GeoteknikPro

STUDIO — SLOPE ANALYSIS

Version 3.0.1

USER GUIDE

and Calculation Methods Document

Four ways to build the model · Materials · Groundwater · Loads · Support elements · Slip surface generators · Limit equilibrium solution · Sensitivity · Report and drawing · PLAXIS

In accordance with TBDY 2018 Chapter 16 and KDYY

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Contents

Contents 2
List of Figures 5
List of Tables 6
Symbols and Units 7
1. Introduction 8
1.1 What the program does 8
1.2 Scope 8
1.3 Who it is for 8
2. Screen and Tools 9
2.1 Two modes: PLAN and SECTION 9
2.2 Screen areas (SECTION mode) 9
2.3 Drawing modes 10
3. Building the Model — Four Ways 11
3.1 First way — manual drawing 11
Snapping — where a click is taken to be 11
The Extend command 11
Water and ignore — chain selection 11
Other manual edits 12
3.2 Second way — the slope wizard 12
Geometry produced by the wizard 12
3.3 Third way — DXF import 14
Entities that are read and not read 14
The unit question — the one that changes the result 14
After the import 15
3.4 Fourth way — taking a section from contour lines 16
Step 1 — importing the contour map 16
Step 2 — assigning elevations 16
Step 3 — drawing the section line 16
Step 4 — automatic completion of elevations 16
Step 5 — extracting the section 17
4. The Region Concept 18
4.1 Building the regions 18
4.2 Region types 18
4.3 Two safeguards 19
Split structure 19
Soil floating in the air 19
4.4 Ground surface 19
5. Materials 21
5.1 Strength models 21

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

5.2 Soil library 21
5.3 Stiffness values 22
6. Groundwater 22
7. Loads 22
7.1 Defining a surcharge 22
7.2 Multi-point distribution 22
7.3 Action type 23
7.4 Load transfer elevation 23
8. Structures and Support Elements 23
8.1 Retaining structures 23
8.2 Support element types 24
8.3 Tendon rupture and grout pull-out 25
9. Soil Nail Walls 26
9.1 Scope and design basis 26
9.2 Geometry: ground, excavation face and lifts 27
9.3 Lift convention 27
9.4 Nail rows and skin friction 28
9.5 Partial factors and capacities 28
9.6 Tensile demand: T_{max} and $T_{i,max}$ 29
9.7 Shotcrete facing 29
9.8 External checks and serviceability 30
9.9 Test programme, quantities and PLAXIS 31
9.10 Verification: KDYY P20 example project 32
9.11 Design wizard 32
9.12 Solution method and tension crack 34
9.13 Required nail force and length optimisation 34
9.14 Results and outputs 35
10. Analysis 37
10.1 Slip surface generators 37
10.2 TBDY 2018 Equation 16.30 39
10.3 Other methods 39
10.4 Loading cases 39
10.5 Staged analysis 40
10.6 Slope face shotcrete facing 40
10.7 Calculation method, cohesion term and warnings 41
10.8 Pore water: transfer from the Seepage module 41
10.9 Saving settings as defaults 42
10.10 Back analysis 42
11. Design Bases 43
11.1 KDYY Table 2.1 factors 43

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

11.2 Single-source assumption 43
12. Reading the Results 45
12.1 Slice table 45
12.2 Method comparison 45
12.3 Toe failure and global failure 45
13. Sensitivity Analysis 46
14. Verification 47
14.1 Comparison with reference slope analysis reports 47
14.2 Comparison of the solvers with each other and with the classical solution 48
14.3 Comparison with independent solutions 50
14.4 Internal consistency checks 50
14.5 Known differences and limitations 50
15. Report and Drawing 52
15.1 Word report 52
15.2 DXF drawing 52
16. Export to PLAXIS 2D 53
16.1 Important differences for a slope model 53
17. Points to Watch 54
17.1 When building the model 54
17.2 When analysing 54
18. Code References 55
19. Known Limitations and Out-of-Scope Topics 55
20. Frequently Asked Questions 56
Why do some methods give no result? 56
Why did the wedge generator produce no surfaces at all? 56
Does the result change when the model box is enlarged? 56
Why are some methods not shown in the report? 56
Why does the program not find the elevations by itself on the contour route? 56
21. References 57

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

List of Figures

Figure 1. PLAN mode: contour lines and two sections drawn on the plan 9
Figure 2. SECTION mode: toolbars, material–support–region panels on the left, section in the centre, slice table at the bottom 10
Figure 3. Slope wizard: bench and face definitions, layer splitting and preview of the generated geometry 14
Figure 4. Drawing unit window: the question “How many meters is 1 drawing unit?” and the check of the metric equivalent 15
Figure 5. Region type selection: the menu that opens on right-clicking a region 19
Figure 6. Structure load window: marking a region as a structure and its base pressure 19
Figure 7. Soil library window: ready-made soil and rock records, type filter and search 21
Figure 8. Surface loads window and a multi-point surcharge distribution on the slope 23
Figure 9. Menu that opens on right-clicking a line: turning the line into an anchor, soil nail or pile / wall, and defining a shotcrete facing 24
Figure 10. Section containing a pile / wall support element and a structure load (q = 110 kPa) 25
Figure 11. Settings window, Slope › Soil nail tab: design code, basis and solution, nail demand options 27
Figure 12. Soil nail design wizard, step 4 · Nails: layout parameters, row table and section preview 34
Figure 13. Soil nail design wizard, step 9 · Preview and analysis: staged analysis result and lifts table 35
Figure 14. Settings window, Slope › General analysis tab: factors of safety, earthquake, search and sensitivity analysis settings 37
Figure 15. Critical slip surface found with the weak-seam following generator, and the slice table 38
Figure 16. All surfaces view: evaluated slip surfaces coloured by factor of safety 38
Figure 17. Safety map: centre grid points coloured by factor of safety 39
Figure 18. Method comparison window: results of the methods for the same slip surface and the classical Bishop information row 45
Figure 19. Evaluated slip surfaces window: the surface list and the selected surface highlighted on the section 46
Figure 20. Sensitivity analysis window: variation of the factor of safety with parameter deviation 46
Figure 21. Prepare report window: the “Include sensitivity analysis” option and the sensitivity scan running while the report is prepared 47
Figure 22. Export to PLAXIS 2D window: soil model selection and script preview 53

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

List of Tables

Table 1. Symbols used in the guide 7
Table 2. Screen areas 9
Table 3. Drawing modes and their functions 10
Table 4. Comparison of the four ways 11
Table 5. Slope wizard inputs 12
Table 6. Rules of automatic elevation assignment 16
Table 7. Region types and their behaviour 18
Table 8. Strength models 21
Table 9. Support element types 24
Table 10. Lift convention options 28
Table 11. Source of τ_{bu} 28
Table 12. KDYY partial factor chain (default values) 28
Table 13. Nail test programme (KDYY 3.3.4, Table 3.21) 31
Table 14. Comparison with KDYY P20 32
Table 15. Slip surface generators 37
Table 16. Design bases 43
Table 17. KDYY Table 2.1 partial factors 43
Table 18. Basis of the verification 47
Table 19. Comparison with reference reports (γ_{Rk}) 48
Table 20. Comparison of the solvers on the same surface 48
Table 21. TBDY solver (classical term) compared with GLE-Bishop 49
Table 22. Polygonal surface compared with its circular equivalent (Janbu) 49
Table 23. Cohesion term: TBDY 2018 formulation versus classical Bishop 49
Table 24. Pile with shear capacity: sensitivity to the capacity 50
Table 25. Soil-nailed vertical cut and planar block: comparison with closed-form solutions 50
Table 26. Global failure: methods compared with PLAXIS (1.56) (single section) 51
Table 27. Code references 55
Table 28. Out-of-scope and limited topics 55

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Symbols and Units

Throughout the guide the unit of force is kN, the unit of length is m and the unit of stress is kPa. The analysis assumes plane strain and is carried out for a unit width (1 m) perpendicular to the section.

Table 1. Symbols used in the guide

Symbol	Description	Unit
γ_{Rk}	Characteristic factor of safety (TBDY 2018, Eq. 16.30)	—
γ	Natural unit weight of the soil	kN/m³
γ_{sat}	Saturated unit weight of the soil	kN/m³
γ_w	Unit weight of water	kN/m³
c'	Effective cohesion	kPa
c_u	Undrained shear strength	kPa
ϕ'	Effective angle of internal friction	°
b	Slice width (horizontal)	m
ℓ	True length of the slice base, $\ell = b/\cos\alpha$	m
α	Angle of the slice base with the horizontal	°
W_s	Slice weight (including surcharge)	kN/m
u	Pore water pressure at the slice base	kPa
h_w	Water height at the slice base, $h_w = u/\gamma_w$	m
m(α)	Bishop factor, $m(\alpha) = \cos\alpha + \sin\alpha \cdot \tan\phi'/\gamma_{Rk}$	—
q	Surcharge load intensity	kPa
R	Radius of the slip circle	m
H	Slope height	m
k_h	Horizontal pseudo-static seismic coefficient	—
k_v	Vertical pseudo-static seismic coefficient	—
S_{DS}	Short-period design spectral acceleration coefficient	—
ST	Local site effect coefficient	—
T	Design axial force of the support element	kN
V	Design shear force of the support element	kN
M_p	Plastic moment capacity of the support element	kNm
q_s	Unit pull-out resistance	kN/m
$\gamma_{G,dst}$	Partial factor on permanent action, destabilising	—
$\gamma_{G,stb}$	Partial factor on permanent action, stabilising	—
$\gamma_{Q,dst}$	Partial factor on variable action, destabilising	—
γ_{Re}	Partial factor on soil resistance	—
Msf	Strength reduction factor in finite element analysis	—

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

1. Introduction

1.1 What the program does

GeoteknikPro Studio Slope is a module that examines slope stability on two-dimensional sections by the limit equilibrium method. The user defines a section — ground surface, soil layers, groundwater, loads and any support elements; the program generates thousands of possible slip surfaces, solves each of them and takes the surface giving the lowest factor of safety as critical.

The result is given as the characteristic factor of safety γ_{Rk} in accordance with TBDY 2018 (Turkish Building Earthquake Code) Chapter 16 and is compared with the limit value prescribed by the code. Optionally, a design calculation can also be made with the partial factor set of Table 2.1 of KDYY (Turkish Excavation Support Structures Regulation).

1.2 Scope

The module focuses on slope stability by the limit equilibrium method. The following topics lie outside this module and are handled in other modules or in separate checks:

- Bearing capacity calculation. Local failure beneath a load is a separate check.
- Settlement and deformation calculation. The limit equilibrium method examines only the moment of failure.
- Staged construction modelling that tracks the stress history. In the Slope module, excavation and construction stages are solved one by one as frozen sections (Section 10.5); the stress and strain history during construction is not tracked. This kind of modelling is the subject of the Excavation Support Structures module.
- Structural member design. Wall reinforcement or shotcrete thickness are not calculated here.
- Three-dimensional effects. The analysis is based on the plane strain assumption.

1.3 Who it is for

The module is intended for users with adequate engineering training who know the assumptions of the limit equilibrium method. Interpreting the results and selecting the soil parameters are the user’s responsibility. The program performs the calculation; it does not make the engineering decision.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

2. Screen and Tools

2.1 Two modes: PLAN and SECTION

The module is split into two modes, and you switch between them with the two buttons on the top toolbar. PLAN mode is for contour lines, elevation assignment and section lines; no analysis is done in this mode. SECTION mode is where the model is built and solved. The fourth way of building the model starts in PLAN mode; the other three run directly in SECTION mode.

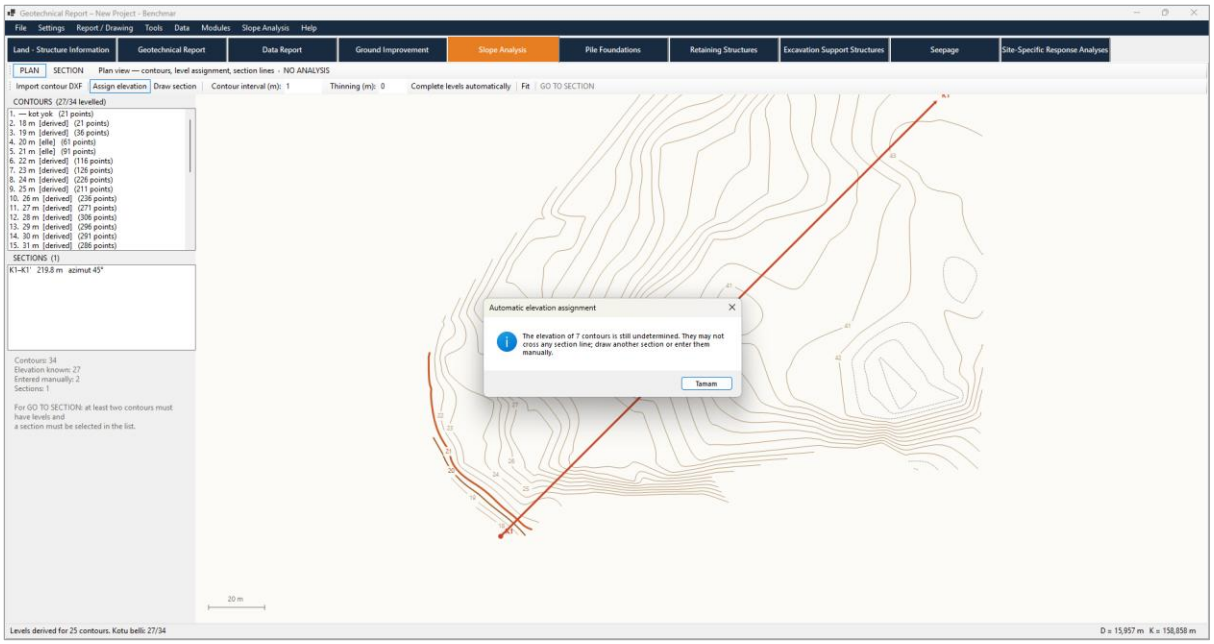


Figure 1. PLAN mode: contour lines and two sections drawn on the plan

2.2 Screen areas (SECTION mode)

The window has four areas: a two-row toolbar at the top, material and element lists on the left, the section view in the centre, and the results summary and slice table at the bottom.

Table 2. Screen areas

Area	Content
Top toolbar — row 1	Model building: wizard, DXF, drawing modes, surcharge, undo
Top toolbar — row 2	View, result display, analysis (including soil nail and back analysis), PLAXIS; the settings and report commands are in the main menu at the top
Left panel	Materials, support elements, regions; selected point and earthquake
Section view	Geometry, slip surfaces, slices; ruler and crosshair cursor
Bottom area	Loading cases, governing case and slice calculation table

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

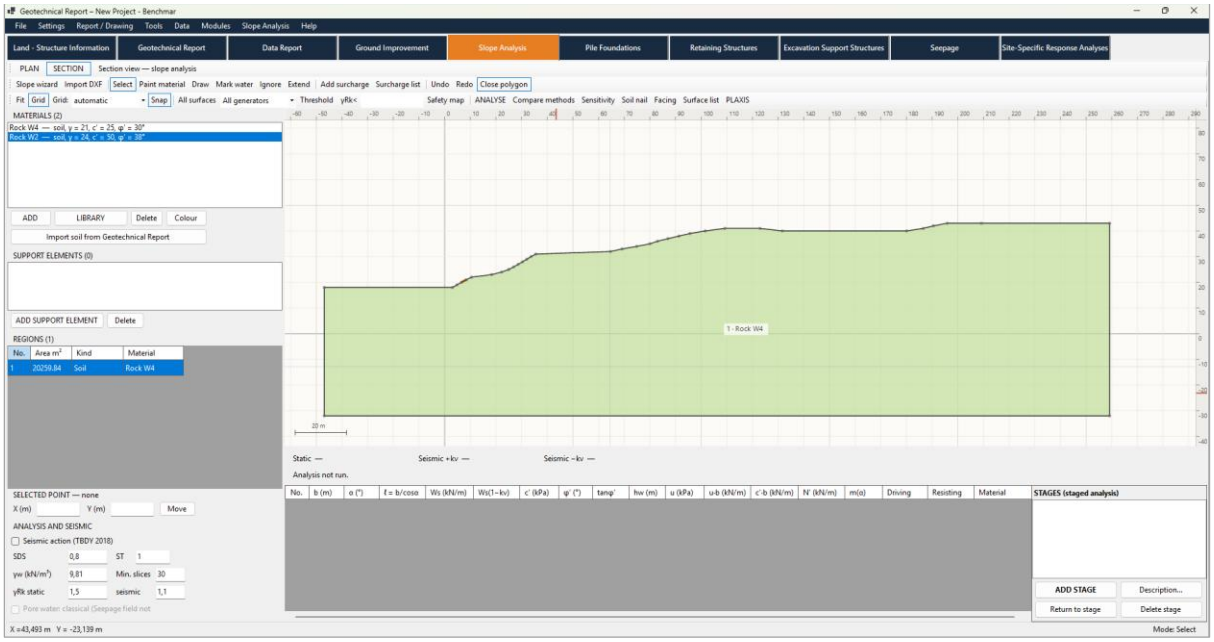


Figure 2. SECTION mode: toolbars, material–support–region panels on the left, section in the centre, slice table at the bottom

2.3 Drawing modes

The mode buttons on the toolbar determine what the mouse does. While in a mode, a right-click or ESC cancels an unfinished operation. The modes work in the same way regardless of how the model was built.

Table 3. Drawing modes and their functions

Mode	Function
Select	Selecting nodes and regions; moving a node by dragging
Paint material	Assigns the selected material to the clicked region
Draw manually	Adds lines with successive clicks; a right-click or Enter finishes
Mark water	Makes the clicked line chain the groundwater level
Ignore	Excludes the clicked chain from the analysis
Extend	Extends a line up to the first line on its extension
Add surcharge	Defines a load strip by clicking two points on the ground

In addition there are buttons for grid on/off and grid spacing (automatic · 0.5 · 1 · 2 · 5 · 10 · 25 m), snapping on/off, drawing a closed polygon, fit, undo and redo. The undo stack holds fifty steps.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

3. Building the Model — Four Ways

A section can be built in four ways: manual drawing, the slope wizard, DXF import and taking a section from contour (isohypse) lines. All four arrive at the same place: a geometry made of closed line loops. Whichever way you start, the model can be edited by hand afterwards; the four ways are not alternatives to one another but different starting points for the same job.

Table 4. Comparison of the four ways

Way	Input	When to prefer it
Manual drawing	None — mouse and grid	Small and simple sections; correcting a model built in another way
Slope wizard	Number of benches and dimensions	Regular, benched slopes matching a design section; horizontally layered profiles
DXF import	ASCII DXF section drawing	When the section has already been drawn in CAD — the most common way
Contour lines	Contour DXF map	No section at hand, but a map is available; when the section direction itself is to be investigated

All four end in the same list

The product of all four ways is a single line list. A wall coming from DXF can therefore be added to a slope generated by the wizard, and a layer boundary can be drawn by hand on ground extracted from contours. The program keeps no record of the kind “this model came from that way”.

3.1 First way — manual drawing

This is the most basic way and also complements the other three. In “Draw manually” mode, successively clicked points form a polyline; a right-click or Enter finishes the line, and ESC cancels an unfinished one. A draft with fewer than two points is discarded. When the “Close polygon” button is on and there are more than two points, the last point is joined to the first and the loop is closed.

Snapping — where a click is taken to be

The clicked location is not taken as it is; it is snapped first. The snapping order is: first existing geometry nodes, then line ends, and last the grid. Which of them was caught is written to the status bar — “Point added (x; y) — node/end/grid snapped”. The snap radius is fixed on screen; as you zoom in, the tolerance in real dimensions gets smaller, so zooming in is enough for precise work.

While a node is being dragged, only grid snapping is active. Otherwise, the dragged node would stick to every node it passes over and moving it to another point would become impossible.

The Extend command

The coordinates of an intersection point may be something like (14.4384; 27.3218), and such a point cannot be caught by hand. In “Extend” mode you click on the line near the end to be extended; the program advances that end along the line’s own direction and stops it at the first line it meets. The result is reported in the status bar as “Extended: … m → (x; y)”. In this way a line drawn approximately finds the intersection exactly.

If there is no line to intersect in the extension direction, the program says so and does nothing; parallel lines are not counted as targets.

Water and ignore — chain selection

In the “Mark water” and “Ignore” modes it is enough to click a single line: starting from that line, the program selects and marks the whole end-to-end connected chain. The chain stops when it reaches a branching point,

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

because there it is ambiguous which branch to follow. Lines marked as water or ignore do not form regions; water lines represent the groundwater level profile, and ignored lines represent drawing leftovers that do not enter the model.

