PLAXIS

CONNECT Edition V22.01

PLAXIS 2D-Reference Manual

Bentley®
Advancing Infrastructure

Last Updated: March 07, 2022

Table of Contents

Chapter 1: Introduction 11

1.1 Features and Licence levels 11

Chapter 2: General Information 13

2.1 Units and sign conventions 13
2.1.1 Units 13
2.1.2 Sign convention 16
2.2 File handling 16
2.3 Help facilities 17

Chapter 3: Input program - General overview 18

3.1 Starting the Input program 18
3.1.1 New project 20
3.1.2 Existing project 27
3.2 Layout of the Input program 27
3.3 Menus in the Menu bar 30
3.3.1 File menu 30
3.3.2 Edit menu 31
3.3.3 Soil menu 31
3.3.4 Structures menu 32
3.3.5 Mesh menu 32
3.3.6 Flow conditions menu 32
3.3.7 Phases menu 33
3.3.8 Options menu 33
3.3.9 Expert menu [GSE] 33
3.3.10 Help menu 34
3.4 Input program structure - Modes 35
3.5 Model in the drawing area 36
3.5.1 Supporting tools 36
3.5.2 Model view - Zooming 37
3.5.3 Selecting geometry components 37
3.6 Explorers 38
3.6.1 Model explorer 38
3.6.2 Selection explorer 43
3.6.3 Phases explorer 44
3.7 General information on command line 44
3.7.1 Indexing in commands 46
3.7.2 Accessing Output commands in Input command line 46

Chapter 4: Ground modelling - Soil mode 48

4.1 Adjustment of the soil contour 48
4.2 Creating boreholes 48
4.2.1 Creating new boreholes 49
4.2.2 Editing boreholes 50
4.3 Soil layers 51

4.3.1 Creating soil layers in boreholes ... 51
4.3.2 Defining water conditions ... 53
4.3.3 Initial Conditions of Soil ... 54
4.3.4 Preconsolidation ... 55
4.3.5 Generating soil stratigraphy from CPT logs [GSE] ... 57
4.4 Importing soil layers [GSE] ... 62
4.5 Site response analysis [ULT] + [GSE] ... 65

Chapter 5: Modelling loads and structures - Structures mode ... 68

5.1 Assisting tools ... 68
5.1.1 Selecting geometry components ... 68
5.1.2 Move object ... 69
5.1.3 Copy object - Array ... 69
5.2 Geometric entities ... 70
5.2.1 Create point ... 71
5.2.2 Create line ... 71
5.2.3 Create polygon ... 72
5.2.4 Advanced geometric modelling options ... 73
5.3 Importing geometry [GSE] ... 76
5.3.1 Importing polycurves as polygons ... 77
5.3.2 Importing point clouds ... 78
5.3.3 Importing IFC files ... 79
5.4 Loads ... 81
5.4.1 Point load ... 82
5.4.2 Distributed load ... 83
5.5 Prescribed Displacements ... 83
5.5.3 Point prescribed displacement ... 85
5.5.4 Distributed prescribed displacement ... 85
5.5.5 Line contraction ... 85
5.6 Dynamic loading [ULT] ... 86
5.6.1 Definition of multipliers ... 87
5.6.2 Harmonic signal ... 87
5.6.3 Signal from table ... 88
5.6.4 Input accelerograms ... 91
5.6.5 Added mass ... 95
5.7 Structural elements ... 97
5.7.1 Fixed-end anchors ... 98
5.7.2 Node-to-node anchors ... 99
5.7.3 Embedded Beam Rows ... 100
5.7.4 Plates ... 103
5.7.5 Geogrids ... 105
5.7.6 Interfaces ... 106
5.7.7 Discontinuities ... 111
5.7.8 Connections ... 114
5.8 Tunnels [GSE] ... 115
5.8.1 Tunnel designer ... 116
5.8.2 Definition of tunnel cross section geometry ... 117
5.8.3 Import tunnel geometry ... 121
5.8.4 Properties ... 123
5.9 Design approaches [ADV] ... 126
5.9.1 Definition of design approaches ... 127

5.9.2 Definition of partial factors for loads 129
5.9.3 Definition of partial factors for materials 129
5.10 Hydraulic conditions [ADV] 131
5.10.1 Well 132
5.10.2 Drain 133
5.10.3 Groundwater flow boundary conditions 134
5.11 Flow functions [ULT] 141
5.12 Thermal conditions [ADV] 146
5.12.1 Thermal flow boundary conditions [ADV] 147
5.13 Thermal functions [ULT] 154

Chapter 6: Material properties and material database 160

6.1 Modelling soil and interface behaviour 162
6.1.1 General tabsheet 162
6.1.2 Mechanical tabsheet 169
6.1.3 Hoek-Brown pre-processing tool 203
6.1.4 Cyclic accumulation and optimisation tool [ADV] 209
6.1.5 Groundwater tabsheet 224
6.1.6 Thermal tabsheet [ADV] 237
6.1.7 Interfaces tabsheet 242
6.1.8 Initial tabsheet 251
6.2 Modelling undrained behaviour 255
6.2.1 Undrained A 257
6.2.2 Undrained B 257
6.2.3 Undrained C 257
6.2.4 Undrained behaviour for unsaturated soil 258
6.3 Material data sets for discontinuities 259
6.3.1 General properties 259
6.3.2 Mechanical properties 260
6.3.3 Groundwater properties 262
6.3.4 Thermal properties [ADV] 263
6.4 Material data sets for plates 264
6.4.1 General properties 265
6.4.2 Mechanical properties 266
6.4.3 Thermal properties [ADV] 269
6.5 Material data sets for geogrids 269
6.5.1 General properties 270
6.5.2 Mechanical properties 270
6.5.3 Thermal properties [ADV] 274
6.6 Material data sets for embedded beams 275
6.6.1 General properties 275
6.6.2 Mechanical properties 277
6.6.3 Interaction properties: bearing capacity 280
6.6.4 Interface stiffness factor 283
6.7 Material data sets for anchors 285
6.7.1 General properties 285
6.7.2 Mechanical properties 286
6.7.3 Thermal properties [ADV] 288
6.8 Assigning material data sets to geometry components 289

Chapter 7: Meshing and calculation 291

7.1 Mesh generation - Mesh mode ... 291
7.1.1 Elements ... 292
7.1.2 Global settings ... 292
7.1.3 Local refinement ... 293
7.1.4 Enhanced mesh refinements (EMR) ... 294
7.2 Defining calculation phases ... 295
7.2.1 Phases explorer ... 295
7.2.2 Order of calculation phases ... 297
7.2.3 Phases window ... 298
7.3 Types of analysis ... 299
7.3.1 Initial stress generation ... 300
7.3.2 Plastic calculation ... 305
7.3.3 Consolidation calculation [ADV] ... 306
7.3.4 Fully coupled flow-deformation analysis [ULT] ... 307
7.3.5 Safety calculation (phi/c reduction) ... 307
7.3.6 Dynamics calculation [ULT] ... 311
7.3.7 Dynamics with consolidation calculation [ULT] ... 313
7.3.8 Plastic nil-phase ... 314
7.3.9 Updated mesh analysis ... 314
7.4 Loading type ... 316
7.4.1 Staged construction ... 317
7.4.2 Minimum excess pore pressure [ADV] ... 320
7.4.3 Degree of consolidation [ADV] ... 320
7.4.4 Target ΣMsf ... 321
7.4.5 Incremental multipliers ... 321
7.5 Water pressure calculation ... 321
7.5.1 Calculation type ... 322
7.5.2 Global water level ... 323
7.6 Temperature calculation [ADV] ... 324
7.6.1 Calculation type ... 324
7.6.2 Coupled analysis [ULT] ... 325
7.6.3 Non-linearity during Phase-changes [ADV] ... 326
7.7 Load stepping procedures ... 326
7.7.1 Automatic step size procedure ... 328
7.7.2 Load advancement - Ultimate level ... 328
7.7.3 Load advancement - Number of steps ... 330
7.7.4 Automatic time stepping (consolidation [ADV] and fully coupled analysis [ULT]) ... 330
7.7.5 Automatic time stepping (Dynamics) [ULT] ... 331
7.8 Calculation control parameters ... 331
7.8.1 General phase parameters ... 331
7.8.2 Deformation control parameters ... 334
7.8.3 Numerical control parameters ... 337
7.8.4 Flow control parameters [ADV] ... 348
7.8.5 Dynamics control parameters [ULT] ... 348
7.8.6 Reached values ... 349
7.9 Flow conditions ... 350
7.9.1 Water levels ... 351
7.9.2 Assigning water conditions to soil polygons ... 353
7.9.3 Water pressure generation and previewing ... 355
7.9.4 Flow boundary conditions [ADV] ... 356
7.9.5 Precipitation [ADV] ... 356

7.9.6 Climate [ADV] 358
7.10 Geometry configuration - Staged Construction mode 359
7.10.1 Changing geometry configuration 359
7.10.2 Staged construction procedure in calculations 360
7.10.3 Changing loads 361
7.10.4 Connections in Staged construction 361
7.10.5 Reassigning material data sets 361
7.10.6 Applying volumetric strains in clusters 362
7.10.7 Prestressing of anchors 363
7.10.8 Advancing a tunnel [GSE] 363
7.10.9 Model conditions 364
7.11 Special option for User Defined Soil Models [ADV] 373
7.12 Starting a calculation 374
7.12.1 Previewing a construction stage 374
7.12.2 Selecting points for curves 374
7.12.3 Execution of the calculation process 374
7.12.4 Aborting a calculation 375
7.12.5 Output during calculations 375
7.12.6 Selecting calculation phases for output 380
7.12.7 Adjustment to input data in between calculations 381
7.12.8 Automatic error checks 381

Chapter 8: Output program - General overview 384

8.1 Layout of the Output program 385
8.2 Menus in the Menu bar 386
8.2.1 File menu 386
8.2.2 View menu 387
8.2.3 Project menu 389
8.2.4 Geometry menu 390
8.2.5 Mesh menu 390
8.2.6 Deformations menu 391
8.2.7 Stresses menu 391
8.2.8 Forces menu 391
8.2.9 Interface stresses menu 391
8.2.10 Discontinuity stresses menu 392
8.2.11 Tools menu 392
8.2.12 Options 393
8.2.13 Expert menu [GSE] 393
8.2.14 Window menu 394
8.2.15 Help menu 394
8.3 General information on command line 394
8.3.1 Indexing in commands 395
8.4 Tools in the Output program 395
8.4.1 Accessing the Output program 395
8.4.2 Exporting output data 396
8.4.3 Curves manager 398
8.4.4 Store the view for reports 398
8.4.5 Zooming the plot 399
8.4.6 Relocation of the plot 399
8.4.7 Scaling the displayed results 399
8.4.8 Tables 400

8.4.9 Selection of results ... 401
8.4.10 Display type of results ... 401
8.4.11 Select structures ... 403
8.4.12 Partial geometry ... 403
8.4.13 Viewing results in cross sections ... 404
8.4.14 Plot annotations ... 405
8.4.15 Miscellaneous tools ... 407
8.5 Display area ... 410
8.5.1 Legend ... 410
8.5.2 Modifying the display settings ... 411
8.6 Views in Output ... 414
8.7 Report generation ... 415
8.7.1 Configuration of the document ... 419
8.8 Creating animations ... 420
8.9 Paraview Export [GSE] ... 421

Chapter 9: Results available in Output program ... 423

9.1 Connectivity plot ... 423
9.2 Deformations ... 424
9.2.1 Deformed mesh ... 424
9.2.2 Total displacements ... 424
9.2.3 Phase displacements ... 424
9.2.4 Sum phase displacements ... 425
9.2.5 Incremental displacements ... 425
9.2.6 Extreme total displacements ... 426
9.2.7 Velocities [ULT] ... 426
9.2.8 Accelerations [ULT] ... 426
9.2.9 Accelerations in 'g' [ULT] ... 426
9.2.10 Total cartesian strains ... 426
9.2.11 Phase cartesian strains ... 427
9.2.12 Incremental cartesian strains ... 427
9.2.13 Total strains ... 427
9.2.14 Phase strains ... 428
9.2.15 Incremental strains ... 428
9.3 Stresses ... 428
9.3.1 Cartesian effective stresses ... 429
9.3.2 Cartesian total stresses ... 429
9.3.3 Principal effective stresses ... 429
9.3.4 Principal total stresses ... 431
9.3.5 Initial conditions ... 431
9.3.6 State parameters ... 433
9.3.7 Pore pressures ... 435
9.3.8 Groundwater flow [ADV] ... 437
9.3.9 Heat flow [ADV] ... 438
9.3.10 Plastic points ... 438
9.3.11 Fixed end anchors ... 440
9.3.12 Node to node anchors ... 440
9.3.13 Wells [ADV] ... 441
9.3.14 Drains [ADV] ... 441
9.4 Structures and interfaces ... 441
9.4.1 Deformation in structures and interfaces ... 441

9.4.2 Resulting contraction in plates and geogrids ... 442
9.4.3 Resulting stresses in interfaces ... 442
9.4.4 Resulting stresses in discontinuities ... 443
9.4.5 Resulting forces in plates ... 444
9.4.6 Resulting forces in geogrids ... 444
9.4.7 Resulting forces in embedded beam rows ... 444
9.4.8 Resulting force in anchors ... 445
9.4.9 Results in connections ... 445
9.4.10 Structural forces in volume plates ... 445

Chapter 10: Curves ... 451

10.1 Selecting points for curves ... 451
10.1.1 Mesh point selection ... 451
10.1.2 Pre-calculation points ... 452
10.1.3 Post-calculation points ... 453
10.2 Generating curves ... 453
10.2.1 Load-displacement curves ... 455
10.2.2 Force-displacement curves ... 456
10.2.3 Displacement-time or force-time curves ... 456
10.2.4 Stress and strain diagrams ... 457
10.2.5 Curves in Dynamics calculations [ULT] ... 457
10.3 Formatting curves ... 467
10.3.1 Menus for curves ... 468
10.3.2 Editing curve data in table ... 469
10.3.3 Value indication ... 471
10.4 Formatting options ... 471
10.4.1 Chart settings ... 472
10.4.2 Curve settings ... 473
10.5 Regeneration of curves ... 475
10.6 Multiple curves in one chart ... 475

Chapter 11: PLAXIS Advanced Tools ... 477

11.1 Sensitivity Analysis and Parameter variation [GSE] ... 477
11.1.1 Select parameters ... 478
11.1.2 Sensitivity analysis ... 479
11.1.3 Parameter variation ... 481
11.2 Simulation of soil lab tests [GSE] ... 482
11.2.1 Triaxial test ... 485
11.2.2 Cyclic triaxial test ... 486
11.2.3 Oedometer test ... 488
11.2.4 CRS ... 489
11.2.5 DSS ... 489
11.2.6 CDSS ... 490
11.2.7 PMT ... 492
11.2.8 General ... 494
11.2.9 Results ... 494
11.2.10 Parameter optimisation ... 496
11.3 Calculation using design approaches [ADV] ... 503
11.4 Plaxis Remote Scripting Interface [GSE] ... 504
11.4.1 Anatomy of a command ... 504
11.4.2 Geometry and calculation modes objects differences ... 506

11.5 PLAXIS 2D to 3D Converter [GSE] ... 506
11.5.1 Remote scripting server configuration [GSE] ... 507
11.5.2 Running the PLAXIS 2D to 3D converter tool ... 508
11.5.3 PLAXIS 3D model outcome ... 512
11.6 Packing a project ... 513
11.7 Exporting geometry [GSE] ... 515
11.8 PLAXIS 2D LE Model Importer [GSE] ... 516
11.9 Bentley Cloud Services [GSE] ... 518
11.10 ProjectWise Integration Commands [GSE] ... 519
11.10.1 Add File to ProjectWise ... 519
11.10.2 Open File stored in ProjectWise ... 521
11.10.3 Save to ProjectWise ... 522
11.10.4 To View ProjectWise properties ... 523

Chapter 12: Engineering Applications ... 525

12.1 Modelling ground anchors ... 525
12.1.1 Geometric modelling of ground anchors ... 525
12.1.2 Properties of ground anchors ... 525
12.1.3 Activation and pre-stressing of ground anchors ... 526
12.2 Transverse forces in embedded beams ... 526
12.2.1 Abutment pile ... 527
12.2.2 Quay wall with relieving platform ... 527
12.3 Modelling vacuum consolidation [ADV] ... 528
12.3.1 Vacuum consolidation in a one-dimensional soil column ... 528
12.3.2 Vacuum consolidation in a 2D or 3D model ... 528
12.3.3 Other requirements ... 528
12.3.4 Calculation options ... 529
12.3.5 Switching-off vacuum ... 529

Chapter 13: References ... 530

Appendices ... 533

Appendix A: Import CPT data: required file format [GSE] ... 534
Appendix B: Cyclic accumulation Tool - Contour File Formats [ADV] ... 536
Appendix C: Summary of the creation of General Water Levels ... 541
Appendix D: Conventional and unsaturated soil behaviour in PLAXIS ... 543
Appendix E: Shortcuts Input program ... 553
Appendix F: Shortcuts Output Program ... 555
Appendix G: Python HTTP REST API wrapper [GSE] ... 558
G.1 Python HTTP REST API wrapper ... 558
G.2 Starting the remote scripting server ... 558
G.3 Connecting to a scripting server ... 559
G.4 Creating standalone scripts ... 560
G.5 Calling new_server without arguments ... 561
G.6 Disabling the user interface ... 561
G.7 Identifying the type and version of PLAXIS ... 562
G.8 Creating a simple project ... 563
G.9 Performance and logging ... 565

G.10 External manipulations during script execution 565
G.11 Error handling mode 565
G.12 Installing third-party Python modules 566
G.13 Tokenizer 567
G.14 Sample Applications 568
Appendix H: HTTP REST API [GSE] 584
Appendix I: Calculation warning and errors in PLAXIS 602
Appendix J: Overview Features and Licence levels - 612

1 Introduction

PLAXIS 2D is a special purpose two-dimensional finite element program used to perform deformation, stability and flow analysis for various types of geotechnical applications.

Real situations may be modelled either by a plane strain or an axisymmetric model. The program uses a convenient graphical user interface that enables users to quickly generate a geometry model and finite element mesh based on a representative vertical cross section of the situation at hand.

Users need to be familiar with the Windows environment. To obtain a quick working knowledge of the main features of PLAXIS, users should work through the example problems contained in the Tutorial Manual.

The Reference Manual is intended for users who want more detailed information about program features. The manual covers topics that are not covered exhaustively in the Tutorial Manual. It also contains practical details on how to use the PLAXIS program for a wide variety of problem types. The user interface consists of two sub-programs (Input and Output).

The Input program is a pre-processor, which is used to define the problem geometry, to create the finite element mesh and to define calculation phases.

The Output program is a post-processor, which is used to inspect the results of calculations in a two-dimensional view or in cross sections, and to plot graphs (curves) of output quantities of selected geometry points.

The contents of this Reference Manual are arranged according to the sub-programs and their respective options as listed in the corresponding modes and menus. This manual does not contain detailed information about the constitutive models, the finite element formulations or the non-linear solution algorithms used in the program. For detailed information on these and other related subjects, users are referred to the various chapters and papers listed in the Scientific Manual and the Material Models Manual. Additional information can be found in Bentley Communities (Bentley Communities).

1.1 Features and Licence levels

Features of PLAXIS are offered under certain licence levels (See General Information Manual). In this manual, features falling under each licence level can be identified as follows:

- Features with no identification - generally available for PLAXIS 2D licence.
- [ADV] - features available only for users with PLAXIS 2D Advanced licence.
- [ULT] - features available only for users with PLAXIS 2D Ultimate licence.
- [GSE] - features available only for users with Geotechnical SELECT subscription.

An abbreviated list of features and corresponding licence level can be explored in Overview Features and Licence levels - (on page 612).

Note:

- Symbol for features offered as a Technology Preview.
- Features (subsections in the manual) with no identification but that are within a bigger capability (identified with [ADV] or [ULT]) inherit the licencing of the later (e.g Added mass (on page 95) inherits [ULT] since it is under Dynamic loading [ULT] (on page 86)).
- Features using consolidation, steady-state ground water or steady-state thermal flow analysis require minimum an [ADV] licence. Features using dynamics, transient flow or fully coupled analysis require minimum an [ULT] licence.

2 General Information

Before describing the specific features in the different parts of the PLAXIS 2D user interface, the information given in this chapter applies to all parts of the program.

2.1 Units and sign conventions

2.1.1 *Units*

It is important in any analysis to adopt a consistent system of units. At the start of the input of a geometry, a suitable set of basic units should be selected. The basic units comprise a unit for length, force, time, temperature, energy and mass. These basic units are defined in the project properties inside the Input program (go to **File menu** > **Project Properties** > **Model**; see Input program - General overview (on page 18)).

The default units are meters [m] for length, kilonewtons [kN] for force, days [day] for time, seconds [s] for dynamics time, kelvins [K] for temperature, kilojoules [kJ] for energy, kilowatts [kW] for power and tonnes [t] for mass.

Given below an overview of all available units, the [default] settings and conversion factors to the default units. All subsequent input data should conform to the selected unit system and the output data should be interpreted in terms of the same unit system.

From the basic set of units, as defined by the user, the appropriate unit for the input of a particular parameter is generally listed directly behind the edit box or, when using input tables, above the input column. In all of the examples given in the PLAXIS manuals, the standard units are used.

Units	Value	Conversion
Length	1 mm	0.001 m
	1 cm	0.01 m
	1 m	1 m
	1 km	1000 m
	1 in (inch)	0.0254 m

Units	Value	Conversion
	1 ft (feet)	0.3048 m
	1 yd (yard)	0.9144 m
Force	1 N	0.001 kN
	1 kN	1 kN
	1 MN	1000 kN
	1 lbf (pounds force)	0.004448 kN
	1 kip (kilo pound)	4.4482 kN
Time	1 s (sec)	1/86400 day
	1 min	1/1440 day
	1 h	1/24 day
	1 day	1 day
Temperature	1 [K]	1 K
	0 °C	273.15 K
	0 °F	255.37 K
	1 °C	274.15 K
	1 °F	255.93 K
	100 °C	373.15 K
	212 °F	373.15 K
Energy	1 J	0.001 kJ
	1 kJ	1 kJ
	1 MJ	1000 kJ
Power	1 W	0.001 kW
	1 kW	1 kW
	1 MW	1000 kW
Mass	1 kg	0.001 t
	1 t (tonne)	1 t

Note: Note that for calculations involving thermal effects, it is strongly recommended to use the default units.

For convenience, the units of commonly used quantities in two different sets of units are listed below:

		International system	Imperial system
Basic units	Length	[m]	[in] or [ft]
	Force	[N]	[lbf] or [kip]
	Time	[day]	[day]
Geometry	Coordinates	[m]	[in] or [ft]
	Displacements	[m]	[in] or [ft]
Material properties	Young's modulus	$[kN/m^2]=[kPa]$	$[psi]=[lbf/in^2]$ or $[ksf]=[kip/ft^2]$
	Cohesion	$[kN/m^2]$	[psi] or [ksf]
	Friction angle	[deg.]	[deg.]
	Dilatancy angle	[deg.]	[deg.]
	Unit weight	$[kN/m^3]$	[lbf/cu in] or [kip/ft]
	Permeability	[m/day]	[in/day] or [ft/day]
Forces & stresses	Point loads	[kN]	[lbf] or [kip]
	Line loads	[kN/m]	[lbf/in] or [kip/ft]
	Distributed loads	$[kN/m^2]$	$[psi]$ or $[kip/ft^2]$
	Stresses	$[kN/m^2]$	$[psi]$ or $[kip/ft^2]$

Note:

- The user can change the default PLAXIS units in the **Project properties** window. This will automatically convert, in the input program, the defined input values (e,g. geometry data, loads, and prescribed displacements) .
- PLAXIS supports the combination of model properties with different units systems (e.g Forces in [KN] while Length in [ft]). Therefore, the user needs to ensure the consistent selection of these unit systems.

In a plane strain analysis, the calculated forces resulting from prescribed displacements represent forces per unit length in the out-of-plane direction (*z*-direction; see Figure 1 (on page 16)). In an axisymmetric analysis, the calculated forces (*Force-X*, *Force-Y*) are those that act on the boundary of a circle subtending an angle of 1 radian. In order to obtain the forces corresponding to the complete problem therefore, these forces should be multiplied by a factor 2π. All other output for axisymmetric problems is given per unit width and not per radian.

The generation of a 2D finite element model in PLAXIS 2D is based on the creation of a geometry model. The geometry model involves a composition of , areas, lines and points. Multiple vertical boreholes can be defined to determine the soil stratigraphy at different locations. In between the boreholes the soil layer positions are interpolated. Soil layers and ground surfaces may be non-horizontal.

2.1.2 *Sign convention*

The generation of a two-dimensional (2D) finite element model in PLAXIS 2D is based on the creation of a geometry model. This geometry model is created in the *x*-*y* plane of the global coordinate system (Figure 1 (on page 16)), whereas the *z*-direction is the out-of-plane direction. In the global coordinate system the positive *z*-direction is pointing towards the user.

In all of the output data, compressive stresses and forces, including pore pressures, are taken to be negative, whereas tensile stresses and forces are taken to be positive. Figure 1 (on page 16) shows the positive stress directions.

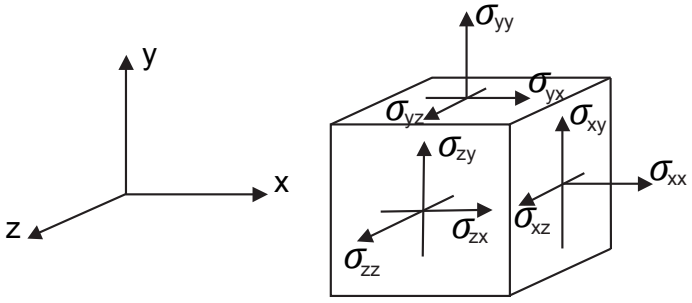


Figure 1: Coordinate system and indication of positive stress components

Although PLAXIS 2D is a 2D program, stresses are based on the 3D Cartesian coordinate system. . In a plane strain analysis σ_{zz} is the out-of-plane stress. In an axisymmetric analysis, *x* represents the radial coordinate, *y* represents the axial coordinate and *z* represents the tangential direction. In this case, σ_{xx} represents the radial stress and σ_{zz} represents the hoop stress.

