Modifying any property relies on knowing the appropriate **Object Key**. These, along with access to the command-line functionality, are available to users holding an appropriate license.

Object Keys are unique numerical identifiers for the individual parts (materials, geometry objects etc.) of a LimitState:GEO project that, when used on the command-line, will allow modification of the value for a particular property of said part. Project-level properties can also be edited using the command-line interface but require no **Object Key**.

The syntax for modifying any property with an object key is:

```
-p:<PROPERTY>[:<KEY>]=<VALUE>
```

For example, revisiting the command-line call from Section F.9:

```
> geo -p:cu:44=30 -x -sf:_my_test test.geo
```

Here, the undrained cohesion (c_u) of the material with **Object Key** 44 is modified.

Object Keys are displayed in the **Property Editor** when in GUI mode. They are hidden by default - to show them, go to **Tools > Preferences > General** and select the option to **Show additional Property Editor attributes**. Figure F.3 shows the **Object Keys** displayed in the **Property Editor** for a selected **Solid** object. Note that the **Materials** field for the object is expanded, showing the **Object Key** associated with the material present in the **Solid**.

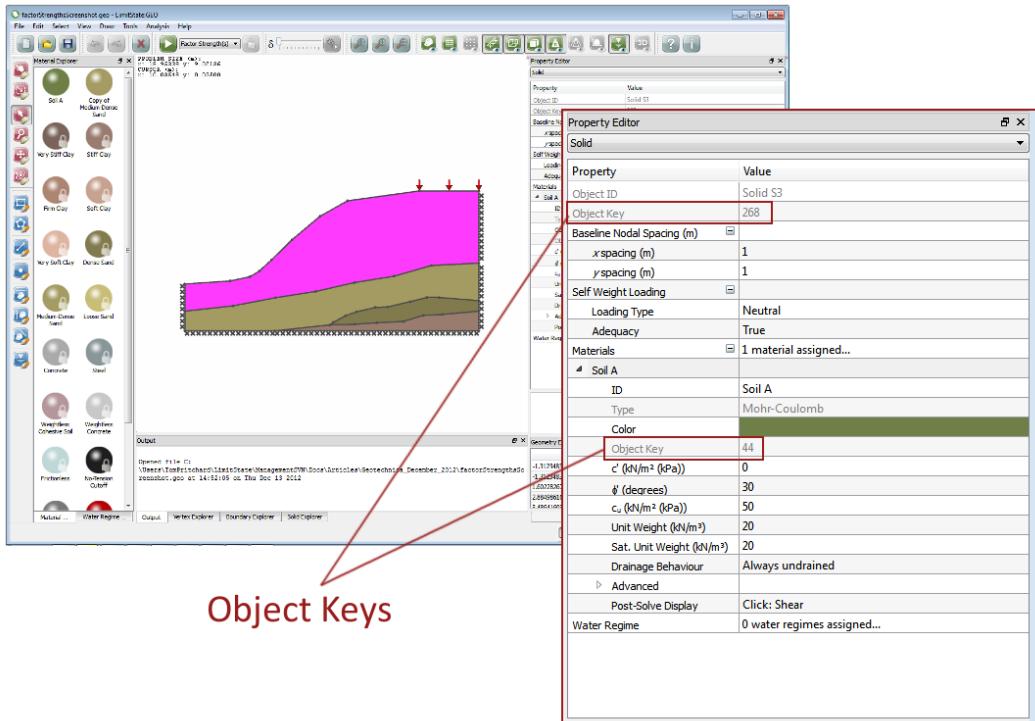


Figure F.3: Object keys for a selected solid and it's constituent material, displayed in the Limit-State:GEO **Property Editor**

When directly examining the XML code for a file, the object keys are specified by the text as:

```
k=""
```

For example:

`k="268"` is the key for the solid highlighted in Figure F.3

The properties that can be altered using **Object Keys** fall in to the following categories:

- Project properties
- Scenario properties
- Material properties
- Water regime properties
- Loading
- Geometry

F.10 Properties

The following sections detail some of the more commonly used properties that can be modified using the command-line. Many have short codes that are used to represent longer property names as seen in the XML. Not all properties are listed - others can be identified by examining the uncompressed XML directly.

F.10.1 Project Properties

In GUI mode, the project-level **Object Key** will be displayed in the **Property Editor** but it is not necessary to use this with command-line options in order to force a change.

Short Name	XML Property Name	Allowed Values	Description
refinement	refinement	Any number	Determines how closely the nodes are spaced during an analysis.
rotate	rotationFlag	0 or 1	Specify whether or not rotational failure mechanisms should be modelled. 0=Translational failures only, 1=Rotational and translational failures.
water	isWaterTableActive	true / false	Indicates whether water is taken into account during the analysis.
seisacch	seismicHorizAccel	Any number	Horizontal base acceleration (units: g). Specify a positive value for a left-to-right base acceleration.
seisaccv	seismicVertAccel	Any number	Vertical base acceleration (units: g). Specify a positive value for an upward base acceleration.
seisaqv	seismicVertAccelAdequacy	true / false	Whether or not the adequacy factor is to be applied to the specified vertical base acceleration.
seisaqh	seismicHorizAccelAdequacy	true / false	Whether or not the adequacy factor is to be applied to the specified horizontal base acceleration.

Table F.2: Common properties accessible when modifying Project properties using the command-line interface.