Other manual edits

- In “Select” mode the geometry is changed by clicking a node and dragging it; clicking in empty space selects a line or region, and the Delete key deletes the selected line.
- Right-clicking a line opens options to delete that line or to turn it into a support element.
- The “Clean up duplicate lines” command removes identical lines lying on top of each other; direction is disregarded, so A→B and B→A count as the same line.
- Every change is written to the undo stack (fifty steps), and the regions are rebuilt after every change.

3.2 Second way — the slope wizard

The wizard generates a regular slope from its dimensions. It is the fastest way for benched foundation excavations, mine slopes, highway cuts and fills, and regular slopes read from a design section. The screen has input fields on the left and a live preview on the right; with every change the geometry is rebuilt and the preview and summary notes are refreshed.

If the wizard is opened while there are lines in the drawing, you are told that it will replace the existing drawing and asked to confirm. If you confirm, the lines and support elements are deleted; you can go back with Undo.

Table 5. Slope wizard inputs

Input	Meaning and limit
Number of benches	0 – 8; 0 means a single-face slope. The number of faces is one more than the number of benches.
Face dimension input	One of two forms: horizontal and vertical dimension (m), or vertical dimension and slope (1 vertical : n horizontal).
Crest and toe flats	Length of the flat parts above and below the slope (m).
Slope faces table	For each face, the horizontal/vertical dimension and the width of the next bench. The bench cell of the last row is empty; there is no bench below it.
Back and front slope	Slope of the ground rising behind the crest and falling beyond the toe; if left empty, the ground is extended flat.
Model box	Extension to the left, extension to the right and depth below (m). If left empty, each is taken as twice the slope height.
Elevation assignment	Optional. The elevation of the toe flat is entered; all other elevations are derived from it.
Layer splitting	0 – 20 layers; thicknesses are measured downwards from the top point of the excavation. Each layer boundary is a horizontal line cutting across the full width of the model box.

Geometry produced by the wizard

The ground line starts at the top-left corner and develops descending to the right; the closed model box is completed by adding a left edge, a right edge and a base to this ground line. Because the layer boundaries are added as model segments, the box is automatically divided into as many regions as there are layers when the regions are built.

The wizard summary may show the following notes: that the model box was auto-sized, that the back or front slope was applied, the total slope height and average slope, the assumption with which the elevation assignment

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

was made and — if any — how many layer boundaries could not be drawn because they fall below the base of the box. The last note is important: you are told about any layer that could not be drawn.

The model is not ready when the wizard finishes

If the material list is empty, the wizard defines a default soil and assigns all regions to it. This is a convenience, not a recommendation: before running the analysis you must edit the materials with your own values, assign separate materials to the layers, and define the water level and the loads.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

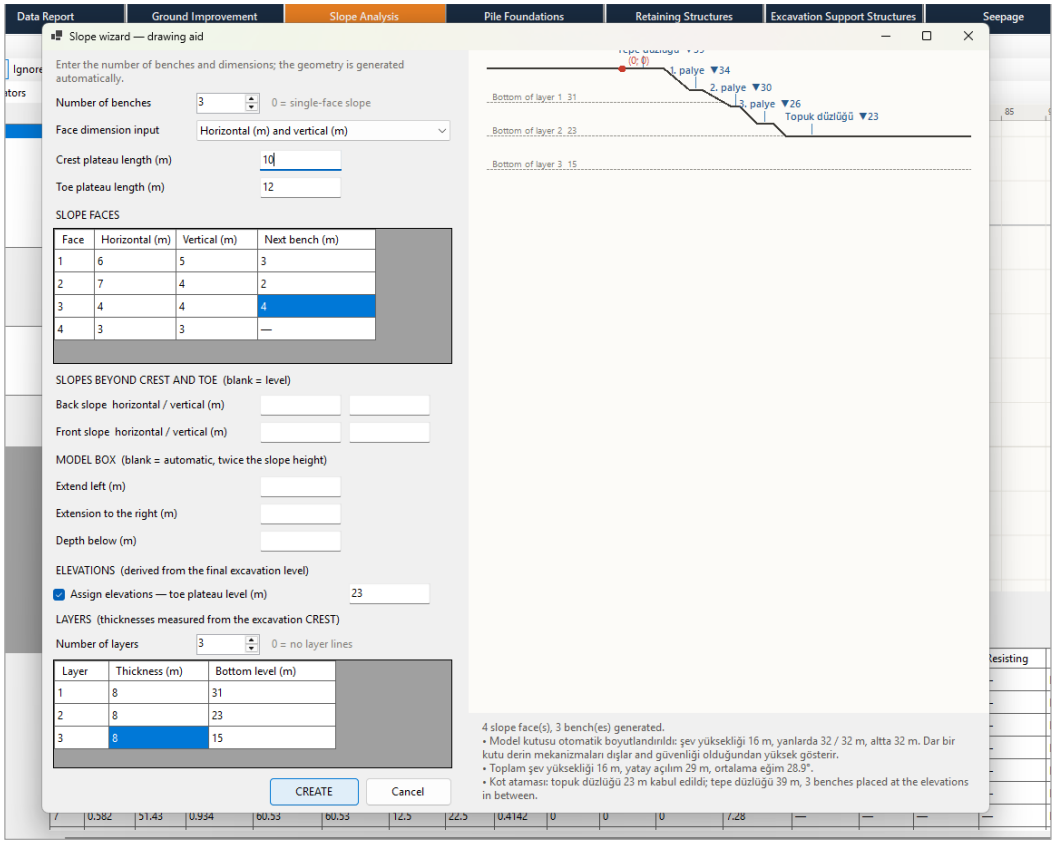


Figure 3. Slope wizard: bench and face definitions, layer splitting and preview of the generated geometry

3.3 Third way — DXF import

If the section has already been drawn in CAD, the most accurate way is to take it as it is. The “Import DXF” button lets you pick a file, reads the lines, asks for the unit and places the geometry in the model.

Entities that are read and not read

The reader does not depend on an external library and expects ASCII DXF; binary DXF is not read. The entities read are LINE, LWPOLYLINE and the old-style POLYLINE. Polylines are split into segments, and the closing segment of closed ones is added as well. ARC, SPLINE, CIRCLE, TEXT and block (INSERT) entities are not read; the arc (bulge) information on an LWPOLYLINE is also ignored and that segment is taken as straight. The Z coordinate is ignored — reading is entirely two-dimensional.

If not a single readable line is found in the file, the program stops, saying so and stating which entities it read. Sections containing an arc or curve must first be broken into straight segments (exploded) on the CAD side.

The unit question — the one that changes the result

After the lines have been read you are asked “How many meters is 1 drawing unit?”. The options are meter, centimeter, millimeter, feet, inch and a factor of your own. If the largest extent of the drawing is greater than two thousand, centimeter comes preselected. The window shows live the model size converted with the selected unit and writes a verdict: it warns if the size looks too small or too large for a slope section.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

A wrong unit spoils the result, not only the display

Cohesion scales linearly with length, weight with the square of length. Importing a model with the wrong unit therefore changes the factor of safety not proportionally but in a distorted way: in one measured example, choosing centimeter instead of meter lowered the factor of safety by more than half. If the unit window is cancelled, the whole import is cancelled; the option of continuing with a wrong unit has been left out on purpose.

After the import

The coordinates are multiplied by the selected factor, the lines are added to the model and duplicate lines are discarded immediately afterwards — overlapping lines are common in CAD drawings and they spoil the region building. The status bar reports how many lines were read, how many duplicates were discarded, how many regions were found and the dimensions of the model.

If there are unclosed corners, the program states their number and marks them with a red cross on the section. An unclosed corner means that the region adjacent to it is not formed at all; corners are closed with the “Extend” command or by adding a line by hand.

Unlike the wizard, no material is assigned to any region on the DXF route. All regions arrive “unclassified”, and the analysis does not run until all of them are classified. Which region is which soil is information that cannot be read from the drawing; the program does not guess it.

In this version no layer selection is made during import; all readable lines in the file are taken. For a crowded drawing there are two routes: save a file containing only the necessary layers on the CAD side, or mark the surplus after import with the “Ignore” mode. Ignored lines are visible in the drawing but do not enter the model.

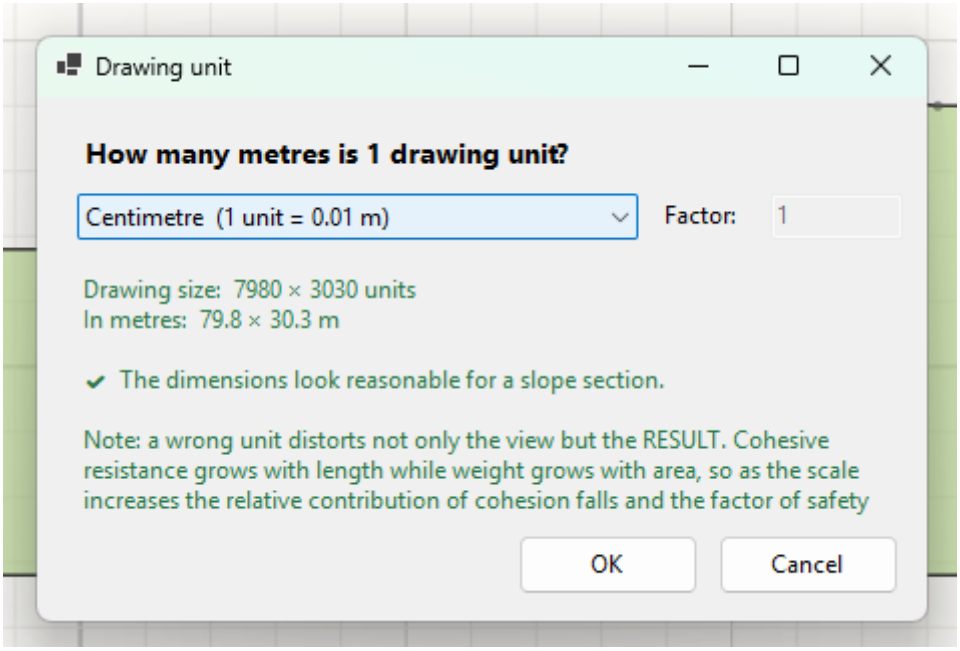


Figure 4. Drawing unit window: the question “How many meters is 1 drawing unit?” and the check of the metric equivalent

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

3.4 Fourth way — taking a section from contour lines

This way is used when there is no section at hand but a map is available. The contour (isohypse) lines are imported as a plan, elevations are assigned to the lines, a section line is drawn on the map, and the program extracts the ground profile along that line and builds the two-dimensional model. The whole of this route runs in PLAN mode; once the section has been extracted, you switch to SECTION mode.

Step 1 — importing the contour map

The “Import contour DXF” button uses the same reader as the section import; the same entity limits and the same unit window apply. The line segments read are joined end to end into chains, and each chain becomes a contour line. If the map already contains lines, the program asks whether the new ones should replace the old ones or be added on top. The second option is for those who load the map piece by piece.

Step 2 — assigning elevations

None of the imported lines has a known elevation; DXF does not carry this information. With the “Assign elevation” tool you click a line and type its elevation. Every elevation entered by hand is an anchor and is never changed by the program. Lines are drawn differently according to their status: those given an elevation by hand are thick and dark, derived ones are of medium thickness, and those with unknown elevation are faint and dashed. In this way you can see at a glance how much of the map is known.

Step 3 — drawing the section line

With the “Draw section” tool you click two points on the map. The line is given a name and an azimuth measured clockwise from north. As soon as the line is drawn, the program searches for the direction of steepest descent of the ground around the middle of the section and reports the deviation of the line from that direction. If the deviation exceeds twenty degrees it warns: an oblique section shows the slope flatter than it is and gives a factor of safety that is high on the unsafe side.

Step 4 — automatic completion of elevations

A map can have hundreds of lines, and entering elevations for all of them by hand is both long and error-prone. The “Auto-complete elevations” command orders the lines along the section line and derives the intermediate ones from the anchors. The reason the command requires a section line is this: because contour lines are nested in the plane, they have no natural order such as “left to right”; an order is defined only along a section line. The command needs at least two anchors, because direction and interval can only be obtained from two points.

Table 6. Rules of automatic elevation assignment

Rule	Behaviour
Anchors are kept	No elevation entered by hand is overwritten; derived elevations, on the other hand, are regenerated from scratch every time the command is used.
Between two anchors	The elevation difference is divided by the number of steps. If the interval found conflicts with the interval entered in the “Contour interval (m)” box on the toolbar, the lines in between are left without elevation and the inconsistency is reported — the program does not pick one of the two values and hide the other.
Outside the anchors	The direction of the nearest pair of anchors is continued outwards; thus the sequence 299–300–299 continues outside as 298, not as 301.
Two anchors with the same elevation	The lines in between indicate a local hill or hollow; they are filled symmetrically and the direction is estimated from the neighbouring anchors. If it cannot be estimated, the lines are left without elevation and the situation is reported.
The same line cut twice	The section crosses a hill or a hollow: the elevation stays the same and the direction is reversed.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Every case in which an elevation cannot be derived is reported with a message and the line concerned is left without elevation. Lines that do not cross any section line also remain without elevation; for these, either another section is drawn or the elevation is entered by hand.

Step 5 — extracting the section

The “GO TO SECTION” button becomes active when the elevations of at least two lines are known and a section is selected in the list. The program finds the points where the section line crosses lines with defined elevations, sorts them by distance along the line and builds the ground profile from the (distance, elevation) pairs. Lines without elevation are skipped; the profile is built only from the crossing points with lines of known elevation. The “Thinning (m)” box on the toolbar determines the number of points in the profile. The default value is 0, in which case all crossing points of the section line with lines that have elevations are taken into the profile. If a tolerance greater than zero is entered, the profile is simplified with the Douglas–Peucker method; the number of points before and after simplification is reported.

As soon as a section is selected on the plan, its crossings with lines that have no elevation are shown with a red ring and “?”, and their number is written in the status bar. These crossings are not added to the profile when the section is transferred. The marks are refreshed as elevations are entered.

If the profile is not one-directional, that is, if it contains a ridge or a valley, the program says so. The analysis can solve such a profile; the purpose of the warning is to have you check on which slope the critical surface comes out.

Beyond the section ends the ground is extended flat

The model box is extended to the left, to the right and downwards by twice the elevation difference. Because there are no contour data beyond the ends of the section line, the ground slope is not continued but extended flat, and a note on this is added to the report. If the critical surface reaches into this flat zone, the section line should be lengthened.

By default the section elevations are the true elevations; the profile is not shifted to zero. This means the vertical axis of the model lies, for example, in the 239–260 m range, which makes comparison with the map easier. As on the wizard route, all regions are assigned here to a default soil; editing the materials, drawing the layer boundaries and defining the water and the loads are left to the user.

The plan work — lines, elevations and section lines — is not stored in the main project file; only the plan north direction used in the drawing is saved with the project. If you want to work with the map again, the contour DXF is imported again.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

4. The Region Concept

The program works with regions rather than a layer stack. Every closed line loop is a region, and each region is assigned a type and — if it is soil — a material. This arrangement allows inclined layers, lenses, embedded structures and excavated zones to be defined naturally; a horizontal layer stack is a special case of it.

4.1 Building the regions

The regions are rebuilt after every geometry change. Building goes through these steps: intersecting lines are split at the intersection points; if the end of one line touches the middle of another, a node is opened there too (a T junction — the case where layer boundaries meet the model box); very close ends are merged into a single node, dangling edges connected to nothing are cleaned up, and closed cycles are extracted from the remaining edges. The cycle with the largest area is the project boundary and is not counted as a region; the rest are numbered.

At the end of the build you are told how many segments were split into how many edges, how many dangling edges were discarded and how many regions were found. Nodes of degree one — that is, lines with a free end — are marked as unclosed corners, and two free ends close to each other are reported as a gap together with the distance between them.

4.2 Region types

Table 7. Region types and their behaviour

Type	Behaviour
Unclassified	Not yet classified. The analysis does not run while such a region exists, and the missing regions are listed by name.
Soil	Has a material; the slip surface can pass through it.
Rigid block	Retaining wall, concrete or stone mass. Its weight enters the calculation; the slip surface cannot pass through it and has to go around it.
Structure	Its load (base pressure q) is transferred as a strip load along the base edges where the structure bears on the soil, each base segment at its own level, as a permanent (G) load; the drawn height has no physical role. Within the footprint the ground is the structure base; a slip surface may start from the structure base, but its exit cannot lie in the footprint or on a side face (Section 7.4). The base pressure is asked for.
Void	Has no weight and no strength.
Ignore	Visible in the drawing, does not enter the model — an excavated zone or a drawing leftover.

The region type comes from the type of the material assigned: a region assigned a retaining structure material becomes a rigid block, a region assigned a building material becomes a structure, and the others become soil. Assignment is made in two ways: by clicking the region in “Paint material” mode, or by right-clicking the region and choosing from the list that opens. The heading of the right-click list shows the number and the area (m^2) of the region.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

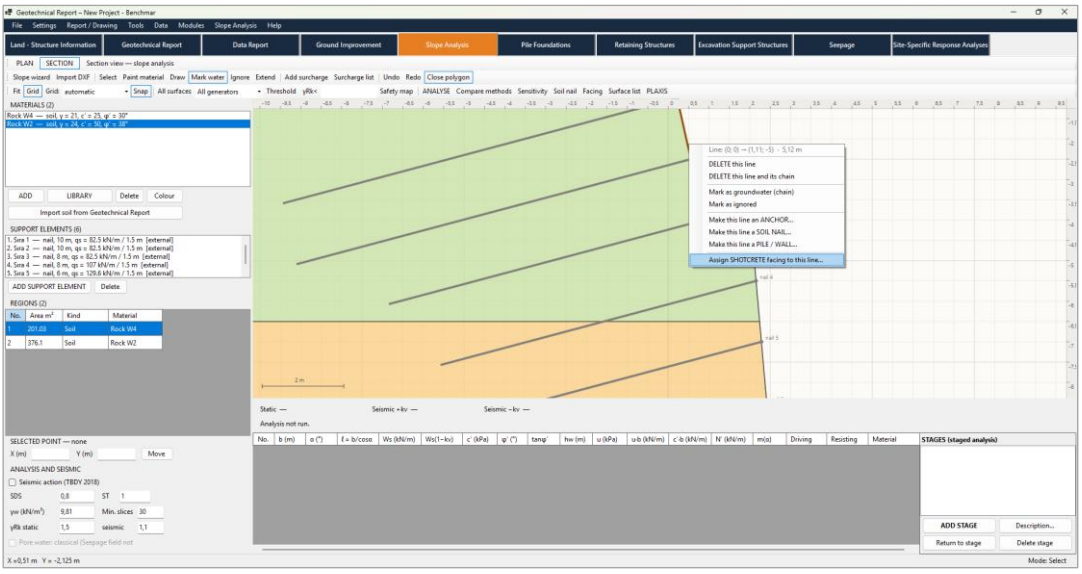


Figure 5. Region type selection: the menu that opens on right-clicking a region

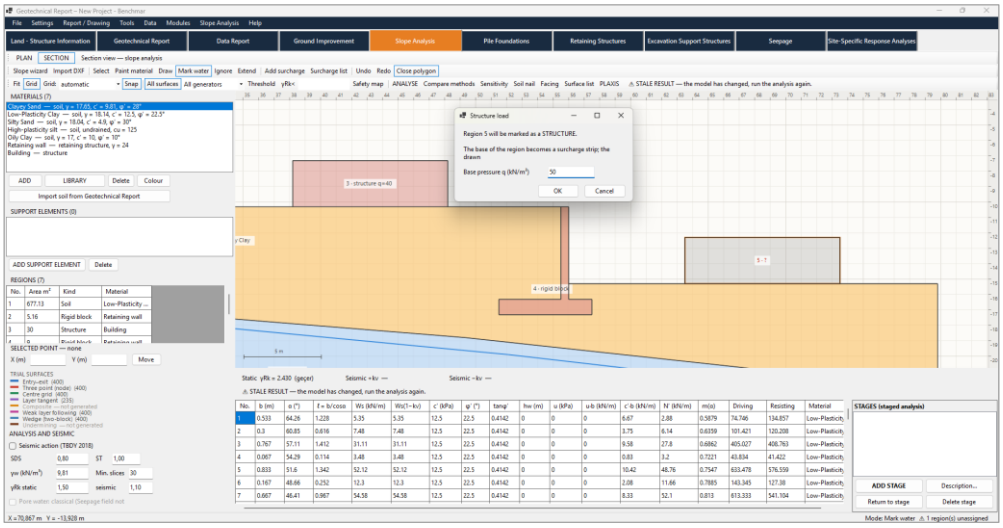


Figure 6. Structure load window: marking a region as a structure and its base pressure

4.3 Two safeguards

Split structure

A retaining wall may be split into several parts in the drawing — stem and footing as separate loops. If a retaining structure material is assigned to only one part, the slip surface passes through the other part and pierces the structure. The program detects parts that touch each other vertically within the same X range and asks whether they should all be assigned together.