2.2 *File handling*

All file handling in PLAXIS 2D is done using a modified version of the general Windows® file requester Figure 2 (on page 17).

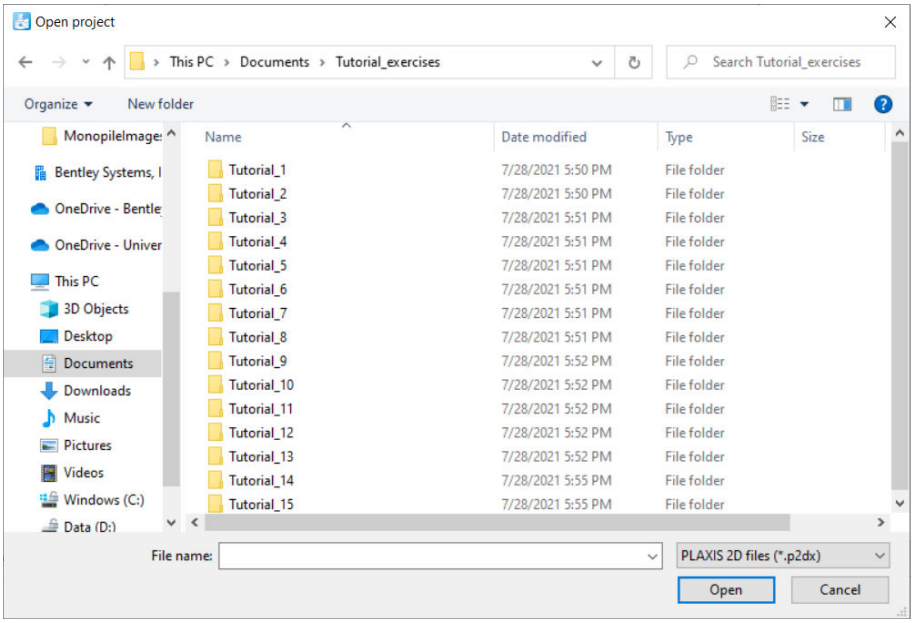


Figure 2: PLAXIS 2D file requester

With the file requester, it is possible to search for files in any admissible folder of the computer (and network) environment. The main file used to store information for a PLAXIS 2D project has a structured format and is named <project>.p2dx, where <project> is the project title. Besides this file, additional data is stored in multiple files in the sub-folder <project>.p2dxdat. It is generally not necessary to enter such a folder because it is not possible to read individual files in this folder.

If a PLAXIS 2D project file (*.p2dx) is selected, a small bitmap of the corresponding project geometry is shown in the file requester to enable a quick and easy recognition of a project.

2.3 Help facilities

To inform the user about the various program options and features, PLAXIS 2D provides a link in the **Help** menu to a digital version of the Manuals. A link is provided to the reference document of the commands used in the program as well. Moreover, the **Help** menu may be used to view and configure licences. A more detailed description of the **Help** menu of the Input and Output program is given in Help menu (on page 34) and Help menu (on page 394) respectively.

Many features are available as buttons in a toolbar. When the mouse pointer is positioned on a button for more than a second, a short description ('Note') appears, indicating the function of the button. For some input parameters side panels appear to help the user decide which value to select.

3 Input program - General overview

To carry out a finite element analysis using the PLAXIS 2D program, the user has to create a 2D geometry model composed of points, lines and other components, in the xy-plane and specify the material properties and boundary conditions. This is done in the first two tabsheets (**Soil** and **Structures**) of the Input program (see Figure 11 (on page 28)). The mesh generation and the definition of the calculation phases is done in the last three tabsheets (**Mesh**, **Flow conditions** and **Staged construction**) of the Input program(see Figure 11 (on page 28)).

Note:

1. The **Soil** and **Structures** tabsheets are known as ***Geometry*** modes.
2. The **Mesh**, **Flow conditions** and **Staged construction** tabsheets are known as ***Calculation*** modes.

3.1 Starting the Input program

This icon represents the Input program.

Start PLAXIS 2D by double clicking the shortcut on your desktop or clicking the entry in the **Windows** > **Bentley Geotechnical** menu.

PLAXIS 2D will first check if you are entitled to use the software and present you with the available options, see Figure 3 (on page 18).

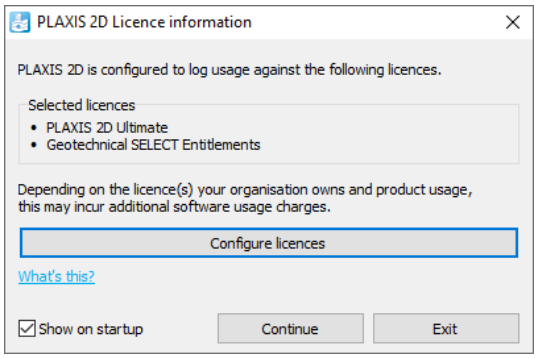


Figure 3: Licence check

If you need to change the licence configuration, e.g. if you have to add some modules, click **Configure licences**.

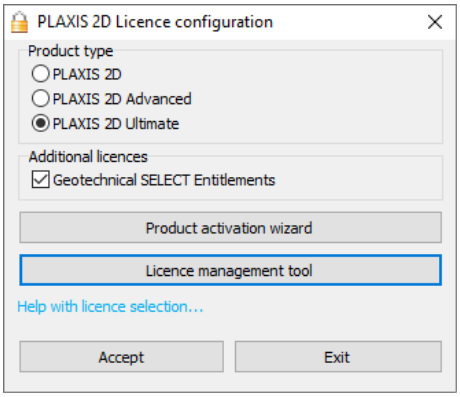


Figure 4: Licence configuration

In the configuration dialog, see Figure 4 (on page 19), you may select the modules that you want to use.

Note:

1- The licence configuration is also available from the menu **Windows** > **Programs** and from the menu **Help** > **Licence configuration**.

2- If you change the license configuration while PLAXIS 2D is open, you have to restart PLAXIS 2D for the changes to take effect.

The licence administrator grants access to software licences. If you miss licences in this list, contact the licence administrator of your company.

For more information on the Product activation wizard and the Licence management tool, see this info on Bentley communities: Info on Product Activation/License Management.

When you have chosen the licences to use, click **Continue** to start using PLAXIS 2D.

At the start of the Input program the **Quick select** window appears in which a choice must be made between the selection of an existing project and the creation of a new project (Figure 5 (on page 20)).

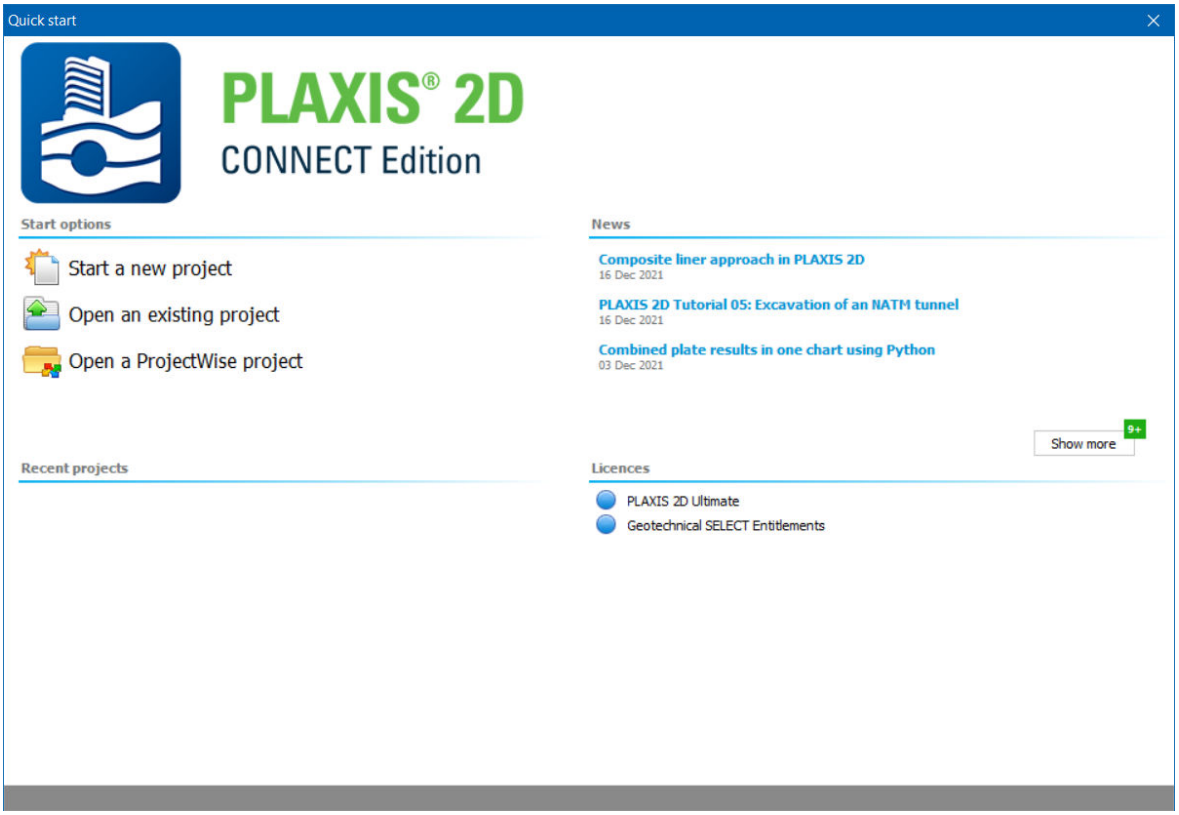


Figure 5: Quick start window

3.1.1 *New project*

When the **Start a new project** option is selected, the **Project properties** window appears in which the basic model parameters of the new project can be defined. The **Project properties** window contains the **Project** and the **Model** tabsheets. The **Project** tabsheet (Figure 6 (on page 21)) contains the project name and description and offers the possibility to select a company logo.

The **Model** tabsheet (Figure 7 (on page 22)) contains the type of the model, the basic units for length, force and time (see Units (on page 13)), the unit weight of water and the initial dimensions of the model. The default values can be replaced by the current values when selecting *Set as default* and clicking the **OK** button. A more detailed description of all these options is given below.

Project properties

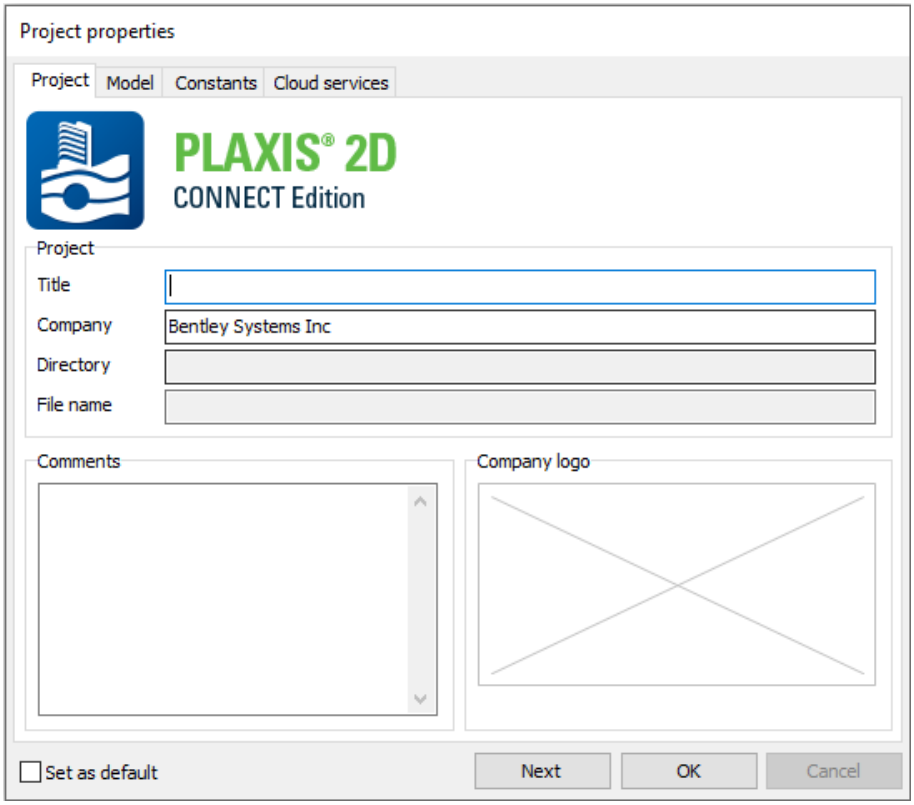


Figure 6: Project properties window (Project tabsheet)

1. **Project**

   The title, folder and the file name of the project are available in the *Project* group box available in the **Project** tabsheet.

Title	The defined title appears as a default name for the file of the project when it is saved.
Company	The company that holds the licence for this software.
Directory	The address to the folder where the project is saved is displayed. For a new project, there is no information shown.
File name	The name of the project file is displayed. For a new project, no information is shown.

2. **Comments**

   The *Comments* box in the **Project** tabsheet gives the possibility to add some extra comments about the project.

3. **Company logo**

   Clicking the *Company logo* box in the **Project** tabsheet will open a file requester from which the desired file can be selected. This logo can be included in the output plots.

General model properties

The general options of the project are available in the *Model* tabsheet of the **Project properties** window.

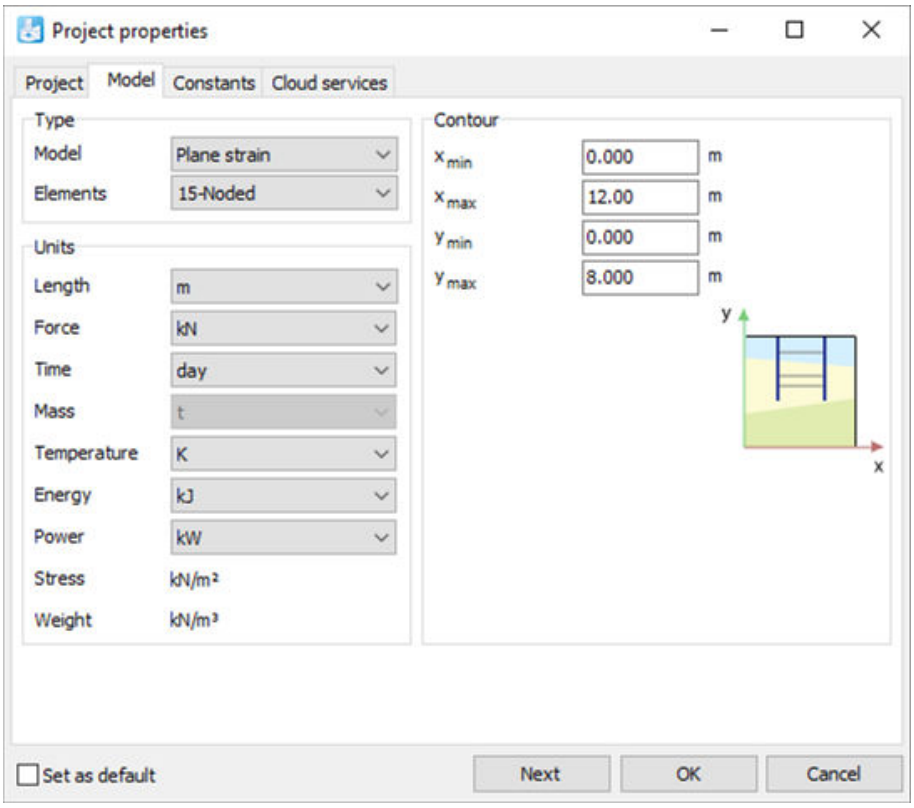


Figure 7: Project properties window (Model tabsheet)

1. **Model**

   ProductName may be used to carry out two-dimensional finite element analysis. The finite element model is defined by selecting the corresponding option in the *Model* drop down-menu in the **Project** tabsheet.

Plane strain	A *Plane strain* model is used for geometries with a (more or less) uniform cross section and corresponding stress state and loading scheme over a certain length perpendicular to the cross section (*z*-direction). Displacements and strains in *z*-direction are assumed to be zero. However, normal stresses in *z*-direction are fully taken into account. In earthquake problems the dynamic loading source is usually applied along the bottom of the model resulting in shear waves that propagate upwards. This type of problem is generally simulated using a plane strain model.

Axisymmetric	A *Axisymmetric* model is used for circular structures with a (more or less) uniform radial cross section and loading scheme around the central axis, where the deformation and stress state are assumed to be identical in any radial direction. Note that for axisymmetric problems the *x*-coordinate represents the radius and the *y*-axis corresponds to the axial line of symmetry. Negative *x*-coordinates cannot be used. Single-source vibration problems are often modelled with axisymmetric models. This is because waves in an axisymmetric system radiate in a manner similar to that in a three dimensional system. In this case, the energy disperses leading to wave attenuations with distance. Such effect can be attributed to the geometric damping (or radiation damping), which is by definition included in the axisymmetric model.

The selection of *Plane strain* or *Axisymmetric* results in a two dimensional finite element model with only two translational degrees of freedom per node (*x*- and *y*-direction).

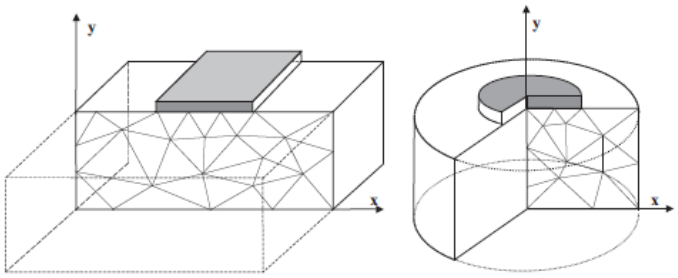


Figure 8: Example of a plane strain (left) and axisymmetrical problem (right)

2. Elements

The user may select either 6-node or 15-node triangular elements to model soil layers and other volume clusters.

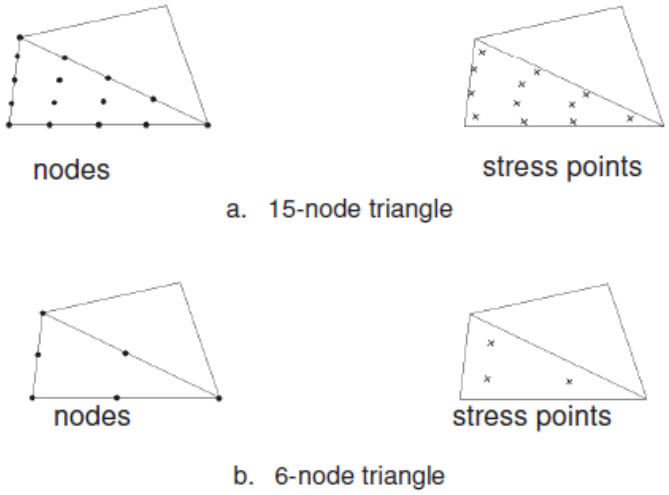


Figure 9: Position of nodes and stress points in soil elements

15-Node	The 15-node triangle is the default element. It provides a fourth order interpolation for displacements and the numerical integration involves twelve Gauss points (stress points). The type of element for structural elements and interfaces is automatically taken to be compatible with the soil element type as selected here. The 15-node triangle is a very accurate element that has produced high quality stress results for difficult problems, as for example in collapse calculations for incompressible soils Sloan, (1981) (on page 532); Sloan & Randolph, 1982 (on page 532); Nagtegaal, Parks & Rice, 1974 (on page 531). The 15-node triangle is particularly recommended to be used in axi-symmetric analysis. The use of 15-node triangles leads to more memory consumption and slower calculation and operation performance. Therefore a more simple type of elements is also available.
6-Node	The 6-node triangle provides a second order interpolation for displacements and the numerical integration involves three Gauss points. The type of element for structural elements and interfaces is automatically taken to be compatible with the soil element type as selected here. The 6-node triangle is a fairly accurate element that gives good results in standard deformation analyses, provided that a sufficient number of elements are used. However, care should be taken with axi-symmetric models or in situations where (possible) failure plays a role, such as a bearing capacity calculation or a safety analysis by means of *phi-c reduction*. Failure loads or safety factors are generally overpredicted using 6-noded elements. In those cases the use of 15-node elements is preferred. One 15-node element can be thought of a composition of four 6-node elements, since the total number of nodes and stress points is equal. Nevertheless, one 15-node element is more powerful than four 6-node elements.

In addition to the soil elements, compatible plate elements are used to simulate the behaviour of walls, plates and shells (Plates (on page 103)) and geogrid elements are used to simulate the behaviour of geogrids and wovens (Geogrids (on page 105)). Moreover, compatible interface elements are used to simulate soil-structure interaction (Interfaces (on page 106)). Finally, the geometry creation mode allows for the input of embedded beam rows, fixed-end anchors and node-to-node anchors (Fixed-end anchors (on page 98), Node-to-node anchors (on page 99)).

3. **Units**

   Units for length, force, time, temperature, energy, power and mass to be used in the analysis need to be specified. These basic units are entered in the *Model* tabsheet of the **Project properties** window.

   The default units, as suggested by the program, are m (metre) for length, kN (kilonewton) for force, day for time, K (kelvin) for temperature, kJ (kilojoule) for energy, kW (kilowatt) for power and t (tonne) for mass. The corresponding units for stress and unit weights are listed in the box below the basic units.

   In a dynamics analysis, the time is usually measured in [seconds] rather than the default unit [days]. In PLAXIS 2D *Time* and *Dynamics time* are different parameters. The time interval in a dynamics analysis is always the dynamics time and PLAXIS 2D always uses seconds [s] as the unit of *Dynamics time*.

   Note: Note that for calculations involving thermal effects, it is strongly recommended to use the default units.

4. **Contour**

At the start of a new project the user needs to specify the contours of the geometry model. The initial setting of the x_{min}, x_{max}, y_{min} and y_{max} parameters set the outer boundaries of the geometry model. The initial view of the drawing area is such that the model dimensions are fully visible within the drawing area. The aforementioned parameters are entered in the *Model* tabsheet of the **Project properties** window.

5. ***Constants*** *[ADV]*

In thermal calculations, pore water may appear in different states, i.e. in liquid state (water), solid state (ice) and gas state (vapour). Thermal properties of soils highly depend on the state of water in the pores. Rather than specifying these properties in material data sets, the thermal properties of pore water are specified for the project as a whole in the *Constants* tab sheet of the **Project properties** window:

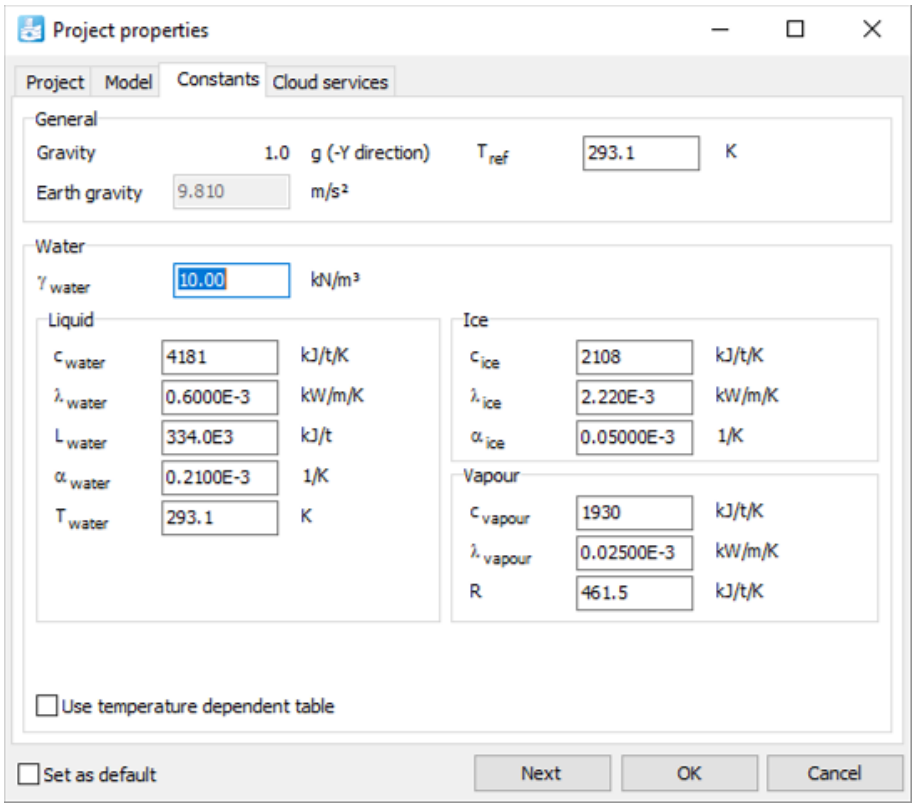


Figure 10: Project properties window (Constants tabsheet

Gravity	By default, the earth gravity acceleration, g, is set to 9.810 m/s^2, assuming the default basic length unit is [m]. The direction of gravity coincides with the negative y-axis. Gravity is implicitly included in the unit weights given by the user (Material properties and material database (on page 160)). In this way, the gravity is controlled by the total load multiplier for weights of materials, *ΣMweight* (General phase parameters (on page 331)). In dynamics calculations, the value of the gravity acceleration, g, is used to calculate the material density, ρ, from the unit of weight, γ ($\rho=\gamma/g$).