F.10.2 Scenario Properties

In GUI mode, the **Object Key** associated with each **Scenario** will be displayed in the relevant heading in the **Scenario Manager** table.

Note that a new **Scenario** is only created once the **OK** button has been pressed. Therefore, to determine the **Object Key** for a newly-created **Scenario**, it is necessary to press **OK** and then re-open the **Scenario Manager** dialog.

Short Name	XML Property Name	Allowed Values	Description
drained	isLongTerm	True / False	When a long term (typically drained) analysis is being performed, this should be set to 'True'.

Table F.3: Common properties accessible when modifying Scenario properties using the command-line interface.

F.10.3 Material Properties

In GUI mode, the **Object Key** associated with each material can be accessed in the **Property Editor** by clicking the appropriate icon in the **Materials Explorer** or by selecting a **Boundary** or **Solid** object containing the material of interest.

Note that, when examining the XML source, the **Object Key** that needs to be used to affect any material modifications is that associated with the `<eng_physicalDefinition>`. For example, when modifying the material 'Soil A' described in Figure F.1, the **Object Key** required is `k="43"`, eg:

```
<eng_PhysicalDefinition isUsed="true" color="110, 128, 71, 255">
clf_Object k="43" P="1" C="Soil A"/>
</eng_PhysicalDefinition>
```

F.10.3.1 Mohr-Coulomb Materials

Short Name	XML Property Name	Allowed Values	Description
cdash	drainedCohesion	Any number	c' , drained cohesion intercept.
phidash	drainedPhi	Any number	ϕ' , drained angle of shearing resistance.
cu	undrainedCohesion	Any number	c_u , undrained shear strength.
uw	uW	Any number	Unit weight of material located above the water table and in a zone where no water regime is applied .
satuw	saturatedUnitWeight	Any number	Unit weight of material located below the water table or within a zone containing a water regime.
drain	drainageBehaviour	"Drained/undrained", "Always drained", "Always undrained"	Determines the behaviour of the material in long and short term analysis modes.

Table F.4: Common properties accessible when modifying Mohr-Coulomb materials using the command-line interface.

F.10.3.2 Derived Mohr-Coulomb Properties

Short Name	XML Property Name	Allowed Values	Description
master	master	Any valid Object key	The Object Key of the Mohr-Coulomb material that is to act as the parent material.
cmult	CohesionMultiplier	Any number	The factor by which parent c_u and c' values are multiplied by for this material.
phimult	tanPhiMultiplier	Any number	The factor by which parent $\tan\phi'$ values are multiplied by for this material.
drain	drainageBehaviour	"Drained/undrained", "Always drained", "Always undrained"	Determines the behaviour of the material in long and short term analysis modes.

Table F.5: Common properties accessible when modifying Derived Mohr-Coulomb materials using the command-line interface.

F.10.3.3 Cutoff Material Properties

Short Name	XML Property Name	Allowed Values	Description
csig	limitingCompressiveStress	Any number	Limiting compressive stress.
tsig	limitingTensileStress	Any number	Limiting tensile stress.
uw	uW	Any number	Unit weight of material located above the water table and in a zone where no water regime is applied.
satuw	saturatedUnitWeight	Any number	Unit weight of material located below the water table or within a zone containing a water regime.

Table F.6: Common properties accessible when modifying Cutoff materials using the command-line interface.

F.10.3.4 Rigid Material Properties

Short Name	XML Property Name	Allowed Values	Description
uw	uW	Any number	Unit weight of material located above the water table and in a zone where no water regime is applied.
satuw	saturatedUnitWeight	Any number	Unit weight of material located below the water table or within a zone containing a water regime.

Table F.7: Common properties accessible when modifying Rigid materials using the command-line interface.

F.10.3.5 Engineered Element Properties

Short Name	XML Property Name	Allowed Values	Description
tc	slipFactor	Any number	Pullout Factor, T_c used to compute the pullout resistance (T) of the element.
tq	slipFactorStressMultiplier	Any number	Pullout Factor, T_q used to compute the pullout resistance (T) of the element.
nc	bearingFactor	Any number	Lateral Factor, N_c used to compute the resistance (N) of the element to lateral displacement.
nq	bearingFactorStressMultiplier	Any number	Lateral Factor, N_q used to compute the resistance (N) of the element to lateral displacement.
mp	mp	Any number	Plastic moment of resistance.
tlim	tensileLimit	Any number	R , rupture strength. Maximum tensile force that can be carried by the element.
clim	compLimit	Any number	C , undrained shear strength. Maximum compressive force that can be carried by the element.
subdiv	isFailureBetweenVertices	true, false	Specify whether or not to automatically subdivide the Engineered Element at intermediate nodes. (ADVANCED)

Table F.8: Common properties accessible when modifying Engineered Element materials using the command-line interface.

F.10.4 Water Regime Properties

In GUI mode, the **Object Key** associated with each water regime can be accessed in the **Property Editor** by clicking the appropriate icon in the **Materials Explorer** or by selecting a **Solid** object containing the regime of interest.

Note that, when examining the XML source, the **Object Key** that needs to be used to affect any water regime modifications, like for materials, is that associated with the `<eng_physicalDefinition>`. For example, consider the following XML code:

```
<eng_PhysicalDefinition isUsed="true" color="215, 253, 253, 255">
  clf_Object k="471" P="1" C="Constant Potential A"/>
</eng_PhysicalDefinition>
```

To modify ‘Constant Potential A’, the **Object Key** required is `k="471"`.