Soil floating in the air

If two soil blocks that do not touch each other are drawn, the upper one floats in the air. The program scans the soil mass with vertical sections and, if it finds a gap without material, warns you, stating where it is and how thick. This warning does not stop the analysis — the user may have built such a model on purpose — but it should not be ignored.

4.4 Ground surface

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The ground surface is not marked by the user; for each X it is derived as the upper boundary of the topmost soil or rigid block region. This is why the Y coordinate of surcharges is not asked for — the load sits on the surface and the surface is already known. When the geometry changes, the surface changes with it automatically.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

5. Materials

5.1 Strength models

One of two strength models is selected for each soil.

Table 8. Strength models

Model	Parameters	When
Drained	c′, ϕ′	Long term; pore water pressure dissipated
Undrained	c_u (ϕ = 0)	Short term; rapid loading in clays

The material list also contains retaining structure and building type materials; these also determine the type of the region to which they are assigned (see Section 4.2).

5.2 Soil library

The library button on the material panel opens ready-made soil and rock records. The search is insensitive to Turkish characters and multiple selection is possible. Materials added from the library can be changed freely within the project — the library is a starting point, not an imposition.

The library can be replaced with the user's own list: if a file named zemin_kutuphanesi.json is placed in the program (installation) folder, next to the program file, it is used instead of the embedded list. In this way organisations can work with their own standard soil sets.

If the values entered go outside reasonable ranges, the program warns but does not block; for example, entering a rock mass cohesion that is low compared with published values is reported.

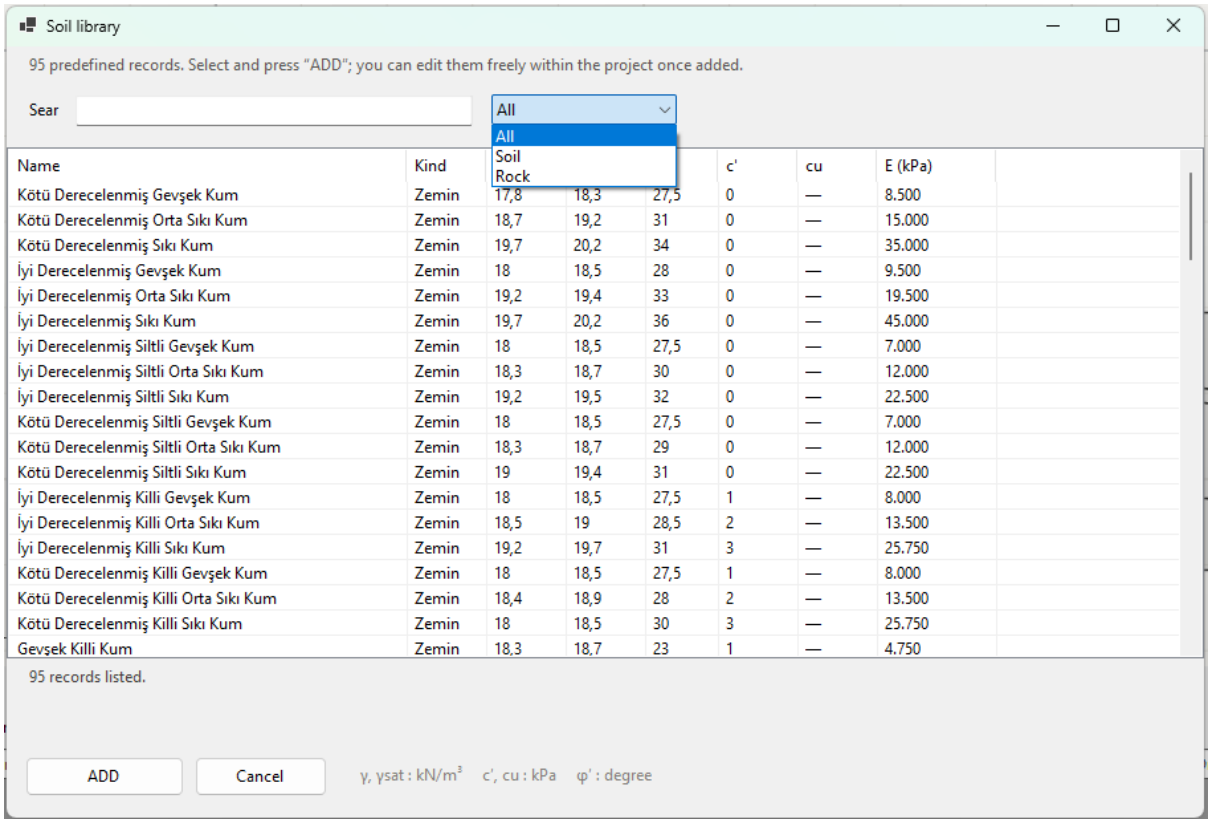


Figure 7. Soil library window: ready-made soil and rock records, type filter and search

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

5.3 Stiffness values

The modulus of elasticity does not affect the limit equilibrium calculation

The E [MPa], v and E_{oed} [MPa] fields in the material window are optional and are used only in the PLAXIS export. The limit equilibrium method examines the moment of failure; deformation is not calculated, so stiffness does not enter the result. The program does not estimate stiffness: during export, the user is asked about layers with missing E or v (Section 16).

In an undrained layer, E and v are effective values (PLAXIS Undrained B). Values coming from the soil library or from the Geotechnical Report (E_s, v) fill these fields automatically and can be corrected.

6. Groundwater

The water level is defined as a line chain: in “Mark water” mode it is enough to click the chain. The pore water pressure is calculated as the product of the depth of the slice base below the water level and the unit weight of water.

$$u = \gamma_w \cdot h_w$$

The natural unit weight is used for slices above the water level and the saturated unit weight for those below it. This distinction is automatic and can be followed in the slice table.

The water level may be inclined; it need not be horizontal. In models coming from the contour route the water line is also drawn by hand, because the contour map carries no water level information.

The water level is one of the most decisive variables

The water level is one of the variables with the greatest effect on the factor of safety. It is recommended that the water level also be examined in the sensitivity analysis; a calculation based on a single measured level does not see the seasonal variation.

7. Loads

7.1 Defining a surcharge

The “Add surcharge” button switches to drawing mode. Click the start point on the ground; then either click the end point or type the length. The Y coordinate is not asked of the user; it is read from the ground surface.

7.2 Multi-point distribution

After the strip has been drawn, its distribution is edited by double-clicking it in the list. The ends are fixed; a piecewise distribution can be built by adding points in between. Ready-made shapes — uniform, trapezoidal, triangular, ramp — fill the table with a single click.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

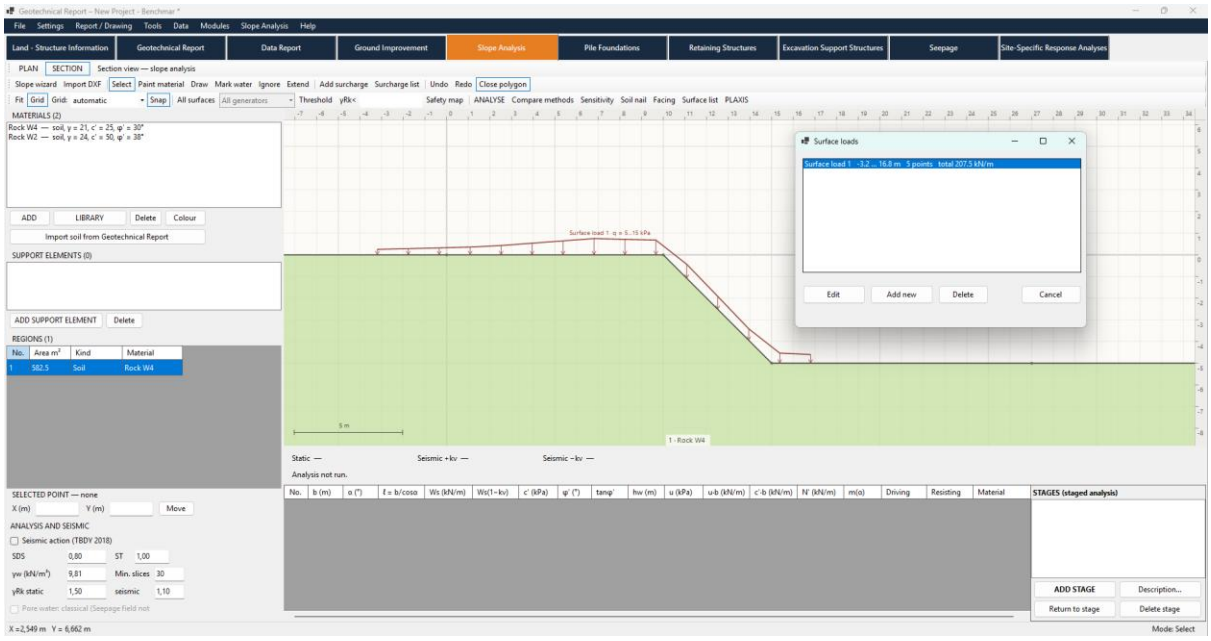


Figure 8. Surface loads window and a multi-point surcharge distribution on the slope

7.3 Action type

Each load is marked as permanent (G) or variable (Q). A building dead load is permanent; traffic, storage and construction loads are variable. This distinction does not affect the characteristic calculation, but it determines which factor is applied in the partial factor calculation.

7.4 Load transfer elevation

The load of a structure with basements acts not at the ground surface but at the foundation base level. When a load transfer elevation is entered, only the slip surfaces passing below that elevation feel the load. A surface passing above it does not carry the load — because the structure stays above that surface and does not move with it.

Within the footprint of a structure region with basements the ground surface is the structure base; slip surfaces may start at the structure base. Surfaces exiting within the footprint or on the side faces of the structure are invalid: soil cannot slide into the basement volume. The structure load is taken as a permanent load; with partial factors, the permanent-load factor applies. In sections with a basement structure close to the slope crest these assumptions lower the factor of safety; this is the expected and correct result.

The structure (building) load is applied as a strip load along the base edges where the structure bears on the soil. Each base segment is a separate strip at its own level; in a stepped basement each step carries load at its own level. Where the upper floors are drawn wider than the base (overhang, eaves), no load is applied beneath the overhang if the space below it is empty; if that space is a fill or soil region, it is treated as contact. q is the base pressure you enter; the total load is q × the actual contact width. If a structure is drawn wider than its base, the program warns once (the analysis is not stopped): “The structure is drawn wider than its base; the load was applied only along the base width.” The report lists each load strip’s start, end, width, q and transfer level. The footprint used in the “sliding into the basement” check is likewise the structure’s base, not its widest drawn extent.

8. Structures and Support Elements

8.1 Retaining structures

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Massive structures such as retaining walls are defined as rigid blocks: the slip surface cannot pass through them and has to go around. This is the counterpart of modelling a concrete mass in finite element analysis.

This rule is also applied to thin and wide rigid blocks (for example the base slab of a retaining wall): if a slip surface cuts the block, it is counted as invalid and eliminated in the search; the surface passes below or around the block. This check is made at closely spaced points along the block, so that steep polygonal surfaces that skip the slab between two points are caught as well. Because the unrealistic surfaces piercing the slab are eliminated, the factor of safety is equal to or higher than the value that would be found without this check.

Thin members are not rigid blocks

Thin members such as bored pile walls, sheet piles or diaphragm walls are defined as support elements; they have finite shear and bending capacities, and the slip surface can pass through them by cutting them. Treating a thin wall as a rigid block forbids shallow mechanisms and inflates the factor of safety on the unsafe side.

8.2 Support element types

Table 9. Support element types

Type	Form of resistance	Values entered
Anchor	Axial tension; pull-out in the bond zone	Design force, bond length, pull-out resistance, horizontal spacing
Soil nail	Fully embedded; tension limited by pull-out	Tensile capacity, unit pull-out resistance, spacing
Pile / wall	Shear and bending; resists when it cuts the surface	Shear capacity, moment capacity, diameter, spacing

Support elements are entered together with their horizontal spacing and converted to resistance per meter; this conversion is compulsory because the section is solved for a unit width. A CAD line in the drawing can be converted to a support element from the right-click menu.

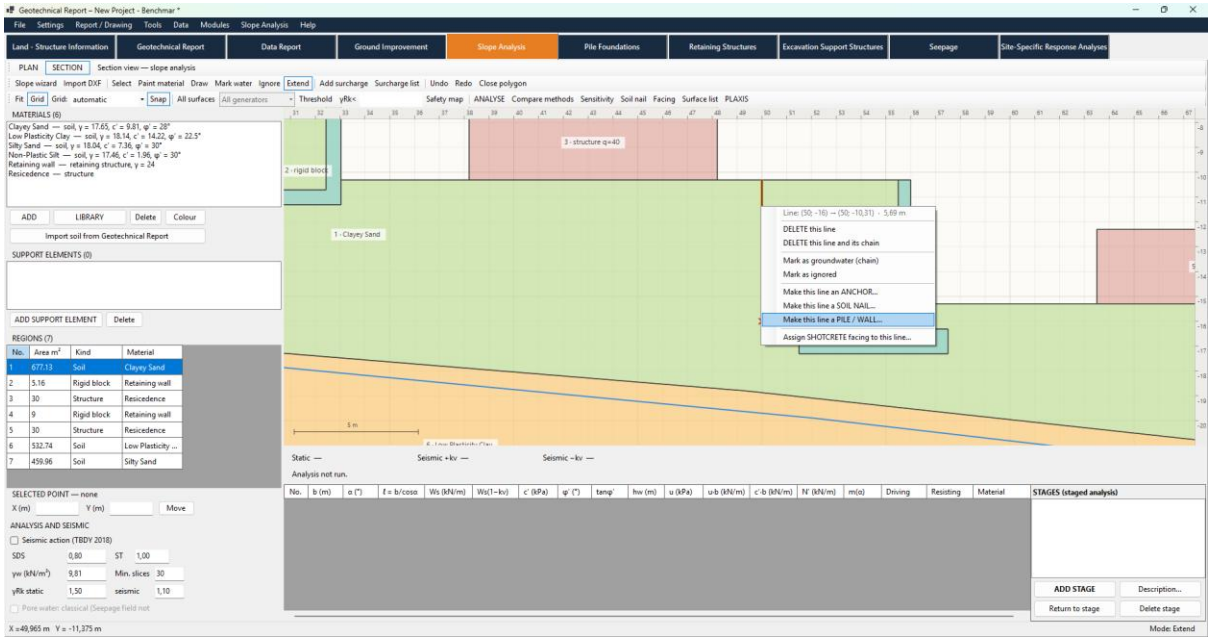


Figure 9. Menu that opens on right-clicking a line: turning the line into an anchor, soil nail or pile / wall, and defining a shotcrete facing

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

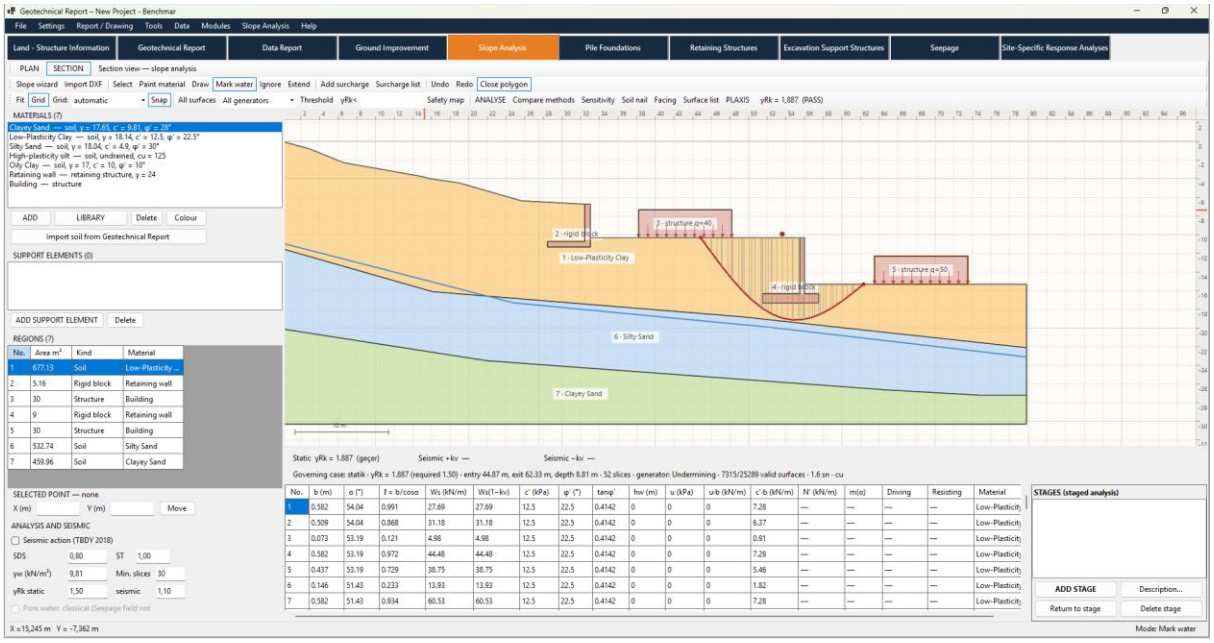


Figure 10. Section containing a pile / wall support element and a structure load (q = 110 kPa)

8.3 Tendon rupture and grout pull-out

An anchor has two failure modes: rupture of the tendon and pull-out of the bond zone from the soil. In accordance with the KDYY Design Handbook 9.2.2, both must be checked and the lower one taken as governing.

The program calculates the pull-out resistance with the bond length lying behind the slip surface; the part in front of the surface produces no resistance, because it moves with the sliding mass. If no pull-out resistance is entered, only the tendon capacity is taken into account and this is stated in the report.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

9. Soil Nail Walls

A soil nail wall is a support system formed by applying a shotcrete facing at each lift as the excavation advances from the top down in lifts, and by installing non-prestressed (passive) steel bars in the ground. The design is done in the Slope module, because a soil nail problem is very often a ground problem before it is an excavation problem, and the routes of drawing the ground surface by hand or taking it from DXF or from contours are set up in this module. The calculation uses the limit equilibrium solver described in Section 10 itself; there is no separate solver.

Scope and basis

Soil nail design has been compared figure by figure with the P20 example project of the KDYY Design Handbook (Section 9.10). The calculation covers the provisions of KDYY 3.3 and the calculation methods of FHWA GEC-7 (FHWA-NHI-14-007): taking the nail demand from the solver, basal heave, FHWA 5.7.3 base sliding, serviceability (SLS) and the g.1/g.2 decision, the final facing and headed studs of a permanent wall, pull-out test evaluation and the ULS summary. The design wizard, result screens, Word report, DXF drawing sheet, transfer to the Slope drawing and project saving are described in Sections 9.11–9.14.

The single “Soil nail” support element of Section 8.2 continues to be used for the general slope check. The design of a soil-nailed wall, on the other hand, is the staged analysis in the wizard; the two routes need not give the same result, because the wizard applies the rules specific to soil nailing (Section 9.12).

9.1 Scope and design basis

- Temporary and permanent nails are designed together; the service type is declared for each row.
- The basic design approach is the KDYY partial factor method; all factors can be changed and the values used are written to the report.
- The facing is shotcrete only (temporary and permanent thicknesses are entered separately).
- The skin friction comes from the library and can always be overridden by hand (Section 9.4).
- All method choices relating to soil nails are in one place: Settings → Soil nail tab. The choices are stored in the main project file; the wizard displays them, and changes are made in the Settings window.

The Settings menu lies between View and Analysis; the “Soil nail settings…” command (or the “Settings…” button in step 1 of the wizard) opens the window directly on the Soil nail tab. The tab consists of four groups:

- Design code — in this version only KDYY 2026 can be selected; FHWA GEC-7 and BS 8006-2 cannot be selected yet and appear greyed out (Section 11). The choice is written to the report.
- Basis and solution — design basis (KDYY partial factors, default / characteristic); soil resistance factor γ_{Re} (KDYY Table 2.1: 1.10, default / KDYY 3.3 Table 3.16: 1.00); abnormal risk option (all factors greater than 1.0 ×1.10; Table 3.16 note A); model factor γ_{sd} (default 1.00; applied to the driving action on the partial basis); solution method, lift convention and tension crack (Sections 9.3 and 9.12).
- Nail demand and external checks — source of T_{max} (Manual / Solver, default for a new wall / PLAXIS export; Section 9.6); K_a for base sliding (Coulomb, default / inclined Rankine, only $\alpha < 8°$); earthquake mode for base sliding (block inertia, default / Mononobe–Okabe / both); basal heave duration (automatic: short term for a temporary wall, long term for a permanent wall, default / short term, $\phi_{BH} = 0.65$ / long term, $\phi_{BH} = 0.50$) and N_c (default 5.14; a different value requires a justification).
- Pull-out tests — KBD γ_d (1.5–2.0; default 2.0), KBD ξ basis (1.1 based on the lowest result, default / 1.3 based on the mean) and KD load factor k (1.1–1.5; default 1.25).