Reference temperature	In projects that involve temperature, the input of reference temperature is required. The reference temperature is the temperature at the ground surface in the case that no specific thermal conditions are given here. The reference temperature (T_{ref}) can be entered in the *Constants* tabsheet of the **Project properties** window. By default, the reference temperature is set to 293.1 K(= 20 °C).
Unit weight of water	In projects that involve pore pressures, the input of a unit weight of water is required to determine the effective stresses and pore pressures. The water weight (γ_{water}) can be entered in the *Constants* tabsheet of the **Project properties** window. By default, the unit weight of water is set to 10.00, assuming the default basic units of [kN] and [m] are used.
Specific heat capacity	The specific heat capacity of water, c_{water}, is a parameter that describes the amount of energy (heat) that can be stored in the pore water per unit of mass. It is specified in the unit of energy per unit of mass per unit of temperature (Material properties and material database (on page 160)). Similarly, specific heat capacity can be given for given for ice and vapour.
Thermal conductivity	The thermal conductivity of water, λ_{water}, is a parameter that describes the rate of energy (heat) that can be transported in the pore water. It is specified in the unit of energy per unit length per unit of temperature. Similarly, thermal conductivity can be given for ice and vapour.
Latent heat of water	The latent heat of water is the amount of energy that is required to change the state of water from liquid state to solid state and vice versa. It is specified in the unit of energy per unit of mass.
Thermal expansion coefficient	The thermal expansion coefficient, α is the (change of) volumetric strain per unit of temperature. The thermal expansion coefficient can be given for water and ice. However, thermal expansion of vapour is generally suppressed and therefore ignored. Note that the thermal expansion for ice starts to *increase* from the freeze temperature downward.
Water temperature	In the case that water is entering into the soil (by means of inflow boundary conditions or infiltration wells), this water is assumed to have a temperature equal to the specified water temperature, T_{water}.
Specific gas constant for vapour	The specific gas constant for vapour, R, is a parameter that is used in the equation of relative humidity (see Scientific Manual). It is recommended to use the default value.

Temperature-dependent pore fluid flow	To enable *Temperature-dependent* pore fluid properties, click on the **Use temperature-dependent table** checkbox. A table is displayed and the values are visualised in small graphs. The table contains five columns: *Temperature (T)*, *Density (ρ)*, *Thermal conductivity (λ)*, *Specifyc head capacity (c)* and *Dynamic viscosity(μ)*. The table is pre-filled with default values for water and its phase transition to ice. If this option is selected, the constant parameters for liquid water and ice are not editable to emphasise that the *Temperature-dependent* water properties are taken into account. The table editor allows you to edit the table directly, or to import and export tables from and to a text file.
	Note: Note that the viscosity below 273.15 K is not relevant in the case of the default properties. It is set to a constant value to ease visualisation and allow for modifications if needed, in the case of brines for example, where the freeze point is low.

3.1.2 *Existing project*

When the Input program is started, a list of the recent projects appears in the **Quick select** window. In the case when a project other than the listed recent ones is required, the *Open an existing project* option should be selected. As this selection is made, the Windows ® file requester Figure 2 (on page 17) pops up. It enables the user to browse through all available folders and to select the desired PLAXIS 2D project file (*.p2dx). After the selection of an existing project, the corresponding geometry is presented in the main window.

An existing PLAXIS 2D AE project can also be read by selecting the *Open project* option in the *File* menu. In the file requester, the type of the file is, by default, set to 'PLAXIS 2D AE files *.p2dx '.

3.2 Layout of the Input program

The general layout of the Input program for a new project is shown in Figure 11 (on page 28). The main window of the Input program contains the following items:

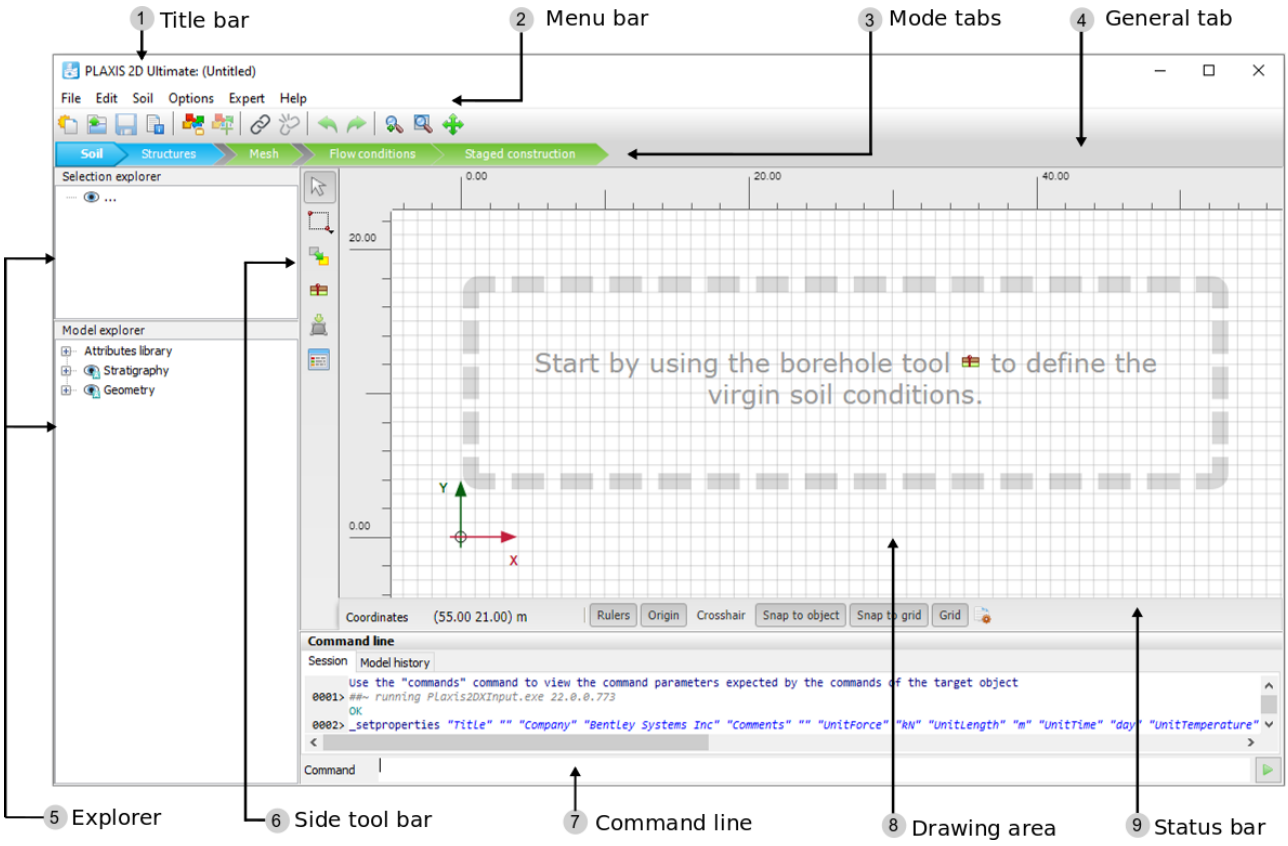


Figure 11: Layout of the Input program

1. ***Title bar***

   The name of the program and the title of the project is displayed in the title bar. Unsaved modifications in the project are indicated by a '*' in the project name.

2. ***Menu bar***

   The menu bar contains drop-down menus covering the options available in the Input program.

3. ***Mode tabs***

   The mode tabs are used to separate different workflow steps. The following tabs are available:

Soil	To define the soil stratigraphy.
Structures	To defined the structural components of the model.
Mesh	To define the mesh properties and generation of the finite element mesh from the geometry model.
Flow conditions	To define water- and thermal conditions through for instance water levels and boundary conditions.
Staged construction	To define the construction phases of the project and calculate the project.

Note: The first two modes (**Soil** and **Structures**) are referred to as the **Geometry** modes, whereas the latter three modes (**Mesh**, **Flow conditions** and **Staged construction**) are referred to as the **Calculation** modes.

For a more detailed description of the modes in the Input program see Input program structure - Modes (on page 35).

4. ***General toolbar***

   The general toolbar contains buttons for general actions such as file operations and model display arrangement. It also contains a button to start the Output program.

   Note: If the mouse is moved over a button in a toolbar, a hint about the function of this button is displayed.

5. ***Explorers***

   Information about the model and the project is given in the explorers. The availability and the functionality of the explorers depend on the active mode.

   The explorers available in the input program are:

Selection explorer	Displays the properties of the selected geometry objects and the features assigned to them. Additionally it gives the possibility to hide/show, activate/deactivate these objects (Selection explorer (on page 43)).
Model explorer	Displays the properties of all the geometry objects and the features assigned to them in the entire model. Additionally it gives the possibility to hide/show, activate/ deactivate these objects (Model explorer (on page 38)).
Phases explorer	Displays the list of the calculation phases, allowing to add, insert, delete and modify a calculation phase as well as the dependencies of all calculation phases (Phases explorer (on page 44)).

   Note: *Selection explorer* and *Model explorer* are referred as *Object explorer*

6. ***Side toolbar***

   The buttons of the features available in the active mode are located in the side toolbar. A hint on the function of the button is displayed when the pointer is hovered over it for a few seconds. The side toolbar also contains options to select one or more objects in the drawing area.

7. ***Command line***

   PLAXIS 2D enables to carry out actions using keyboard input by typing the corresponding commands in the command line. On the other hand, most of the actions carried out using the mouse are translated into commands. All the commands executed in the project are displayed when the **Model history** tab is clicked, whereas only the commands executed in the active session together with the program's feedback are displayed when the **Session** tab is clicked.

   Information about the available commands is provided when the *Command reference* option of the *Help* menu is selected.

8. ***Drawing area***

   The drawing area is the workspace on which the geometry model is created and modified. The geometry model can be created by means of the mouse and using the buttons available in the side toolbar. The buttons

in the side toolbar vary according to the active mode. A more detailed description on the drawing area is given in Model in the drawing area (on page 36).

9. ***Status bar***

   The status bar displays information about the location of the mouse cursor in the drawing area. The display of rulers, origin, crosshair, grid and the snapping to grid or objects can be switched on/off by clicking the corresponding buttons in the status bar.

3.3 Menus in the Menu bar

The menu bar of the Input program contains drop-down menus covering most options for handling files, transferring data, viewing graphs, creating a geometry model, generating finite element meshes and entering data in general.

The availability of the menus depends on the active mode (Input program structure - Modes (on page 35)). The menus available in the Input program are:

3.3.1 *File menu*

New project	To create a new project. In case of a new project, the **Project properties** window is automatically displayed to define its properties.
Open project	To open an existing project. The file requester is displayed.
Recent projects	To quickly open one of the most recent projects.
Save project	To save the current project under the existing name. If a name has not been given before, the file requester is presented.
Save project as	To save the current project under a new name. The file requester is displayed.
Packing a project (on page 513)	To compress the current project.
Close project	To close the current project.
Bentley Cloud Services [GSE] (on page 518)	To access to the different cloud services.
ProjectWise Integration Commands [GSE] (on page 519)	To integrate PLAXIS projects with ProjectWise for infrastructure design collaboration.

PLAXIS 2D LE Model Importer [GSE] (on page 516)	To open the PLAXIS 2D LE model importer allowing the transfer of PLAXIS 2D LE projects into PLAXIS 2D.
Exporting geometry [GSE] (on page 515)	To export the current project. Available in **Soil** and **Structures** modes only.
Project properties	To activate the **Project properties** window (Project properties (on page 21)).
Print	To print the geometry model on a selected printer.
Exit	To leave the Input program.

3.3.2 *Edit menu*

Undo	To restore a previous status of the geometry model (after an input error).
Redo	To redo an action that was undone.
Copy screen image	To quickly open one of the most recent projects. To copy the model image to the Windows clipboard.
Delete	To remove an object.
Select all	To select all the selectable entities in a mode.
Deselect all	To deselect the selected entities.

3.3.3 *Soil menu*

Modify soil layers	To modify the soil layers in the model using the borehole facility.
Import soil [GSE]	To import predefined soil volumes.
Show material	To open the material data base with material data sets.
Design approaches [ADV]	To open the **Design approaches** window.

The **Soil** menu is available only in the **Soil** mode.

3.3.4 *Structures menu*

Import geometry [GSE]	To import predefined geometry.
Show dynamics multipliers [ULT]	To open the **Dynamics multipliers** window.
Show materials	To open the material data base with material data sets.
Show flow functions [ULT]	To open the **Flow functions** window.
Show thermal functions [ULT]	To open the **Thermal functions** window.
Design approaches [ADV]	To open the **Design approaches** window.

The **Structures** menu is available only in the **Structures** mode.

3.3.5 *Mesh menu*

Check geometry	To check the geometry consistency
Generate mesh	To generate the mesh for the defined geometry.

The **Mesh** menu is available only in the **Mesh** mode.

3.3.6 *Flow conditions menu*

Import water levels [GSE]	To import predefined water levels. More information on importing geometry is available in Importing geometry [GSE] (on page 76).
Show materials	To open the material data base with material data sets.
Show flow functions [ULT]	To open the **Flow functions** window.

Show thermal functions [ULT]	To open the **Thermal functions** window.

The **Flow conditions** menu is available only in the **Flow conditions** mode.

3.3.7 *Phases menu*

Edit phases	To open the **Phases** dialog window.
Show materials	To open the material data base with material data sets.
Show dynamics multipliers [ULT]	To open the **Dynamics multipliers** window.
Show flow functions [ULT]	To open the **Flow functions** window.
Show thermal functions [ULT]	To open the **Thermal functions** window.

The **Phases** menu is available only in the **Flow conditions** and **Staged construction** modes.

3.3.8 *Options menu*

Display numbers using	To select the *Significant digits* for the project.
Autocomplete [GSE]	To activate or deactivate the autocomplete function in the command line.
Licencing information	To switch on or off showing the licence information at startup.

3.3.9 *Expert menu* [GSE]

Examine commands	To display the commands executed in the project and to enable their examination.
Run commands	To open the **Commands runner** window.

Macro library	To modify and run macros. Macros can be defined and indexed in the **Macro library** window, displayed as the corresponding option is selected in the sub-menu. To run a macro click the corresponding option in the sub-menu.
Run Python script	To configure remote scripting server and open python script to run them (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)). Visit Plaxis Remote Scripting Interface [GSE] (on page 504) for more information.
Run Python tool	To configure remote scripting server and run python scripts (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Python	To **Configure Python Interpreter**, open and use **Interpreter**, **Editor**, **Command prompt**, **Jupyter notebook** and to run python scripts (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Configure remote scripting server	To specify an available port and open it for connections by local or remote clients (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Sensitivity Analysis and Parameter variation [GSE] (on page 477)	To evaluate the influence of model parameter on calculation results.
Plaxis 2D to 3D Converter	To automatically convert the geometry and soil materials of a PLAXIS 2D model into a PLAXIS 3D model.
Calculation manager	To easily queue up PLAXIS 2DPLAXIS 3D calculations run unattended calculations on projects (Typically less than 10 projects), or use manual control to run calculations manipulating them with the command line or python scripting. For more information visit Bentley communities.
View files	To display the contents of the (binary) files used in the current project.

3.3.10 *Help menu*

Manuals	To display the manuals.
Command reference	To display information about commands in the program.
Scripting reference	To open a Jupyter notebook with examples of PLAXIS commands using the Python scripting capabilities

Instruction movies	To reach the Bentley communities website where instruction movies are displayed.
Request support	To send a request for support.
Licence configuration...	To view, configure and update licences.
Visit website	To reach the PLAXIS 2D product page.
News Feed	To launch the PLAXIS 2D latest news
Disclaimer	The complete disclaimer text is displayed.
About	Information about the program version and licence are displayed.

3.4 Input program structure - Modes

The modelling process is completed in five modes. The mode tabs are shown in the mode bar. The modes are separated into **Geometry** and **Calculation** modes.

3.4.1 *Geometry modes*

The geometric configuration of the project is defined in the **Geometry** modes which are indicated by blue coloured tabs in the Input program. All the changes of geometry (such as creation, relocation, modification or removal of entities) are only possible in the **Geometry** modes. Features, such as structures (plates, beams), interfaces or loads, can be assigned to geometric entities only in the **Structures** mode.

The **Geometry** modes are:

Soil	The soil stratigraphy, the general water levels and the initial conditions of the soil layers are defined in the **Soil** mode. The features required to define the soil material and stratigraphy in the geometry are available. A detailed description on modelling soil stratigraphy is given in Material properties and material database (on page 160). Note that the soil stratigraphy can only be edited in this mode.
Structures	The geometric entities as well as the structural elements and forces in the project are defined in the **Structures** mode. Note that features, such as structures (plates), interfaces or loads, can be assigned to geometric entities only in the **Structures** mode.

3.4.2 *Calculation modes*

The calculation process is defined in the **Calculation** modes which are indicated by green coloured tabs in the Input program. In these modes entities cannot be created and new features cannot be assigned to existing

geometry entities. However, the properties of the defined features (material dataset, load values) can be modified.

The **Calculation** modes are:

Mesh	The geometry model is discretised and a finite element mesh is generated in the **Mesh** mode. The geometric configuration cannot be modified in this mode. The mesh should be regenerated whenever the geometry of the project is modified.
Flow conditions	Water levels for a simple hydrostatic pore pressure distribution as well as boundary conditions for a groundwater flow analysis can be defined. Furthermore, boundary conditions for a thermal flow analysis can be defined in the **Flow conditions** mode.
Staged construction	Parts of the geometry model can be activated/deactivated and properties can be modified. The project is calculated in the **Staged construction** mode.

Note: The **Flow conditions** and **Staged Construction** modes are known as ***Phase definition*** modes.

3.5 Model in the drawing area

The drawing area displays the physical model and is automatically updated after any change in the geometry. The buttons in the toolbar located at the left side of the drawing area (side toolbar) can be used to create and modify the model. The availability of the tools depends on the active mode. These features are described in the following chapters depending on their function.

3.5.1 *Supporting tools*

Supporting tools such as *Rulers*, *Coordinate axes*, *Snapping* and *Grid* can be used to simplify the definition of the model geometry. The buttons under the drawing area can be used to switch on/off these tools.

Rulers

At both the left and the top of the drawing area, rulers indicate the physical *x*- and *y*-coordinates of the geometry model. This enables a direct view of the geometry dimensions. The rulers can be switched off by clicking the corresponding button under the drawing area.

Origin and global axes

The physical origin of the model is indicated by the intersection of the *x*- and *y*-axes. Each axis is displayed in a different colour and their positive directions are indicated by arrows. The axes and the origin can be switched off by clicking the corresponding button under the drawing area.

Crosshair

A pair of fine lines, representing the *x*- and *y*-axes can be displayed at the cursor location in the drawing area when the corresponding button is clicked under the drawing area.

Grid and snapping

To facilitate the creation of the geometry model, the user may define a grid for the drawing area. This grid may be used to snap the pointer into certain 'regular' positions. The grid and snapping, either to the grid or to objects in the model, can be activated by clicking the corresponding buttons under the drawing area. To define the grid and the snapping options:

- Click the *Snapping options* button in the botton toolbar. The **Snapping** window pops up (Figure 12 (on page 37)). The options available are:
  - **Snap to object:** To switch on/off snapping to nearby objects.
  - **Snap to grid:** To switch on/off snapping to grid points defined by the **Spacing** and **Snap interval** settings.
  - **Spacing:** To set up the size of the grid cells, indicated by crossing perpendicular lines on the drawing area.
  - **Number of snap intervals:** To set intermediate snapping points inside the grid.

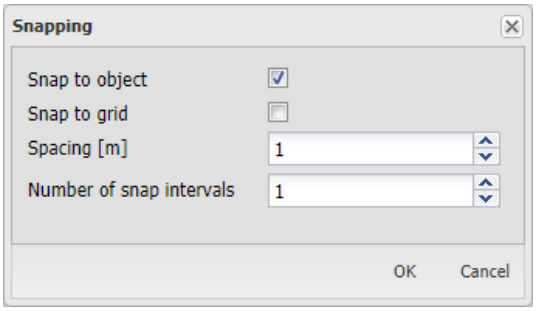


*Figure 12: The **Snapping** window*

3.5.2 *Model view - Zooming*

To zoom in and out of the view, scroll the middle button (wheel) of the mouse. The location of the mouse cursor is the centre of the zoom.

- To define a local zooming rectangle, click the *Zoom in* button. The zoom area must be indicated using the mouse. Click the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will zoom into the selected area. The zoom option may be used repetitively.
- To restore the original view, click the *Reset zoom* button or select the corresponding option in the *View* menu.

3.5.3 *Selecting geometry components*

- When the *Selection* tool is active, a geometry component may be selected by clicking once on that component in the geometry model. Multiple selection is possible by holding down the *Shift* key on the keyboard while selecting the desired components.

- In addition to selecting geometry components one by one by using the *Selection* tool, several geometry components can be selected at once by using the *Select multiple objects* tool. A rectangle can be drawn in the drawing area by clicking on the first corner of the desired rectangle and moving the cursor while holding down the mouse button. After release of the mouse button, all visible geometry components in this rectangle will be selected.

  The order in which the rectangle is created influences the selection type. Drawing from the upper left corner to the lower right corner selects all components that fall completely inside the rectangle. Drawing from the lower right corner to the upper left corner selects all components of which parts fall inside the rectangle.

  It is also possible to select multiple geometry components of one type (for example points, lines or surfaces) or multiple structural elements of one type (for example node-to-node anchors or plates) at once by selecting the desired sub-tool from the *Select rectangle* menu.

Note: Right-clicking can be used to select entities in the model. After right-clicking an entity or a multiple selections of entities either in the drawing area or explorers, a menu appears displaying the name of the selection and different options depending on the selection. Clicking the name of the object in the menu copies it to clipboard. The copied name can be used in the command line (General information on command line (on page 44)).

3.6 Explorers

Explorers display information about the physical model, the calculation phases and their configuration.

Model explorer	Information related to all the objects in the physical model is given.
Selection explorer	Information related to the object (or group of objects) selected in the drawing area is given. From a group selection, only the information valid for all the objects in the group.
Phases explorer	A list of the calculation phases defined in the project is displayed. The **Phases explorer** is available only in Calculation modes (on page 35). However, as phases are defined only in the *Phase definition* modes (see notes of Calculation modes (on page 35)), it is greyed out in the **Mesh** mode.

Note: The **Model explorer** and the **Selection explorer** are referred to as ***Object explorer***.

3.6.1 *Model explorer*

Model explorer displays information related to the physical entities composing the model. It is automatically updated as the model is modified (Figure 13 (on page 39)).

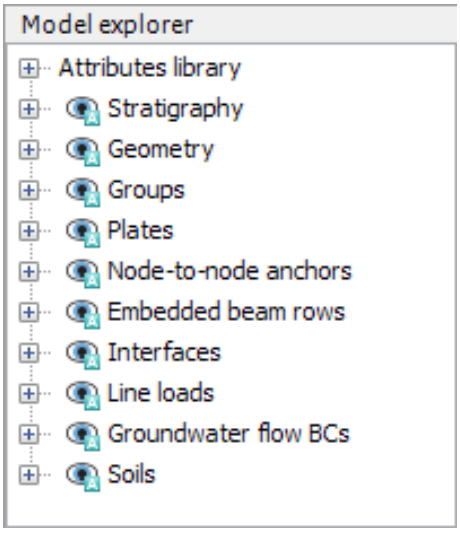


Figure 13: Layout of the Model explorer in different modes: Structures Mode

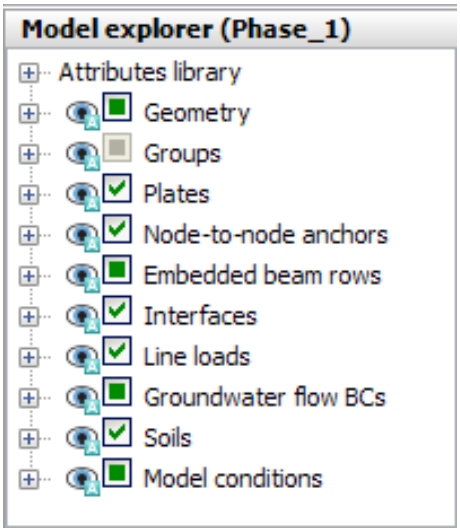


Figure 14: Layout of the Model explorer in different modes: Staged construction Mode

The information displayed in the *Object explorers* varies according to the active mode. Information related to geometry, such as location coordinates, is given in all modes, however it can only be edited in *Geometry* modes.

The visibility of the model components is indicated and can be modified in the *Object explorers* in all the modes of the program. However, activation/deactivation is possible only in the *Staged construction* mode.

The information displayed in the *Object explorers* depends on the active mode. The information is grouped under different categories.

Attributes library: The *Attributes library* contains global attributes that have been defined and that have been (can be) assigned to individual objects. An example of such an attribute is a material data set with properties. Other types of attributes are functions that describe time-dependent conditions, such as *Dynamics multipliers* or *Flow functions*. These functions can be assigned to dynamic loads and hydraulic boundary conditions, respectively, in order to describe the variation of a condition with time. Finally, the *Attributes library* contains

the set of water levels that have been created from the information in the boreholes, the soil clusters and the user-defined water levels.

Dynamics multipliers [ULT]	All the displacement and load dynamics multipliers defined in the project are listed under *Dynamics multipliers*.
Flow functions [ULT]	All the flow functions (*Head functions* and *Discharge functions*) defined in the project are listed under *Flow functions*.
Thermal functions 2D [ULT]	All the thermal functions (*Temperature functions*, *Heat flux functions* and *Heat total functions*) defined in the project are listed under Thermal functions.
Materials	All the material sets assigned to the entities in the model are listed under *Materials*. The identification and the colour representing the material data set are displayed.
Design approaches 2D [ADV]	All the design approaches defined in the project are listed under *Design approaches*.
Water levels	All the water levels created in the model are listed. • **Borehole water levels:** The water levels generated according to the water conditions defined in the boreholes are listed. • **User water levels:** The water levels created in the *Flow conditions* mode are listed. Note that the *Water levels* subtree is available only in the *Calculation* (green) modes.

Since these attributes are *Global*, any change in such an attribute will affect the model as a whole. If it is the intention to change an attribute only in a particular calculation phase, it is advised to duplicate the attribute using the corresponding option in the right-hand mouse menu, and make the change in the copy.