F.10.4.1 Aquiclude Properties

The Aquiclude water regime type contains no commonly modified properties.

F.10.4.2 Constant Potential Properties

Short Name	XML Property Name	Allowed Values	Description
pot	potential	Any number	Potential of water relative to Datum (Datum is defined in the Project level Property Editor).
gammaw	gammaWFactor	Any number	Factor on the unit weight of water. This can be used to model different fluids or model e.g. vertical flow of water.

Table F.9: Common properties accessible when modifying Constant Potential water regimes using the command-line interface.

F.10.4.3 Constant Pressure Properties

Short Name	XML Property Name	Allowed Values	Description
press	constantPressure	Any number	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).

Table F.10: Common properties accessible when modifying Constant Pressure water regimes using the command-line interface.

F.10.4.4 Interpolated Grid Properties

Short Name	XML Property Name	Allowed Values	Description
xorig	xOrigin	Any number	The x position of the origin of the water pressure grid.
yorig	yOrigin	Any number	The y position of the origin of the water pressure grid.
x	x	Any number	The x position of a value in the grid.
y	y	Any number	The y position of a value in the grid.
val	value	Any number	The value of the water pressure.

Table F.11: Common properties accessible when modifying Interpolated Grid water regimes using the command-line interface.

F.10.5 Boundary Object Properties

Loads applied to **Boundary** objects are split into three types - **Permanent**, **Variable** and **Accidental**. Each load has a unique **Object Key** (not to be confused with that of the boundary itself), which can be accessed in the **Property Editor** by clicking the appropriate **Boundary** and expanding the **Load** fields.

Short Name	XML Property Name	Allowed Values	Description
shear	FX	Any number	The stress applied along the local x axis of the Boundary (i.e. the Shear load).
norm	FY	Any number	The stress applied along the local y axis of the Boundary (i.e. the Normal load)
aq	adequacy	true, false	Set this to 'true' to include this load during the computation of the global problem adequacy factor.

Table F.12: Common properties accessible when modifying boundary loading using the command-line interface.

A number of non-loading options are also available:

Short Name	XML Property Name	Allowed Values	Description
resx	ResX	Any number	The baseline nodal spacing between nodes along the Boundary.
supp	SupportType	Free, Fixed, Symmetry	Defines the type of support given to this boundary.

Table F.13: Common properties accessible when modifying boundaries using the command-line interface.

F.10.6 Solid Object Properties

Loading from **Solid** objects is considered to act in the vertical, downwards direction (i.e. with gravity). As with boundary loading, self-weight loading from **Solid** objects can be set as being favourable (increasing the adequacy factor) or unfavourable (decreasing the adequacy factor). The **Object Key** associated with the self-weight loading can be accessed in the **Property Editor** by clicking the appropriate **Solid** and expanding the appropriate **Self-Weight Loading** row.

Short Name	XML Property Name	Allowed Values	Description
aq	adequacy	true, false	Set this to 'true' to include this self-weight during the computation of the global problem adequacy factor.

Table F.14: Common properties accessible when modifying solid self-weight loading using the command-line interface.

A number of non-loading options are also available:

Short Name	XML Property Name	Allowed Values	Description
resx	ResX	Any number	The baseline nodal spacing between nodes in the x direction.
resy	ResY	Any number	The baseline nodal spacing between nodes in the y direction.

Table F.15: Common properties accessible when modifying solid objects using the command-line interface.

F.10.6.1 Geometry Properties

The position of any **Vertex** in the problem can be modified using the command-line. The **Object Key** can be accessed in the **Property Editor** by clicking the appropriate **Vertex**.

Short Name	XML Property Name	Allowed Values	Description
x	x	Any number	Defines the x coordinate of the vertex.
y	y	Any number	Defines the y coordinate of the vertex.

Table F.16: Common properties accessible when modifying solid self-weight loading using the command-line interface.

F.10.7 Examples

To open a file named ‘test.geo’ that is located in the current directory, enter:

```
> geo test.geo
```

To execute a file named ‘test.geo’ that is located in the current directory, but not save the report output, enter:

```
> geo -x -nr test.geo
```

To modify the undrained cohesion of a material in file ‘test.geo’, execute and save the default output files with the suffix ‘_mod_cohesion’, enter:

```
> geo -p:cu:44=35 -x -sf:"_mod_cohesion" test.geo
```

To modify the undrained cohesion of a material in file ‘test.geo’, execute, save the default output files and also output a solution file named ‘solution_file.csv’, enter:

```
> geo -p:cu:44=35 -x -sl:"solution_file.csv" test.geo
```

To modify the value of a constant potential water regime (with key=76) in file ‘test.geo’, execute and save the output files with the default suffix (which would be, in this case, “pot:76=1.5”) enter:

```
> geo -p:cu:76=1.5 -x test.geo
```

To modify the loading on e.g. the footing in a file named ‘footing.geo’, execute and save only the modified input file, enter:

```
> geo -p:shear:100=1.25 -x -nr -nl footing.geo
```

To modify the loading on the footing in a file named ‘footing.geo’, change the support type of the boundary with key=257, execute and save the default output files, enter:

```
> geo -p:shear:100=1.25 -p:supp:257=Fixed -x footing.geo
```

To set the adequacy on the self-weight of a solid body (with key=83) to ‘true’ in file ‘test.geo’ and make the analysis drained, enter:

```
> geo -p:aq:83=true drained=True -x test.geo
```

F.11 Creating and Running a Batch File

In addition to single command-line calls, it is possible to run LimitState:GEO from a batch file. This allows multiple files to be amended and solved without the requirement to be present at the machine.