The design basis, lift convention, solution method and tension crack choices are also on this tab; they are not in step 1 of the wizard.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

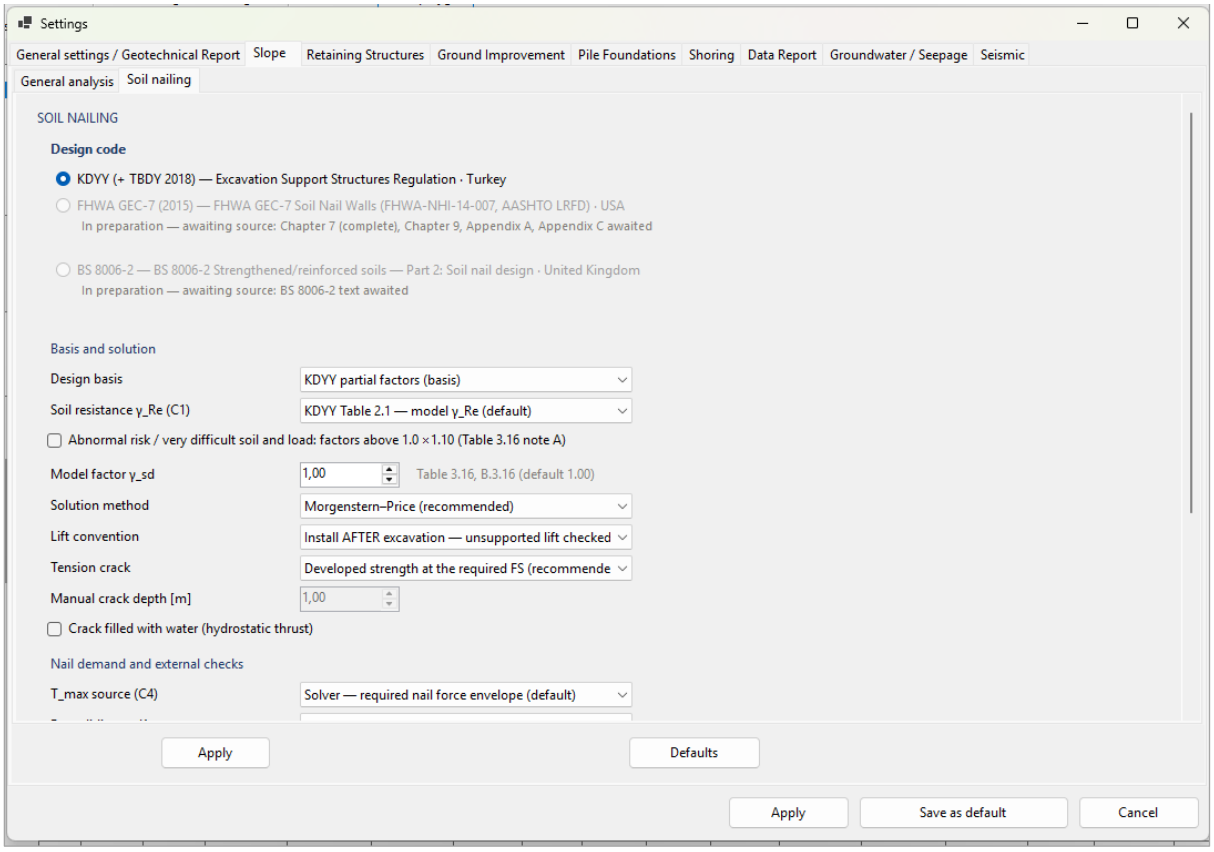


Figure 11. Settings window, Slope › Soil nail tab: design code, basis and solution, nail demand options

9.2 Geometry: ground, excavation face and lifts

The ground surface is the surface before excavation; its excavated state is generated by the program separately for each lift. The excavation face is a free polyline: it may be vertical, inclined or stepped. The lift elevations are entered by the user.

The lift surface is the point-by-point lower envelope of the original ground and the lift excavation profile. The lift excavation profile consists of the part of the excavation face down to that elevation and the horizontal bench extending from that elevation towards the excavation side. The bench is the plane on which the construction machine stands at the end of the lift.

The excavation direction is not guessed

If the excavation face is exactly vertical, the geometry does not tell on which side the excavation lies. In this case the program reports the direction as “undetermined” and asks the user for it; the direction in which the nails extend (towards the retained soil) depends on this information.

9.3 Lift convention

With a prestressed anchor the sequence is clear: excavate, drill, stress. Stressing is an event and separates the phase by itself. In soil nailing there is no such event; the nail is passive. So when the nail comes into action is not a physical fact but a declaration by the modeller. The program asks for this declaration as an explicit setting and writes the chosen option verbatim to the report and to the PLAXIS script.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Table 10. Lift convention options

Option	Analysis cases generated	Note
Install after excavation (recommended, default)	Two cases for each lift: n-K unsupported excavation (the nail of that lift is not yet installed) and n-C nail installed. Four lifts give eight cases.	Unsupported lift stability (FHWA “excavation lift stability”) is checked; this check often governs the lift height.
Install with excavation	A single case for each lift: excavation and installation together.	The unsupported lift is not checked; the displacement occurring before installation is not taken into account either. This is written in the note of each lift.

9.4 Nail rows and skin friction

The design is row-based: for each row the elevation, horizontal spacing S_h, vertical spacing S_v, length, inclination, drill hole diameter, bar diameter (or the bar cross-sectional area directly), steel yield strength, service type and corrosion protection are entered. Individual nails are derived from the row. A nail has no free length; bond acts over the whole length.

The ultimate skin friction τ_{bu} comes from the nail library based on KDYY Table 3.18. This library is separate from the anchor bond friction library: the two quantities are read from different tables and reduced with different factors. The library has been verified and consists of 26 rows (auger 6, open hole 7, rotary 8, driven casing 3, jet grouting 2). Each row shows the method, soil, condition, minimum, maximum and middle value, together with an FHWA cross-check (FHWA Table 4.4/4.5). The value transferred is the middle value by default and can be corrected; a value outside the range is flagged “OUT OF RANGE” in the report. A source note is written together with the value.

Table 11. Source of τ_{bu}

Source	Condition
Library	From the verified library; the source note is written automatically. KDYY Table 3.17 empirical value factors: $\gamma_k = 1.80$, $\gamma_{rb} = 1.10$.
Manual	A justification (source note) is compulsory; a manual value without justification is rejected. It is reduced like an empirical value (1.80 · 1.10).
Comparison	Value based on the pull-out test result (Section 9.9); reduced with ξ (1.1 lowest result / 1.3 mean) and γ_{rb} (1.3 coarse-grained / 1.7 medium and high plasticity soil). If the soil class is not selected, the row is invalid.
Effective stress	$\tau_{bu} = \sigma'_v \cdot \tan\delta$ (from the τ_{bu} helper); $\gamma_k = 1.35$, $\gamma_{rb} = 1.10$.
Total stress	$\tau_{bu} = \alpha_u \cdot c_u$ (KDYY 3.3.3.6-f; from the τ_{bu} helper); $\gamma_k = 1.80$, $\gamma_{rb} = 1.10$.
None	The row is counted as invalid; no calculation is produced.

For rock the limit, in accordance with KDYY 3.3.3.6-h, is the relation $\tau_{bu} \le \min(4\ \text{MPa}; 10\% \cdot \text{UCS})$; it is a relation, not a table value, and is applied as a relation in the program. For this, the rock UCS [MPa] is entered for the layer. If a nail passes through more than one layer, the τ_{bu} value of each layer can be entered; the program samples the bond length at 100 points and takes a weighted average according to the layers it passes through.

9.5 Partial factors and capacities

Table 12. KDYY partial factor chain (default values)

Quantity	Relation	Factor
Pull-out resistance	$\tau_{bk} = \tau_{bu} / \gamma_k$; $\tau_{bd} = \tau_{bk} / \gamma_{tb}$	$\gamma_k = 1.80$; $\gamma_{tb} = 1.10$
Bar rupture	$R_{tk} = A_s \cdot f_{yk}$; $R_{td} = R_{tk} / \gamma_s$	$\gamma_s = 1.15$
Action (tensile force)	$T_{d,max} = T_{max} \cdot \gamma_G$	1.35 static ; 1.00 earthquake

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The limit equilibrium solver applies a single resistance factor to the support resistance; in KDYY, however, pull-out and rupture are reduced with different factors. The program therefore produces the KDYY design value for each nail, multiplies it by the solver's factor and passes it to the solver; when the solver divides, exactly the KDYY design value is left. The other support elements in the model are not affected by this.

The γ_k = 1.80 in Table 12 applies to an empirical (library or manual) τ_{bu}. In accordance with KDYY Table 3.17 the factors vary with the source of τ_{bu}: empirical 1.80 · 1.10; effective stress 1.35 · 1.10; total stress 1.80 · 1.10; pull-out test ξ (1.1 / 1.3) · γ_{rb} (1.3 / 1.7). If abnormal risk is selected in the settings, all factors greater than 1.0 are multiplied by 1.10. The bar yield strength f_y cannot be entered lower than 420 MPa (S420 or higher ribbed bar; KDYY 3.3.3.7-a).

Corrosion: a permanent nail does not pass the calculation unless its protection is declared. If a sacrificial thickness is chosen, the loss can be derived from the relation $X = 80 \cdot t^{0.8}$ µm (t: service life, years); the diameter decreases from all sides (diameter loss 2X). With double protection the section is not reduced. If the bar area has been entered directly, the area after corrosion is reduced by the square of the diameter ratio, and the area used is written to the report.

9.6 Tensile demand: T_{max} and $T_{i,max}$

KDYY requires two separate tensile forces: the maximum force along the nail T_{max} for the rupture check, and the force $T_{i,max}$ in the length lying outside the slip surface for the pull-out check. Where these forces come from is selected with "Source of T_{max}" on the Settings → Soil nail tab (KDYY 3.3.3.7-c); the selected source is written to the report. No force distribution along the nail is produced, and no assumed distribution is used.

- Solver (default for a new wall) — from the required nail force in Section 9.13; for each row the largest of all lifts is taken and $T_{i,max} = T_{max}$ is assumed. On the partial factor basis this value is already a design value: $T_{d,max} = \lambda \cdot T_d$; the action factor of 1.35 is not applied a second time. On the characteristic basis $T_{max} = \lambda \cdot \min(q_s \cdot L_e; R_{tk}) / FS_{required}$ is taken and multiplied by 1.35.
- PLAXIS export — from the PLAXIS output pasted into step 4 of the wizard; force per nail $N_{max} \cdot S_h$, the largest over all phases. It is a characteristic force: $T_{d,max}$ = action factor·T_{max}.
- Manual — the values are entered by the user in the row table; the T_{max} and $T_{i,max}$ columns are visible only with this source.

FHWA's empirical relation $0.75 \cdot K_a \cdot \gamma \cdot H \cdot S_v \cdot S_h$ (half of that for rows in the lower third) is written only as a check row; it does not enter the check.

If no demand is entered

If a row has no demand (not entered with the Manual source, or the solver or PLAXIS has produced no value for that row), no check is produced; the capacities are reported and the note "KDYY rupture and pull-out checks require the tensile loads produced by the limit equilibrium solution" is added to the row.

For the order of magnitude, an estimate is reported as mobilised force ≈ design capacity / factor of safety; this value does not enter the check. Using the estimate in the check is an explicit choice, off by default, and when it is switched on the warning "THE DEMAND IS AN ESTIMATE" is written on every row. If the real demand is entered, the estimate is disabled.

When the demand is entered, the rupture and pull-out check ratios and the required bond length $L_{required} = T_{d,max} / (\tau_{bd} \cdot \pi \cdot D)$ are produced for each row.

9.7 Shotcrete facing

The head force T_0 is derived from T_{max} with the KDYY relation ($S = \max(S_v, S_h)$): for $S < 1$ m, $T_0/T_{max} = 0.60$; for $1 \le S < 3$ m, $0.5 + (S - 0.5)/5$; for $S \ge 3$ m, 1.00. It can be entered by hand if desired.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

By default the facing bending is calculated with the TS 500 strip method: $P = T_0/(S_v \cdot S_h)$, $M_0 = P \cdot S^3/8$; span moment $0.35 \cdot M_0$, support moment $0.65 \cdot M_0$; governing column strip moment $0.75 \cdot M_{support} + 0.60 \cdot M_{span}$; middle strip $M_0 - M_{column}$. The required reinforcement is $A_s = k_s \cdot M_{column}/d$. k_s is calculated from the TS 500 rectangular stress block; a k_s entered by hand takes precedence. The TS 500 minimum and maximum reinforcement limits are not checked; the report says "not checked". The simple span method remains as an option, but the report states that it gives a lower moment than the TS 500 strip method.

In accordance with KDYY-3.3.3.8.d, the punching check uses not T_0 but $T_{d,max}$. The head plate is also checked separately. The temporary facing is checked with the temporary facing thickness only.

In a permanent wall the final (permanent) facing is defined separately: concrete class, mesh, cover and headed studs (number of studs N_H, default 4; shank diameter D_{SC}; head diameter D_{SH}; stud length L_S; head thickness t_{SH}; yield strength $f_{y\text{-}hs}$; stud spacing S_{SH}). The temporary and final facings are checked separately:

- Bending — TS 500 strip method; each facing with its own thickness and reinforcement.
- Punching — for the temporary facing the rule above; for the final facing the cone geometry of FHWA Eq. 6.24/6.25, $D'_c = \min(S_{SH} + h_c; 2h_c)$, $h_c = L_S + t_P - t_{SH}$, concrete tensile strength TS 500 f_{ctd}; demand $T_0 = T_{max}$.
- Stud tension — $R_{FH} = N_H \cdot A_S \cdot f_y / \gamma_s$; in addition the conditions head area $A_H \geq 2.5 \cdot A_S$ and $t_{SH} \geq 0.5 \cdot (D_{SH} - D_{SC})$ are checked.

9.8 External checks and serviceability

Global stability is checked for each lift case with the slip surface search of Section 10. For base sliding the nailed block is treated like a gravity wall (KDYY-3.3.3.5; FHWA 5.7.3, Eq. 5.17–5.23). The block width B_L extends from the toe of the face to the farthest nail tip. Thrust height $H_1 = H + (B_L - H \cdot \tan\alpha) \cdot \tan\beta_{eq}$; β_{eq} is found from the ground within 2H behind the crest. K_a is calculated with the Coulomb relation and $\delta = \beta_{eq}$ is taken (a simplification permitted by FHWA); inclined Rankine can be selected only when $\alpha < 8°$. The resistance is $R_s = c_b \cdot B_L + (W + P_a \cdot \sin\delta) \cdot \tan\phi_b$.

A variable surcharge on top of the block is not counted in the resistance; a permanent surcharge is counted favourably with a factor of 1.0. On the partial factor basis the earth thrust is multiplied by 1.35 (1.00 in an earthquake), the variable surcharge thrust by 1.5 and the permanent surcharge thrust by 1.35; the resistance is divided by the γ_{Re} selected in the settings. In an earthquake, block inertia $k_h \cdot W$, Mononobe–Okabe K_{AE} (FHWA 6.40–6.43) or both are used. Missing input is not taken as zero: no check is produced and the quantity that is missing is stated.

Basal heave (FHWA 5.6.6) is applied only when the soil below the excavation base is undrained clay; otherwise "Not applicable" is written together with the reason. B_e, L_e and D_B are entered in step 2 of the wizard. $B'_e = 0.707 \cdot B_e$; resistance $R_S = S_u \cdot N_c \cdot B'_e$ or $S_u \cdot N_c \cdot D_B$; load $q = (B'_e \cdot H \cdot \gamma + Q - S_{u1} \cdot H) / B'_e$. The check is $\phi_{BH} \cdot N_c \cdot S_u \geq 1.35 \cdot q$; ϕ_{BH} is 0.65 short term and 0.50 long term (FHWA Table 5.5; in FHWA Table 6.3 these values are printed shifted by one row, and a note on this is added to the report). In addition $FS = 5.14 \cdot S_u / (H \cdot \gamma) \geq 2.0$ (short term) / 2.5 (long term) is written. If $S_u < 0.15 \cdot \gamma' \cdot H$ (trigger), the note "heave possible" appears in the report.

For serviceability (KDYY 3.3.3.9), the horizontal and vertical displacements are calculated with $\delta_h = \delta_v = (\delta/H)_i \cdot H$ and the zone of influence $D_{DEF} = C \cdot (1 - \tan\beta_4) \cdot H$ (β_4: angle of the face with the vertical). $(\delta/H)_i$ and C are read from KDYY Table 3.20 (= FHWA Table 5.12) according to the SLS soil class selected in step 3 of the wizard: weathered rock and stiff soil / sandy soil / fine-grained soil. The relation is valid only under these conditions: $L/H \geq 0.7$ (length of the top row), no surcharge, last lift $FS \geq 1.5$ (characteristic) or ≥ 1.0 (partial). If one of the conditions is not met, "stress–deformation analysis required" is written.

The limit is $\delta_h \leq 0.005 \cdot H$ (KDYY 3.3.3.9-f); if it is exceeded, the measures to be taken are written to the report. If the maximum horizontal displacement found with PLAXIS (stress–deformation analysis, GDA) is entered in step

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

1 of the wizard, it is shown as well. A creep warning is issued for a layer with PI > 20 and LI > 0.2; if there is a displacement-sensitive structure within D_{DEF}, a numerical analysis is compulsory; for cohesive soil the 1.5H note is written. The 0.1% / 0.2% / 0.3%·H values of FHWA 6.7.1 are shown for reference only. The D_{DEF} dimension is drawn on the drawing sheet in the CIVI-SLS layer.

The calculation approach is determined according to KDYY 3.3.3.1-g. If the excavation category is 2 and the neighbouring structure is not displacement-sensitive, or its distance is d > H, a ULS calculation alone is sufficient (g.1). In all other cases (g.2) a stress–deformation analysis (GDA, e.g. PLAXIS) is compulsory in addition to the ULS calculation; if the source of T_{max} is not the PLAXIS export, the KDYY 3.3.3.7-c warning is written. The category, d and the sensitive structure are entered in step 1 of the wizard.

The nail layout is checked according to KDYY 3.3.3.1-i and FHWA 6.3.3; every rule that is not met is written in the “Input warnings” block and in the report: S_v or S_h > 2.0 m (information note between 1.5 and 2.0 m); spacing < 1.07 m; $S_v \cdot S_h$ > 3.9 m^2; first row deeper than 1.07 m from the top; last row higher than 0.9 m above the base; inclination < 10° or > 20° (above 40° the nail may go into compression); length < 0.5H or > 1.2H (≈ 0.7H recommended).

All checks are collected in a ULS summary table (KDYY 3.3.3.1-j, Figure 3.20): internal stability, external stability, basal heave, base sliding, pull-out, rupture, facing bending (temporary / final), punching (temporary / final), stud and SLS. Each row shows E_d, R_d, ratio, verdict and the relevant clause. External stability (KDYY 3.3.3.4, FHWA 6.5.2) is found with a separate search: at every lift with nails installed the nails are treated as ineffective and only surfaces enclosing all the nails are searched — entry behind the farthest nail tip, exit at or in front of the toe, no nail cut. The solver, factors, tension crack and required factor of safety are the same as in the lift search. The type of the governing surface is stated: A/B deep circular or J forward translation (flat-based). If the external search finds a lower factor of safety than the general search at a lift, the report states this as a warning. The external search is not made in the intermediate analyses during length optimisation; it is made in the final analysis and fills row 2 of the ULS summary; the external critical surface is also drawn separately on the drawing sheet and in the preview. The following design notes are also written to the report: drainage (3.3.3.1-e; not included in the quantities), durability of the final facing in a permanent wall (3.3.3.1-d) and that anchor values cannot be used for nails (3.3.3.6-b).

9.9 Test programme, quantities and PLAXIS

Table 13. Nail test programme (KDYY 3.3.4, Table 3.21)

Comparison	Test load	Number
Short-term sizing test (KBD)	$P = T_{d,max} \cdot \gamma_d \cdot \xi$ (γ_d and ξ from the settings; defaults 2.0 and 1.1); test bond length $L = P/(\tau_{bd} \cdot \pi \cdot d)$	Category 2: ≥ 3 (≥ 1 in each soil type) · Category 3: ≥ 5 (≥ 2 in each soil type)
Acceptance test (KD)	$P = T_{d,max} \cdot k$ (k from the settings; default 1.25)	Category 2: 2%, ≥ 3 (≥ 1 at each excavation stage and in each soil type) · Category 3: 3%, ≥ 5 (≥ 1 at each excavation stage); the wall length is needed for the percentage

KBD nails are not production nails; they are installed separately between the production nails when the excavation reaches the relevant nail level. This warning is written verbatim to the report. The quantities give the number and length of nails, drilling length, bar weight and shotcrete area.