Stratigraphy: The soil layers and boreholes created in the model are listed under *Stratigraphy*. This section displays Borehole information such as coordinates, head, pore pressure values, etc. and the associated soil layers, information that is especially beneficial for remote scripting in PLAXIS 2D. The user can view the data stored in an entity by right-clicking on the same and clicking on the tabulate option. Stratigraphy information consists of *Soil layer* and *Boreholes* information that provide the following details:

Zones	Zones are the vertical positions of layer boundaries in the boreholes. For every zone the top and bottom of the layer is given as well as the thickness and the borehole corresponding to this zone.
Pore pressure	Includes information about pore pressure per layer and borehole.
Boreholes	Includes all boreholes information: coordinates, head and field data.

Geometry: All the geometric entities created in the model are listed under *Geometry*. Note that when a geometric entity is created, its sub-entities are automatically created by the program. When a line is created, the first (start) point and the second (end) point are automatically created. New geometric entities are also automatically created when a geometric entity is decomposed.

The available sub-groups :

Points	The points created in the model are listed. For each point, the coordinates of its location, the features assigned and their properties are given. Features such as *Point load*, *Point prescribed displacement* and *Fixed-end anchor* can be assigned to points. More information on points is given in Create point (on page 71).
Lines	The lines created in the model are listed. For each line the coordinates of its first and second points, the features assigned and their properties are given. Features such as beam, embedded beam, line load, line prescribed displacement and node-to-node anchor can be assigned to lines. More information on lines is given in Create line (on page 71).
Polygons	The polygons created or imported in the *Structures* mode are listed. For each polygon the coordinates of its reference point and the material data set assigned to it is given. More information on polygons is given in Create polygon (on page 72).
Soil polygons	The soil polygons generated according to the soil layers in boreholes or imported in the *Soil* mode (Material properties and material database (on page 160)) and the material data sets assigned are given.

Note: In the *Geometry* modes, coordinates indicating the location of geometric entities are in the *Model explorer*. It is possible to relocate the points by changing these coordinates. Note that this is not possible in the *Calculation* modes, where geometry modifications are impossible (Input program structure - Modes (on page 35)).

Note: In the *Calculation* modes, the *Coarseness factor* is given indicating the local refinement of the mesh at the geometric entity. The mesh can be refined or coarsened locally by modifying this value.

Connections: All the connections created explicitly by the user are listed.

Embedded beam rows: All the embedded beam rows created in the model are listed. Information about the assigned material and the connection type is given for each embedded beam row.

Soils: All the soils created in the model are listed. Information about the assigned material, volumetric strain and water conditions is given for each soil.

Plates: All the plates created in the model are listed. Information about the assigned material is given for each plate.

Geogrids: All the geogrids created in the model are listed. Information about the assigned material is given for each geogrid.

Tunnels [GSE]: All the tunnel cross sections created in the model are listed. Information about the segments composing the cross section as well as the insertion point of the cross section is given.

Line (distributed) loads: All the line (distributed) loads created in the model are listed. The distribution and the values of the load components can be defined.

Prescribed displacement: All the line prescribed displacements created in the model are listed. The distribution and the values of the prescribed displacement components can be defined.

Point loads: All the point loads created in the model are listed. The values of the load components can be defined.

Point prescribed displacement: All the point prescribed displacements created in the model are listed. The values of the prescribed displacement components can be defined.

Interfaces: All the positive and negative interfaces created in the project are listed. Either the material of the adjacent soil or a new material can be assigned. Permeability condition can be specified.

Node-to-node anchors: All the node-to-node anchors created in the model are listed. Information about the assigned material is given for each node-to-node anchor.

Fixed-end anchors: All the fixed-end anchors created in the model are listed. Information about the assigned material is given for each fixed-end anchor. The components and the equivalent length can be defined for each fixed-end anchor.

Groundwater flow BC's [ADV]: All the groundwater flow boundary conditions created in the model are listed. Information about the assigned behaviour (*Seepage*, *Closed* or *Head*), is given for each boundary condition. The assigned behaviour can be changed by selecting the option in the *Behaviour* drop-down menu.

Thermal flow BC's [ADV]: All the thermal flow boundary conditions created in the model are listed. Information about the assigned behaviour (*Closed*, *Temperature*, *Inflow*, *Outflow* or *Convection*), is given for each boundary condition. The assigned behaviour can be changed by selecting the option in the *Behaviour* drop-down menu.

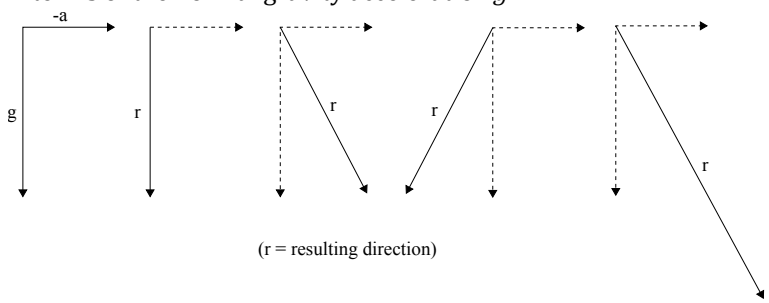
Wells [ADV]: All the wells created in the model are listed. The behaviour (*Extraction*, *Infiltration*), the discharge ($|q_{well}|$) and the minimum head (h_{min}) can be defined for each well.

Drains [ADV]: All the drains created in the model are listed. The head can be defined for each line drain.

Model conditions: The general boundary conditions of the model as a whole for each calculation phase can be specified in the *Model conditions* subtree in *Model explorer*. The conditions at the extremities of the model are considered. This makes it possible to make a quick selection of general boundary conditions that apply to the model as a whole. The *Model conditions* subtree is available in the *Calculation* modes in the Input program. Modification of the information given in the *Model conditions* subtree is possible only in the *Flow conditions* and *Staged construction* modes. Note that any change in the model boundaries is applied only to the phase selected in Phases explorer.

The options available are:

Climate [ADV]	This option can be used for thermal calculations to specify a general thermal convective condition due to weather conditions to all boundaries that represent the ground surface. Other types of thermal boundary conditions can be defined using the corresponding feature in the Structures mode (Thermal conditions [ADV] (on page 146)).
Deformations	A set of general fixities is automatically applied to the boundaries of the geometry model for the selected calculation phase. More information on default fixities is given in Model conditions (on page 364). The default general fixities are removed when the corresponding option is set to *False*. In that case it is required to set the appropriate boundary conditions manually.
Dynamics [ULT]	The model conditions for a dynamics analysis can be defined at the extreme boundaries of the model. The options available are *None*, *Viscous*, *free-field*, *compliant base*, and *tied degrees of freedom*. Besides the boundary conditions, the relaxation coefficients (C_1 and C_2) can be defined for each phase. For more information on boundary conditions for *Dynamics* calculations see Model conditions (on page 364).
Groundwater Flow - steady state [ADV]	The model conditions for a steady state groundwater flow calculation or a consolidation analysis can be defined at the extreme boundaries of the model. The options available are *Open* and *Closed*.

Groundwater Flow - transient [ULT]	The model conditions for a transient groundwater flow calculation or fully coupled flow-deformation analysis can be defined at the extreme boundaries of the model. The options available are *Open* and *Closed*.
Precipitation [ADV]	This option can be used to specify a general vertical recharge or infiltration (q) due to weather conditions to all boundaries that represent the ground surface. Other types of flow boundary conditions can be defined using the corresponding feature in the *Structures* mode (Hydraulic conditions [ADV] (on page 131)).
PseudoStatic [ADV]	This option can be used to specify a global acceleration to model dynamic forces in a pseudo-static way. The input values of the x- and y-acceleration components are expressed in terms of the normal gravity acceleration g  *Figure 15: Resulting force direction r due to combinations of gravity and pseudo-static acceleration*
Thermal Flow - steady state [ADV] / *transient* [ULT]	This option can be used for thermal calculations for which the *Thermal* calculation type in the **Phases** window is set to *Earth gradient*, to specify an (initial) temperature distribution in the ground. The temperature distribution is defined by means of a reference temperature, T_{ref}, a reference level, h_{ref}, and an earth gradient (increase of temperature per unit of depth). This temperature distribution which is used in each soil cluster in which the thermal conditions parameter is set to *None*.
Water	The global water level for a selected calculation phase can be assigned in the *Water* subtree under *Model conditions* in *Model explorer*. More information of the usage of the global water level is given in Global water level (on page 323).

Note: The reference level, h_{ref}, is likely to coincide with the ground surface, but may also be taken above or below the ground surface. In case h_{ref} is below the ground surface, the temperature at the ground surface is set equal to the reference temperature, as defined in the *Constants* tabsheet of the **Project properties** window (Project properties (on page 21)). The temperature distribution from the ground surface to h_{ref} is then obtained from a steady-state thermal flow calculation, whereas the temperature distribution below h_{ref} is obtained from T_{ref} and the earth gradient.

3.6.2 *Selection explorer*

The *Selection explorer* displays information about the selection made in the drawing area. For multiple objects, the common information is displayed. The structure of the *Selection explorer* is the same as the one of the *Model explorer*.

3.6.3 *Phases explorer*

The *Phases explorer* is available in the *Calculation* modes in the Input program. It visualises the order of the calculation phases and it provides information for each calculation phase (e.g. calculation status, calculation type, loading type, etc.). Modification of the information given in the *Phases explorer* is possible only in the *Flow conditions* and *Staged construction* modes. More information on the *Phases explorer* is given in Figure 16 (on page 44).

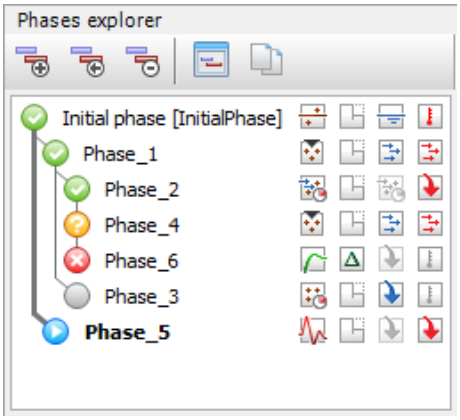


Figure 16: Phases explorer

3.7 General information on command line

At the bottom of the input application, a command line panel is available. The command line provides an alternative way of executing operations, by entering text commands. In fact, all operations in the input application generate a command as text, that is sent to an interpreter, before being processed. It is possible to make the command line panel into a new window, by right clicking in the command window and choosing the option *Pop out*.

It is also possible to copy *All* commands, *Successful commands* or *Successful responses*, by selecting the desired commands in the command line panel and right clicking the selected commands for menu options.

A feedback panel is situated above the command line panel. Here, the executed commands are shown, together with the feedback of the command. Successful commands return green coloured feedback, consisting of details on newly generated geometry or elements. Erroneous commands return red coloured feedback, containing an error report.

A command consists of a command name, followed by a number of parameters. Together, they constitute the signature of the command. The number and type of parameters depends on the particular command, and can be zero or larger.

Commands can be divided into two types:

Global commands	Have a global scope (like the project or the geometry) that are not necessarily named in the command, and may or may not have parameters. An example is the undo command.

Target commands	Have a local scope restricted to the selected (target) objects, such as geometric entities and materials. An example of such command is the `set` command.

Command names consist of a simple string. Many commands have a short name in addition to the standard, explicit long name (for example `point` and `pt`). Any command name can be prefixed with one underscore in order to distinguish it from any objects bearing the same name (the commands `undo` and `_undo` are identical). The application will always generate the commands with an underscore prefix. It is safe to omit the prefix in hand written code. There are also commands that start with two underscores. These are mostly intended for debugging, troubleshooting or advanced automation purposes. The underscores in those cases are not optional.

Command parameters can be of two different types:

References to objects	Represented by strings containing the name of an object. The names of such objects may not start with an underscore (_).
Value	Can be of many data types, such as strings, integers, floating point numbers, enumerations and booleans. String value parameters must be between single (') or double (") quotes, and may start and end with either one, two or three of these quotes. Examples of valid string value parameters are: "hello", "hello 'world'!", "Young's "modulus"". Floating point numbers must always use the dot decimal notation, even on systems that normally use comma as decimal separator. Enumerations may be represented either by a string value, or by the corresponding integer index of that value. Booleans are represented as True or False. In most cases where a value is expected, a property of that type is also allowed - this is indicated by a trailing single quote. For example when trying to set the x coordinate of a point, this can be done as`set point_1.x 5.2` or as `set point_1.x point_2.x`.

In some cases it is allowed to place parameters between parentheses, which is indicated by a trailing single quote in the parameter signature of a command. For example a point can be made with the command `point 1 2` or as `point (1 2)`. The parentheses are optional and are mostly useful for legibility, but may in some cases be necessary in order to distinguish between different types of calls to a given method.

In PLAXIS 2D, a global command is included that shows information on the commands that can be executed: `cms`. This command lists the signatures of all commands available in PLAXIS 2D.

As an example, the signature for the `delete` command is shown here:

```
delete (del)
    Material'
    <1,...: Feature'>'
    <1,...: Point' |Line' |GeoPolygon'>'
    <1,...: Borehole'>'
    Soillayer
```

In the first line, the command name and its abbreviation are shown. Each of the following lines shows a different set of parameters that can be used with this command. In this example there are five different signatures for the `delete` command. The `delete` command can be either used to delete a single material (referenced by its name),

a set of features, a set of geometric elements, a set of boreholes, or a single soil layer. Note that the signatures do not allow a feature, a geometric element or a borehole to be deleted in one single call of delete.

Below are several examples of the `delete` command:

- `delete SoilMat_1`
- `delete Polygon_1 Polygon_2`
- `delete (Polygon_1 Polygon_2)`

3.7.1 *Indexing in commands*

Array indexing syntax can be used in the command line. This consists of square brackets behind an object name, followed by either an integer or a string and a matching closing bracket. Integer indexing is zero-based and works on any listable object, i.e. any object on which you can use a `filter` or `tabulate` command. Both positive and negative numbering is used in indexing. The positive numbering begins at the start of the list, in which 0 is the first item from the list, while negative numbering starts from the end and goes backwards through the list. As an example:

`>line (1 2)(5 1)(5 3)(4 7)`	`#Create points with coordinates specified in the brackets and lines connecting them`
`>tabulate Points "x y"`	`#List the created points and their coordinates`

`Object x y`
`Point_1 1 2`
`Point_2 5 1`
`Point_3 5 3`
`Point_4 4 7`	

`>move points[0] -1 -2`	`#Relocate point_1 to (0 0)`
`>move points[-1] -1 -2`	`#Relocate the last added point, point_4 to (3 5)`
`>move points[-4] 1 2`	`#Relocate the first added point, point_1 to (1 2)`

Note that the program indexes the entities in the order they are created. The indexing of features such as soil, structural elements, loads or prescribed displacements, is NOT done according to the order of their assignment to geometry but according to the order of creation of the geometric entities they are assigned to.

3.7.2 *Accessing Output commands in Input command line*

The prefix `/output` allows redirecting commands to Output when they are entered in Input. This may be useful for generating one-file scripts that also call Output, for example, selection of nodes for curves. Examples:

`>/output addcurvepoint "Node" (-2.5 25)`	`#Adds curve point (-2.5 25 5) before starting calculations in Input.`
`>/output update`	`#List the created points and their coordinates`
`>/output getresults InitialPhase "UTOT___R"`	`#Displays the minimum and maximum or uniform value of the calculation results of block 'UTOT___R' in the initial phase.`

4 Ground modelling - Soil mode

The soil stratigraphy is defined in the soil mode using the *Borehole* feature of the program. When a new project is created, the soil contour defined in the **Project properties** window is displayed in the drawing area.

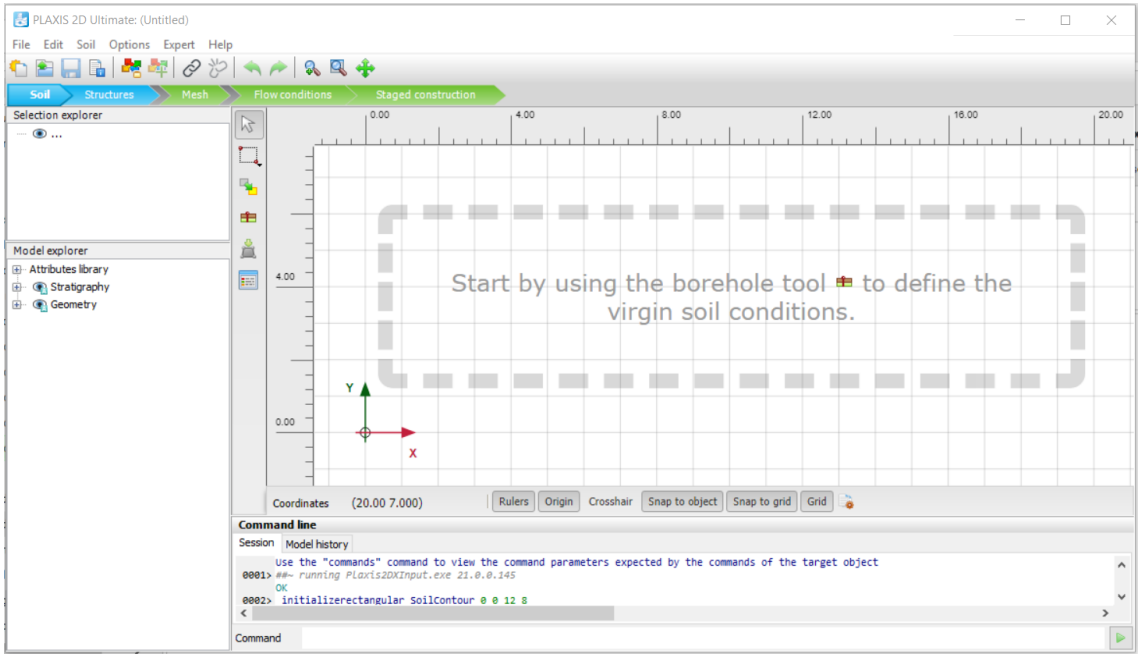


*Figure 17: View of a new project in the **Soil** mode*

4.1 Adjustment of the soil contour

The extension of the model can be specified in the **Model** tabsheet of the **Project properties** window. The **Project properties** window can be activated by selecting the corresponding option in the **File** menu.

4.2 Creating boreholes

Boreholes are locations in the drawing area at which the information on the position of soil layers and the water table is given. If multiple boreholes are defined, PLAXIS 2D will automatically interpolate between boreholes, and derive the position of the soil layers from the borehole information. Each defined soil layer is used throughout the whole model contour. In other words, all soil layers appear in all boreholes. The top and the bottom boundaries of the layers may vary through boreholes, making it possible to define non-horizontal soil layers of non-uniform thickness as well as layers that locally have a zero thickness.

4.2.1 *Creating new boreholes*

To create a borehole click *Create borehole* and click on its location in the drawing area. The **Modify soil layers** window pops up (Figure 18 (on page 49)). Boreholes and soil layers can be modified in this window.

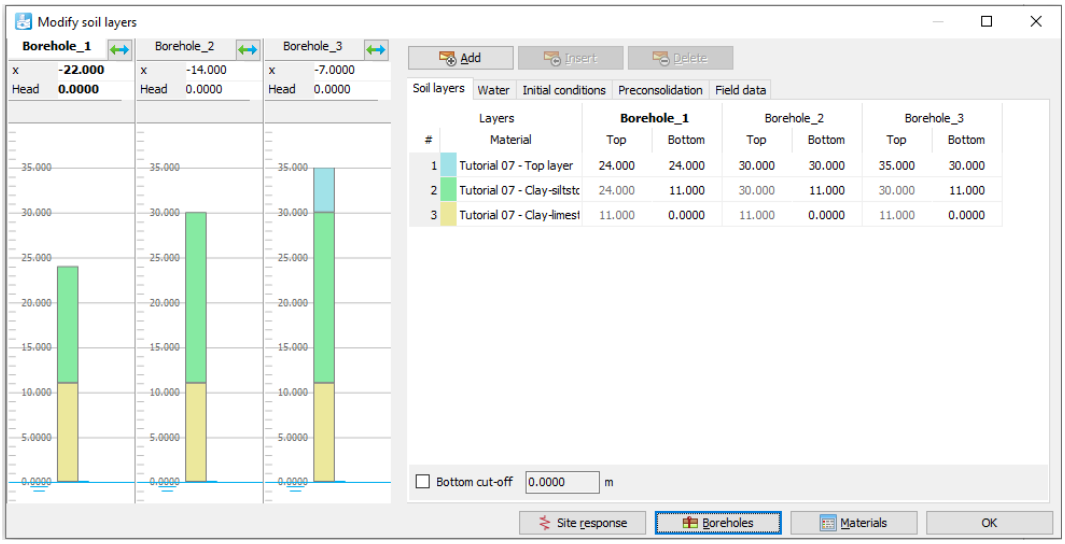


*Figure 18: **Modify soil layers** window*

To add new boreholes in the **Modify soil layers** window:

- Click *Boreholes* (lower button) and select the *Add* option. The **Add** window pops up (Figure 19 (on page 50)).
- In the **Add** window define the position of the new borehole (*x* coordinate).
- Define the source borehole to copy initial layer boundaries from. By default, the source borehole is the nearest one.

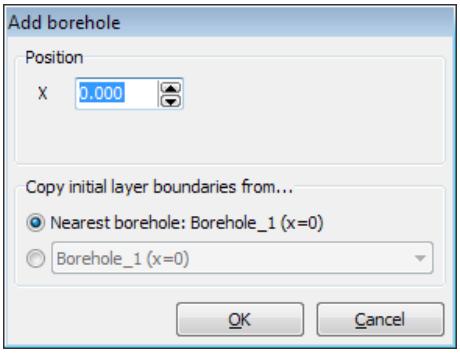


*Figure 19: **Add borehole** window*

Note: A warning message appears in the **Modify soil layers** window when multiple boreholes are located at the same position.

4.2.2 *Editing boreholes*

Boreholes are named consecutively by the program. Boreholes can be renamed in the *Model explorer* and in its soil column in the **Modify soil layers** window. When changing the name, use just letters, digits, and the only allowed special character (underscore '_'). The location of a borehole can be changed by specifying the coordinates of the new location in its soil column in the **Modify soil layers** window.

To relocate a borehole in the **Drawing area**:

- Click the *Move borehole* button in the side toolbar.
- In the drawing area select the borehole and drag and drop it to the new location.

The interpolation of layer boundaries is automatically adapted according to the new borehole position.

The display of the boreholes in the **Modify soil layers** window is managed using the options in the menu appearing as the *Boreholes* button is clicked.

Add...	To create a new borehole.
Add array...	To create multiple boreholes organised in a rectangular pattern.
Select	The boreholes to be displayed in the **Modify soil layers** window are selected.
Show all	All the created boreholes are shown.
Hide all	All the created boreholes are hidden.
Invert visibility	The view of the **Modify soil layers** can be inverted to selected or unselected boreholes.
Sort by	The appearance order of the boreholes in the **Modify soil layers** window can be arranged according to the *Creation order*, *Name* and *Position*.

4.3 Soil layers

The first borehole, created at the start of a new project, contains no soil layers. Soil layers in boreholes are defined in the **Modify soil layers** window.

The **Modify soil layers** window contains the following items (Figure 18 (on page 49)):

Soil column	Graphs of all boreholes with indication of their locations, water head, layer boundary levels and soil material.
Upper buttons	Buttons to add, insert or delete soil layers.
Soil layers tabsheet	Table showing the boundaries and the materials of the soil layers.
Water tabsheet	Table showing the water conditions in every layer and the top and bottom pressure values per borehole in each layer boundary.
Initial conditions tabsheet	Table showing the name, material model and the initial stress ratio, $K0_x$, for all soil layers.
Preconsolidation tabsheet	Table where pre-overburden pressure (POP) can be defined.
Field data tabsheet [GSE]	This tabsheet enables the import of CPT measurements. More information on the usage of CPT logs to generate the soil stratigraphy is given in Generating soil stratigraphy from CPT logs [GSE] (on page 57).
Bottom buttons	Buttons to add, select, visualize or sort boreholes, to open the material database and to accept (*Ok*) the soil layers. Pressing *Ok* closes the **Modify soil layers** window.

4.3.1 *Creating soil layers in boreholes*

Soil layers are created using the top buttons in the **Soil layers** tabsheet.

Add	To add a new layer below the lowest layer in the model.
Insert	To insert a new layer above the selected one.

The thickness of the new layers is zero length units by default. The thickness of the top soil layer in a borehole can be changed by modifying its *Top* and *Bottom* boundaries. The top boundary of an underlying layer is defined by the lower boundary of the overlying layer. To change the thickness of such a layer, the user can alter the *Thickness* option in the *Zones* of the soil layer under *Stratigraphy* tree in the **Model explorer** or simply by changing the bottom boundary of this soil layer.

If a new borehole should contain a layer that does not exist yet, this additional soil layer may be added using the *Add* or *Insert* buttons. In principle, this action creates a new soil layer in all existing boreholes, but the new layer

has a zero thickness in all the boreholes that have been defined earlier to ensure that existing layer distributions are not influenced by this action. In the current borehole the layer thickness can be modified according to the description given in the previous paragraph.

An existing layer may be removed by right-clicking it, either in the tabsheet or in the **Soil column**, and select the option *Delete* in the appearing menu. It should be noted that the layer will be deleted in all the boreholes of the project. If a layer is selected and the *Delete* button is clicked, the **Delete soil layer** window will appear (Figure 20 (on page 52)).