To create a batch file, open a text editor such as MS Notepad and enter the commands. Save the file with the extension ‘.bat’ while ensuring that the ‘Save type’ is set to ‘All files (*.*)’.

To run the batch file, open a command prompt in the location of the file and enter the name, followed by the .bat extension.

At its simplest, the syntax of a batch file need only resemble a number of individual command-line calls on separate lines, for example, see Figure F.4, which shows a batch file for running ‘test.geo’ with undrained cohesion varying between 30 and 40 kPa in steps of 2kPa:

```
modify_cohesion.bat - Notepad
File Edit Format View Help
geo -property:"cu:44=30" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
geo -property:"cu:44=32" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
geo -property:"cu:44=34" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
geo -property:"cu:44=36" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
geo -property:"cu:44=38" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
geo -property:"cu:44=40" -x -sf:"_mod_cohesion" -s1:"c:\users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
```

Figure F.4: Example batch file modifying the undrained cohesion of a material.

Alternatively, for those familiar with batch file programming, other commands can be introduced. For example, the commands in Figure F.4 can be replaced with a single-line ‘FOR loop’ as in Figure F.5:

```
modify_cohesion.bat - Notepad
File Edit Format View Help
FOR /L %%G IN (30,2,40) DO geo -property:"cu:44=%G" -x -sf:"_mod_cohesion" -s1:"c:\Users\AnneEngineer\My Documents\cohesion_study.csv" test.geo
```

Figure F.5: Example ‘FOR loop’ batch file modifying the undrained cohesion of a material.

Appendix G

Benchmark Solutions

G.1 Introduction

This appendix contains the derivation of selected closed form solutions that are unavailable in the literature, but are used as verification tests (see Appendix A).

G.2 Simple Soil Nail in Retaining Wall (Undrained)

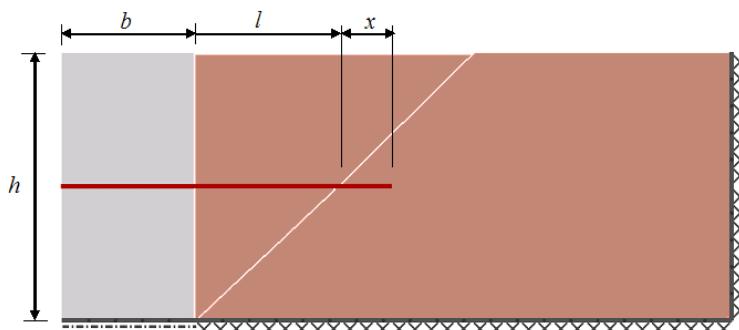


Figure G.1: Reinforcing element (soil nail) in retaining wall (undrained case)

In Figure G.1, let the wall displace to the left by a distance δ . Take the unit weight (weight density) of the soil as γ and the undrained shear strength of the soil be c_u . Let T_w be the pullout resistance per unit length in the wall, N_w the lateral resistance per unit length in the wall, T_s be the pullout resistance per unit length in the soil, and N_s the lateral resistance per unit length in the soil. The wall rests on a smooth foundation and has a smooth interface with the soil. Assume that the failure mechanism consists of a simple 45° sliding wedge as depicted i.e. that the values of T_s and N_s are small enough not to influence the failure mode, and that the reinforcement moves with the wall i.e. that the values of T_w and N_w are large enough to ensure the reinforcement remains locked within the wall.

Work equation:

$$\frac{1}{2}\gamma h^2\delta = c_u \sqrt{2}h\sqrt{2}\delta + xT_s\delta + lN_s\delta \quad (\text{G.1})$$

If the reinforcement is placed at mid depth in the wall then $l = h/2$. Thus:

$$\gamma = 4\frac{c_u}{h} + 2T_s\frac{x}{h^2} + \frac{N_s}{h} \quad (\text{G.2})$$

A selection of sample solutions are presented in Table G.1.

c_u	h	T_s	N_s	x	γ
1.0	1.0	0.0	0.0	0.25	4.0
1.0	1.0	0.1	0.0	0.25	4.05
1.0	1.0	0.1	1.0	0.25	5.05

Table G.1: Solutions to equation G.2

Appendix H

Frequently Asked Questions

1. Is the Adequacy Factor calculated by LimitState:GEO the same as the ‘Factor of Safety’ calculated by other software?

In LimitState:GEO, an Adequacy Factor may be applied to any load or to the self weight of any body of material. The Adequacy Factor that is returned by LimitState:GEO when it has completed solving is the factor by which all the specified loads/self weights must be multiplied by to cause collapse. The Adequacy Factor is similar to a Factor of Safety on load. However in geotechnics, the Factor of Safety is often a value applied to the material strengths (c_u and $\tan\phi$). In this case the Adequacy Factor is not the same as the Factor of Safety.

2. How do I change the accuracy of the solution?

Solution accuracy in LimitState:GEO is modified by changing the nodal resolution. This in turn alters the number and range of the set of potential slip-lines from which the solver will select the critical solution. Increasing the nodal resolution will increase the solution accuracy (except in rare cases - see FAQ later in this section). Further information on changing the nodal resolution may be found in Section 22.3.2.