The numbers of tests come from KDYY Table 3.21 according to the excavation category (notes: 3.3.3.6-k, 3.3.4.1-c/d). The test load cannot exceed the structural limit of the nail: $P_{test} \le 0.80 \cdot R_{tk}$ (KBD) / $0.90 \cdot R_{tk}$ (KD), $\gamma_s = 1.0$; if it is exceeded, the required bar area is given. If the free test length L_{df} is entered, the minimum elastic elongation $\delta_n \ge 0.80 \cdot P \cdot L_{df} / (E \cdot A_s)$, E = 200 GPa is written.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

A loading schedule is produced for each test type: initial load P_0 = 10%; in KBD, load increment ≤ 20%, at least 2 cycles, first cycle ≤ T_d; in KD, at least 5 increments; readings at 0–1–2–5–10–15–20–60 min; criteria Δ ≤ 0.5 mm and creep v < 2 mm.

The test readings entered in step 8 of the wizard are evaluated according to KDYY 3.3.4.3 (B.3.18, B.3.31–32). For each test P_{res} (Δ > 0.5 mm) and P_{cr} (v ≥ 2 mm; t_1 the first non-zero reading of the increment, t_2 the last reading) are found; P_{ult} is the smaller of them and $\tau_{bu,t} = P_{ult} / (\pi \cdot D \cdot L_{b,t})$. The lowest or mean value of the KBD tests is reduced with ξ and γ_{rb}; B.3.18 is taken as the basis for design, and the B.3.32 result is shown alongside. KD tests are evaluated against the acceptance criterion. The result is written to the layer with the "Test" source via "Transfer to selected layer".

A complete command script is produced for PLAXIS; the command templates are the same as those of the PLAXIS export of the Excavation Support Structures module. The script sets up the model boundaries, the Hardening Soil soil materials (derived from E_{oed} or E_s and ν; if these are not entered no script is produced, and stiffness is not estimated), the soil body as polygons (cut at the layer and lift elevations on the excavation side; the excavation at each lift is made automatically by deactivating the relevant polygon), a separate shotcrete plate for the face segment of each lift, the nails (embedded beam row, behaviour "Rock bolt"), the surcharge, the groundwater and the mesh. The phases are written in the order of the lift convention; no element is activated in the excavation phase, and the nail and facing of that lift are activated in the "n-C" phase; a separate Safety branch is started from every lift phase, so that the PLAXIS factor of safety can be compared lift by lift with the staged limit equilibrium table. Only the command lines are copied to the clipboard (Section 16).

9.10 Verification: KDYY P20 example project

The program's soil nail calculation reproduces the figures of the P20 example project in the KDYY Design Handbook. In the comparison, the values to which KDYY rounds the bar areas (800 mm² for ϕ32, 600 mm² after corrosion) were used by entering the bar area directly.

Table 14. Comparison with KDYY P20

Quantity	Program	KDYY P20
Stiff clay τ_{bk} / τ_{bd} (kPa)	25.000 / 22.727	25.0 / 22.7
Sandstone τ_{bk} / τ_{bd} (kPa)	105.556 / 95.960	105.6 / 96.0
ϕ32 R_{tk} / R_{td} (kN)	336.0 / 292.2	336.0 / 292.2
50-year diameter loss 2X (mm)	3.658	3.6
$T_{d,max}$ rupture / pull-out, static (kN)	204.12 / 160.65	204.1 / 160.7
Required bond length, nails 1 / 8 / 11 (m)	5.27 / 6.39 / 4.10	5.3 / 6.4 / 4.1
T_0 (kN) ; P (kPa)	112.49 ; 50.00	112.5 ; 50
M column strip (kNm)	14.71	14.7
Required A_s (mm²/m)	233.4	233
Punching capacity (kN)	300.0	300
δ_h limit (m)	0.0875	0.0875

The remaining parts of the soil nail calculation have also been checked against independent verification examples and hand calculations.

9.11 Design wizard

In the Slope module, the "Soil nail" button on the Analysis toolbar opens the wizard. The wizard does not rewrite any calculation rule: it converts the user's plain design inputs into the program's calculation model (pre-excavation ground, layers, water, surcharge, earthquake and the nailed wall) and runs the staged analysis. The method choices come from the Settings → Soil nail tab (Section 9.1). It has nine steps:

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

- 1 · General — service (temporary / permanent), corrosion protection, service life, excavation side, excavation category (2 / 3), distance d to the neighbouring structure, displacement-sensitive structure, maximum horizontal displacement found with PLAXIS (GDA) [mm], and the resulting g.1/g.2 explanation (Section 9.8). The design basis, lift convention, solution method and tension crack are not selected here: a read-only summary of the settings is shown, and the "Settings…" button opens the Soil nail tab.
- 2 · Geometry — excavation depth, face angle, benches, slope behind the crest and the plan length of the wall (for the quantities, the number of tests and the elevation drawing; if not entered, total values are not produced). The "Basal heave (FHWA 5.6.6)" field contains a status sentence and B_e, L_e and D_B; the field is enabled only if heave applies (undrained clay below the excavation base); otherwise it states the reason as "Not applicable: …".
- 3 · Soil — horizontal layers and groundwater level (YASS). Layers are entered by hand, taken with their depths from the borehole selected in the Geotechnical Report (only inside the main program), or read from the current Slope drawing (along the vertical column at the highest point of the ground). The table also has PI, LI, $(N_1)_{60}$ and rock UCS [MPa] columns; when taken from the Geotechnical Report, $(N_1)_{60}$ is the average of the measured SPTs within the layer depth (refusals are excluded), PI is carried over, and LI is calculated only if w, PL and PI are entered. The source of τ_{bu} is Library, Manual, Effective stress, Total stress or Test (Table 11); the SLS soil class is selected according to KDYY Table 3.20. E_s, ν and E_{oed} are only for the PLAXIS export; E_s and ν come from the Geotechnical Report, and if E_{oed} is empty it is derived from them.
- 4 · Nails — automatic layout parameters (first row depth, S_v, S_h, last row–base clearance, length, inclination, drill hole and bar diameter, f_y, sacrificial thickness, PLAXIS equivalent E) and the row table. If the table is empty, the rows are laid out from the parameters; "Auto layout" fills the table with this arrangement, and the depth, length, inclination, drill hole and bar diameter, f_y, S_h and PLAXIS equivalent E of each row can be changed individually, and rows can be added and deleted. If the table is filled, each row is a nail row. The T_{max} / $T_{i,max}$ columns are visible only when the source of T_{max} is Manual; the demand source is shown on a separate line. With the PLAXIS source, the PLAXIS output is pasted into the "Paste from PLAXIS" box, and "Read and match →" reads the phase, EmbeddedBeamRow_N and N_{max} [kN/m] values and matches them to the rows; the force per nail becomes $N \cdot S_h$ (that the unit is per meter should be confirmed in PLAXIS). f_y < 420 MPa is rejected (KDYY 3.3.3.7-a).
- 5 · Excavation lifts — automatic (each lift goes down a little below a row) or manual.
- 6 · Loads — surcharge, surcharge type (variable q_v: 1.5 / favourable 0 · permanent q_p: 1.35 / favourable 1.0) and earthquake (S_{DS}).
- 7 · Facing — temporary / permanent shotcrete, concrete class, mesh, head plate; for a permanent wall also the "Final (permanent) facing" section: concrete, mesh, cover and headed studs (Section 9.7).
- 8 · Pull-out tests — free test length L_{df} and γ_{rb} class; the test nails table (name, type KBD/KD, D, $L_{b,t}$, L_{df}, A_s) and the readings table (P, t, s). "Evaluate" evaluates the tests, "Transfer to selected layer" writes the result to the layer (Section 9.9).
- 9 · Preview and analysis — section, staged analysis, the "Input warnings" block and the result tabs (Section 9.14).

The "τ_{bu} helpers…" window in step 3 derives a value for the selected layer and writes it to the row together with its source: total stress $\alpha_u \cdot c_u$ (α_u 0.25–0.75); effective stress $\sigma'_v \cdot \tan\delta$, $\tan\delta = (0.75–1.00) \cdot \tan\phi$; SPT → τ (FHWA Table 4.6; only if $(N_1)_{60}$ has been entered); rock ceiling $\tau_{bu} \le \min(4\ \text{MPa}; 10\% \cdot \text{UCS})$; test ξ/γ_{rb}. "Stiffness reference table…" shows FHWA Table 4.3a/b/c; it is read-only and fills no box. The library window shows the 26 rows of KDYY Table 3.18 (Section 9.4).

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The wizard input is stored in the main project file; the wizard opens with the same input next time. During the first seven steps a small section preview sits on the right of the window: as the input changes the model is rebuilt (no analysis is run) and the element the step deals with is highlighted — the excavation face in Geometry, coloured layer bands in Soil, the nails in Nails, the lift elevations in Excavation lifts, the surcharge and water in Loads, and the facing in Facing. If there is an error in the input, it is written in the heading of the preview.

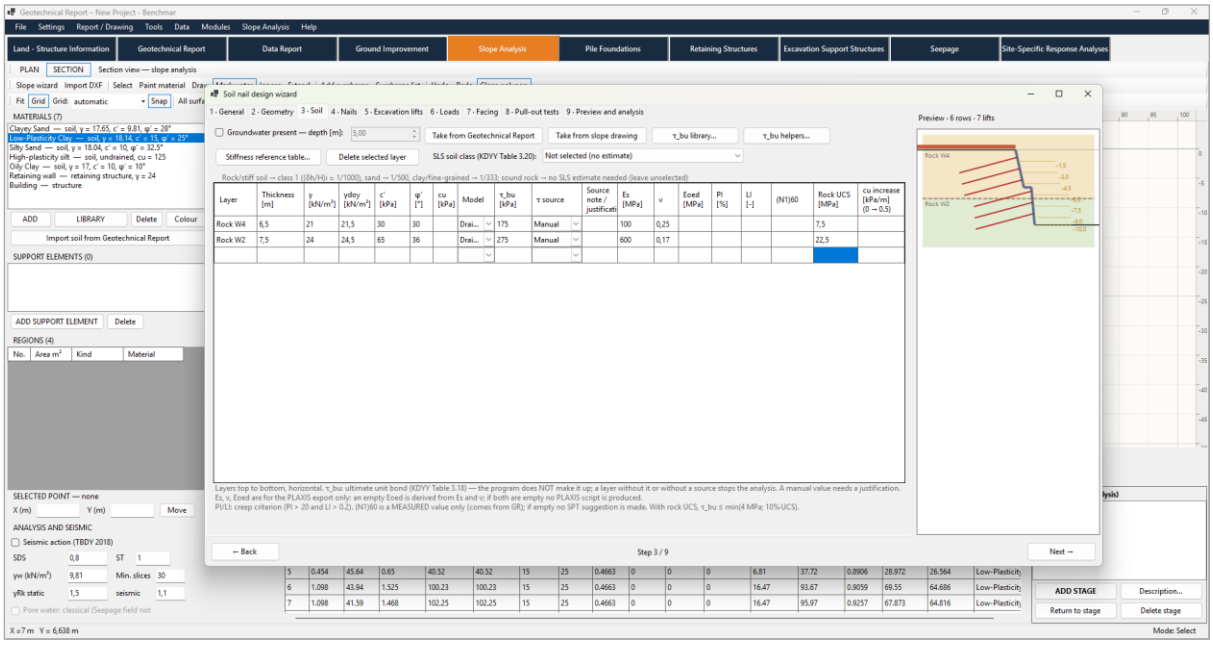


Figure 12. Soil nail design wizard, step 4 · Nails: layout parameters, row table and section preview

9.12 Solution method and tension crack

In a soil-nailed wall the nail forces enter the force equilibrium. TBDY 2018 Eq. 16.30, which considers moment equilibrium only, is therefore not used for soil nails, and this sentence is written to the report. The search is made with Morgenstern–Price (default) or Spencer; the nail force enters the slice whose base it cuts as an external force (its vertical component in the base normal, its horizontal component in the interslice equilibrium, its moment in the moment equilibrium). To ensure convergence on vertical faces, the solution scheme of Zhu, Lee and Jiang (2005) is used.

For a solution to be accepted, the interslice forces must be admissible (Morgenstern and Price 1967) and the factor of safety must lie within 0.5–2.0 times the Janbu value of the same surface. A surface that does not converge is eliminated without falling back to another method, and the number of eliminated surfaces is reported.

The crest zone of a cohesive soil works in tension; counting cohesion as strength in this zone inflates the factor of safety of a vertical face in a non-physical way. For this reason a vertical tension crack is assumed in the crest zone: $z_c = 2 \cdot c_d / (\gamma \cdot \sqrt{K_{a,d}})$, $c_d = c'/F_{ref}$. F_{ref} is the required factor of safety on the characteristic basis and γ_{Re} on the partial factor basis (developed strength). The depth can also be given by hand; if the crack is selected as water-filled, the hydrostatic thrust $\tfrac{1}{2} \cdot \gamma_w \cdot z_c^2$ is added. If the open face height does not exceed $2 \cdot z_c$, no mechanism forms; the verdict “adequate” is given and no factor of safety is written.

These rules are applied to soil nails only. The normal analysis of the Slope module (Section 10) does not change; for ordinary slopes TBDY 16.30 and the slice width assumption continue unchanged.

9.13 Required nail force and length optimisation

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The required nail force is obtained from the solver: the critical surface of each lift is held fixed, all nail forces entering the surface are scaled by the same factor λ, and the λ giving FS(λ) = required FS is found. $T_{required} = \lambda \cdot T_d$. λ < 1 shows that the nails are stronger than needed, λ > 1 that they are inadequate. This value is not KDYY's T_{max} / $T_{i,max}$ (which comes from the force distribution along the nail) and does not enter the KDYY check; it is reported as a separate table.

Length optimisation produces two proposals. Uniform length: the shortest common length for which all lifts are adequate, searched by bisection in 0.5 m steps. Graded length: starting from the uniform length, each row from the bottom upwards is shortened in 0.5 m steps as long as all lifts remain adequate; a row cannot be longer than the row above it. Acceptance criterion: no lift gets worse as the length increases. Both proposals are verified by re-running at full quality; the user applies whichever he or she wants.

When length optimisation is started, the "R pattern" window opens first (FHWA 6.3.3e): the rows in the upper half have the same length L, and the length of the rows in the lower half decreases linearly from mid-height to R·L at the base. The range of R is given according to the selected soil: very dense coarse-grained soil 0.15–0.30 · sand 0.25–0.40 · fine-grained soil 0.30–0.45. The shortest L is found as in the uniform length search and verified at full quality.

9.14 Results and outputs

The results are shown on tabs: lifts (FS, tension crack depth, required value and verdict for each lift), row checks in the governing case, required nail force, facing · base sliding · test programme · quantities and notes; there are also the ULS summary, "Sliding · heave · SLS", "Test programme" (with the loading schedules) and "Final facing" tabs. The screen, the Word report and the drawing sheet read from the same calculation.

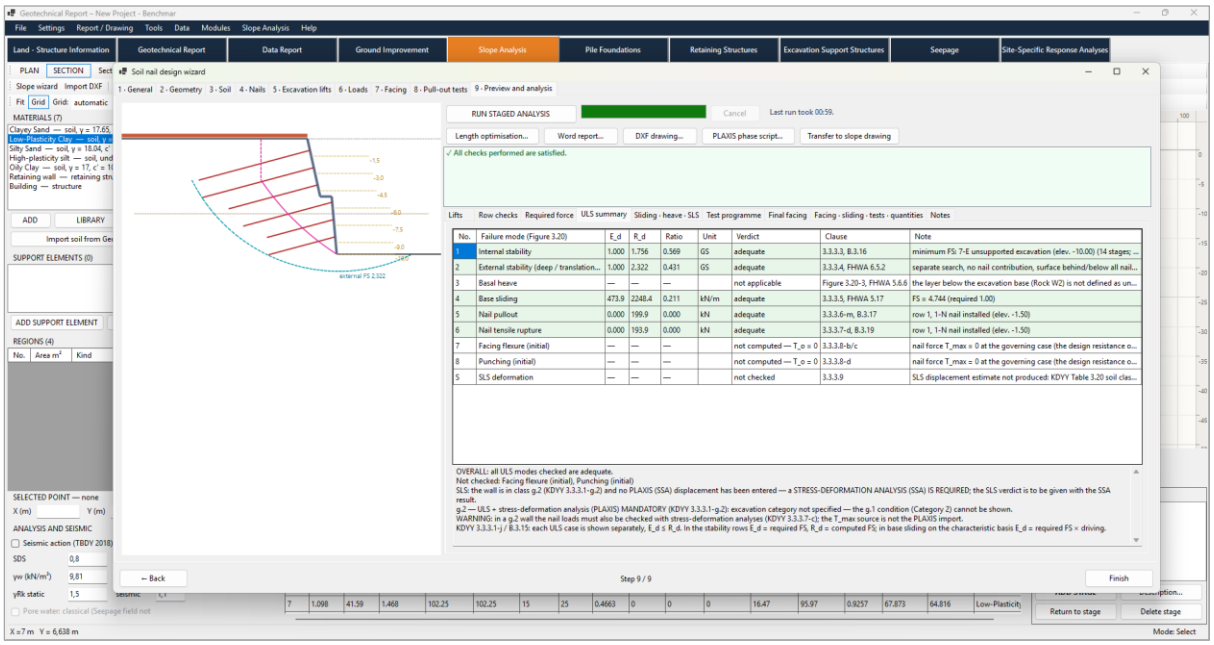


Figure 13. Soil nail design wizard, step 9 · Preview and analysis: staged analysis result and lifts table

- Word report — 1 service, basis and method · 2 geometry and soil (with the section image) · 3 nails and excavation lifts · 4 staged analysis · 5 row checks · 6 required nail force · 7 facing, nail demand, factors and quantities · 8 ULS summary · 9 SLS and g.1/g.2 · 10 basal heave · 11 base sliding · 12 final facing and studs · 13 test programme and loading schedules · 14 test evaluation · 15 design notes and drainage · Conclusion. The conclusion looks at all rows of the ULS summary and at SLS; if there is an item that could not be checked, it says "the design is NOT FINAL".

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

- DXF drawing sheet (A1) — section (final excavation surface, layers, water, facing, nails, lift elevations, bench elevations and widths, governing slip surface; the drill hole is drawn to true scale with two parallel lines and the bar with its centreline, and the label of each nail — row, length, inclination, drill hole/bar diameter — is written along the nail direction), elevation (head plates drawn to true scale as squares, each row with its own S_h spacing; bench elevations with dashed lines; a typical segment if the wall length is not entered), nail table, analysis result, 1/10 head detail and the common title block. The sheet also contains the D_{DEF} dimension and δ_h in the CIVI-SLS layer, the final facing face for a permanent wall, the studs in the head detail, and the heave, SLS, calculation approach (g.1/g.2), final facing and overall verdict rows in the result table.
- PLAXIS script — model, materials, soil polygons, facing, nails, surcharge, water, mesh and a Safety branch from every lift (Section 9.9); only the commands are copied to the clipboard, and the annotated version and the notes on the derived values are on a separate tab.
- Transfer to Slope drawing — the final state (lines, layer regions and their materials, water, surcharge, nails) replaces the Slope drawing; you are asked first, and you can go back in a single step with Undo. The nails are transferred with their nominal resistances. The transferred drawing is for editing and general checking; the normal Slope analysis does not use the rules in Section 9.12.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

10. Analysis

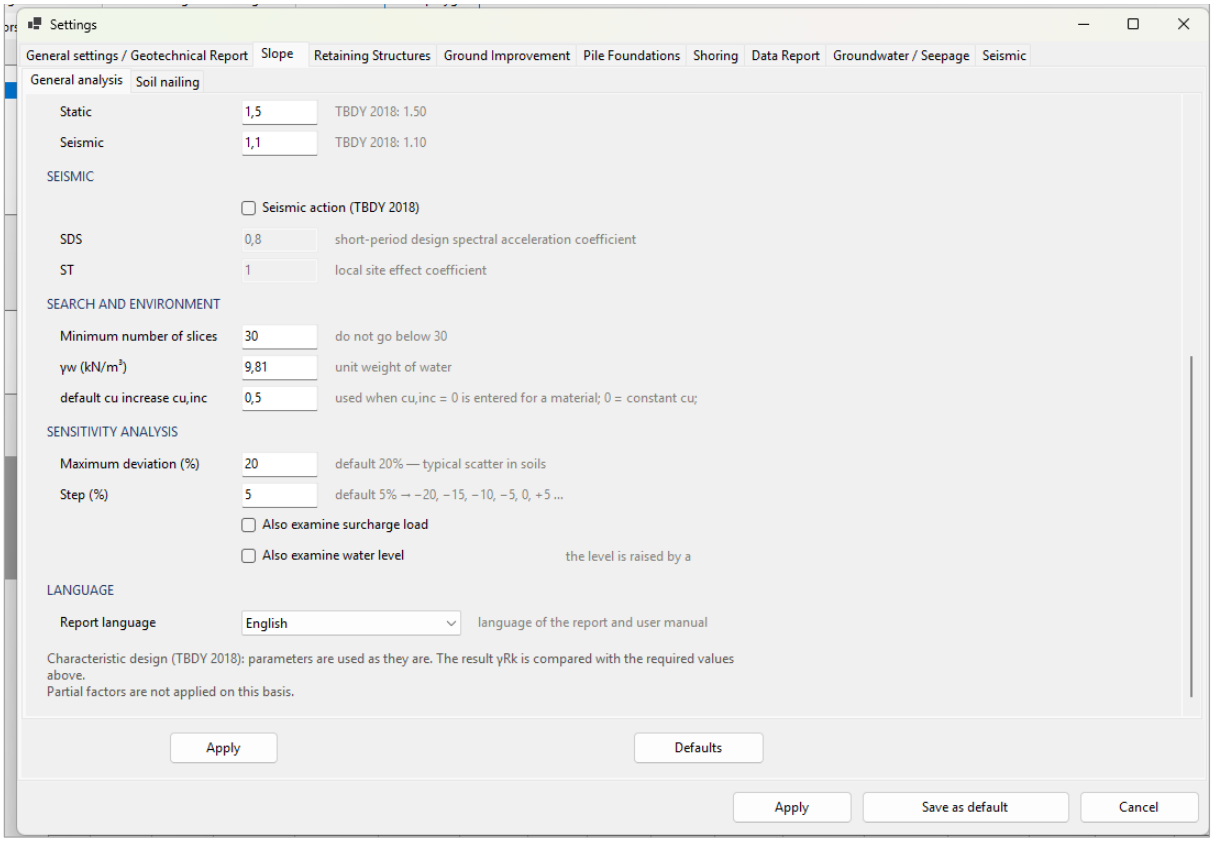


Figure 14. Settings window, Slope › General analysis tab: factors of safety, earthquake, search and sensitivity analysis settings

10.1 Slip surface generators

The program generates slip surfaces from eight different geometric families. Each family represents a different failure mode; all of them are scanned and the most critical is selected. Relying on a single family means missing a mechanism that lies outside that family.