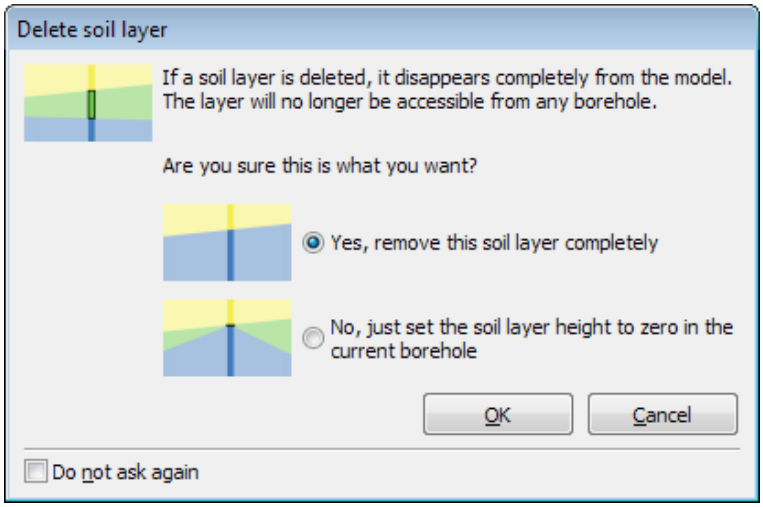


Figure 20: ***Delete soil layer*** *window*

Assigning soil properties

If a certain soil layer does not exist at the position indicated by a model borehole, it should be assigned a zero thickness (bottom level equal to top level) in the relevant borehole of the model.

Different soil layers will have different properties. Individual layer properties can be defined in material data sets. Material data sets are contained in the material database. There are several ways to assign material properties to a soil layer in the **Modify soil layers** window. You can:

- Click the *Materials* button. The **Material sets** window pops up. Drag and drop the material to the corresponding layer either in the tabsheet or in the soil column.
- Click the material either in the **Soil layers** tabsheet or in the **Initial conditions** tabsheet. Select the material from the drop-down menu of defined material sets.
- Right-click the soil layer in the tabsheet, point to *Set material* and then click the material you want to assign.

The colour of the layer in the geometry indicates the material set assigned. The procedure of assigning material datasets to soil layers should be repeated until each layer has its appropriate data set. The names and colours of the material data sets for all layers are shown in the **Soil layers** tabsheet. If the material data set of a layer is changed in a borehole, the same change will be applied to the same layer in all other boreholes.

To assign material to soil layers in the drawing area in the *Soil mode*:

- Click the *Show materials* button in the side toolbar. The **Material sets** window pops up.
- Drag and drop the material to the corresponding layer.

4.3.2 *Defining water conditions*

Groundwater and pore pressures play an important role in the soil behaviour. PLAXIS 2D is based on effective stress principles, in which total soil stresses are divided into effective stresses (in the grain skeleton) and pore pressures (in the pores of the soil). This requires a proper definition of water conditions. In many cases groundwater flow can be neglected and the (steady-state) pore pressure distribution in the ground is more or less known from in-situ soil investigation data. In this case the definition of water conditions can be conveniently related to the creation of boreholes, which forms the basis for a direct generation of pore pressures. In situations where groundwater flow occurs, the pore pressure distribution is not necessarily a priori known, and it may be required to perform a groundwater flow calculation in order to generate the pore pressures in the soil. More details on groundwater flow calculations and the definition of hydraulic boundary conditions are given in Hydraulic conditions [ADV] (on page 131). The current section deals with the definition of water conditions in boreholes for a direct generation of pore pressures.

The water conditions of the soil layers can be defined for each borehole in the **Water** tabsheet by selecting one of the options available in the drop-down menu for each soil layer. There are two columns for each borehole showing the values of the pore pressure distribution at the top (p_{top}), and the bottom (p_{bottom}) of each soil layer. Note that pressure is considered to be negative.

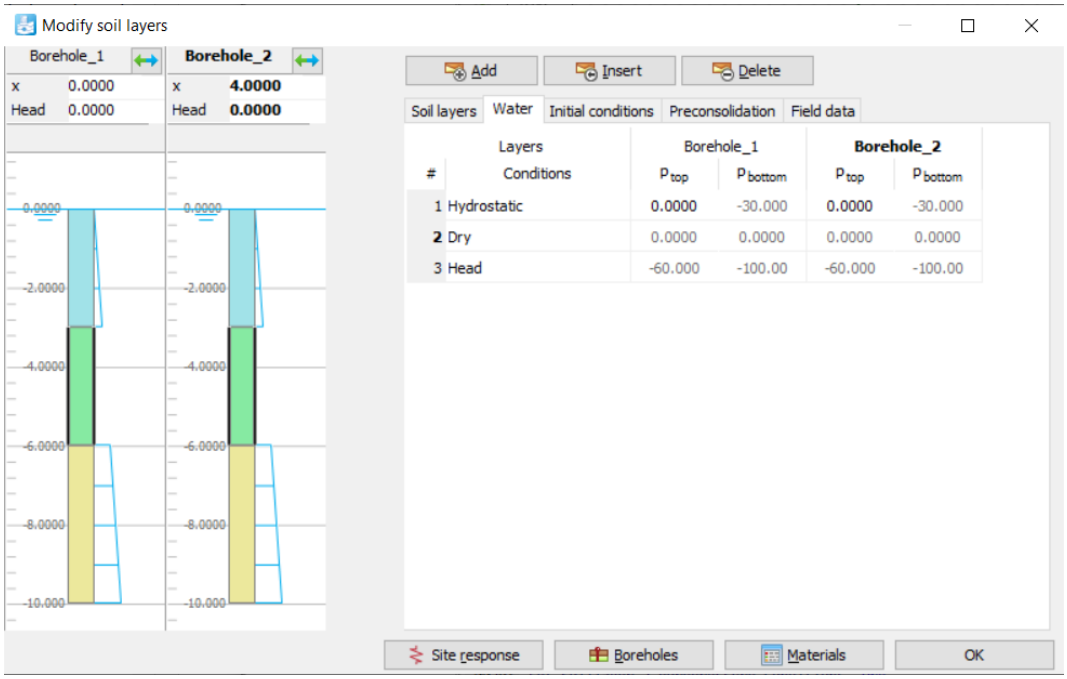


*Figure 21: **Water** tabsheet of the **Modify soil layers** window*

The options for water conditions are:

Head	The pore pressures are generated according to the specified head (phreatic level or water level) in each borehole. The generated pore pressure distribution is visualised in the soil column of the borehole. A head level lower than the top level of the layer, is indicated by a positive value (suction) at the top (p_{top}). However, positive pore stresses are not shown.
Hydrostatic	The pressure value at the top boundary of the soil layer is to be defined. The program will calculate the pore pressure distribution in the soil layer accordingly, taking into account the unit weight of water.
Interpolate	It enables linear vertical interpolation of the pore pressure distribution of the layer from the pressure at the boundary with the layer above to the pressure at the boundary with the layer below the current layer.
Dry	To remove water pressures from a soil layer select the *Dry* option. The value of pore pressure at the upper level of the layer (p_{top}) and the value of the pore pressure at the bottom level of the layer (p_{bottom}) will be zero and no pressure will be generated in the layer.
User-defined	Positive and negative values can be assigned to the top and bottom levels of the soil layer. The pore pressure distribution in the soil layer is interpolated linearly from these values. However, in the borehole only negative pressure values will be visualised. Note that PLAXIS 2D can deal with positive pore water stresses (suction) in calculations. Defining a positive value at the top of the layer and a negative value at the bottom means that there is a phreatic level in the layer.

The program generates water levels according to the information provided in the **Water** tabsheet. Water levels can represent external water levels as well as phreatic levels in the soil. The water level created according to the Head levels in the boreholes is automatically assigned as *Global water level* in the calculation phases. A more detailed description is given in Flow conditions (on page 350).

By default, positive pore water stress (suction) in the unsaturated zone above the phreatic level is ignored. However, PLAXIS 2D can deal with suction in calculations. This requires the selection of an appropriate soil water retention curve in the material data set (Groundwater tabsheet (on page 224)). It also requires that suction is allowed during the calculations (Flow control parameters [ADV] (on page 348)).

4.3.3 *Initial Conditions of Soil*

The **Initial conditions** tabsheet in the **Modify soil layers** window offers (Figure 22 (on page 55)) shows the soil model and the parameters. Double clicking one of these values in the **Initial conditions** tabsheet opens the corresponding material dataset in the material database.

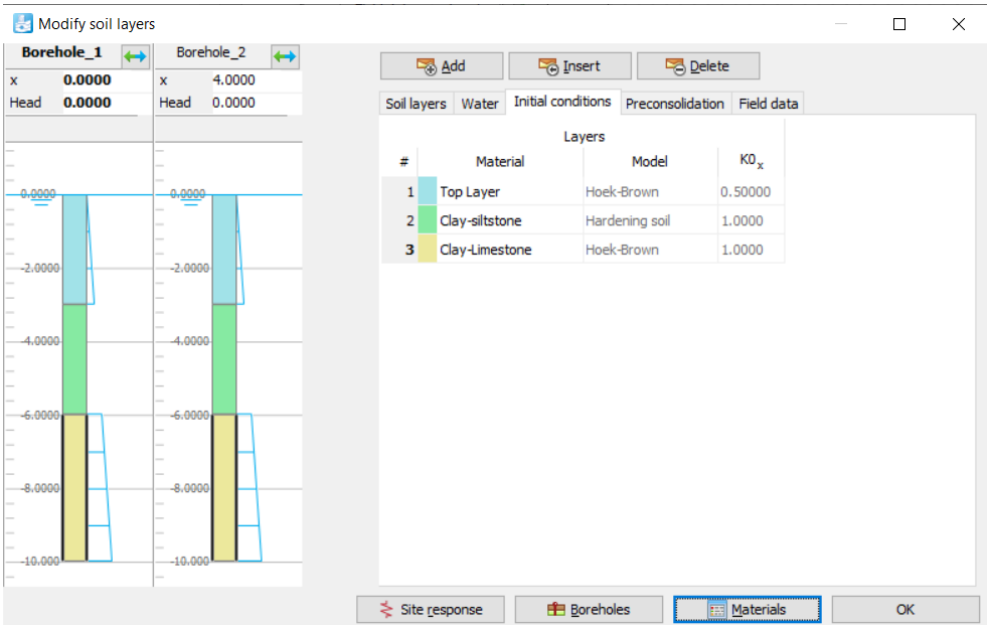


*Figure 22: **Initial conditions** tabsheet of the **Modify soil layers** window*

The initial stresses in a soil body are influenced by the weight of the soil, the water conditions and the history of its formation. This stress state can be generated using either the *K0 procedure* or *Gravity loading*. The generation of initial state stresses is described in Initial stress generation (on page 300).

If the *K0 procedure* is used, a proper $K0_x$value i.e. the initial ratio between horizontal effective stress and vertical effective stress needs to be specified for all layers.

4.3.4 *Preconsolidation*

The way to generate initial preconsolidation stress is defined in the **Preconsolidation** tab in the **Modify soil layers** window. It is on the POP or OCR data provided in the **Initial** tab in the material data set (Figure 23 (on page 56) and Initial tabsheet (on page 251)). Preconsolidation is most relevant when using advanced constitutive models that include preconsolidation pressure as a state parameter to distinguish between primary loading behaviour and unloading or reloading behaviour. In addition, preconsolidation may also be used to initialise a state parameter $\sigma'_{1,max}$ that is later used to account for the stress history to (re)initialise other (state) parameters. This is, for example, the case with SHANSEP type models, which are available as user-defined soil models in PLAXIS.

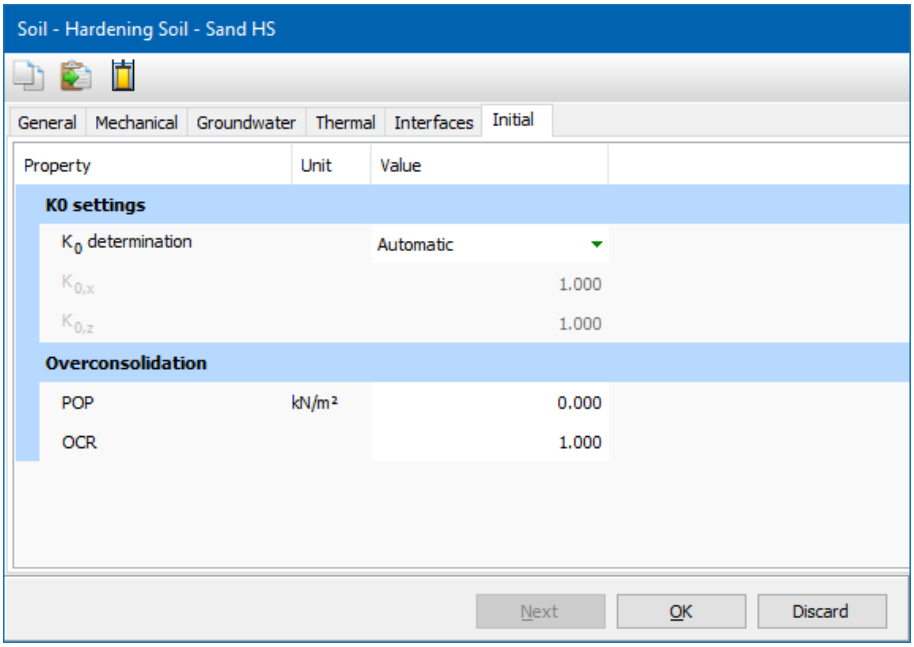


*Figure 23: **Initial** tabsheet in the **Soil** window*

The POP values for each stress point are used for the initialisation of the preconsolidation stress ($\sigma'_{1,max}$), which is initialised in the initial phase and kept updated over all calculation phases as a general state parameter). In the SHANSEP NGI-ADP and the SHANSEP MC models, the undrained shear strength is calculated according to the preconsolidation stress. In the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and Sekiguchi-Ohta model, the yield surface is defined consistently with the defined stress history of the soil. The POP (or OCR) is used to calculate the $\sigma'_{1,max}$ at the end of the *Initial phase* calculation. In the subsequent calculation phases, $\sigma'_{1,max}$ is compared to the current $\sigma'_{1,max}$ and updated if exceeded.

Currently, the only option available to define preconsolidation is *From material* and *From boreholes* (Figure 24 (on page 57)).

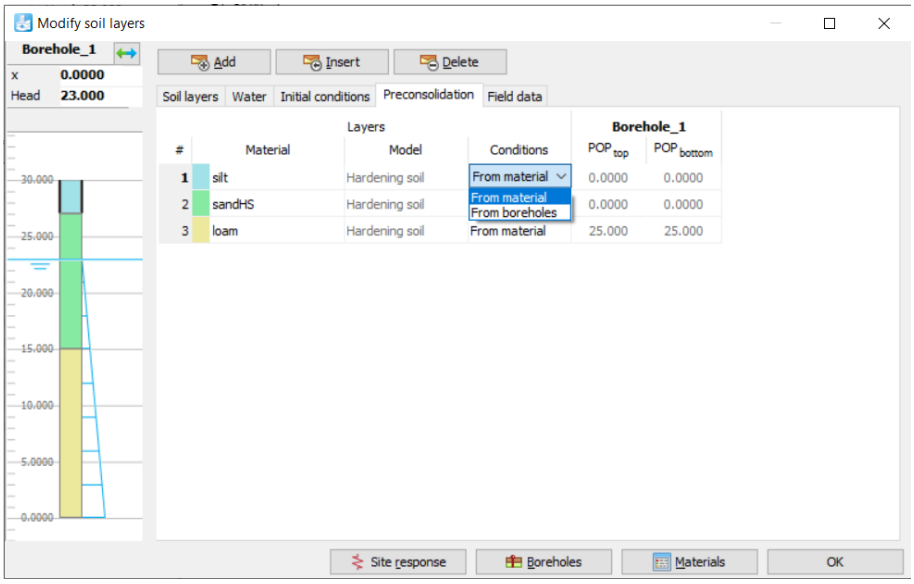


*Figure 24: The **Preconsolidation** tabsheet of the **Modify soil layers** window*

From material: The preconsolidation stress is calculated based on the information (OCR or POP) as defined in the material data sets.

From boreholes: In case the definition of the preconsolidation stress requires a spatial distribution of the pre-overburden pressure POP, such distribution may be defined by selecting the *From boreholes*. In the **Preconsolidation** tabsheet, there are two columns for each borehole showing the values of the POP distribution at the top (POP_{top}), and the bottom (POP_{bottom}) of each soil layer (Figure 24 (on page 57)). The spatial variation is linearly interpolated between the defined values and between the boreholes (in case of multiple boreholes).

Note:

- All soil layers defining POP_{top} and $POP_{bottom}$for one borehole result in a POP variation only in vertical direction.
- All soil layers defining different POP_{top} and POP_{bottom} for multiple boreholes result in a POP variation both vertically and horizontally.

4.3.5 *Generating soil stratigraphy from CPT logs* [GSE]

Cone Penetration Tests (CPT) are commonly used in geotechnical engineering practice to determine the stratigraphy and the type of materials, to estimate geotechnical parameters and to provide results for direct geotechnical design.

The test offers continuous measurements of the cone resistance (q_c) and the sleeve friction (f_s) along the depth. In the case of Cone Penetration Test with pore pressure measurement (*CPTu*) also the pore pressure (usually behind the cone, u_2) is measured.

In PLAXIS 2D, it is possible to load the data of a CPT test and use the Robertson method for the interpretation of the soil stratigraphy or the CUR 3 layers system for data interpretation. The CPT-based chart (Robertson et al., 1986 (on page 532)) is predictive of the soil behaviour type (SBT) and not directly of the soil classification

criteria based on grading and plasticity (USCS, Unified Soil Classification System) but there is often good agreement between the two types of classification.

Note: The soil stratigraphy in a borehole can be specified using available CPT logs that can be loaded in PLAXIS 2D. More information on the required format is given in Import CPT data: required file format [GSE] (on page 534).

Robertson Method

The *Robertson* classification is based on the non-normalised cone resistance which allows to identify 9 different areas with different behaviour. A simplified approach to assign the *SBT* to each measurement point consists of calculation of the index I_{SBT}. This index allows to identify only the soil layers with *SBT* from 2 to 7 (Figure 25 (on page 59)).

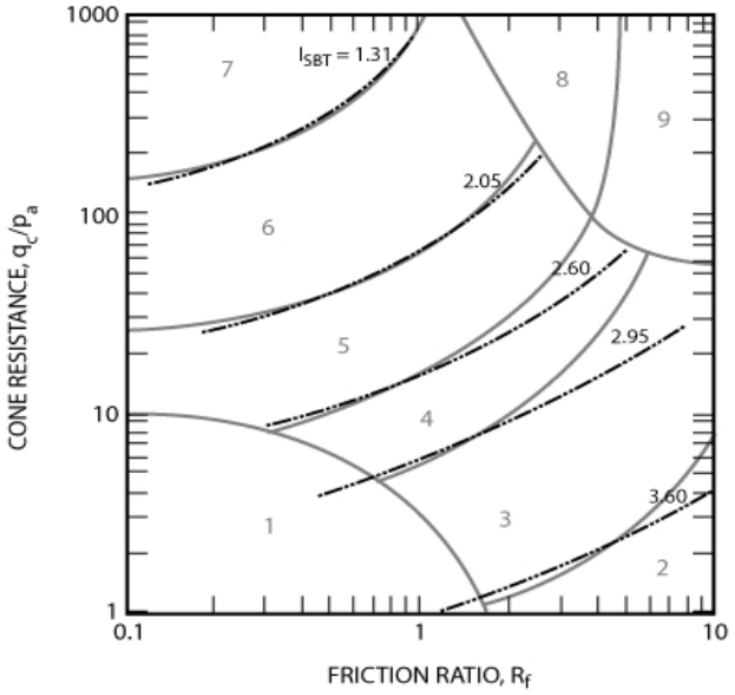
To simplify the application of the CPT charts, the cone resistance and the friction ratio can be used to calculate the *SBT* index, I_{SBT}, where the index is the radius of the concentric circles that represent the boundaries between the zones. Based on predefined ranges, it is possible to determine the *SBT* zone for each point.

The index I_{SBT} is given by the following equation:

$$I_{SBT} = \left[\left(3.47 - log\left(\frac{q_c}{p_a}\right)\right)^2 + \left(log\left(R_f\right) + 1.22\right)^2\right]^{0.5} \qquad Eq.\ [1]$$

where

q_c	=	Cone resistance
R_f	=	Friction ratio (=f_s/q_c)
f_s	=	CPT sleeve friction
p_a	=	Atmospheric pressure.



Zone	Soil Behaviour Type (SBT)
1	*Sensitive fine-grained*
2	*Clay - organic soil*
3	*Clays: clay to silty clay*
4	*Silt mixtures: clayey silt & silty clay*
5	*Sand mixtures: silty sand to sandy silt*
6	*Sands: clean sands to silty sands*
7	*Dense sand to gravelly sand*
8	*Stiff sand to clayey sand**
9	*Stiff fine-grained**

** Overconsolidated or cemented*

Figure 25: SBT chart based on cone resistance (q_c/p_a) and friction ratio R_f, showing contours of I_{SBT} (after Robertson et al., 1986)

Note:

- CPT data can be loaded in PLAXIS 2D in the form of simple text files. These files need to be constructed following the format (e.g. header and units) illustrated in Import CPT data: required file format [GSE] (on page 534).

The soil layers can be subdivided considering the following intervals:

I_{SBT}	Soil type	Description
$I_{SBT} > 3.60$	2	Organic soil - peat
$2.95 < I_{SBT} < 3.60$	3	Clays: clay to silty clay

I_{SBT}	Soil type	Description
$2.60 < I_{SBT} < 2.95$	4	Silt mixture: clayey silt and silty clay
$2.05 < I_{SBT} < 2.60$	5	Sand mixtures: silty sand to sandy silt
$1.31 < I_{SBT} < 2.05$	6	Sands: clean sand to silty sands
$I_{SBT} < 1.31$	7	Dense sand to gravelly sand

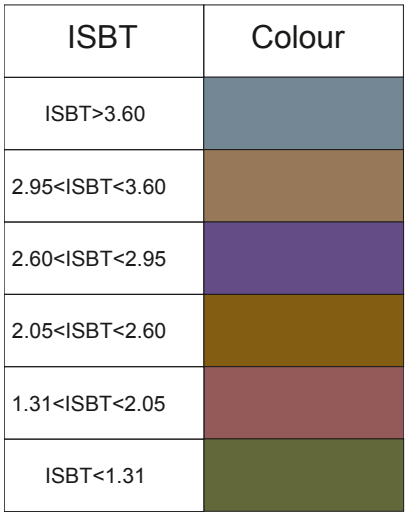


Figure 26: Colour scheme adopted in PLAXIS 2D representing different layers for different I_{SBT}

Importing CPT data

To import CPT data:

- Create a borehole in the model corresponding to the location at which CPT data are available.
- Proceed to the **Field data** tabsheet by clicking the corresponding tab in the **Modify soil layers** window.
- Add field data by clicking the green plus button and select the file containing the data in the **Open Field data** window. The available logs are listed in the **Field data** tabsheet.
- Click the expansion button next to the borehole.
- Select the data to be considered from the **Field data** drop-down menu.
- Select the *Interpretation* method, either the *Robertson* option or the *CUR 3 layers* method. The *CUR 3 layers* criteria are shown in Figure 26 (on page 60)
- Enter the minimal layer thickness (*Minimal thickness* option) to be considered in layer generation to avoid multiple thin layers. The expanded **Modify soil layers** window is shown in Figure 28 (on page 61).
- Click *Apply layers* to subdivide the soil cluster into the layers shown in the borehole.

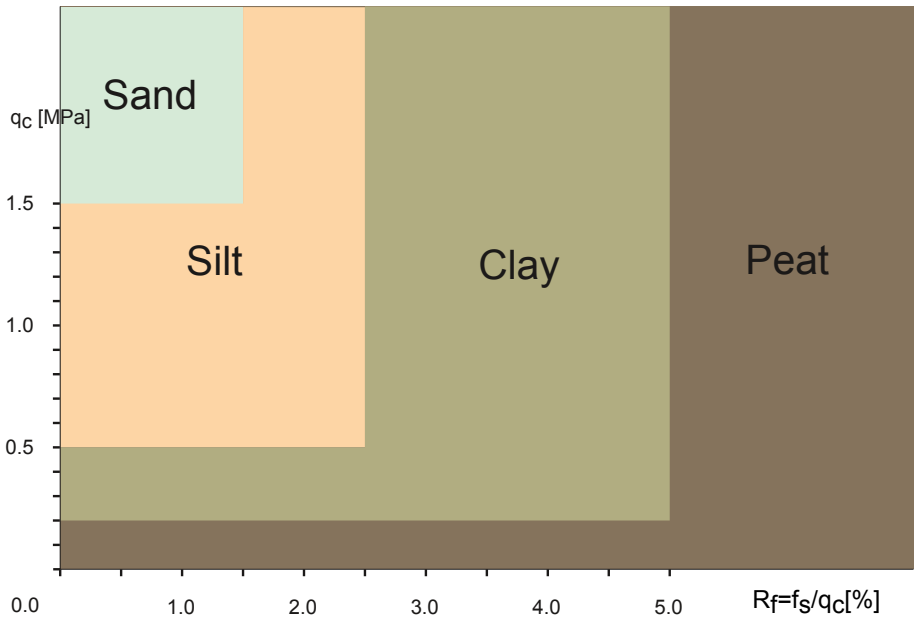


Figure 27: Layering criteria for CUR 3 layers methodCUR 3 layers method

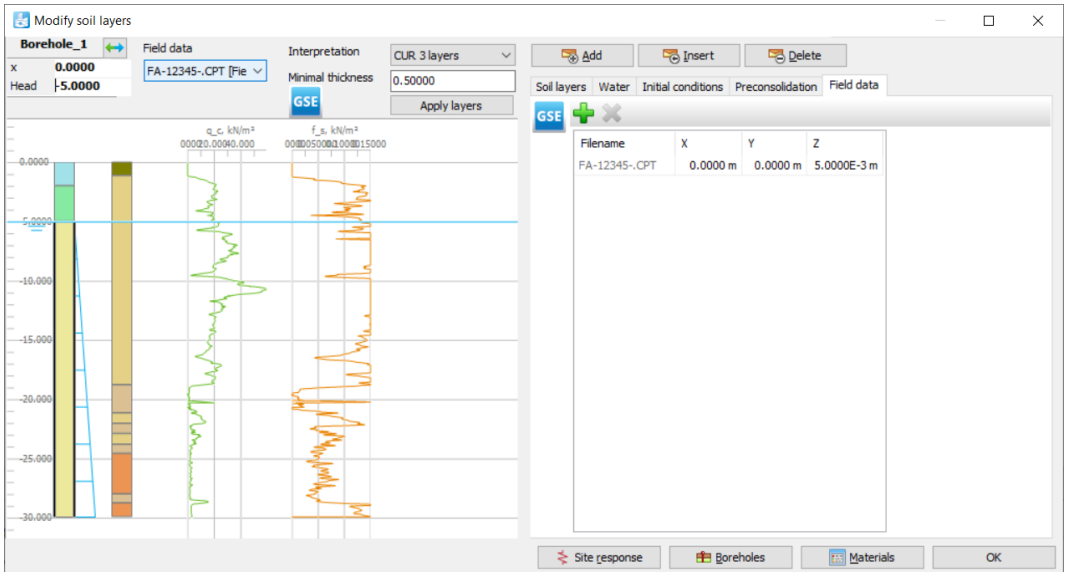


Figure 28: CPT data in the Modify soil layers window

Note:

- When soil stratigraphy is generated from CPT logs the created layers will be applied to the whole model. Boreholes can be used to modify the layer thicknesses at their location.
- However only one CPT log can be used in a project. Generation of layers resulting from a new CPT log will overwrite the existing ones.
- For more information about the importing process of CPT data please visit Bentley Communities.