3. I should get a symmetrical failure mode but I don’t, or the slip-lines are symmetrical but the animation is not.

For certain classes of problems (e.g. certain bearing capacity problems) the plastic collapse load for a symmetrical failure mechanism is identical to an asymmetrical mechanism. In many cases LimitState:GEO will generate the symmetrical failure mechanism, however any asymmetry in the initial set up may skew the result to the asymmetrical case (but generate the same collapse load). Were the problem geometry is symmetrical, the most likely cause of asymmetry will be due to numerical tolerance issues.

4. How accurate is LimitState:GEO?

This is a common issue for numerical analysis codes. Uncertainties exist all the way through geotechnical design calculations from the site investigation data through to construction. The common question at the stage of the numerical calculation is ‘what is the accuracy’. This depends on two issues: (1) the

theoretical model underpinning the numerical analysis (2) the accuracy of the numerical method itself.

The theoretical model underpinning the numerical analysis is the theory of plasticity which has had a long history of application in Geotechnical Design. The majority of text book stability calculations are based on this theory or on simplifications of this theory (see Section 5.2). Selected known issues with limit analysis are discussed in Section 7.

LimitState:GEO is regularly benchmarked against a known set of limit analysis solutions from the literature. These tests are described in more detail in Appendix A and may be accessed via the internet. Reference to these results can provide useful guidance as to the expected accuracy of the software over a range of problem types.

Accuracy does of course depend on nodal resolution. The core analysis method used in LimitState:GEO is based on finding the critical *translational* mechanism that will cause collapse. For a wide range of problem types this leads to good results. For problems where rotational failure mechanisms are likely to be critical (e.g. consider a wall rotating around a pivot), LimitState:GEO currently provides the (optional) capability to model rotations at the edges of pre-defined solid objects. However, though this will lead to a more conservative prediction of the collapse load, as rotations are not permitted to occur within pre-defined solid objects, in certain highly confined problems this may still overestimate the true collapse load significantly. Further details regarding solution accuracy may be found in Section 8.4.

5. I have made a change to the problem geometry that should theoretically have no effect on the collapse load, yet the adequacy factor computed by LimitState:GEO has changed. Why is this?

This is a common issue that relates to the nodal distribution used in the model. It should in general only result in small (a few %) change in adequacy factor. The default setting for LimitState:GEO is to utilise a fixed number of nodes for the problem. If the geometry is altered, for example by deleting a body of soil that plays no part in the collapse mechanism, then the nodes that would have been placed in this body are redistributed elsewhere and should increase the accuracy of the solution (cause a minor reduction in adequacy factor). Nodes are also used on boundaries and with the water table, so modification of these can lead to similar issues. To avoid this issue, change the nodal settings in LimitState:GEO to fixed spacing rather than a constant overall number. For further details concerning setting nodal spacing see Section 22.3.2.

6. I have increased the number of nodes but the adequacy factor has increased rather than decreased.

This will normally only occur when a coarse nodal resolution is being used. With a small increase in the number of nodes, the available slip-line locations may move to slightly less favourable locations for collapse, thus leading to an increase in adequacy factor. A large increase in nodal resolution should lead to the expected decrease in adequacy factor.

7. I am benchmarking LimitState:GEO results against known precise solutions, but am unable to get a good match.

Some problems e.g. the bearing capacity problem in cohesionless soils are extremely sensitive to change in parameters. A change in the angle of shearing

resistance by a few degrees can double the capacity in certain circumstances. The results generated by LimitState:GEO are subject to similar sensitivity. A more logical question to ask is what is the change in input soil strength parameters required to generate the result produced by LimitState:GEO. Considered this way, the discrepancies become much smaller (typically of the order of a few percent). For further discussion of this issue see Section 8.4.

8. Can LimitState:GEO model 1D structural elements or loads?

LimitState:GEO works in general with 2D bodies. To model what is essentially a 1D strut or rod that is able to transmit loads, there are two options: (i) it can be represented as a thin 2D **Solid** (ii) It can be modelled as a **Engineered Element**.

9. What units does LimitState:GEO work in?

LimitState:GEO is designed to work in either Metric or Imperial units and these are displayed in the Wizards, Property Editor, and Reports. If parameters are only known in other units, then the built in **Calculator** provided common conversions for all relevant units.

10. I think I have set the problem geometry and loads up correctly, but the problem will not solve

This is typically due to one of two problems:

- The problem is geometrically locked. Either there is nowhere for the soil to deform into or the soil has a high angle of shearing resistance and no feasible mechanism can be found for the given number of nodes. To correct this problem, check that the boundary conditions are correctly specified and/or increase the number of nodes. The problem can also arise when a slope is modelled that is shallower than the angle of shearing resistance of the soil. In this case no factor on self weight will cause collapse.
- There is no fixed (or possibly symmetry) boundary to which the soil is anchored. Any applied load thus causes the whole soil mass to fly off into space. To correct this problem, check that there is at least one fixed boundary (in certain circumstances a symmetry boundary condition can fulfil this function, in that it is, in effect, a fixed but perfectly smooth boundary). The problem can also arise when a slope is modelled that is steeper than the angle of shearing resistance of the soil. In this case no factor on self weight can prevent collapse.

Further guidance on Troubleshooting problems can be found in Section 8.6.