Table 15. Slip surface generators

Generator	When it becomes critical
Circular — entry/exit	The most common mechanism in homogeneous soils
Circular — centre grid	Deep-seated mechanisms
Circular — three-point	Surfaces passing through ground break points
Tangent to layer	Surfaces grazing the top of a weak layer
Composite	When there is sound bedrock below a shallow cover
Weak-seam following	When there is a thin, low-strength band
Wedge (two-block)	Planar weakness; rotation around the pile tip
Undercut	When the soil squeezes out from beneath a structure. The surface starts just below the underside of the structure; as exits, equally spaced points on the ground, the corner points of the ground (including the excavation base corner) and the ¼, ½, ¾ points of vertical faces are tried; for each entry–exit pair a straight wedge and downward-sagging surfaces are generated.

The report contains only the mechanisms that were generated. Families that could not be generated because of the model geometry — for example composite when no bedrock is defined — do not appear in the list. A family

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

missing from the list does not mean that the mechanism was found to be safe; it means that it was not generated at all.

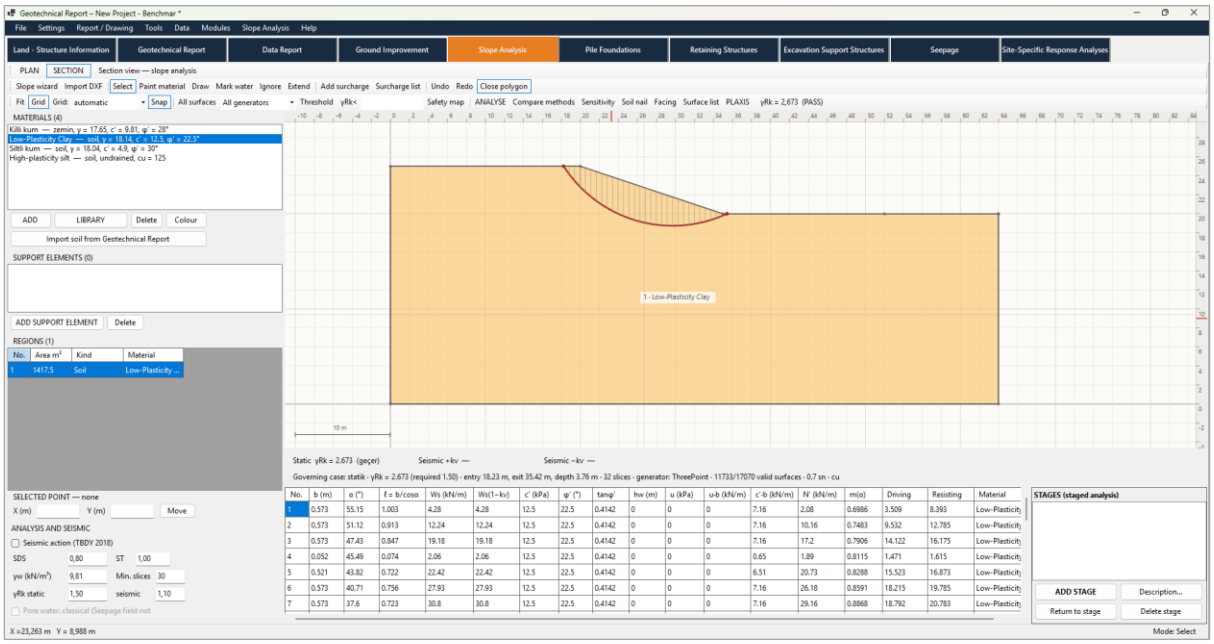


Figure 15. Critical slip surface found with the weak-seam following generator, and the slice table

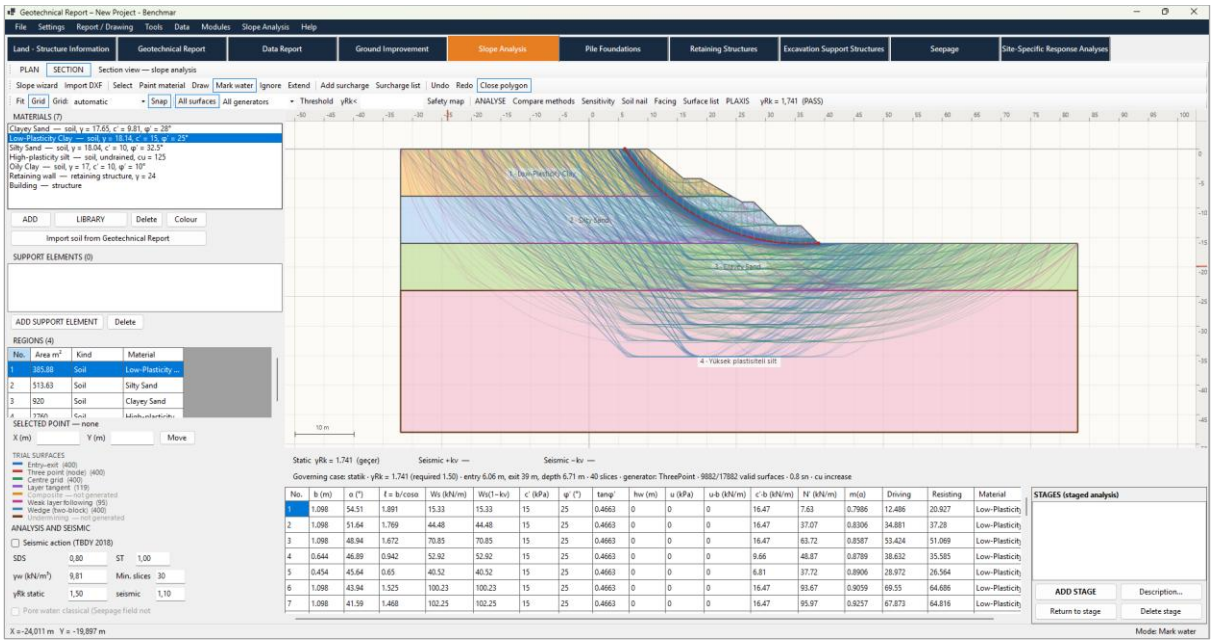


Figure 16. All surfaces view: evaluated slip surfaces coloured by factor of safety

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

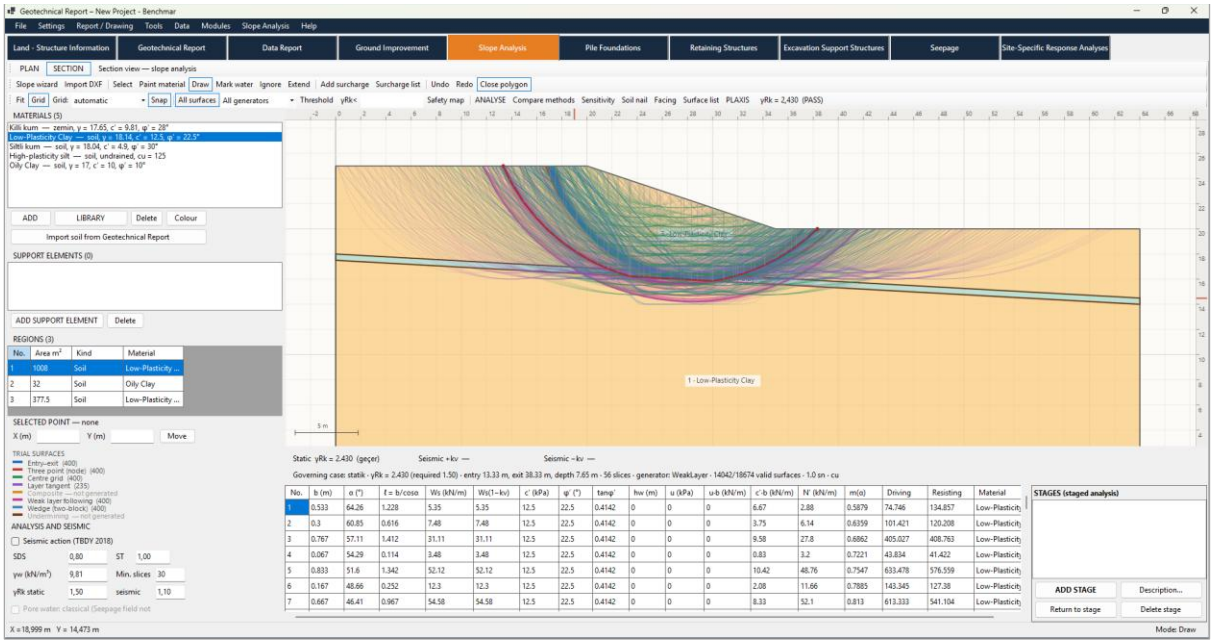


Figure 17. Safety map: centre grid points coloured by factor of safety

10.2 TBDY 2018 Equation 16.30

For circular surfaces, the moment equilibrium equation prescribed by the code is used. The logic of the equation is simple: the ratio of the moments resisting to the moments trying to rotate the sliding mass is taken.

$$\gamma_{Rk} = \Sigma \left[c'\cdot b + (W_s(1-k_v) - c'\cdot b\cdot \sin\alpha/\gamma_{Rk} - u\cdot b)\cdot \tan\varphi' / m(\alpha) \right] / \Sigma \left[W_s\cdot \sin\alpha + k_h\cdot W_s\cdot(\cos\alpha - H_c/R) \right]$$

$$m(\alpha) = \cos\alpha + \sin\alpha\cdot\tan\varphi' / \gamma_{Rk}$$

The equation contains γ_{Rk} on both the left and the right side and is therefore solved iteratively. The program starts from an initial value and repeats until it converges.

10.3 Other methods

On polygonal (broken) surfaces moment equilibrium is undefined — there is no centre of rotation. These surfaces are solved with the Janbu method, which is based on force equilibrium. The program itself selects the right method according to the surface shape. The general limit equilibrium (GLE) solver also takes the interslice forces into account and produces the Bishop, Spencer and Morgenstern–Price results.

A specific method can be forced from the Settings window. If “TBDY 2018 only” is selected, polygonal surfaces are not scanned; the report states this clearly.

10.4 Loading cases

If the earthquake action is on, three cases are solved: static, earthquake $+k_v$ and earthquake $-k_v$. The vertical acceleration is examined separately in the upward and downward directions, because which of them is more critical depends on the geometry and cannot be known in advance.

The governing case is not the one with the lowest factor of safety but the one with the lowest ratio to the required value. This distinction matters because the required value is lower in the earthquake case: a section giving 1.60 in the static case is more comfortable even if it gives 1.20 in the earthquake case.

The horizontal and vertical pseudo-static seismic coefficients are derived from S_{DS} and the local site effect coefficient according to the TBDY 2018 principles; they are entered in the settings and written in the report.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

10.5 Staged analysis

After the analysis, the ADD STAGE button and the stage list are in the bottom right corner. A stage is a frozen copy of the current state: the geometry, materials, support elements, facing, water, loads, the (excavated) lines marked with “Ignore” and the analysis result (factor of safety, critical surface and its image) are stored together. When a stage is added, a description is typed by hand (for example “1st excavation: down to elevation +2.00”).

As the excavation advances or the structure is built, the model is changed, analysed again and a new stage is added. Lines excavated by marking them with “Ignore” are saved to the stage; which excavation was made in which stage can be seen in the stage list and in the report.

Each stage is solved separately with its own frozen section and with the solver and search rules of the normal analysis. No stress or strain history is carried between stages; the stage results are independent of one another and are compared to see the weakest stage.

The stage list has three commands: “Description…” changes the description of the stage; “Return to stage” loads the section of the stage into the working area (you are asked first, because it replaces the working area); “Delete stage” removes the stage from the list. The stages are stored together with the main project file; no stage is stored inside another stage.

If stages have been added, two separate reports can be produced: the “Slope analysis report” contains only the current section; the “Staged slope analysis report” gives each stage as a separate section and compares the stages (Section 15.1). The DXF drawing of each stage can also be produced separately (Section 15.2).

10.6 Slope face shotcrete facing

The shotcrete shell of an unnailed slope face is defined with the “Facing” button on the Analysis toolbar. In the window you enter whether the facing is taken into account, the thickness [cm], the concrete class, the mesh (in the form Q221/221 or R257/131), whether there is drainage (weep holes) under the shell, the local pocket span [m] and the slope width for the quantities [m]. The facing is off by default; while it is off, no calculation, report or drawing changes. The facing of a soil nail wall is separate from this (Section 9.7).

The face to be covered is specified line by line: right-click the slope face line and choose “Define SHOTCRETE facing on this line…”; “REMOVE shotcrete facing from this line” deletes the assignment. The assigned line is shown on screen as a thick orange line with a thickness label. If an assignment has been made, only those ground surface segments that coincide with the assigned line and whose slope exceeds the threshold (default 15°) are covered; if there is no assignment, the whole face whose slope exceeds the threshold is covered. If an assignment is made to a flat or nearly horizontal line, a warning is issued; the assignment is not blocked.

In the analysis the facing enters only as weight (thickness × unit weight; a vertical distributed load along the surface). No shear, bond or tensile contribution is counted. The factor of safety is given side by side without and with the facing; the value with facing is presented both on the critical surface without facing and on the facing model’s own critical surface. In this way the contribution or harm of the facing load to the slope can be seen.

The shell is also checked with two preliminary checks: (1) comparison of the strip force required by a shallow sliding wedge parallel to the face (0.5 m and 1.0 m depth) with the tensile capacity of the mesh and the compressive capacity of the concrete; (2) a bending and shear check of the shell (TS 500) when a local pocket forms beneath it. The factors and formulas adopted are written in the report. If the facing is active, the report also gives the facing quantities and the SEV_PUSKURTME_KAPLAMA layer is added to the DXF drawing.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

10.7 Calculation method, cohesion term and warnings

The “Calculation method” list in the analysis settings selects the solver that calculates the factor of safety of each candidate surface: Automatic (recommended; TBDY 2018 on circular surfaces, Janbu on polygonal surfaces), TBDY 2018 only, Fellenius, Bishop simplified, Janbu simplified, Janbu corrected, Spencer, Morgenstern–Price, Corps of Engineers and Lowe–Karafiath. The shape of the surface, on the other hand, is determined by the generators in Section 10.1. Method and generator are separate concepts but affect each other: if the selected method does not converge on a surface, that surface is counted as invalid; the program does not switch to another method without notifying the user. For this reason, when the method changes, the set of valid surfaces and the critical surface may also change.

In a search with Spencer and Morgenstern–Price, the admissibility of the interslice forces is checked, together with whether the result stays within a reasonable band relative to the other method results for the same surface; a root failing this check is considered unreliable and eliminated. If there are candidate surfaces that could not be solved with Spencer, their number and the lowest Bishop/Janbu value of these surfaces are written as a warning; the verdict is given with the Spencer result and there is no fallback to a backup method. In the global failure check of the Excavation Support module, under-building (undercutting) wedges on which Spencer gives no root are evaluated with corrected Janbu and, if confirmed by Lowe–Karafiath on the same surface within the band [0.5; 2], are included in the factor of safety; this fallback is not switched on in the Slope module.

In Automatic mode a circular surface is solved with TBDY 2018 and a polygonal surface with Janbu; these two numbers are the products of different solver families. The program additionally solves the best circular and the best polygonal surface with Spencer; if the order changes or the two numbers differ by more than 10%, it writes a warning to the result line and the report. The verdict is not affected by this warning. To check the critical mechanism with a single solver family, Spencer or Morgenstern–Price should be selected. If the effective normal force is negative (tension zone) on half or more of the slices of the critical surface, the surface is not eliminated, but a warning is written.

The cohesion term (Analysis settings) is applied in one of two formulations: TBDY 2018 (default; the formulation in Eq. 16.30/16.31, b the slice width) or classical (Bishop; $c' \cdot l \cdot \cos\alpha$, l the base length). On flat slopes the two formulations are close to each other; as the slice base gets steeper (steep excavation, steep rock face), the TBDY formulation counts less cohesion and gives a lower factor of safety on the same surface than the classical formulation (Table 23). The choice is written in the report summary and in the search message; when classical is selected, you are told that it differs from the code formulation and that code compliance is the user’s decision. The Method comparison window additionally shows the classical Bishop value on a circular surface as an information row; this row does not affect the governing factor of safety.

10.8 Pore water: transfer from the Seepage module

If the u(x,y) field of the Seepage module has been transferred to Slope through the main program, the “Pore water: use the Seepage u(x,y) field” box below the Analysis settings panel becomes available. If no transfer has been made, the box is dimmed and reads “Pore water: classical (Seepage field not transferred)”. When the box is ticked, the pore water pressure at the slice base is read by linear interpolation over the triangular mesh of the Seepage solution; for slices outside the field, the classical hydrostatic pressure based on the water line is used. By default suction ($u < 0$) is taken as zero (safe side). If “Suction transfer to Slope ($u < 0$)” is turned on in the Seepage settings, $u_eff = \chi \cdot u$ is used in Slope: the global χ ($\chi \approx \tan\phi b / \tan\phi'$) starts at 0.50 when the box is ticked, and a material with “χ Slope” entered in the Seepage material table uses its own χ. With $\chi = 0$ suction is taken as zero (default rule). Suction transfer is a USER ASSUMPTION. When the box is unticked, the calculation is identical to the classical water line calculation. The choice and the field are saved with the main project file; the report states the field used (number of nodes and triangles, transfer date).

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The “Seepage → Slope: Transfer Water Line…” command in the Data menu of the main program, on the other hand, deletes the water line in Slope and writes the free water surface (phreatic line) of the Seepage solution; you can go back with Undo. If the two sections overlap horizontally by less than 10%, the transfer is rejected.

10.9 Saving settings as defaults

The “Save as default” button in the Settings window writes, on the Slope tab, the analysis settings (factors of safety, earthquake, partial factors, cohesion term, etc.), the calculation method and the sensitivity limits (maximum deviation and step) to the settings file in the %LocalAppData%\GeoteknikPro Studio\Ayarlar folder; the program does not write to the Documents folder on its own. The saved values are applied only to newly opened projects; the settings of existing projects do not change. The button is also found on the settings pages of the other modules.

10.10 Back analysis

Back analysis back-calculates the drained strength parameters (c′ and ϕ′) of an unknown layer in a failed slope. The “Back analysis” button on the Analysis toolbar (next to the “Soil nail” button) opens the “Back analysis — slope” window. The window works on a copy of the section; the original model does not change until “Write to material” is used. The calculation runs in the background and can be stopped with Cancel.