4.4 Importing soil layers [GSE]

In the *Soil* mode, the geometry of the soil can be imported by clicking the *Import soil* button available in the side toolbar or by selecting the *Import soil* option available in the *Soil* menu. When importing a geometry file, a list of possible file extensions are shown. The supported file types are:

File extension	Description
*.dxf	AutoCAD interchange files.

*.csv	Comma-separated value files. An example for csv files is given:
	`Points`
	`0 , 0.00 , 0.00`
	`1 , 20.0 , 00.0`
	`2 , 20.0 , 10.0`
	`3 , 0.00 , 10.0`
	`Lines`
	`0 , 0 , 1`
	`1 , 1 , 2`
	`2 , 2 , 3`
	`3 , 3 , 0`

*.geo, *.sti	D-series files.

*.txt	Tab-separated txt files. An example for txt files is given:
	`Points`
	`0 0.00 0.00`
	`1 20.0 00.0`
	`2 20.0 10.0`
	`3 0.00 10.0`
	`Lines`
	`0 0 1`
	`1 1 2`
	`2 2 3`
	`3 3 0`

*.brep	OpenCascade shape files.
*.step	STEP files.
*.stp	STP files.

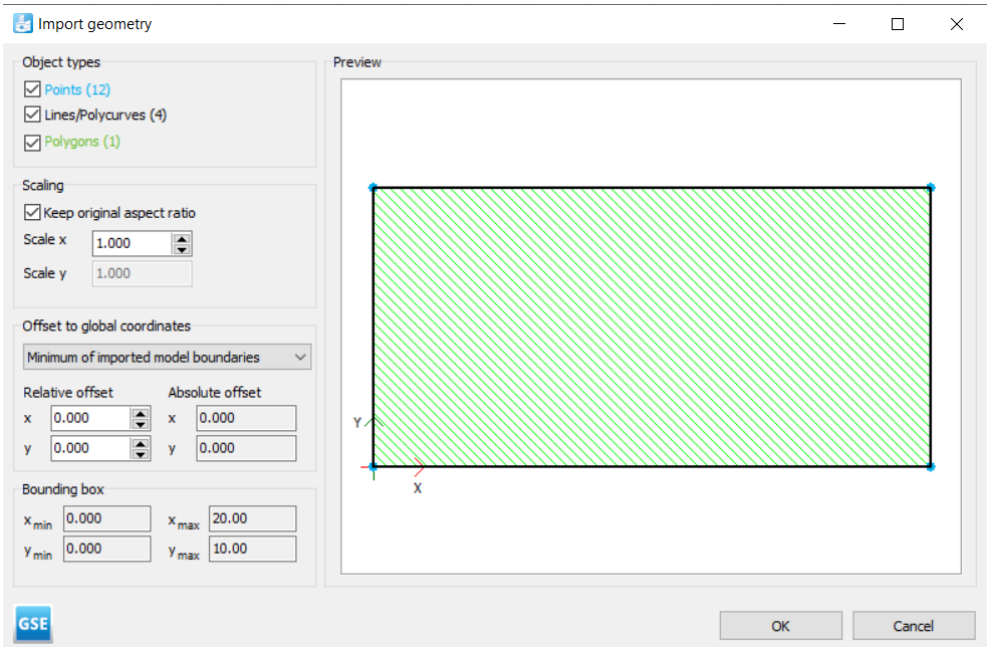


*Figure 29: The **Import geometry** window*

Before importing **.dxf* files, the *INSUNIT* global variable should be set to the desired unit Table 1 (on page 63).

Table 1: Default drawing units for AutoCAD blocks

Unit	*INSUNIT* value
Unitless	0
Inches	1
Feet	2
Miles	3
Millimeters	4
Centimeters	5
Meters	6
Kilometers	7
Microinches	8
Mils	9
Yards	10
Angstroms	11

Unit	*INSUNIT* value
Nanometers	12
Microns	13
Decimeters	14
Decameters	15
Hectometers	16
Gigameters	17

After selecting the file, the **Import geometry** window pops up (Figure 29 (on page 63)), where the geometry can be modified. It is possible to import only some of the geometry entity types by keeping the corresponding option(s) checked in the **Import geometry** window.

Note: The units of imported DXF and STEP files will be automatically converted into the predifined PLAXIS project units. However, for other formats such as BRep no conversion is executed. Therefore, the user needs to ensure that the units of the external file are consistent with the PLAXIS project units before importing. Otherwise, PLAXIS importer will interpret these external files as if they were already in the PLAXIS project units.

4.4.1 *Scaling*

The imported geometry can be scaled by defining a factor for scaling in all directions.

4.4.2 Offset to global coordinates

When a geometric object is imported, its reference point can be chosen in three different ways:

Minimum of imported model boundaries	The insertion point is automatically chosen as the minimum value of the model boundaries.
Maximum of imported model boundaries	The insertion point is automatically chosen as the maximum value of the model boundaries.
Custom	The user can enter a specific value for the insertion point.

Redefining the location of the insertion point will relocate the reference point.

4.4.3 *Bounding box*

The coordinates of the box bounding the imported geometry are shown.

Note:

- Material properties of polygons are not imported and need to be created in the material data base.
- A description of the feature and an instructional movie is given in Bentley Communities (https://communities.bentley.com/products/geotechnical1/w/wiki/41123/plaxis).

4.5 Site response analysis [ULT] + [GSE]

The *Site response* facility in PLAXIS can be used to perform a 1-Dimensional Site Response Analysis prior to a dynamics calculation. The site response analysis of a soil deposit can be considered as a necessary preliminary study for the dynamics analysis of a structure since its seismic response is influenced by the geotechnical properties of the supporting soil. The Site response facility works for any soil model, both standard and user-defined models. Only soil layers defined in the borehole window are considered for the analysis.

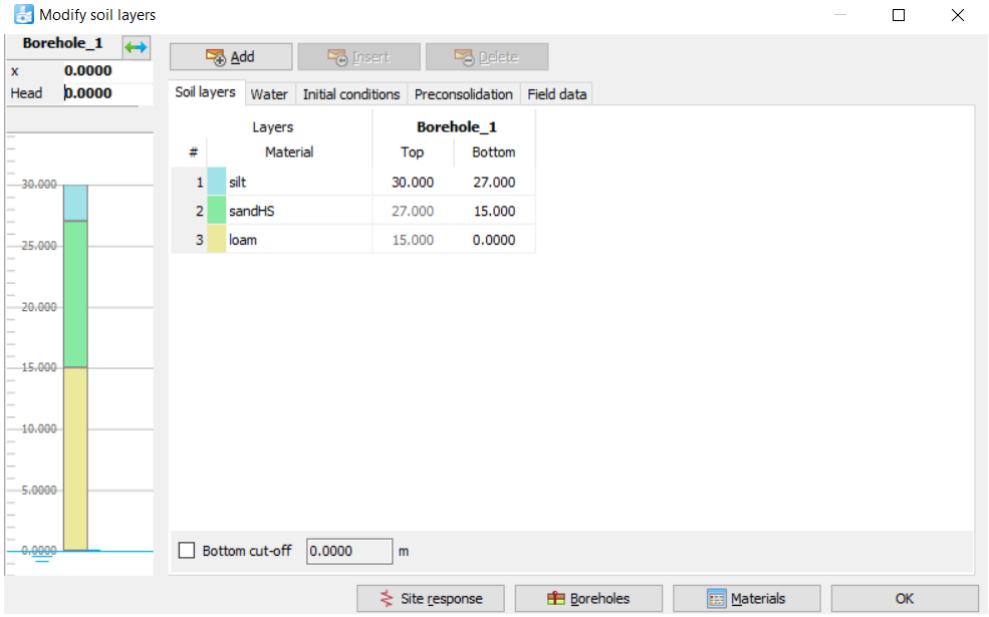


Figure 30: 1D Site Response Analysis Option in Modify Soil Layers window

The *Site response* option is available from the **Modify soil layers** window for any defined borehole (Figure 30 (on page 65)). The analysis will be based on the soils and rocks as defined in a single selected borehole. The user should conduct the analysis multiple times if it is required to perform an analysis for multiple boreholes. The prerequisites for 1D site response analysis are defining a borehole with corresponding head level, soil layers, soil parameters, and at least one accelerogram. It must be noted that only material models which are added to the borehole tab are adopted in the analysis and material models which are changed or modified in staged construction mode are not taken into account. Edit displacement multipliers can be used to define displacement multipliers easily. Any number of multipliers can be added for the calculation under a single project. Users can also toggle drift correction, if necessary for accelerations defined. Additionally, a bottom cut-off value can be defined for the model. The idea is that, in principle, a 1D site response analysis shall consider the entire soil profile all the way down to the bedrock, whereas the intended 2D (or 3D) dynamics analysis may be cut at a particular depth in the soil (for example 30 m depth). The 1D site response analysis may then provide the

required input signal at the base of the 2D (or 3D) model. The program will automatically extract an accelerogram of total motion at this cut-off level. If a bottom cut-off value is specified, the model will be restricted to that particular cut-off level for the project. However, in the 1D site response analysis, all clusters defined for the selected borehole will be considered.

Upon triggering the calculate button, the tool requests to configure the remote scripting server. The user is prompted to set a password and configure an available port. The script will extract all material models assigned for the borehole and all the displacement multipliers. In effect, the script will obtain all necessary data to recreate one borehole with all materials. The present model will be saved and closed. A new project will be automatically created for conducting the site response analysis. The new project is saved in a newly created subdirectory of the original project. The horizontal dimension of the model is chosen as 1 m. The vertical element size is critical for proper shear wave propagation which depends upon the minimum shear wave velocity in the soil layer defined and the higher frequency of interest. Mesh elements must be refined manually if the user expects especially low shear wave velocities or earthquake signals of a higher frequency of interest.

The soil column generated is a 2D plane strain model composed of 6-noded elements. Coordinates defined for the borehole are the same as that of the original model. The units are also copied from the original project. Soil layers, material models as well as the phreatic level are replicated from the original model. A prescribed line displacement is assigned at the base of the model, and an interface is created for assigning a compliant boundary condition. A dummy plate is assigned at the base of the model to retrieve the input signal after calculation. A medium mesh is generated with no local refinements. A K0 calculation will be performed followed by a plastic nil phase with default boundary conditions. For each input signal provided, a dynamic phase is created from the plastic nil phase (Figure 31 (on page 66)). Tied degrees of freedom are assigned for *XMin* and *XMax* boundaries, and a compliant base boundary condition is assigned for the base. Prescribed line displacements are set to 0.5 units for the base since only the upward propagating motion is considered for the compliant base. Dynamic calculation time depends upon the duration of the signal. The number of dynamic steps is determined based on the shortest time step available in the signal and the duration of the signal. The number of substeps is set to one and max-cores for dynamics calculation is also set to one to increase the efficiency in the case of multiple signals (they will run in parallel; each of them using a single core). Nodes are selected for every one meter of the soil column, and at the cut-off level, however, the results are extracted only for the midpoint of each soil cluster.

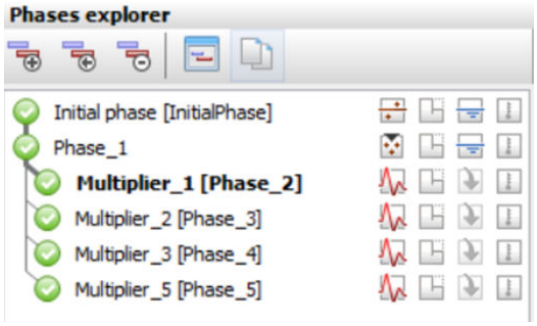


Figure 31: Dynamic phases generated in the new project

Before calculation, a pop-up window is displayed allowing the user to make manual modifications to the project. More information on ground response and liquefaction analysis in PLAXIS 2D can be found in Brinkgreve & Laera (2015) (on page 530). After the analysis, acceleration time histories are extracted for middle points of each cluster as well as at the cut-off level. They are stored with the file name in the following syntax - `OriginalMultiplierNamedepth.txt` as a text file. The extracted multipliers are total motions at a point and should not be interpreted as an upward propagating motion for a compliant base. The original input signal is also extracted at the base using a point selected at the dummy plate. All files are stored along with the project in the subfolder created inside the original project folder.

Finally, the original project is reopened with a pop-up window mentioning the location in which the project is saved. Now, the Edit displacement multipliers option may be used to import the generated multipliers at the cut-off level from the 1D site response analysis.

Note: The tool cannot distinguish between horizontal and vertical multipliers. It interprets multipliers as horizontal multipliers and conducts the analysis.

Note: For more information about Site response analysis in PLAXIS please visit Bentley Communities.

5 Modelling loads and structures - Structures mode

Geometric entities, structural elements and Soil boundary conditions are defined in the **Structures** mode. Click the *Structures* tab to switch to this mode. Boreholes cannot be modified in the **Structures** mode.

5.1 Assisting tools

The assisting tools enable modifying the geometry by changing the location of the object in the model or by creating higher objects with a higher dimension (from lines to surfaces or from surfaces to volumes).

5.1.1 *Selecting geometry components*

When the *Selection* tool is active, a geometry component may be selected by clicking once on that component in the geometry model. Multiple selection is possible by holding down the *Shift* key on the keyboard while selecting the desired components.

In addition to selecting geometry components one by one by using the *Selection* tool, several geometry components can be selected at once by using the *Select multiple objects* tool. A rectangle can be drawn in the drawing area by clicking on the first corner of the desired rectangle and moving the cursor while holding down the mouse button. After release of the mouse button, all visible geometry components in this rectangle will be selected.

The order in which the rectangle is created influences the selection type. When you select from left to right, it selects items completely within the box. When you select from right to left, it selects every item crossing the box boundaries.

It is also possible to select multiple geometry components of one type (for example points or lines) or multiple structural elements of one type (for example node-to-node anchors or plates) at once by selecting the desired sub-tool from the menu that appears when the *Select rectangle* button is clicked.

Note:

Right-clicking can be used to select entities in the model. After right-clicking an entity or a multiple selections of entities either in the drawing area or explorers, a menu appears displaying the name of the selection and different options depending on the selection. Clicking the name of the object in the menu copies it to clipboard. The copied name can be used in the command line (General information on command line (on page 44)).

5.1.2 *Move object*

A selection can be moved in the model. To move a selection:

- Select the objects to be relocated in the model (Selecting geometry components (on page 68))
- Click the *Move object* button and drag the selection to the new location.

5.1.3 *Copy object - Array*

Multiple copies of a selection, arranged in a rectangular or rotated pattern, can be created in the **Structures mode**.

- Select the source object(s) in the model. Multiple selections can be copied.
- Click the *Create array* button. Note that the *Create array* button is active only after the selection is made. The **Create array** window pops up where the pattern of the array can be defined.
- It is possible to choose a *Rectangular* or a *Polar* array pattern.
- For *Rectangular* array pattern, select the necessary option in the *Shape* drop-down menu. Enter the total number of rows (or columns) after copying and define accordingly the distances between adjoining copies.
- For *Polar* array pattern, select the necessary option in the *Plane* drop-down menu. Enter the coordinates for the center point, total number of items after copying, total angle over which the items are distributed equally and whether the objects are to be rotated.

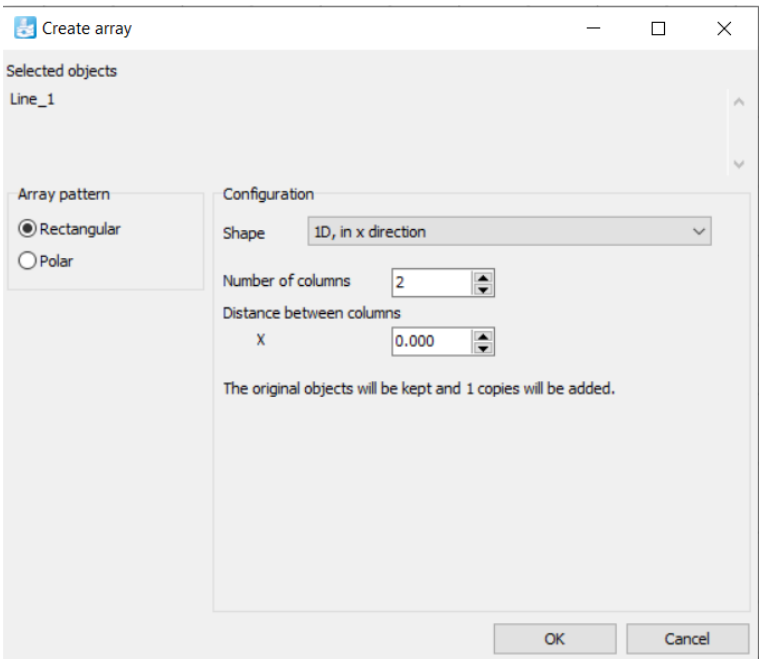


*Figure 32: **Create array** window, Rectangular array pattern*

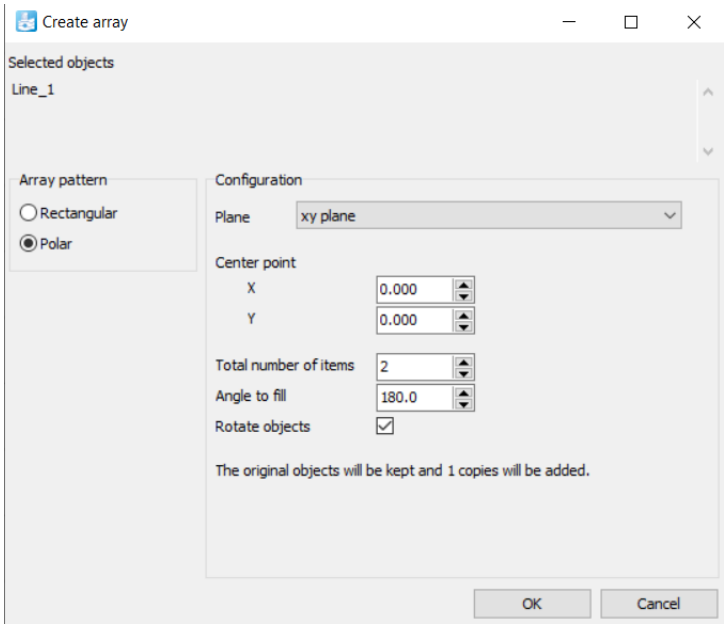


*Figure 33: **Create array** window, Polar array pattern*

Note: Note that when *Array* is used to create copies, besides the geometry entity itself, the assigned features and properties are copied as well. Hence, it is advised to first assign the features and properties before copying geometric entities.

5.2 Geometric entities

The geometric entities are the basic components of the physical model. Features such as structures, loads etc., can be assigned to geometric entities or a geometric entity with a feature assigned to it can be created by selecting the corresponding option available in the expanded menu of the geometric entity. The created entities in the model are automatically named by the program. They can be renamed in the *Object explorers*.

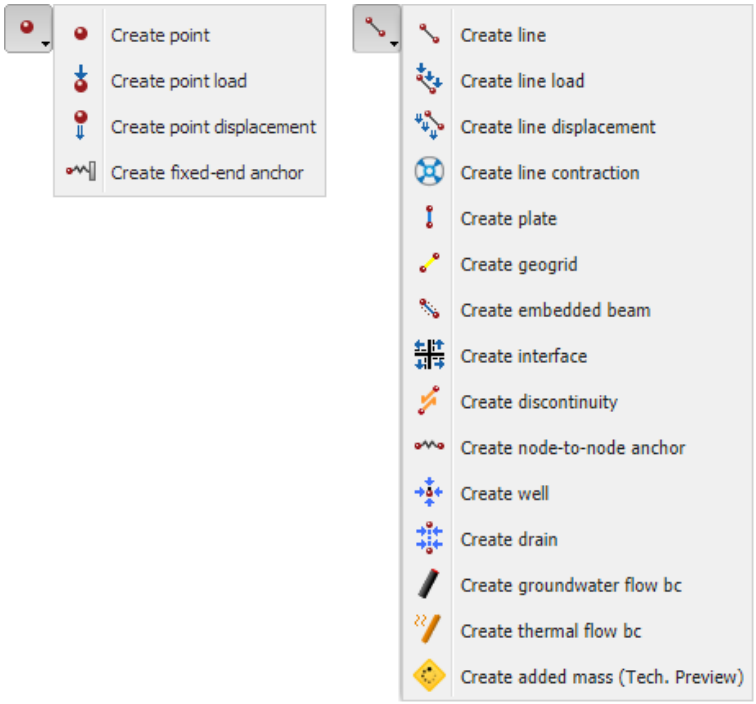


Figure 34: The expanded menus for Create geometric entities

A geometric entity can be relocated by selecting it first using the *Move* tool in the toolbar or by changing the coordinates location can be changed in the *Object explorers*.

There are two options of deleting entities in the model. To delete a geometric entity and all the features assigned to it, right-click it and select the *Delete* option from the appearing menu. To delete only features assigned, right-click it, click on the feature you want to delete in the appearing menu and click *Delete*.

Note: Note that using the *Delete* key, deletes the geometric entity and all the features assigned to it.

5.2.1 *Create point*

One of the basic input items for the creation of a geometry is a point. To create a point click the *Create point* button in the side toolbar and then click at the location of the point. A *Point load*, *Point prescribed displacement* and *Fixed-end anchor* feature can be assigned to a point.

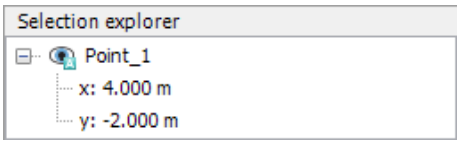


Figure 35: Point in the Selection explorer

5.2.2 *Create line*

Another basic input item for the creation of a geometry is a line. To create a line click the *Create line* button in the side toolbar and then click at the location of the first and the second point of the line in the drawing area. Optionally, additional points may be created resulting in a chain of lines. After creating the last (end) point of the line, right-click or use the *Esc* key to stop drawing. When a line is created, its first and second points are also created.

A *Distributed load*, *Prescribed displacement*, *Plate*, *Geogrid*, *Embedded beam row*, *Interface*, *Node-to-node anchor*, *Well*, *Drain* and *Groundwater flow boundary condition* can be assigned to a line.

Note: A node-to-node anchor or an embedded beam row cannot be assigned to a line to which other features are assigned.

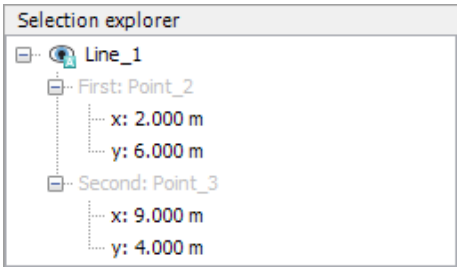


Figure 36: Line in the Selection explorer

5.2.3 *Create polygon*

The *Polygon* feature can be used to create soil clusters in the **Structures** mode. The options available in the menu expanded when the *Polygon* button is clicked are:

Create soil polygon	To create a general polygon by specifying the points that define the polygon select the *Create soil polygon* option and click on the locations of the points in the drawing area.
Create soil rectangle	To quickly create rectangular soil clusters select the *Create soil rectangle* option and click on the points defining the diagonal of the rectangle or click on the first point, drag the rectangle while keeping the mouse button down and release the button of the appropriate point.
Follow contour	To create a polygon which sides are partly shared with the existing geometry select the *Follow contour* option and define the unshared boundary. The program will automatically consider the existing contour as the remaining boundary and a cluster will be created.
Cut polygon	To divide existing soil clusters into subclusters select the *Cut polygon* option and define the shared border of the subcluster in the drawing area.

Add polygon point	To add a point defining a soil polygon select the *Add polygon point* option and click on the location of the point in the drawing area. The program will update the geometry according to the new point configuration.
Delete polygon point	To delete a point defining a soil polygon select the *Delete polygon point* option and click on the point to be deleted in the drawing area. The program will update the geometry according to the new point configuration.
Move polygon point	To relocate a point defining a soil polygon select the *Move polygon point* option and drag the point to a new location. The program will update the geometry according to the new point configuration.
Move polygon line	To relocate a line defining a soil polygon select the *Move polygon line* option and drag the line to a new location. The program will update the geometry according to the new point/line configuration.

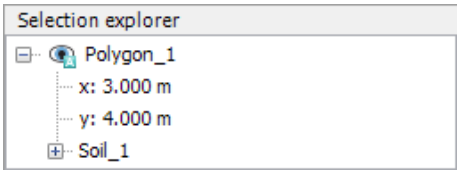


Figure 37: Polygons in the Selection explorer

5.2.4 *Advanced geometric modelling options*

Advanced options for modifying geometric entities in the model are available in the menu appearing as the entity (or multiple entities) is (are) selected by right-clicking in the *Object explorer* or the drawing area.