11. How do I model tension cracks?

LimitState:GEO provides a tension cutoff material type. This can be assigned to either **Boundaries** and **Solids**. A tension cutoff might be added to a **Boundary** of a footing for example to permit breakaway of a cohesive soil from the footing. Assigning tension cutoff to a **Solid** permits tension cracks to form within a body of soil. This might be used for example in a slope stability analysis. Further information on the tension cutoff material type may be found in Section 6.1.3.

12. Can LimitState:GEO model water filled tension cracks?

If a water table lies above any crack that forms, then LimitState:GEO will assume that the crack fills with water and will apply water pressures to the side of the crack. The crack will therefore normally be longer in this case compared to a case with no water present. It is not necessary for the crack to extend to the surface for it to fill with water.

13. Can LimitState:GEO model rotational failures about a single point?

Yes. For example, if you want to model a retaining wall failing by rotation about a single point, LimitState:GEO allows you to do this by placing the retaining wall on a rigid material and applying a 'No-tension cutoff' material in the interface between the wall and the rigid base. Assuming 'Allow rotations' is set to 'Along edges', then this setup will normally ensure a rotational failure mode is predicted.

14. I have checked the free body diagrams in the Report and find that adjacent blocks do not seem to be in equilibrium.

This can occur in rare situations in which a water table is present and in which the **Adequacy factor** is applied to a body self weight in only part of the problem. Where the **Adequacy factor** is applied to the self weight of body, then LimitState:GEO applies it to both the self weight *and* the water pressures in that body. However if the water pressures are factored only in one part of the problem (i.e. where the **Adequacy factor** is applied) these will not be in equilibrium with water pressures in the other part of the problem (unless the **Adequacy factor** = 1.0). For this reason it is not recommended that the **Adequacy factor** is applied to the self weight of isolated bodies. Further discussion of this issue is given in Section 8.2.5.1.

Bibliography

- Babiker, A., Smith, C. & Gilbert, M. 2010 Application of discontinuity layout optimization to problems involving non-associative friction, *Proc. 7th European Conference in Numerical Methods in Geotechnical Engineering*, Trondheim, pp. 45–50.
- BSI 1994 *BS 8002:1994 Code of practice for earth retaining structures*. British Standards Institute.
- Chen, W.-F. 2008 *Limit analysis and soil plasticity*. Elsevier.
- Chen, W.-F. & Liu, X. 1990 *Limit analysis in soil mechanics*. Elsevier.
- Clarke, S., Smith, C. & Gilbert, M. 2010 Analysis of the stability of sheet pile walls using discontinuity layout optimization, *Proc. 7th European Conference in Numerical Methods in Geotechnical Engineering*, Trondheim, pp. 163–168.
- Coulomb, C. A. 1776 Essai sur une application des règles des maximus et minimus à quelque problèmes de statique relatif à l'architecture. *Mémoirs Divers Savants, Académie Science, Paris* 7.
- Gilbert, M., Smith, C. C., Haslam, I. W. & Pritchard, T. J. 2009 Plastic limit analysis using discontinuity layout optimization (DLO), *17th UK Conference on Computational Mechanics (ACME-UK)*, Nottingham.
- Gilbert, M., Smith, C., Haslam, I. & Pritchard, T. 2010 Application of discontinuity layout optimization to geotechnical limit analysis problems, *Proc. 7th European Conference in Numerical Methods in Geotechnical Engineering*, Trondheim, pp. 169–174.
- Gilbert, M. & Tyas, A. 2003 Layout optimisation of large-scale pin-jointed frames. *Engineering Computations* 20(8), 1044–1064.
- Lee, Y., Smith, C. & Cheuk, C. Y. 2008 Bearing capacity of embedded foundations, *2nd International Conference on Foundations, ICOF 2008, Dundee*, pp. 961–972.
- Loukidis, D., Chakraborty, T. & Salgado, R. 2008 Bearing capacity of strip footings on purly frictional soil under eccentric and inclined loads. *Canadian Geotechnical Journal* 45, 768–787.
- Lyamin, A., Salgado, R., Sloan, S. & Prezzi, M. 2007 Two- and three-dimensional bearing capacity of foundations in sand. *Geotechnique* 57(8), 647–662.
- Manzari, M. T. & Nour, M. A. 2000 Significance of soil dilatancy in slope stability analysis. *Journal of Geotechnical and Geoenvironmental Engineering* 126(1), 75–80.
- Matsuzawa, H., Ishibashi, I. & Kawamura, M. 1985 Dynamic soil and water pressures of submerged soils. *ASCE Journal of Geotechnical Engineering* 111(10).