First a material is selected from the “Unknown layer (all others fixed):” list; the values of the other layers stay fixed. There are two modes:

- “Mode 1 — I give the entry / exit points”: the “Entry x [m]:” and “Exit x [m]:” values of the failure surface and, if wanted, its depth with “Give depth [m]:” are entered. For these end points circular, composite, weak-seam following and wedge surfaces are scanned; the type that comes out critical is reported. The undercutting family is not scanned in this mode.
- “Mode 2 — I enter nothing; global circular search”: a global search is made with circular surfaces only; the end points of the critical surface are given for each ϕ′. “Search quality (Mode 2):” is set to Fast, Normal or Detailed.

The “Sought” group has two options. “Fresh failure: c′ and ϕ′ (c′–ϕ′ curve)” finds, for each ϕ′, the c′ that makes the lowest factor of safety of the slope equal to the target. “Old landslide: residual ϕ′r (c′ fixed, kPa):” keeps c′ at the entered value and searches for the residual friction angle. “Target FS (low / mid / high):” is 0.95 / 1.00 / 1.05 by default; “c′ upper limit [kPa]:” is the upper limit of the search. Because the factor of safety increases monotonically with c′, c′ is found by interval narrowing; every point is solved with the search’s own evaluator.

The result is shown on two tabs. “c′–ϕ′ chart” shows the curves of the target factors of safety and the points entered in the “Laboratory / correlation points (overlaid on the chart)” table. The “Searched slip surfaces” tab draws a sample of the searched surfaces, their envelope and the critical surface (entry and exit points in Mode 1, end ranges in Mode 2). “Select pair” picks a (c′, ϕ′) pair from the chart; “FS with this pair” calculates the lowest factor of safety of the slope with this pair; “Write to material” writes the pair to the unknown layer after confirmation.

The “Correlation band…” button in the “Correlation band (optional)” group calculates the lower–upper c′ and ϕ′ range of the Correlations module from the soil and test inputs. If “Overlay on chart and report” is ticked, this range is drawn on the chart as an orange rectangle. The band gives a range only, not a recommendation; the program does not block, it only warns.

“Report (Word)…” writes a separate back analysis report: inputs, fixed layers, c′–ϕ′ chart and table, selected pair, laboratory points, critical surface and warnings. The back analysis result is saved in the project file and also appears as a separate chapter in the slope analysis report (Section 15.1).

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

11. Design Bases

The program can calculate according to two different bases. The difference between them is where the safety margin is held.

Table 16. Design bases

Basis	How it works	Expected
Characteristic (TBDY 2018)	Parameters are used as they are; the safety margin is held in the required value	static 1.50 · earthquake 1.10
Partial factor (KDYY)	Actions are increased, resistances are reduced; the safety margin is in the factors	1.00

Applying both is double counting

Increasing the loads by 1.35 and then also expecting 1.50 effectively demands 2.03; this is an excessive condition found in no code. The program makes this distinction automatically and states in the report on which basis the calculation was made.

The design basis and partial factors are saved with the main project file.

The Settings menu lies between View and Analysis, and the settings window has two tabs: “General analysis” and “Soil nail”. In General analysis the design code is not selected separately; it follows from the design basis and is shown read-only: characteristic basis → TBDY 2018, partial factor basis → KDYY 2026 (Table 2.1). The code for soil nails is selected on its own tab (Section 9.1).

The design code selection follows the same rule in all modules: only codes implemented in the program (Active) can be selected. Codes not yet implemented appear grey, with the note “Not available in this version.” for FHWA GEC-7, Chapter 7, Chapter 9, Appendix A, Appendix C; for EN 1997-1, the text and the national annex). No factor of a code without a source is written into the program. Projects saved with earlier versions can be opened; the code selection is written to the report.

11.1 KDYY Table 2.1 factors

Table 17. KDYY Table 2.1 partial factors

Factor	Description	Value
$\gamma_{G,dst}$	Permanent action, destabilising	1.35
$\gamma_{G,stb}$	Permanent action, stabilising	1.00
$\gamma_{Q,dst}$	Variable action, destabilising	1.50
$\gamma_{Q,stb}$	Variable action, stabilising — not relied upon as stabilising	0
$\gamma_{c'}$, γ_{ϕ}	Material factors (no reduction in DA-2)	1.00
γ_{Re}	Soil resistance	1.10

11.2 Single-source assumption

The weight of the sliding mass produces both driving and resisting effects: where the sign of the base angle changes, the direction of the effect changes too. Slices on one side trigger sliding while those on the other side preserve stability.

The single-source assumption is the application of the same factor to the whole of the weight coming from the same source; its justification is that applying different factors to different parts of the same source is inconsistent. However, this assumption calculates the factor of safety about 10% higher. KDYY Design Handbook

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

9.1.1 states that calculations made without the assumption are more in line with the KDYY principles; in the program it is off by default.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

12. Reading the Results

12.1 Slice table

After the analysis, the table in the bottom area shows all quantities of every slice of the critical surface. The aim is that the calculation can be followed by hand: in the total row at the bottom, the user can do the division Σresisting / Σdriving himself or herself and verify γ_{Rk}. A calculation can be checked to the extent that it can be followed by hand.

On polygonal surfaces the m(α) and moment terms are blank, because moment equilibrium is undefined on those surfaces. The geometry and strength terms remain valid.

12.2 Method comparison

The critical surface is found with one method; the same surface is solved again with all methods. The convergence of the methods towards one another shows the robustness of the result. On a sound circular surface, Bishop and Spencer agree to within a few per mille. A scatter exceeding 25% is a sign that the surface or the model needs to be questioned, and the program warns in this case.

The first row of the window bears the name of the selected solver. If the result of a method falls outside the reasonable band formed by the other methods for the same surface, it is considered unreliable and eliminated, and the reason is written on the row. “Converged” is not written for a method that did not converge; the row states the real situation, such as “did not intersect” or “not applicable to this surface type”. On a circular surface the classical Bishop row is added for information (Section 10.7).

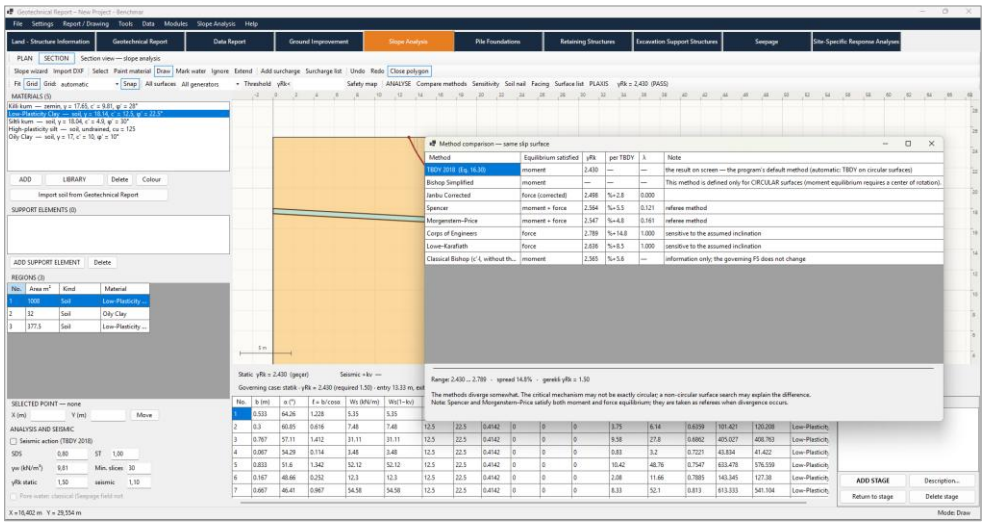


Figure 18. Method comparison window: results of the methods for the same slip surface and the classical Bishop information row

12.3 Toe failure and global failure

Slip surfaces are divided into two groups according to the position of the exit point relative to the toe of the slope. For many retaining structures toe failure is much more critical than global failure; KDYY 9.2.1 requires the two to be examined separately, and the report gives them separately.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

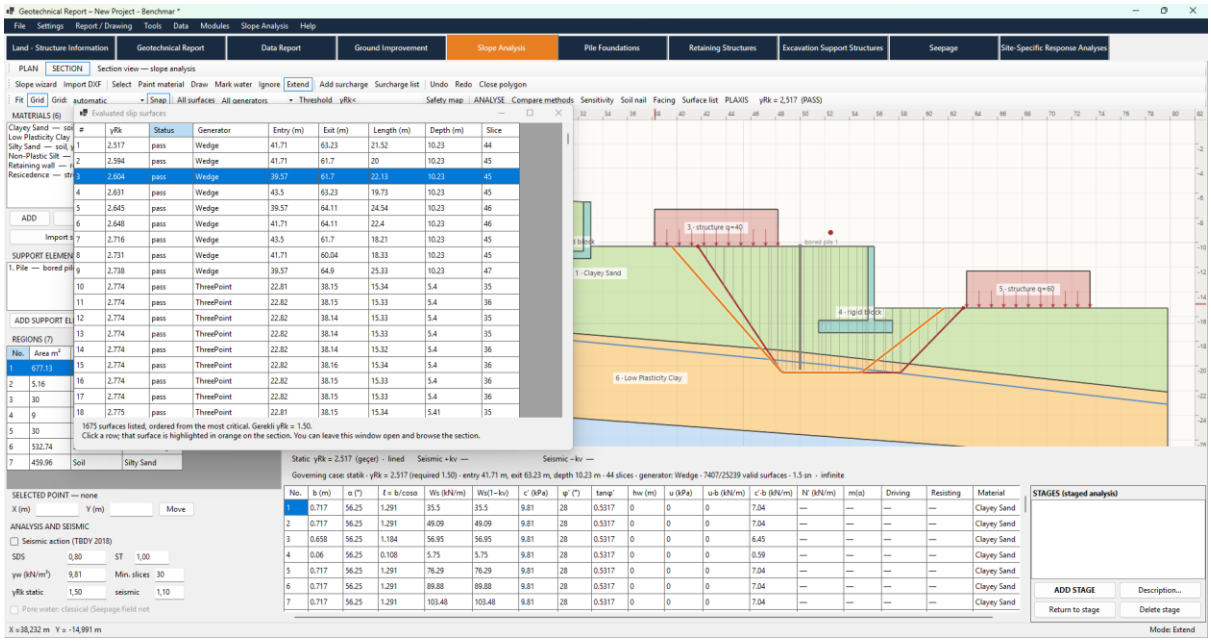


Figure 19. Evaluated slip surfaces window: the surface list and the selected surface highlighted on the section

13. Sensitivity Analysis

The weakest point of a report is the uncertainty of the soil parameters. Giving a single γ_{Rk} creates the impression that those parameters are known exactly. Yet c′ and ϕ′ are derived from tests, and both show scatter.

Showing the sensitivity does not weaken the result; it makes the uncertainty visible. The reader sees which parameter governs and with how much confidence he or she can speak.

Each parameter is varied on its own while the others are held constant. With every change the critical surface is searched again, because the location of the critical surface may change when a parameter changes; solving the old surface with the new parameters would show the sensitivity smaller than it is.

The deviation limits are set in the settings. The default maximum deviation is 20% and the step 5%; this corresponds to an ordinary scatter in soil parameters.

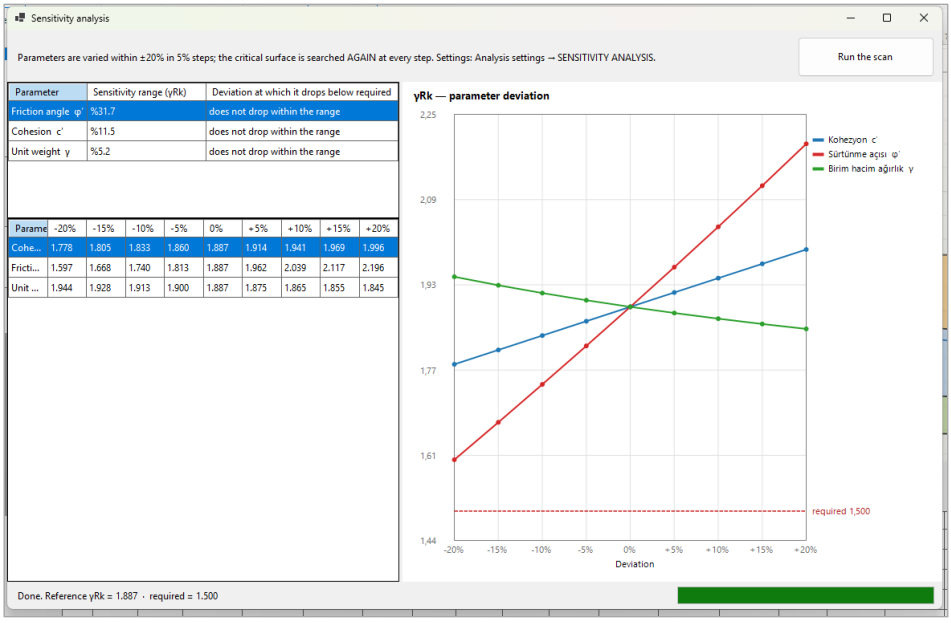


Figure 20. Sensitivity analysis window: variation of the factor of safety with parameter deviation

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

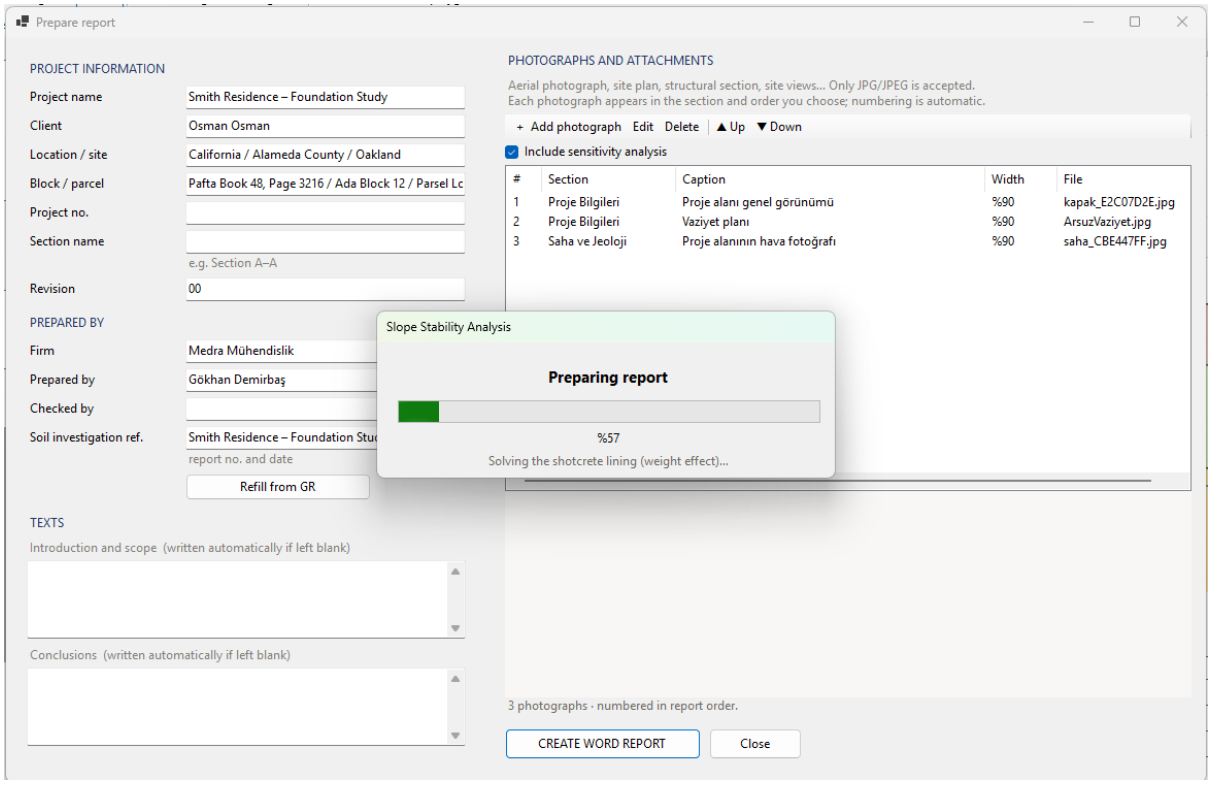


Figure 21. Prepare report window: the “Include sensitivity analysis” option and the sensitivity scan running while the report is prepared

14. Verification

The calculations in this module have been checked against independent verification examples and hand calculations. The verification rests on three kinds of basis: the slice-by-slice figures of reference slope analysis reports, comparison with independent solutions (finite element, closed-form solution, limit solution) and internal consistency rules. In this section the comparisons are given as tables, and the known differences are stated separately.

Table 18. Basis of the verification

Basis	Scope
Reference slope analysis report	Slice width, base angle, slice area and iteration steps are compared figure by figure; geometry, slicing rules and the equation are checked separately.
Finite element comparison	An excavation section supported by a pile with shear capacity is compared with an independent finite element solution; the difference is of the order of 2%, and it is checked that the result increases monotonically with the capacity and saturates.
Internal consistency checks	Region building, node editing, rigid block behaviour, wedge and composite generators, contour solver, surcharge transfer, symmetry and DXF generation; building (structure region) load, sign of the driving moment, suction transfer from Seepage, DXF plan and back analysis.

What verification says and what it does not

The verification shows that the program follows its own rules and the reference figures. It does not show that the soil parameters of a site have been selected correctly or that the model represents that site; these are the responsibility of the engineer.

14.1 Comparison with reference slope analysis reports

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

The program is compared with the figures of two reference slope analysis reports. The first is a circular section with 31 slices; the second is a section with 35 slices including water, surcharge and earthquake. Slice width, base angle, slice area, weight, pore water and iteration steps are checked figure by figure. In the comparison the program also does the slicing itself starting from the raw geometry (end to end), and it is checked whether the surface search finds a surface equal to or more critical than the critical surface of the report.

Table 19. Comparison with reference reports (γ_{Rk})

Comparison	Quantity	Reference	Program	Difference
Report-1, TBDY Eq. 16.30 iteration (with the report's slices)	Iteration 1 / 2 / 3 / 4	1.742 / 1.581 / 1.565 / 1.563	1.7425 / 1.5812 / 1.5646 / 1.5628	At most 0.0005; 4/4 iterations matched
Report-1, total forces	Driving / resisting total	0.8910 / 1.3930	0.8914 / 1.3929	0.0004 / 0.0001
Report-1, end to end from raw geometry (circle-1)	γ_{Rk}; number of slices	1.563; 31	1.5635; 31	0.03%
Report-1, end to end from raw geometry (circle-2)	γ_{Rk}; number of slices	1.563; 31	1.5632; 31	0.01%
Report-2, formula (water, surcharge, earthquake) static	γ_{Rk}	2.351	2.3512	0.0002
Report-2, formula, earthquake $+k_v$	γ_{Rk}	1.555	1.5554	0.0004
Report-2, formula, earthquake $-k_v$	γ_{Rk}	1.535	1.5350	0.0000
Report-2, 35 slices end to end (static)	γ_{Rk}; driving; resisting	2.351; 16.607; 39.047	2.3511; 16.6068; 39.0437	0.00%
Report-2, surface search (static)	γ_{Rk}	2.351	2.350	0.04%
Report-2, surface search (earthquake $+k_v$)	γ_{Rk}	1.555	1.555	0.00%
Report-2, surface search (earthquake $-k_v$)	γ_{Rk}	1.535	1.533	−0.13%

All rows are within the acceptance criterion. In the surface search the critical circle found by the program differs slightly from that of the report (in the static case the centre and radius are different); the factors of safety, however, are within 0.002, and the program's value is equal to or more critical than the report's. The end-to-end slice comparison in the earthquake rows was made on the static critical circle; the report's earthquake critical circles (radius 8.614 m and 9.315 m) were used in the surface search comparison.

14.2 Comparison of the solvers with each other and with the classical solution

The same circular surface is solved with all solvers. The results are in Table 20. The verified classical Bishop value is 2.6365; its difference from the Bishop solution within GLE is 0.0022 (criterion 0.01).

Table 20. Comparison of the solvers on the same surface

Method	Equilibrium	γ_{Rk}	Difference from GLE-Bishop
Fellenius (Ordinary)	Moment	2.4202	−8.3%
Bishop simplified	Moment	2.6387	—
Janbu simplified	Force	2.3151	−12.3%
Janbu corrected	Force (corrected)	2.5134	−4.7%
Spencer	Moment + force	2.6335	−0.2%
Morgenstern–Price	Moment + force	2.6334	−0.2%

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Method	Equilibrium	γ_{Rk}	Difference from GLE-Bishop
Corps of Engineers	Force	2.8023	+6.2%
Lowe–Karafiath	Force	Did not converge	—

The expected orders are satisfied: Fellenius is lower than Bishop, Janbu corrected is higher than Janbu simplified, and Spencer and Morgenstern–Price agree with Bishop to within a few per mille (0.20%; criterion 3%). On a non-circular surface Bishop is rejected. In homogeneous soil the circular surface comes out more critical than the wedge (circular 1.087; wedge 1.359); among the force equilibrium methods, the convergence rate is 100% for Janbu simplified and corrected and 64.7% for Corps of Engineers. On this surface Corps of Engineers gave 2.8023 and Lowe–Karafiath did not converge.