Note: Multiple entities can be selected by keeping the *Ctrl* key pressed while clicking on the entities.

Feature assignment to existing geometric entity

When the geometric entity is already available in the model (e.g. a line) it is advised to assign a feature to it (e.g. plate) without recreating the geometry in order to prevent the model from being unnecessarily large and complicated. An overview of the features assignable to geometric entities is given in Table 2 (on page 73).

Table 2: Assignable features

Loads		
Point	Line	Polygon
Load Prescribed displacement	Load Prescribed displacement	Volume strain
Assignable structures and conditions		

Fixed-end anchor	Contraction Plate Geogrid Embedded beam row Interface (Positive/Negative) Node to node anchor Well Drain Groundwater flow boundary condition Thermal flow boundary condition	

To assign a feature to an existing geometric entity:

- Right click the geometric entity either in the *Object explorer* or in the drawing area.
- In the appearing menu select the option corresponding to the feature that is to be assigned.

Note:

- Note that *Volume strain* can be assigned to soil clusters only in the **Phase definition** modes.
- If an embedded beam row or a node-to node anchor is assigned to a line in the model, no other features can be assigned to it.

Merge equivalent geometric objects

There are different options to create a feature:

- Using the buttons in the side toolbar.
- Assigning the feature to an existing geometric entity.
- Using the command line.

Consider the case when a geometric entity (e.g. a line) already exists in the model. If a node-to-node anchor is created at (nearly) the same position as the existing line, a new line will be added with the *Node-to-node anchor* feature. This means that, in principle, two lines will exist at that position; one with the *Node-to-node anchor* feature and one without. PLAXIS 2D allows for merging equivalent objects in order to remove unnecessary objects from the model. The way this works depends on which option has been selected to create the node-to-node anchor.

- If the above option 1 is used, the new line is automatically merged with the existing line into a single line having the *Node-to-node anchor* feature.
- If the above option 2 is used, no new line is created, and the *Node-to-node anchor* feature is directly assigned to the existing line.
- If the above option 3 is used, the existing line and the new line with the *Node-to-node anchor* feature will both exist in the **Structures** mode.

In the latter case it may be desired to already merge equivalent objects in the **Structures** mode (in order to manually remove unnecessary objects from the model). This can be done by right-clicking on *Geometry*, *Points*,

Lines in the *Model explorer* and selecting *Merge equivalent geometric objects* from the appearing menu, or by using the corresponding command (see command reference), for example:

```
mergeequivalents geometry
```

The `merge equivalent geometric objects` command may also be used to merge objects that are not exactly located at the same position, but very close to each other. By default, the distance tolerance is 0.001 length units, equal to the standard snap distance (see below), but the user can include the distance tolerance in the command, for example:

```
mergeequivalents geometry 0.2
```

When similar geometric objects were created not exactly at the same position but less than 0.2 length units apart, they will be merged into a single object by this command. Any feature assigned to one of the original objects will be assigned to the remaining single object.

Note:

- Boreholes cannot be merged.
- If an embedded beam or a node-to node anchor is assigned to a line in the model it cannot be merged to equivalent lines to which other features are assigned.

Snap

Drawing imperfections such as disconnected or overlapping geometric entities can lead to intersection or meshing difficulties. The *Snap* feature enables the correction of such imperfections in a range of 0.001 length units. The default value of snapping distance can be temporarily modified when snapping is performed using the corresponding command (see *Command reference*).

The *Snap* feature may be used in different ways:

- When the *Snap to grid* option (below the drawing area) is active, drawing of points is restricted to regular grid points. The grid spacing is generally large enough to avoid mispositioning of points and it also facilitates clicking on existing points.
- When the *Snap to grid* option is inactive and the *Snap to object* option is active, the drawing of points is not restricted to the grid points, but new points close to existing geometry entities will automatically be snapped to the existing objects, considering the defined snap distance. Since the snap distance is generally small, this allows for small distances between two separate points (or a point, a line or a polygon), and hence to the possibility that a point that is intended to be snapped to an existing geometric entity turns out to be a separate point.
- When both the *Snap to grid* and *Snap to object* options are NOT active, it is even more likely that two objects which are intended to coincide, are in fact separate.
- When both the *Snap to grid* and *Snap to object* options are active, new points will be snapped to the grid at intersections between grid and geometric entities, in other cases new points will be snapped to geometric entities.

Situations as described in 2 and 3 may be restored by using the `Snap` command in the command line with a larger value for the snap distance. For example:

```
snap geometry 0.1
```

```
snap lines 0.25
```

Points or lines that were supposed to coincide but turned out to be separate, may be merged by using the *Merge equivalent geometric objects* option (see above).

Another snapping option is available in the right mouse button menu enabling extension or trimming of lines. To utilise the feature:

- Multi-select the line and its end point that is to be extended.
- Add to the selection the extension target line (or trimming boundary line).
- Right-click and select the *Snap to object* option in the appearing menu. The geometry will be updated accordingly.

Group

The *Group* option in the right mouse menu enables creating groups of entities in the model. The created group will be listed under the *Groups* subtree in the *Model explorer*. Grouping enables modification of the common properties of the group at the same time. After grouping different entities into a group, the original entities still exist. Utilisation of the *Group* feature is described in Chapter *Dry Excavation using a tie back wall* of the 2D Tutorial Manual.

5.3 Importing geometry [GSE]

In the **Structures mode**, the geometry of the soil can be imported by clicking the *Import structures* button available in the side toolbar under the *Import...* button or by selecting the option available in the *Structures* menu.

Note that only surfaces and volumes can be imported. A detailed description on how geometry is imported is given in Importing soil layers [GSE] (on page 62). When importing a geometry file, a list of possible file extensions are shown. The supported file types are:

Table 3: Supported file types

Extension	File Type
*.dxf	AutoCAD interchange files.
*.csv	Comma-separated value files. An example for csv files is given in Importing soil layers [GSE] (on page 62).
*.geo, *.sti	D-series files.
*.txt	Tab-separated txt files. An example for txt files is given in Importing soil layers [GSE] (on page 62).
*.brep	OpenCascade shape files.
*.step, *.stp	STEP files.

Note: The units of imported DXF and STEP files will be automatically converted into the PLAXIS project units. However, for other formats no conversion is done (e.g BRep);the user needs to ensure the file units are consistent with the PLAXIS project units before the import. Otherwise, PLAXIS importer will interpret these external files as if they were already in the PLAXIS project units.

After selecting the file, the **Import geometry** window pops up, where the geometry can be modified. It is possible to import some of the geometry entity types by keeping the corresponding options checked in the **Import geometry** window.

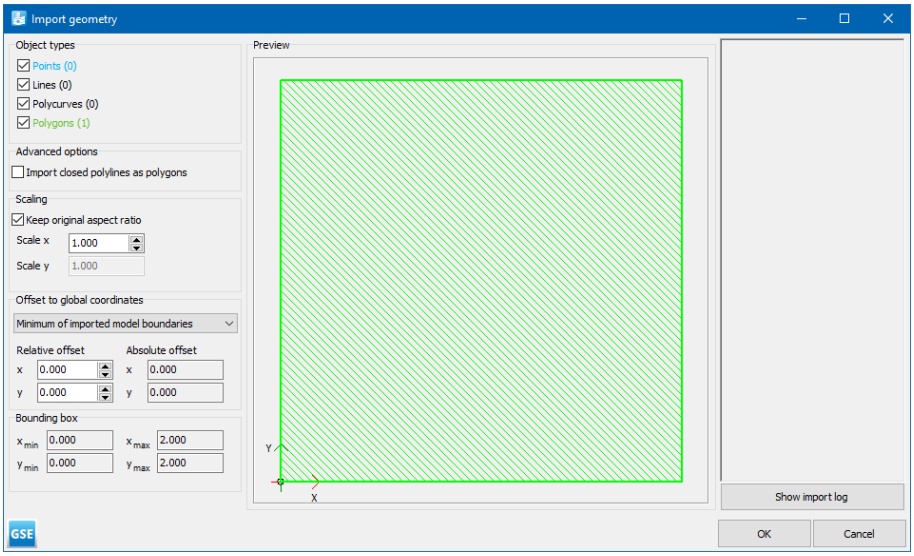


*Figure 38: The **Import geometry** window*

Note: PLAXIS 2D supports the importing of polycurves as polygons taking into account the following conditions:

- The Polycurve need to be not self-intersecting.
- The Polycurve need to be closed.
- The area described by the polycurve needs to be higher than 1E-7.

5.3.1 Importing polycurves as polygons

As displayed in the *Advanced options* of the **Import geomery tool** (see Figure 38 (on page 77)), PLAXIS 2D supports the importing of polycurves as polygons taking into account the following conditions:

- The polycurve needs to be not self-intersecting.
- The polycurve needs to be closed.
- The area described by the polycurve needs to be higher than 1E-7.
- The importing polycurve as polycurve functionality only works in the **Structures mode**.

If a polycurve is not directly imported as polygon, the user can transform a imported PLAXIS 2D polycurve into a polygon following the next steps:

- In the drawing area, on the closed polycurve, **Right click** > **Create** > **Create Polygon**.

Note: This action can be perfomed for individual polycurve objects.

5.3.2 Importing point clouds

The user can import surfaces using the point cloud feature available in GUI. The supported file type is .txt (a plain text file). The program detects a best-fit plane as the reference plane.

The NURBS surface is generated with default generation parameters. After the import, the surface can be selected and the generation properties can be changed. Changing any property will trigger the regeneration of the surface. *Selection explorer* shows the following properties for a NURBS Surface.

x,y,z	Coordinates of the insertion point.

Axis Function	Definition of local axes of surface.
	• Automatic: The axes follow the right-hand rule. Axis1 and Axis2 always define the tangential plane of the surface. Axis3 is perpendicular to this plane. The direction of Axis1 and Axis2 within the tangential plane are defined based on these rules:
	  • If Axis3 is parallel to the *z*-axis then Axis1 points to *+x*-axis and Axis2 points to *+y*-axis.
	  • If Axis3 is not parallel to the *z*-axis then Axis1 is perpendicular to the plane defined by Axis3 and the *z*-axis. Axis2 is defined as the vector product of Axis3 and Axis1.
	• Manual: Axis1 is defined by the user. Axis3 is always the normal. Axis2 is calculated as the vector product of Axis3 and Axis1.

Min/Max degree	The minimum/maximum degree of the polynomials. This concerns the polynomials that join the points (setting both min and max values to 1 only allows for linear functions, and if set between 1 and 3, it allows for lines, parabolas and cubic functions).
Number of grid points rows	The number of grid rows where the point cloud is sampled.
Number of grid points columns	The number of grid columns where the point cloud is sampled. The rows and columns refer to the underlying grid used to define the NURBS surface. The more rows and columns you use, the more control you have over the surface and the more flexible your surface will be, but the heavier and thus less workable your surface will become.
Continuity	• C0: Only geometric continuity • G1: For each point on the curve, the tangent vectors "on the right" and "on the left" are co-linear with the same orientation • G2: For each point on the curve, the normalized normal vectors "on the right" and "on the left" are equal • C2: Continuity of the second derivative. The "C2" curve is also "G2" but, in addition, the main directions of the curves also match. • C3: Continuity of the third derivative.
Status	The status reported by the generator tool.

The `importpointcloud` command can be used from the command line too with more parameters to fine tune how the import is done. For more information on how to use this command, see Command Reference.

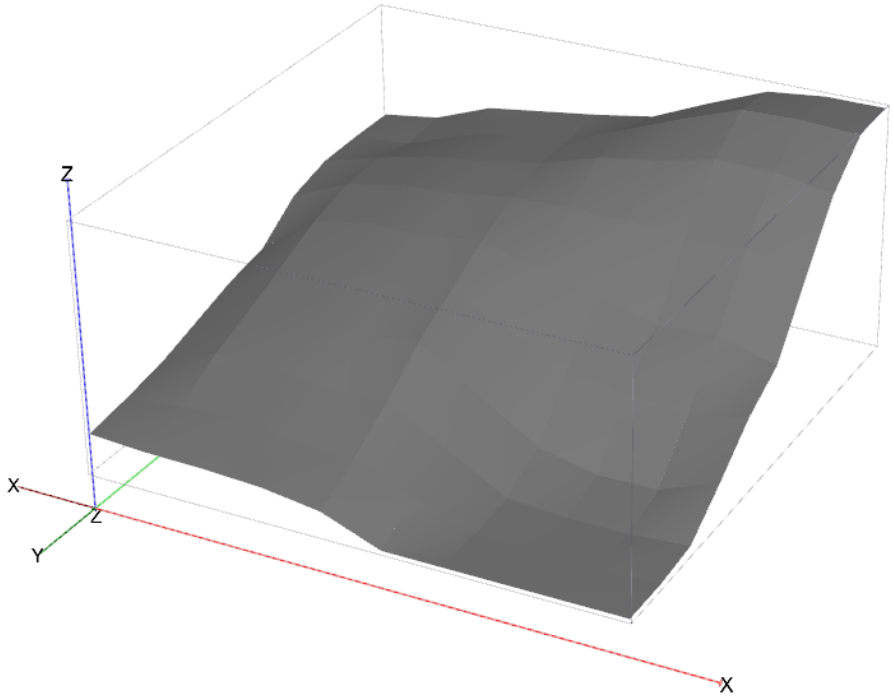


Figure 39: Imported point cloud file

5.3.3 Importing IFC files

The user can also import Industry Foundation Classes (IFC) data using files with .ifc extension. It is possible to import data using *Import structures...* tool in the side toolbar of the **Structures mode** .

To change filtering properties and import, for example, only walls included in the IFC file, the user can use command `importifc`.

The supported types are:

- ifcroof
- ifcslab
- ifcfooting
- ifcpile
- ifcbeam
- ifcplate
- ifcwall
- ifccolumn

Examples of using `importifc` command:

`importifc "structures" "C: directory \3.ifc" "ifcslab"`	`#imports only slab structures into a project`
`importifc "structures" "C: directory \3.ifc" "ifcwall"`	`#imports only wall data into a project`
`importifc "structures" "C: directory \3.ifc" "ifcroof"`	`#imports only roof data into a project`

Note: When using incorrect types or trying to import the structures that do not exist in a file, the program shows warning messages.

For more information on the `importifc` command please visit the *Input command Reference* for PLAXIS 3D (see Help menu (on page 34)).

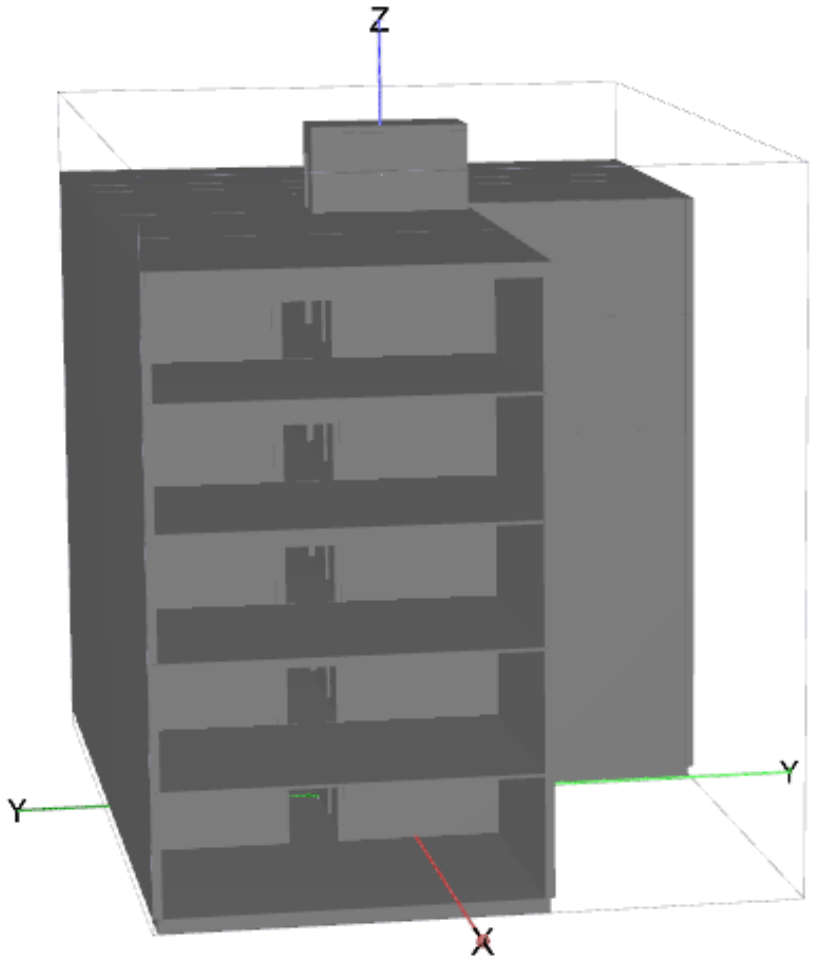


Figure 40: Imported IFC data

5.4 Loads

Loads are features that can be assigned to geometric entities. Loads can be assigned to existing geometric entities by right-clicking the entity and selecting the corresponding option in the appearing menu.

A geometric entity can be created and a load can be assigned to it simultaneously using the options available in the menu displayed as the *Create load* button is selected in the side toolbar. This provides a faster definition of loads. Instead of creating the geometric entity and then assigning a load to it, the process is completed in one step. The process is similar to the creation of a geometric entity with the difference that the load is assigned to it as well.

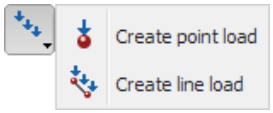


Figure 41: The Create load menu

Although the input values of loads are specified in the **Structures** mode, the activation, deactivation or change of loads may be considered in the framework of *Staged construction* (Changing loads (on page 361)).

On a part of the geometry where both fixities and loads are applied and active, the fixities have priority over the loads during calculations. Hence, it is not useful to apply loads on a fixed geometric entity. However, it is possible to apply a load in the free direction if only one direction is fixed.

The *Load* subtree in the *Object explorers* consists of two parts where the static and the dynamic components of the load can be defined respectively.

1. **Static load**

   The distribution and the components of the static load are assigned in the first part of the *Load* subtree. Note that the *Distribution* option is not available for *Point* loads.

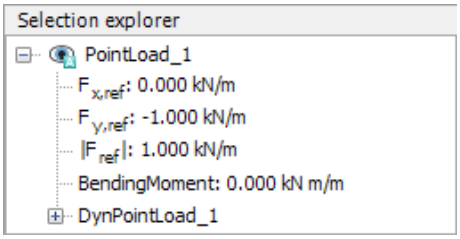


Figure 42: Static point load in the Object explorers

2. **Dynamic load** [ULT]

   The distribution and the components of the dynamic load is assigned in the second part of the *Load* subtree. Note that the *Distribution* option is not available for *Point* loads. Besides the *Distribution* and the components of the load, *Multipliers* can be defined for each of the load components separately. One of the available multipliers defined in *Attributes library* (Definition of multipliers (on page 87)) can be selected from the drop-down menu. The view of the dynamic load in the *Object explorers* is given in figure below.

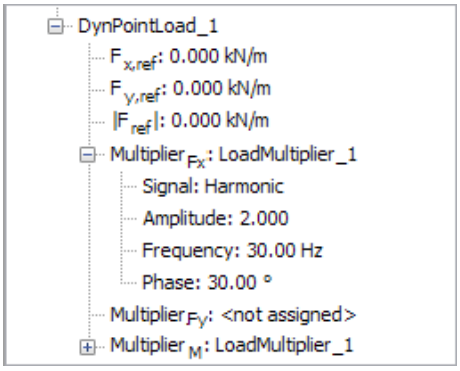


Figure 43: Dynamic point load in the Object explorers

Note: Note that a bending moment value can be specified only for *Point loads. y-direction*

5.4.1 *Point load*

A point load can be created using the *Create point load* button. The input values of a point load are given in the unit of force per unit of width in the out-of-plane direction. The default value of point loads is one unit in negative . The x- and y-components of a point load as well as an optional bending moment can be defined or modified in the *Object explorers*. If a different magnitude value is assigned to the absolute load, the program calculates the individual components accordingly, assuming the initial load direction.

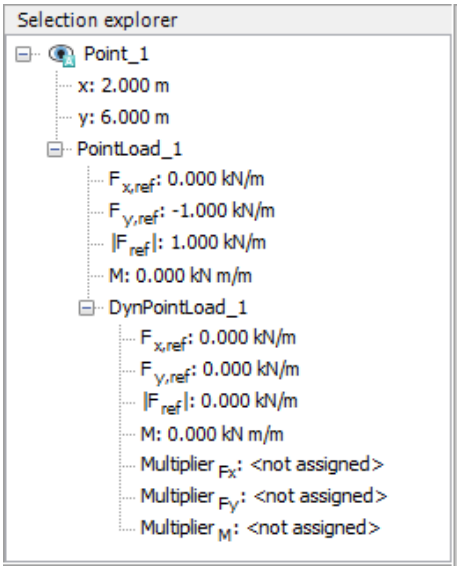


Figure 44: Point load in the Selection explorer

Note:

- Bending moments can only be applied to plates or embedded beam rows.

- Bending moments applied to nodes that do not belong to the plates or embedded beam rows are not included in the calculation.

5.4.2 *Distributed load*

A distributed load can be created using the *Create distributed load* button. The input values of a distributed load is given in the unit of force per length per length of out-of-plane. The default value of a distributed load is one unit in negative y-direction. If a different magnitude value is assigned to the absolute load, the program calculates the individual components accordingly, assuming the initial load direction.

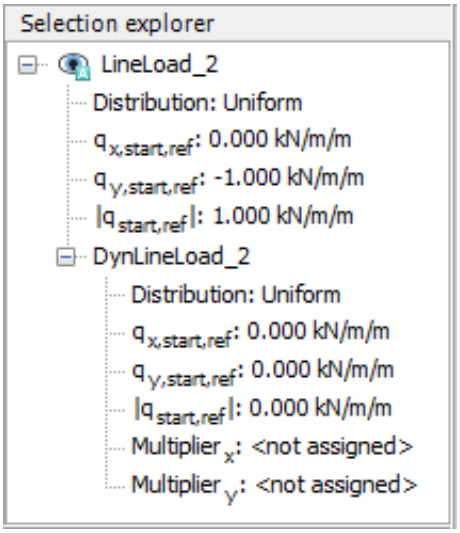


Figure 45: LineDistributed load in the Selection explorer

The distribution options for a line load are:

Uniform	To create a uniform line load.
Linear	A linearly varying distributed load is created by defining the components and the magnitude of the force at the start and at the end points of the line representing it.
Perpendicular	To create a uniformly distributed load perpendicular to the line.
Perpendicular, vertical increment	To create a load perpendicular to the line varying in the direction of gravity (-y) by defining the components and the magnitude of the load at the reference point and its increment in the negative y-direction.

5.5 Prescribed Displacements

Prescribed displacements are special conditions that can be imposed to the model to control the displacements at certain locations. Prescribed displacements can be assigned to existing geometric entities by right clicking the entity and selecting the corresponding option in the appearing menu. To assign prescribed displacements to existing geometry entities:

- Select the button corresponding to the prescribed displacement from the menu appearing when the *Create prescribed displacement* button is selected.
- Drag the tool (keep the left mouse button down) and drop it on the geometric entity.

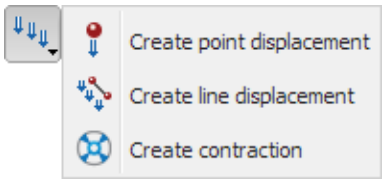


Figure 46: Create prescribed displacement menu

Although the input values of prescribed displacements are specified in the **Structures** mode, the activation, deactivation or change of values may be considered in the framework of **Staged construction** (Changing geometry configuration (on page 359)).

On a geometric entity where both prescribed displacements and loads are applied and active, the prescribed displacement has priority over the load during calculations. If both prescribed and fixed displacement are assigned to a line, the fixed displacement will be taken into consideration. However, when not all the displacement directions are fixed, it is possible to apply a load in the free directions.

Note: *Prescribed displacements* should be interpreted as specified total displacements at the end of the phase instead of additional (incremental) phase displacements. If no prescribed displacement increment should be applied in a phase, the value assigned to the prescribed displacement should be equal to the one in the previous phase. Assigning 0 to the value will implicitly apply an incremental prescribed displacement with the same value but in the opposite direction resulting in a zero displacement at the end of the phase.

The *Prescribed displacement* subtree in the *Object explorers* consists of two parts where the static and the dynamic components of the prescribed displacement can be defined respectively.

5.5.1 *Static prescribed displacement*

The distribution and the components of the static prescribed displacement are assigned in the first part of the *Prescribed displacements* subtree. Note that the *Distribution* option is not available for the *Point prescribed displacements*.

5.5.2 *Dynamic prescribed displacement* [ULT]

Dynamics Multipliers can be defined for each of the static prescribed displacement components, to apply the prescribed displacement as a function of (dynamics) time in a dynamics calculation. One of the available dynamics multipliers defined in *Attributes library* (Definition of multipliers (on page 87)) can be selected from

the drop-down menu. The view of the dynamic prescribed displacement in the *Object explorers* is given in figure below.

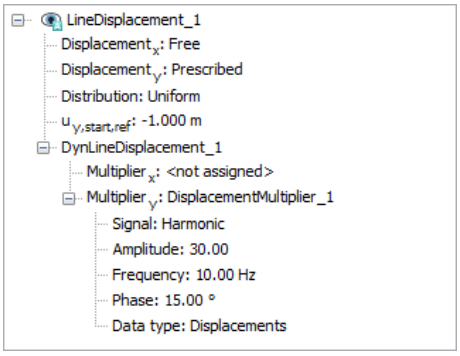


Figure 47: Dynamic prescribed displacement in the Object explorers

5.5.3 *Point prescribed displacement*

A point prescribed displacement can be created using the *Create point prescribed displacement* button. The input of point prescribed displacements in the geometry model is similar to the creation of points (Create point (on page 71)).