- Merifield, R. & Smith, C. 2010 The ultimate uplift capacity of multi-plate strip anchors undrained clay. *Computers and Geotechnics* **37**(4), 504–510.
- Salgado, R., Lyamin, A., Sloan, S. & Yu, H. 2004 Two- and three-dimensional bearing capacity of foundations in clay. *Geotechnique* **54**(5), 297–306.
- Smith, C. C. & Gilbert, M. 2007b New upper bound solutions for layered soil bearing capacity problems using discontinuity layout optimization, *10th Australia New Zealand Conference on Geomechanics, Brisbane*, pp. 250–255.
- Smith, C. & Gilbert, M. 2007a Application of discontinuity layout optimization to plane plasticity problems. *Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences* **463**(2086), 2461–2484.
- Smith, C. & Gilbert, M. 2008 Limit analysis of the stability of foundations on inclined ground, *2nd International Conference on Foundations, ICOF 2008, Dundee*, pp. 1683–1692.
- Smith, C. & Gilbert, M. 2010a Advances in computational limit state analysis and design, *GeoFlorida 2010: Advances in Analysis, Modeling, & Design, Proceedings of the GeoFlorida 2010 Conference, West Palm Beach*.
- Smith, C. & Gilbert, M. 2010b Ultimate limit state design to Eurocode 7 using numerical methods, *Proc. 7th European Conference in Numerical Methods in Geotechnical Engineering, Trondheim*, pp. 947–952.
- Smith, C. & Gilbert, M. 2011a Ultimate limit state design to Eurocode 7 using numerical methods Part I: methodology and theory. *Ground Engineering* pp. 25–30.
- Smith, C. & Gilbert, M. 2011b Ultimate limit state design to Eurocode 7 using numerical methods Part II: proposed design procedure and application. *Ground Engineering* pp. 24–29.
- Taylor, D. 1948 *Fundamentals of Soil Mechanics*. John Wiley and Sons, New York.

Index

1D loads, 313

1D structural elements, 313

3D, 105

abort, 229

accuracy, 103, 311

add-in modules

 command-line interface, 293

adequacy

 modifying, 150

adequacy factor, 29, 88, 134, 202, 231

 water table, 96, 314

analysis, 29

 changing type, 172

 drained, 171

 factor load(s), 172

 factor strength(s), 172

 long term, 171

 rotations, 172

 short term, 171

 undrained, 171

analysis options, 51

analysis type

 setting in wizard, 51

anchor tendons, 125

animation, 31, 231, 248

 export, 247

auto-recovery, 249

behaviour

 long term, 181

 short term, 181

bending moment, 32, 236

boundary, 87, 153

boundary conditions, 199

boundary effects, 105

boundary explorer, 150

boundary object

 nodal spacing, 225

calculator, 151

command-line interface

about, 293

add-in modules, 293

aquiclude, 304

batch files, 308

boundary object, 305

constant potential, 305

constant pressure, 305

cutoff, 303

derived mohr-coulomb, 303

engineered element, 304

examples, 307

file format, 294

file output, 296

geometry, 307

help, 297

input, 294

interpolated grid, 305

license requirements, 293

material properties, 302

mohr-coulomb, 302

object keys, 299

on-screen output, 295

options syntax, 298

output, 295

project, 301

properties, 301

rigid, 303

running, 297

scenario, 301

solid object, 306

solution file, 298

syntax, 297

uncompressing files, 294

units, 295

 water regimes, 304

connect, 165, 166

construction lines, 161

copy and paste, 152

default adequacy, 91

diagrams, 32

disconnect, 167

display
 force, 236
 pressure, 234
 DLO, 57
 DRAG, 149, 165
 draw, 162
 boundary, 162
 line, 162
 menu, 278
 polygon, 162
 rectangle, 162
 vertex, 162
 dry unit weight, 187
 dxf, 159, 160
 dxf import, 157
 earthquake loading, 76, 131, 217
 edit
 menu, 275
 editor
 geometry, 148
 property, 147
 empty project, 154
 engineered element, 77, 117, 119, 188, 309
 diagrams, 192
 lateral resistance, 193
 moment, 192
 nominal stresses, 193
 pullout, 193
 shear force, 192
 tensile force, 192
 working resistance, 193
 example problems, 111, 287
 explorer
 boundary, 150
 material, 144
 solid, 151
 vertex, 150
 water, 145
 explorers, 150
 export, 160, 245, 248
 animation, 247
 geometry, 245
 image, 246
 scenarios, 266
 factor of safety, 88, 311
 default adequacy, 91
 loads, 90
 material strengths, 91
 solution tolerance, 91
 factor strength(s)
 settings, 172
 favourable, 203
 file
 menu, 274
 properties, 23
 search, 23
 tags, 23
 thumbnails, 23
 force
 shear, 236
 tensile, 236
 force distributions, 236
 geometry, 153
 copy / paste, 148
 export, 245
 setting in wizard, 48
 geometry editor, 148
 geometry object
 nodal spacing, 225
 geometry objects
 boundary, 87, 153
 solid, 87, 153
 vertex, 153
 geometry window, 146
 geosynthetics, 122
 geotextiles, 122–124
 graphical interface, 141
 graphical output, 245
 grid, 155
 help, 20, 280
 Hoek-Brown, 71
 horizontal acceleration, 217
 image, 248
 export, 246
 imperial, 153, 176
 import
 scenarios, 266
 insoluble problem, 227
 insoluble problems, 107
 locked, 107
 unknown, 108
 unstable, 107
 installation, 23
 kinematic constraints, 82
 lens, 167
 license
 changing, 261
 dialog, 261

- licensing, 23
- limit analysis
 - advantages and limitations, 79
- LimitState, 20, 21
- line, 167
- line loads, 313
- load type
 - modifying, 150
- loading type
 - modifying, 150
- loads, 201
 - setting in wizard, 49
- lock, 229
- long term, 265
- long term behaviour, 181