Table 21. TBDY solver (classical term) compared with GLE-Bishop

Case	TBDY solver (classical term)	GLE-Bishop	Difference
Characteristic, static	2.6365	2.6387	0.0022
Characteristic, earthquake $+k_v$	1.8340	1.8355	0.0014
Characteristic, earthquake $-k_v$	1.7503	1.7517	0.0014
Partial factor, static	1.8461	1.8476	0.0015
Partial factor, earthquake $+k_v$	1.4241	1.4253	0.0011

The two solvers are algebraically identical; the difference comes only from the chord/arc discretisation (criterion 0.01). On the partial factor basis the factors of safety are lower than on the characteristic basis (static 1.8461 < 2.6365; earthquake 1.4241 < 1.8340).

Table 22. Polygonal surface compared with its circular equivalent (Janbu)

Number of polygon points	Polygonal Janbu	Difference from circular Janbu (1.7219)
8	1.7277	0.0058
16	1.7232	0.0013
32	1.7221	0.0001
64	1.7217	0.0002

As the number of points increases, the polygonal solution converges to the circular solution (difference 0.0002 at 64 points; criterion 0.005). On the same surface the Spencer result is 1.8311 circular and 1.8310 polygonal (deviation 0.00%; criterion 0.5%). Bishop simplified and Fellenius are rejected on a polygonal surface.

Table 23. Cohesion term: TBDY 2018 formulation versus classical Bishop

c′ (t/m²)	TBDY 2018	Classical Bishop	Difference of the classical formulation
0.000	0.838	0.838	0.0%
0.306	1.563	1.720	9.1%
1.000	3.190	3.746	14.8%
3.000	7.870	9.623	18.2%

In the section of reference report-1 (ϕ′ = 21°) the two formulations are identical in cohesionless soil; as cohesion increases, the TBDY formulation stays on the safe side. The difference grows as the slice base gets steeper: in an 8 m excavation with a vertical face (c′ = 30 kPa, ϕ′ = 26°), on the same critical surface the TBDY formulation gave 0.46 and the classical term 1.59 (Culmann plane 1.18; this is a separate surface). This is why there is a cohesion term setting (Section 10.7) and why TBDY Eq. 16.30 is not used for soil-nailed walls (Section 9.12).

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

14.3 Comparison with independent solutions

A 5 m excavation section supported by a pile with shear capacity (pile Ø65 cm, spacing 90 cm, length 12 m, surcharge 15 kPa) is compared with an independent finite element solution (PLAXIS). The independent reference values are PLAXIS 2.351, Janbu 2.416 and Bishop 2.908; the acceptance criteria are to stay within 10% of PLAXIS, to increase monotonically with the capacity and to saturate at a very large capacity.

Table 24. Pile with shear capacity: sensitivity to the capacity

Pile shear capacity (kN)	Per meter (kN/m)	γ_{Rk} (program)	Reference	Difference
0	0	0.234	—	—
100	111	1.525	—	—
233	259	2.306	2.351 (PLAXIS)	−1.9%
400	444	2.939	—	—
800	889	3.279	—	—

The program increases monotonically with the capacity, saturates after 400 kN (3.279 at 800 kN) and is below 1.0 without capacity. For 233 kN the difference from PLAXIS is 1.9%, below the criterion.

Table 25. Soil-nailed vertical cut and planar block: comparison with closed-form solutions

Comparison	Reference	Program	Difference
Vertical cut H = 4 m, c′ = 20 kPa	Chen (1975) 1.414	Morgenstern–Price 1.432	+1.3%
Vertical cut H = 8 m, c′ = 30 kPa	Chen (1975) 1.144	Morgenstern–Price 1.159	+1.3%
Planar block: 2 angles × {no crack, dry crack, water-filled crack, nailed, nailed + water-filled}, 10 cases	Closed-form solution	Janbu = Spencer = Morgenstern–Price	< 0.01%

Chen's (1975) solution is a log-spiral upper bound for the vertical cut; the program's result approaches it from above within 1.3%. These rows verify the KDYY/FHWA-based soil nail solver (Section 9.12). The comparison with the KDYY P20 example project is given in Table 14 (Section 9.10).

14.4 Internal consistency checks

The internal consistency checks of region building, node editing, rigid block behaviour, wedge and composite generators, contour solver, surcharge transfer, symmetry, DXF generation, structure load, sign of the driving moment, suction transfer from Seepage, DXF plan and back analysis have been supported by independent verification examples and hand calculations.

14.5 Known differences and limitations

The comparison below sets the same solvers, in the global failure check of the Retaining Structures module, against the PLAXIS (ϕ–c reduction) result of 1.56 for one section (single-layer stone wall, H = 4 m). The difference has been measured for one section only.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

Table 26. Global failure: methods compared with PLAXIS (1.56) (single section)

Method	γ_{Rk}	Relative to PLAXIS 1.56
Spencer (governing)	1.46	−6.4%
TBDY 16.30 (circular)	1.44	−7.7%
Janbu corrected	1.292	−17.2%
Janbu simplified	1.224	−21.5%

In PLAXIS the sliding mechanism is wedge/block-shaped. Spencer is the method closest to PLAXIS while staying on the safe side; Janbu coming out low is expected behaviour on a surface with a steep upper end (base angle 67°–78°). No PLAXIS comparison has been made with other sections.

Topics not compared: the global factor of safety of a soil-nailed wall has not been compared with the worked example of FHWA GEC-7 or with PLAXIS; the KDYY P20 example covers the internal checks and gives no global factor of safety. An independent hand comparison of two-block force equilibrium for the results of the wedge generator has not been made yet either. In these areas the result should be supported by the user's own independent check.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

15. Report and Drawing

15.1 Word report

The “Slope analysis report…” command in the Report menu opens the report screen. Project information is on the left and photo management on the right. Only the JPG/JPEG format is accepted; each photo is assigned a figure caption, a report section and a page width.

The report consists of a cover, contents, lists of figures and tables, symbols and the chapters. Figure and table numbers are generated with Word’s own field mechanism; after opening the document, the lists are updated with Ctrl+A → F9. If a back analysis has been made, a “Back analysis” chapter is added to the slope analysis report (Section 10.10).

In the main program, Report and Drawing are separate submenus. For Slope there are two commands in the Report menu: “Slope analysis report…” prepares the Word report of the current section only. “Staged slope analysis report…” uses its own layout in projects with stages added: cover (Staged Slope Stability Report), introduction, construction stages (table and excavated lines), soil, calculation method, a separate full chapter for each stage (overview, loading cases, mechanisms, method comparison, notes and the slice table of the governing case), comparison of the stages, conclusion and references. While the report is prepared, each stage is re-solved independently from its own saved section; depending on the number of stages this may take longer, and a progress window is shown. The slice table is given only for the governing loading case. In a project without stages, stages must first be added for the staged report (Section 10.5).

15.2 DXF drawing

The section is exported as a layered AutoCAD drawing. The ten basic layers are: ground, layers, structures, rigid blocks, water, support elements, slip surface, slices, labels and frame. Additional layers that depend on the model (for example shotcrete facing, excavation dimensions, plan layers) are added when needed. A table of material parameters is placed below the drawing.

The slices are not numbered. The purpose of the drawing is verification by hand; users select and number the slice they want to check themselves.

Drawings are produced with the “Get drawing (DXF)…” command in the Drawing menu; the window has three options: current section (with the critical slip surface and slices), A1 drawing sheet, and a separate DXF for each stage. Stage drawings are written to the selected folder with the names ASAMA_01_…, ASAMA_02_… (if the same name occurs, _2, _3 is appended). Each stage is re-solved with its own section (with the excavated lines removed), its critical surface is drawn, and the stage number and description are written in the title. If a facing has been assigned, the drawing contains the SEV_PUSKURTME_KAPLAMA layer (Section 10.6). On the A1 sheet the scale is chosen from the standard series (1/50, 1/100, 1/200, 1/500, 1/1000; largest 1/50). If the section does not fit on A1 even at 1/1000, the scale does not change; the paper grows.

The “Get drawing” window also has the “Plan north direction [°] (0 = up, clockwise +):” box; the value is saved with the project and the north arrow in the plan drawing turns by this angle. If the section was taken from contour lines, in the plan frame beside the drawing the entry–exit range of the critical slip surface is drawn as a thick line on the section line, on the SEV-GOCME-BOLGESI layer: green if the factor of safety meets the required value, red if it does not. Below it the legend “Probable failure zone (critical slip surface entry–exit)”, the colour explanation and the FS of the section are written; if there is no analysis or it is not up to date, the zone is not drawn. The section ends are named after the section (for example K5 and K5’) in the section and in the plan. The failure zone and the legend are also on the A1 sheet.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

16. Export to PLAXIS 2D

The model can be converted into a PLAXIS 2D command script. The PLAXIS button generates the script and copies it to the clipboard; in PLAXIS 2D Input it is pasted and run with Expert → Run commands….

Stiffness is not estimated. Only soils assigned to a region (in use) are written to the script. For each of these layers with missing E or ν, you are asked at the start of the export in a window showing its name and drainage condition (drained / undrained). If you click OK, the values entered are written to the project and the export continues; Cancel stops the export. A layer left empty stops the generation of the script, and the reason is written to the warning list with the layer name.

In the Mohr-Coulomb model the user's E (E_{ref}) and ν are used. In the Hardening Soil model $E_{50}^{ref} = E_{oed}^{ref} = E_{oed}$, $E_{ur}^{ref} = 3 \cdot E_{oed}$, exponent m = 0.5 (c ≤ 3 kPa) or 0.8 (otherwise; in an undrained layer c_u instead of c), $\nu_{ur} = 0.2$ and ψ = max(0; ϕ − 30°) are taken. E_{oed} is either entered or derived with the relation used in the Excavation Support Structures module: $E_{oed} = E \cdot (1 - \nu) / ((1 + \nu) \cdot (1 - 2\nu))$; ν is capped at 0.49. The preview states which one was used.

Anchor and nail behaviour is assigned by object name (EmbeddedBeamRow_N); the water line is linked to InitialPhase with the _setglobalwaterlevel command. Lines not verified in PLAXIS are marked "% NOT VERIFIED" in the preview.

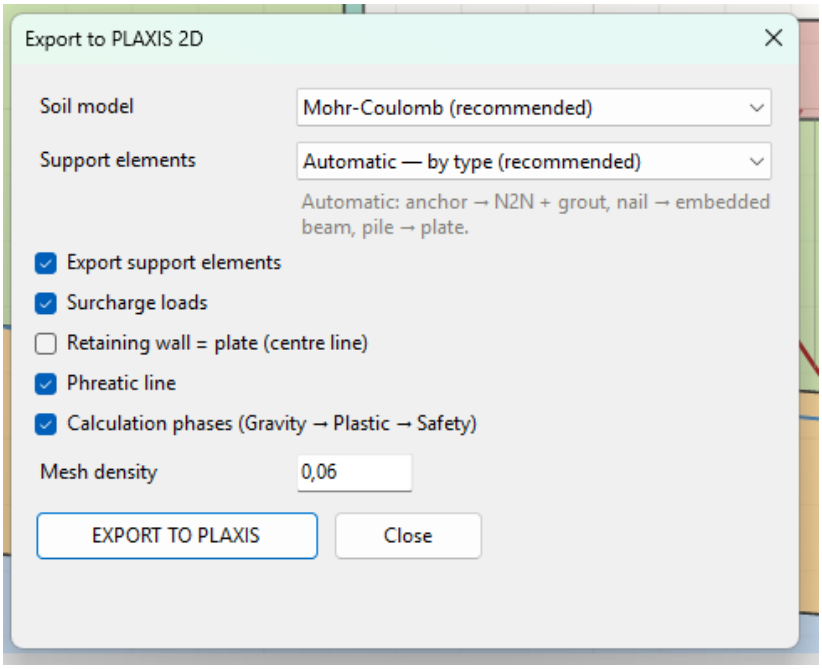


Figure 22. Export to PLAXIS 2D window: soil model selection and script preview

16.1 Important differences for a slope model

- The initial stresses are set up with Gravity loading, not with the K_0 procedure. K_0 assumes a horizontal ground surface and produces unbalanced stresses on sloping ground.
- Flow: Gravity loading → Plastic → Safety (ϕ–c reduction).
- Retaining structures are exported not as walls but as concrete masses; this is the finite element counterpart of the rigid block assumption.
- Structures are exported not with their own weight but as a surcharge at the foundation base level. The basement volume is drawn as a separate region and deactivated. A buried structure receives a foundation slab, two side walls and a top tie beam as plates; a structure sitting on the ground receives

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

only the foundation slab. Because the ground in the footprint is the structure base (Section 7.4), the side wall levels are read just outside the footprint.

Parts of the script not verified in PLAXIS are clearly marked in the preview and in the warning list. Geometry and materials should be reviewed before running.

17. Points to Watch

The following are frequently encountered situations that can lead to a wrong interpretation of the result.

17.1 When building the model

- Choosing the wrong DXF unit spoils the result, not only the display; cohesion scales linearly with length, weight with the square of length.
- On the contour route, deviation of the section line from the direction of steepest descent affects the result on the unsafe side: an oblique section shows the slope flatter. The program measures the deviation and warns if it exceeds twenty degrees.
- On the contour route the ground beyond the section ends is extended flat. If the critical surface reaches into this zone, the section line should be lengthened.
- Unclosed corners prevent regions from forming; the red crosses should not be ignored but closed with the “Extend” command or by hand.
- If two soil blocks that do not touch each other are drawn, the upper one floats in the air. The program detects the gap and warns.
- The wizard and contour routes assign all regions to a default soil. This is a convenience; the analysis should not be run before the materials have been edited.

17.2 When analysing

- If a very small surface immediately below a load comes out critical, this is not slope failure but a local bearing capacity problem. The program eliminates such surfaces by keeping the minimum size criteria proportional to the slope height; even so, the result should be checked visually.
- Slip surfaces that are horizontal from start to end, with entry and exit at the same elevation, are not considered candidates; this prevents bearing-capacity-like flat solutions from governing the slope result under very large point loads. Inclined and curved surfaces are not affected by this rule.
- If the critical surface exits at the model boundary, the box may be too narrow and the mechanism may have been cut off. The box should be enlarged and the analysis repeated; if the result changes, the enlarged version is correct.
- If the scatter in the method comparison exceeds 25%, the surface is numerically unreliable. This is seen especially on polygonal surfaces: force equilibrium methods assume the interslice force angle, and on a broken surface this assumption governs the result.
- Defining a thin wall as a rigid block forbids shallow mechanisms and inflates the factor of safety on the unsafe side.
- If only the tendon capacity of an anchor is entered, grout pull-out is not checked; in many cases it is pull-out that governs.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

18. Code References

Table 28 shows which function of the program corresponds to which code clause.

Table 27. Code references

Topic	Source	Counterpart in the program
Slope stability factor of safety	TBDY 2018, 16.30	Default solution on circular surfaces
Pseudo-static seismic coefficients	TBDY 2018, 16.5	Settings → Earthquake (S_{DS}, ST)
Required factors of safety	TBDY 2018, Table 16.1	Static 1.50 · earthquake 1.10
Partial factor set	KDYY, Table 2.1	Settings → Partial factors
Single-source assumption	KDYY Handbook, 9.1.1	Settings → Single-source assumption (off by default)
Toe failure and global failure	KDYY Handbook, 9.2.1	Report: failure mode section
Tendon rupture and grout pull-out	KDYY Handbook, 9.2.2	Support element window; the lower one governs
Usable load capacity of the anchor	KDYY, 3.2.1.1.e	Taken into account when the design force is entered
Soil nail wall	KDYY 2026, 3.3; FHWA GEC-7 (methods)	Section 9; Settings → Soil nail

19. Known Limitations and Out-of-Scope Topics

This section collects the topics that lie outside the scope of the module or that it treats approximately.

Table 28. Out-of-scope and limited topics

Topic	Status
DXF entity types	Only LINE, LWPOLYLINE and POLYLINE are read. ARC, SPLINE, CIRCLE, TEXT and blocks are not read; arc information is taken as straight. Binary DXF is not supported.
Soil nail wall	Soil nail wall design (calculation, design wizard, result screens, Word report, DXF drawing sheet and PLAXIS script) is described in Section 9. Tmax is taken from the solver, from the PLAXIS export or by hand; no force distribution along the nail is produced. In this version the only design code is KDYY 2026; FHWA GEC-7 and BS 8006-2 cannot be selected yet. That the N_{max} read from PLAXIS is per meter should be confirmed in PLAXIS. The TS 500 minimum / maximum reinforcement limits are not checked for the facing.
DXF layer selection	In this version there is no layer filter on import; the whole file is taken. Surplus lines are marked with “Ignore”.
Storage of the plan work	Contour lines, elevations and section lines are not stored in the main project file (only the plan north direction is saved); if you want to work with the map again, it is imported again.
Map underlay	An aerial photo or map image underlay is not supported; PLAN mode works only with vector contour lines.
Three-dimensional effects	Out of scope. The analysis is based on the plane strain assumption.
Deformation and settlement	Out of scope; the limit equilibrium method examines only the moment of failure.
Staged construction	Staged construction modelling that tracks the stress history is out of scope; it is the subject of the Excavation Support Structures module. In the Slope module the stages are solved independently as frozen sections (Section 10.5).
Probabilistic assessment	Out of scope. Uncertainty is shown with the sensitivity scan.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

20. Frequently Asked Questions

Why do some methods give no result?

If the critical surface has been determined as polygonal, moment equilibrium is undefined and moment-based methods cannot give a result. This is the expected behaviour, not an error.

Why did the wedge generator produce no surfaces at all?

A wedge mechanism may not be kinematically possible; for example, no wedge is generated on flat ground. In addition, surfaces with a negative driving force are rejected (not only for wedges but in all methods), because the mass does not want to slide in that direction.

Does the result change when the model box is enlarged?

Because the scan is concentrated around the slope, the box size does not affect the result. If it changes nevertheless, the critical mechanism is being cut off at the boundary of the box; the box should be enlarged.

Why are some methods not shown in the report?

Fellenius and Janbu Simplified neglect the interslice forces completely and systematically give low results. They have been removed from the display to avoid misinterpretation; the program continues to calculate them.

Why does the program not find the elevations by itself on the contour route?

A DXF file does not carry the elevations of the lines, and contour lines have no natural order in the plane. The program derives direction and interval from at least two elevations entered by hand; without them there is no information to derive from.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.

21. References

Turkish Building Earthquake Code (TBDY 2018). Disaster and Emergency Management Presidency (AFAD), Ankara.

Excavation Support Structures Regulation (Kazı Destek Yapıları Yönetmeliği, KDYY) and Design Handbook.

EN 1997-1:2004. Eurocode 7 — Geotechnical design, Part 1: General rules. CEN.

Bishop, A. W. (1955). The use of the slip circle in the stability analysis of slopes. Géotechnique, 5(1).

Fellenius, W. (1936). Calculation of the stability of earth dams. 2nd Congress on Large Dams, Washington D.C.

Fredlund, D. G. and Krahn, J. (1977). Comparison of slope stability methods of analysis. Canadian Geotechnical Journal, 14(3).

Janbu, N. (1973). Slope stability computations. In: Embankment Dam Engineering, Casagrande Volume, John Wiley and Sons.

Krahn, J. (2003). The 2001 R. M. Hardy Lecture: The limits of limit equilibrium analyses. Canadian Geotechnical Journal, 40(3).

Morgenstern, N. R. and Price, V. E. (1965). The analysis of the stability of general slip surfaces. Géotechnique, 15(1).

Spencer, E. (1967). A method of analysis of the stability of embankments assuming parallel inter-slice forces. Géotechnique, 17(1).

Terzaghi, K. (1943). Theoretical Soil Mechanics. John Wiley and Sons.

Douglas, D. H. and Peucker, T. K. (1973). Algorithms for the reduction of the number of points required to represent a digitized line or its caricature. The Canadian Cartographer, 10(2).

Hoek, E. and Bray, J. W. (1981). Rock Slope Engineering, 3rd ed. Institution of Mining and Metallurgy.

Duncan, J. M., Wright, S. G. and Brandon, T. L. (2014). Soil Strength and Slope Stability, 2nd ed. John Wiley and Sons.

© 2026 Medra Mühendislik Ltd. Şti. — This guide was prepared by Medra Mühendislik Ltd. Şti., which reserves all rights. No part of the guide may be reproduced or distributed without written permission, nor quoted without acknowledgement of the source.