- manage
 - scenarios, 265
- material
 - drag and drop, 144
 - explorer, 144
 - properties, 144
- material explorer, 144
- materials, 67, 175
 - assigning, 177
 - combining, 71
 - create new, 194
 - crushing, 68
 - cutoff, 186
 - deleting, 196
 - derived, 185
 - duplicate, 196
 - engineered element, 69, 77, 117, 119, 188
 - exporting, 197
 - importing, 197
 - linear Mohr-Coulomb, 181, 182
 - Mohr-Coulomb, 67, 178, 179
 - reinforcement, 69, 77
 - rigid, 69, 187
 - setting in wizard, 48
 - soil nail, 69, 77, 188
 - spatially varying Mohr-Coulomb, 183
 - tension cutoff, 68
 - unit weight, 187
 - user-defined, 194
 - zero and negative strength, 182
- menu
 - analysis, 279
 - draw, 278
 - edit, 275
 - file, 274

- help, 280
- select, 275
- tools, 278
- view, 276
- viewer pane, 283
- menu bar, 142
- menus, 274
- merge, 165, 166
- metric, 153, 176
- model definition
 - boundaries, 87
 - solids, 87
- mouse, 273
- multiple scenario, 248
- multiple scenarios, 265

- nodal density
 - setting in wizard, 51
- nodes, 225, 226
 - geometry objects, 225
 - optimization, 226
- non-associativity, 82
- non-yielding zones, 80
- normal displacement, 233
- normal force, 233
- normal stress, 32, 233

- open, 249
- ORTHO, 149, 165
- output
 - adequacy factor, 148
 - copy and paste, 148
 - iteration data, 148
- output pane, 148

- partial factors, 68, 100, 263, 264
 - manage, 265
 - setting in wizard, 50
- post-analysis, 231
- post-solve
 - display context menu, 235, 238
 - display dialog, 234, 238
- preferences, 255
 - export, 260
 - general, 255
 - report, 257
 - solve, 258
 - startup, 256
 - units, 256
- pressure distributions, 234
- previewing strength, 197
- program limits, 20

project details
 setting in wizard, 47
 property editor, 34, 147, 151, 233
 Mohr Coulomb Material, 180
 redo, 168
 report, 241
 customizing, 243
 saving, 243
 viewing, 241
 rotational failure, 60, 105, 172, 314
 saturated unit weight, 187
 save, 249
 scale factor, 225
 scenario manager, 263
 scenarios
 export, 266
 import, 266
 manage, 265
 solving, 266
 scroll wheels, 273
 scrollbars, 273
 search
 project details, 252
 tags, 252
 searching, 252
 Seismic Loading
 Mononobe-Okabe, 132
 seismic loading, 76, 131, 217
 select
 menu, 275
 selection
 multiple, 164
 rectangle, 163
 single, 163
 self weight, 202
 setting units, 153
 shear force, 233
 shear stress, 32, 233
 sheet pile wall
 can yield at vertices, 121
 general, 120, 121
 rigid, 121
 short term, 265
 short term behaviour, 181
 SI units, 153, 176
 slip-lines, 233
 slope stability, 113
 SNAP, 149, 164
 snap, 156
 soil nail, 309
 can yield at vertices, 120
 flexible at vertices, 120
 general, 119
 modelling, 122
 rigid, 120
 soil reinforcement, 77, 117, 119, 188
 solid, 87, 153
 solid explorer, 151
 solid object
 nodal spacing, 225
 solution
 always locked, 228
 always unstable, 228
 locked, 227
 out of space, 229
 slow progress, 228
 terminated, 228
 unknown, 228
 unstable, 227
 solution accuracy, 103, 225, 311
 solution tolerance, 91
 solve, 29, 219, 279
 post-solve information, 192
 solver, 87, 219
 checks, 220
 diagnostics tool, 220
 solving
 scenarios, 266
 starting LimitState:GEO, 24
 status bar, 149
 stem wall, 62
 strain compatibility, 81
 stress
 normal, 234
 shear, 234
 stress displacement, 233
 stress states, 80
 support type
 modifying, 150
 supports, 199
 symmetry, 311
 system requirements, 20
 tags, 252
 target number, 225
 tension crack, 313
 terminology, 19
 thumbnails, 251
 title bar, 142
 toolbars, 281
 default, 281

optional, 282
TRACK, 149, 150
translational failure, 172
troubleshooting, 107

undo, 168
unfavourable, 203
unit weight, 187
units, 152, 313
unlock, 229
user interface
 material explorer, 144
 menu bar, 142
 title bar, 142
 toolbars, 143
 viewer pane, 146
 water explorer, 145

vertex, 153, 167
 identification, 150
vertex explorer, 150
vertical acceleration, 217
view
 menu, 276
viewer pane, 146
 menu, 283
void, 167

walls
 nodal spacing, 226
water
 definition, 209
 drag and drop, 145
 explorer, 145
 pressures, 205
 properties, 145
 regimes, 205
 table, 205
 water explorer, 145
water filled tension crack, 314
water pressures, 72
 aquiclude, 213
 aquiclude, 75
 average pore pressure ratio, 74
 constant potential, 75, 209
 constant pressure, 75, 210
 deleting, 214
 duplicate, 214
 enabling and disabling, 205
 exporting, 214
 hierarchy, 76
 importing, 214

interpolated grid, 211
interpolated pressure, 75
on boundaries, 208
phreatic surface, 73
previewing, 214
regimes, 75, 208, 209
table, 73
within solids, 208
water regime
 modifying, 150
water table
 adequacy factor, 96, 314
 clearing, 206
 drawing, 206
 Ru, 207
weight density, 187
wizards, 45

yield surface, 80

zone, 167
zoom, 33