The default point prescribed displacement is free in x direction and it has a prescribed value of -1 unit in the *y*-direction. The options for the components of point prescribed displacements are *Free*, *Fixed* and *Prescribed*. These options can be selected and the displacement value can be defined if the *Prescribed* option is selected in the *Object explorers*. A rotational fixity can be assigned to a point by checking the corresponding box in the *Object explorers*.

5.5.4 *Distributed prescribed displacement*

In order to define prescribed displacements, the *Create distributed prescribed displacement* option should be selected from the menu appearing as the *Create prescribed displacement* button is clicked.

The default distributed prescribed displacement is free in x direction and it has a prescribed value of -1 in *y*-direction. The options for the components of line prescribed displacements are *Free*, *Fixed* and *Prescribed*. These options can be selected in the *Model explorer*. Either a uniform or a linearly varying prescribed displacement can be assigned to a line *representing the distributed prescribed displacement*. When the *Linear* option is selected in the *Distribution* drop-down menu, the displacement value at the start of the line (the first point) and at the end of the line (the second point) can be defined.

5.5.5 *Line contraction*

Line contraction can be used to simulate an area loss around a tunnel lining or to apply a contraction to a line in general. Contraction is specified as a strain without any unit associated.

The entered value of contraction is not always entirely applied, depending on the stiffness of the surrounding clusters and objects. The specified C_{ref} is applied gradually during the calculation phase (via $\Sigma Mstage$) as a shrinkage strain. In principle, the shrinkage strain does not cause any stress changes by itself. However, due to any surrounding restraints, the applied strain may not be equal to the input contraction, resulting in stress variation.

In order to define a contraction option should be selected from the menu appearing when the *Create prescribed displacement* icon is clicked in the side toolbar. Line contraction may also be defined directly in the *Tunnel designer*.

Line contraction should be interpreted as specified total contraction at the end of the phase instead of additional (incremental) phase contraction. If no contraction increment should be applied in a phase, the value assigned to the contraction should be equal to the one in the previous phase. If a contraction was already applied in the previous phase, assigning 0 to the value will implicitly apply a contraction with the same value but in the opposite direction resulting in a zero contraction at the end of the current phase.

Note:

- For tunnels, the applied contraction is equal to the tunnel contraction to simulate small volume loss (≈3% volume contraction).
- When contraction is used to simulate the area loss around a tunnel, the value of contraction to be applied is equal to the area loss as a percentage of the tunnel area.
- In geogrid elements, a negative contraction can be used to simulate thermal expansion.

5.6 Dynamic loading [ULT]

In PLAXIS 2D, a dynamic load is specified by means of an input value and a multiplier. The actual dynamic load value at each time step equals to the input value times the multiplier. Dynamics multipliers can be assigned to the dynamics component of a load (Loads (on page 81)) or to a prescribed displacement (Prescribed Displacements (on page 83)). The dynamics multipliers to be applied in the model can be defined in the *Dynamics multipliers* subtree under the *Attributes library* in the *Model explorer*.

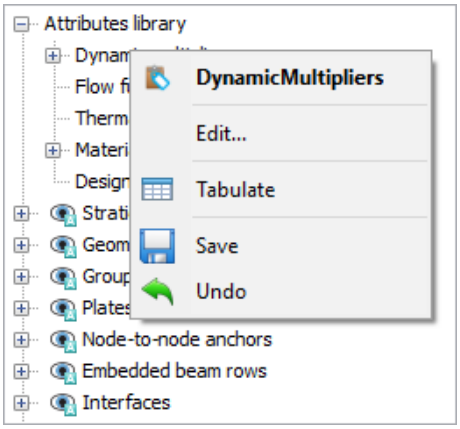


Figure 48: Dynamics multipliers branch in the Model explorer

5.6.1 *Definition of multipliers*

The multipliers can be defined in the **Multipliers** window that pops up when the *Dynamics multipliers* subtree under the *Attributes library* in the *Model explorer* is right-clicked and the *Edit...* option is selected in the appearing menu. The window consists of two tabsheets, i.e **Displacement multipliers** and **Load multipliers** where the multipliers applicable to the prescribed displacements and to the loads can be defined respectively.

The buttons available under the multipliers tabs can be used to add new multipliers or to delete the selected multiplier in the list. When a new multiplier is added, the options to define it are displayed.

Name	The name of the multiplier can be defined.
Signal	The type of the signal can be assigned. The options available are *Harmonic* and *Table*. More information on definition of a harmonic multiplier and definition of the multipliers in a table is given in Harmonic signal (on page 87) and Signal from table (on page 88) respectively.
Type	The options *Displacements*, *Velocities* and *Accelerations* are available. Note that the *Type* drop-down menu is available only for displacement multipliers. For *Load multipliers* the type does not need to be specified.
Drift correction	It is used to correct the displacement drift. Due to the integration of the accelerations and velocities, a drift might occur in the displacements. When selected, the displacement drift is corrected by applying a low frequency motion from the beginning of the calculation and by correcting the acceleration accordingly.

5.6.2 *Harmonic signal*

In PLAXIS 2D harmonic loads are defined as:

$$F = \hat{M}\hat{F}\sin\left(\omega t + \phi_0\right) \qquad Eq.\ [2]$$

in which:

$\hat{M}$*:* Amplitude multiplier

$\hat{F}$*:* Input value of the load

ω*: 2πf* with *f* Frequency in *Hz*

ϕ_0*:* Initial phase angle in degrees

($\hat{M}\hat{F}$ is the amplitude of the dynamic load).

Note: A dynamic load can also suddenly be applied in a single time step or sub step (block load). In case of a *Harmonic load multiplier*, a block load can be modelled by setting the *Amplitude multiplier* equal to the magnitude of the block load, the *Frequency* to *0 Hz* and *Initial phase angle* to *90°* giving the relation $F = \hat{M}\hat{F}$. In case of a *Load multiplier from data file*, a block load can directly be defined.

Figure below shows the **Multipliers** window where a *Harmonic* signal is defined and plotted.

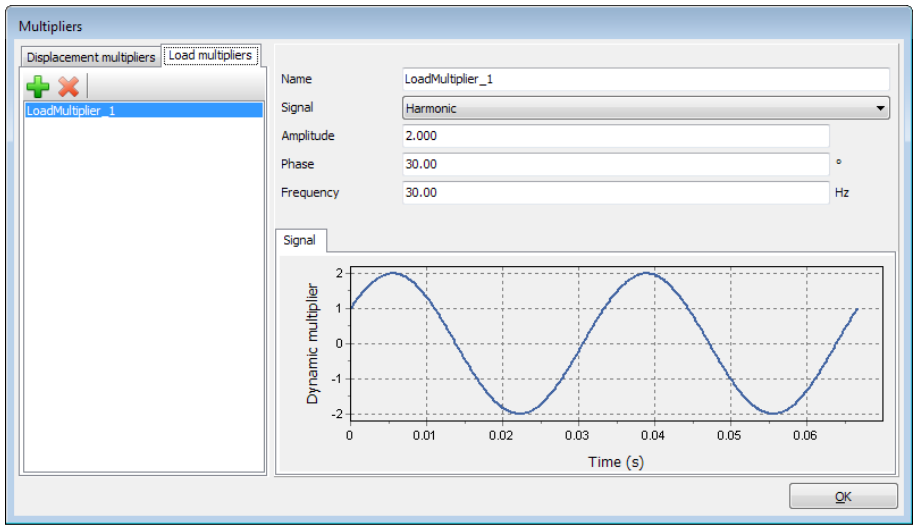


*Figure 49: Definition and display of a Harmonic signal in the **Dynamics multipliers** window*

5.6.3 *Signal from table*

Besides a harmonic signal there is also the possibility to define a signal by specifying the values in the table that appears when the corresponding option is selected in the *Signal* drop-down menu. It is possible to choose *Displacements*, *Velocities* or *Accelerations* from a drop down menu for the *Data type* of a signal. If *Accelerations* are chosen as the data type then the signal can be transformed using *Scaling type* options. For more information refer Input accelerograms (on page 91). The table consists of *Time* and *Multiplier* columns where *Time* relates to the *Dynamic time* in the project. The buttons in the toolbar can be used to modify the table.

- Click the *Add row* button in the toolbar to add a new row in the table.
- Click the *Insert* button to insert a new row before the selected row in the table.
- Click the *Delete* button to delete the selected row in the table.

The values can be defined by clicking the cell in the table and by typing the value.

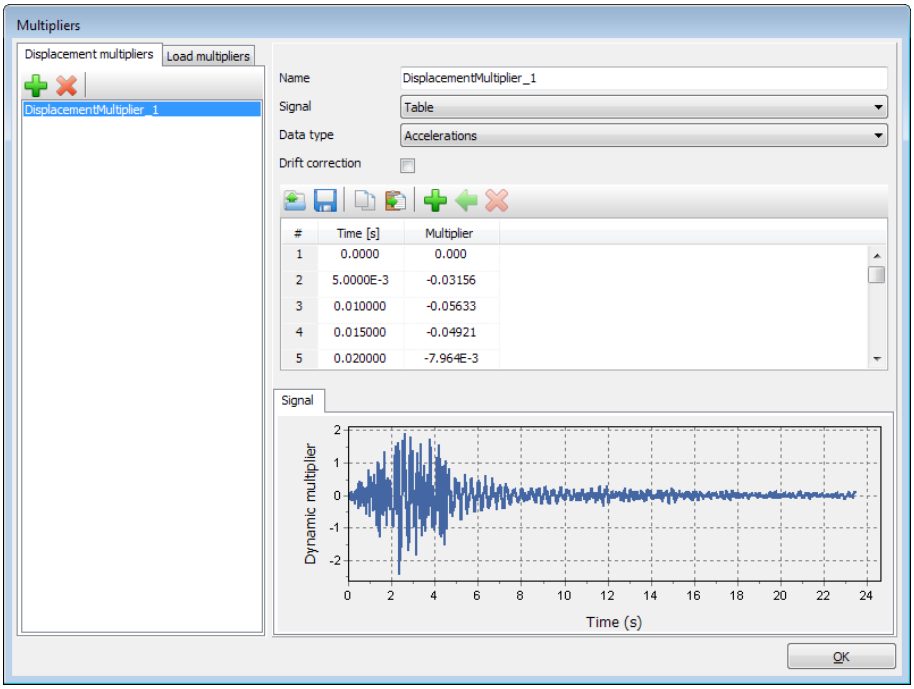


Figure 50: ***Dynamic multipliers*** *window*

Importing data for signals

Besides defining the signal in the table, there is also the possibility to read data from a file with a digitised load signal using the *Open* button in the toolbar. PLAXIS enables reading data from files in plain ASCII or in SMC format.

Note: PLAXIS assumes the data file is located in the current project directory when no directory is specified in the *Dynamic loading* window.

A signal, either defined in a table or received from a file and modified, can be saved using the *Save* button in the toolbar enabling the usage of the signal in other projects or validating the effect of the modifications in the current project.

A signal, either defined in a table or received from a file and modified, can be copied using the *Copy* button in the toolbar.

Copied data from other applications (using *Ctrl+c*) can be imported by using the *Paste* button. The **Import data** window appears as shown below. The starting row of the data to be imported can be defined in the *From row* cell. The data can be parsed as *Plain text files* or *Strong motion CD-ROM files* (SMC). The data and the plot is displayed in the **Dynamic multipliers** window after pressing *OK*.

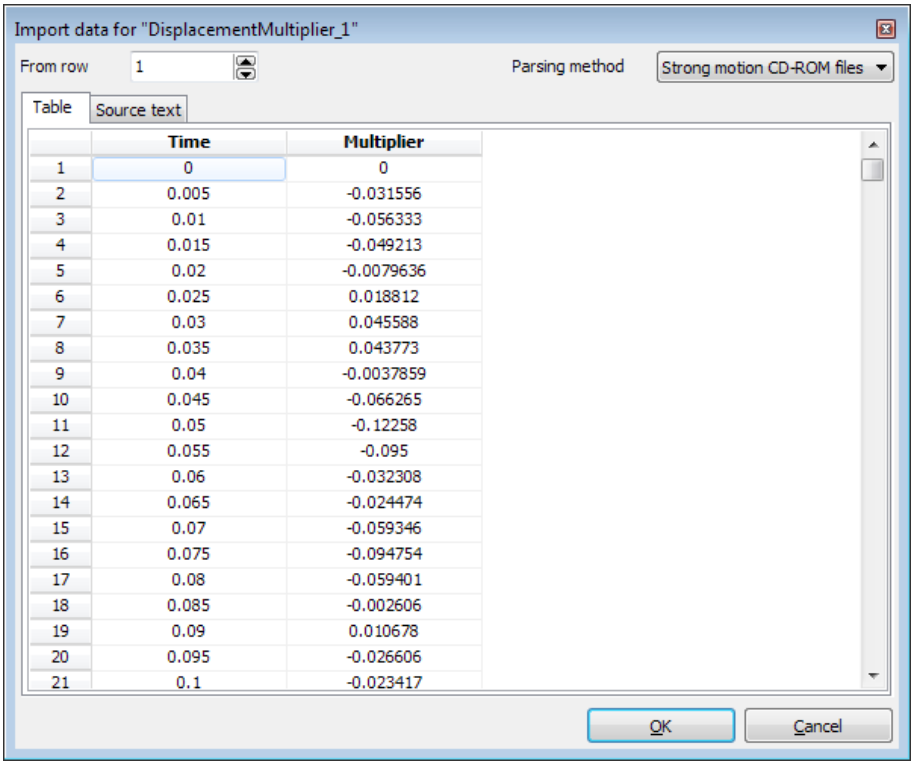


Figure 51: ***Import data*** *window for dynamic multipliers*

ASCII file

An ASCII file can be created by the user with any text editor. In every line a pair of values (*Dynamic* time and corresponding multiplier) is defined, leaving at least one space between them. The time should increase in each new line. It is not necessary to use constant time intervals.

If the time steps in the dynamics analysis are such that they do not correspond with the time series given in the file, the multipliers at a given (*Dynamic*) *time* will be linearly interpolated from the data in the file. If the *Dynamic time* in the calculation is beyond the last time value in the file a constant value, equal to the last multiplier in the file, will be used in the calculations.

SMC file

In addition, it is possible to use earthquake records in SMC-format as input for earthquake loading. The SMC (Strong Motion CD-ROM) format is currently used by the U.S. Geological Survey National Strong-motion Program to record data of earthquakes and other strong vibrations. This format uses ASCII character codes and provides text headers, integer headers, real headers, and comments followed by either digitised time-series coordinates or response values. The header information is designed to provide the user with information about the earthquake and the recording instrument.

Most of the SMC files contain accelerations, but they may also contain velocity or displacement series and response spectra. It is strongly recommended to use corrected earthquake data records, i.e. time series, that are corrected for final drift and non-zero final velocities. SMC files should be used in combination with prescribed boundary displacements at the bottom of a geometry model.

The strong motion data are collected by the U.S. Geological Survey and are available from the National Geophysical Data Center (NGDC) of the National Oceanic and Atmospheric Administration. Information on NGDC products is available on the World-wide Web at http://www.ngdc.noaa.gov/hazard.

Note: The time value in dynamic multipliers always refers to the global dynamic time (in seconds) in the whole list of calculation phases, rather than to the time interval of an individual phase. This means that in series of continuing dynamics calculation phases each phase will only use its consecutive part of the dynamic multiplier.

5.6.4 *Input accelerograms*

If a signal has the data type as *Accelerations*, it is possible to scale the input accelerogram using the following options from the drop-down menu:

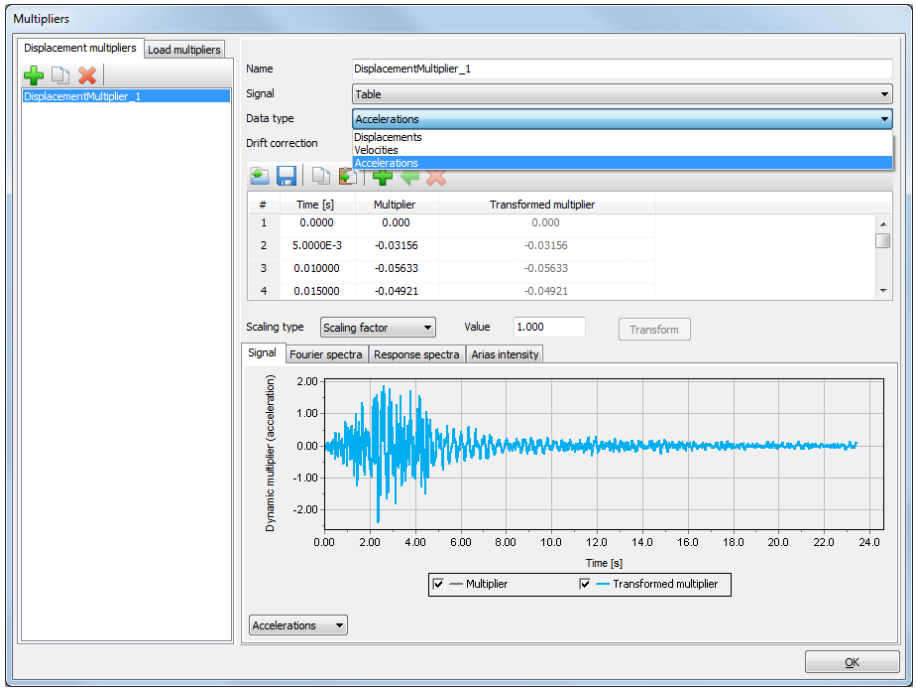


Figure 52: ***Import data*** *window for dynamic multipliers*

Note: Note that if the model is created using the *Command line*, the user must make sure to transform the multiplier, otherwise the transformed multiplier values will not be taken into account during calculation.

Scaling factor	The input multipliers can be transformed by specifying a value. Click to get the newly transformed multipliers.
Maximum multiplier	A desired maximum value of the multiplier can be specified. Click to get the newly transformed multipliers. The scaling factor is calculated as the ratio between the desired maximum value and the maximum value of the multiplier (in absolute value). All the multipliers are then multiplied by this resulting scaling factor.

A user should click on the *Transform** button as shown below, after modifying a value for either the *Scaling factor* or *Maximum multiplier*. PLAXIS then computes the transformed multiplier, and a new column is generated *Transformed multiplier*.

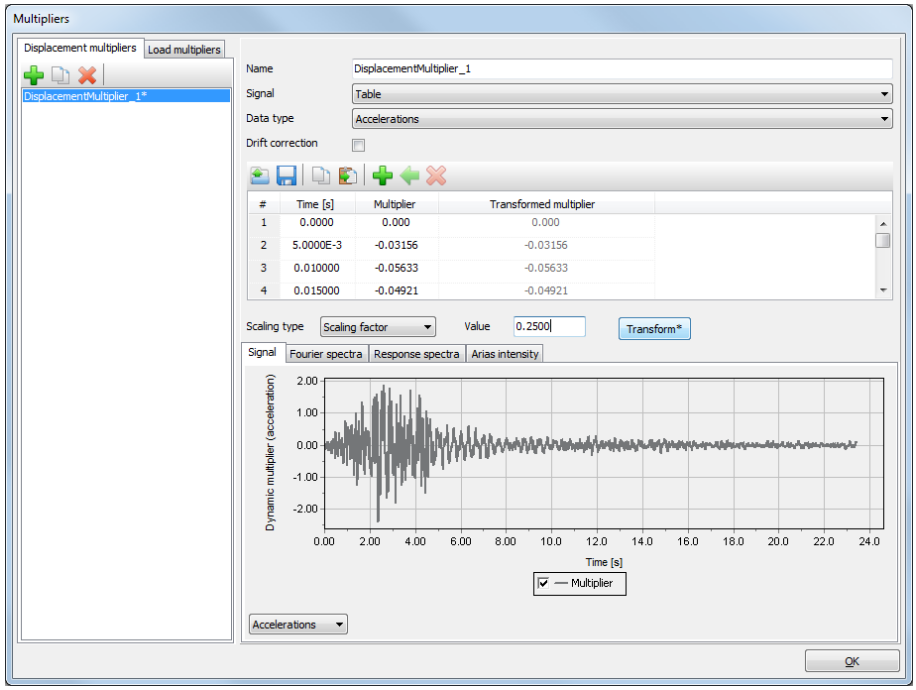


Figure 53: Input accelerograms tranformation

In the table, the column *Transformed multiplier* contains the transformed multiplier values. If the table is modified, the multiplier should be transformed again, as the transformation is not updated automatically. The asterisk * after *Transform* indicates that the table was modified after the last transformation action.

Note: Note that the *Transformed* multiplier only contains the values obtained by applying the scaling factor and not the drift correction.

When the input data type is Accelerations, the signal can be viewed as Accelerations, Velocities or Displacements, by selecting the corresponding option in the drop-down list below the graph. The input accelerogram signal can be visualised in the lower panel in different representative charts: *Fourier spectra*, *Response spectra* and *Arias intensity*. The original multiplier is displayed in grey, and the transformed multiplier is displayed in blue. Both the original multiplier in grey and transformed multiplier in blue are displayed in the *Signal* tab.

Fourier spectra

In this tab, the input multiplier can be visualised as a *Fourier* amplitude spectrum (default) or a power spectrum using the drop-down menu. The spectra are created using the transformation in *Hz*, starting from the time - acceleration multiplier.

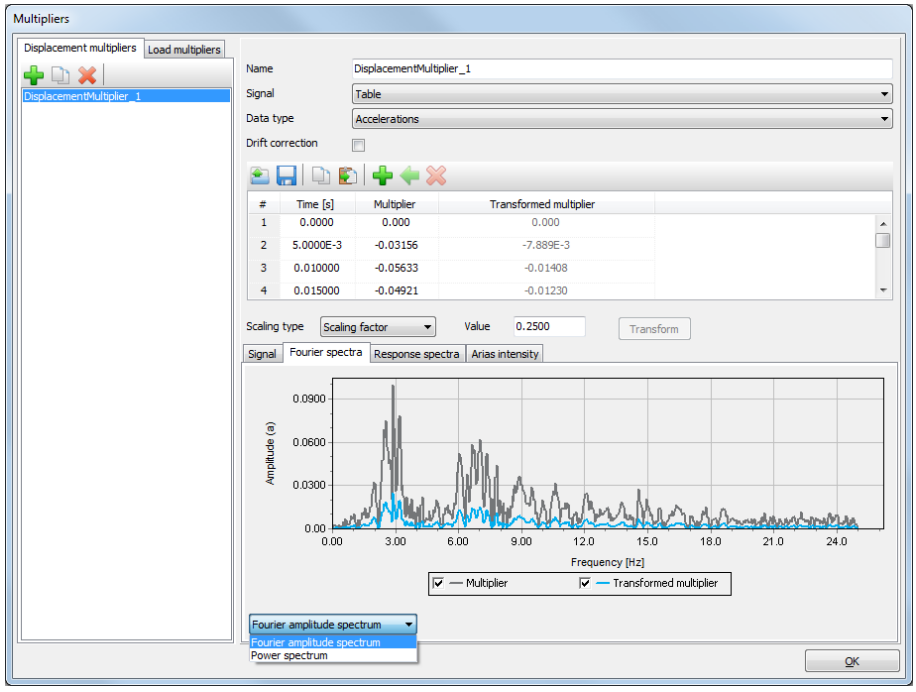


Figure 54: Fourier spectra for input accelerograms

Response spectra

In this tab, the input multiplier can be visualised as a *PSA* (default) and *Relative displacement response* spectrum.

It is possible to set these parameters while using this tab :

- *Damping ratio* is expressed in *%*, has a default value of 5.000 *%* and a range of [0, 100].
- *Max. period* is expressed in *s*, has a default value of 10.00 *s* and a range of [0.1, 1000].

The signal can be viewed with a logarithmic x-axis by clicking on the check box.

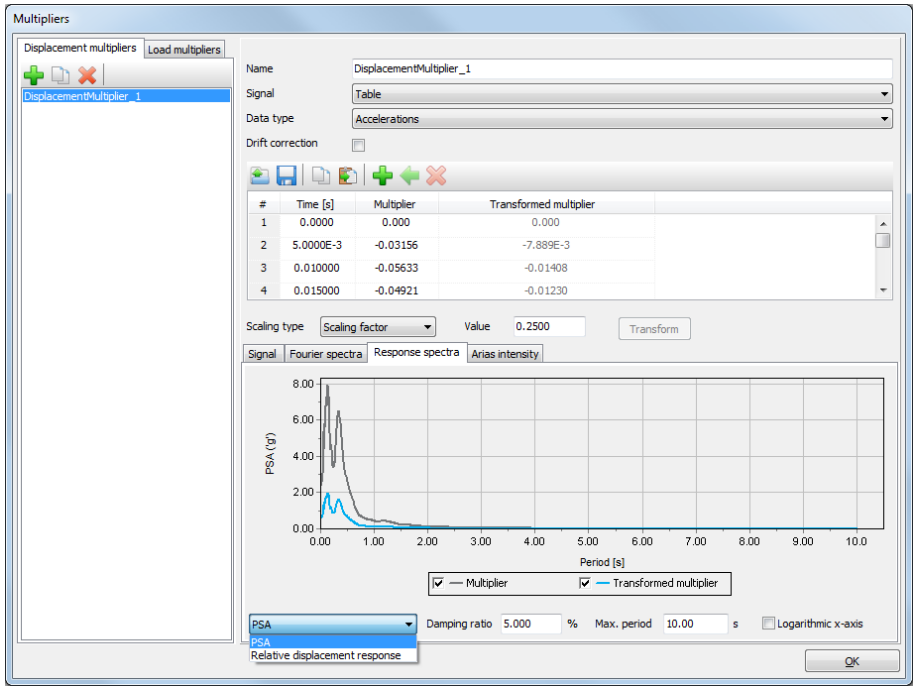


Figure 55: Response spectra for input accelerograms

Arias intensity

In this tab, the input multiplier can be visualised as *Arias intensity*. It is possible to view it in percentage by clicking on the check box *Show Arias intensity in percentage*.

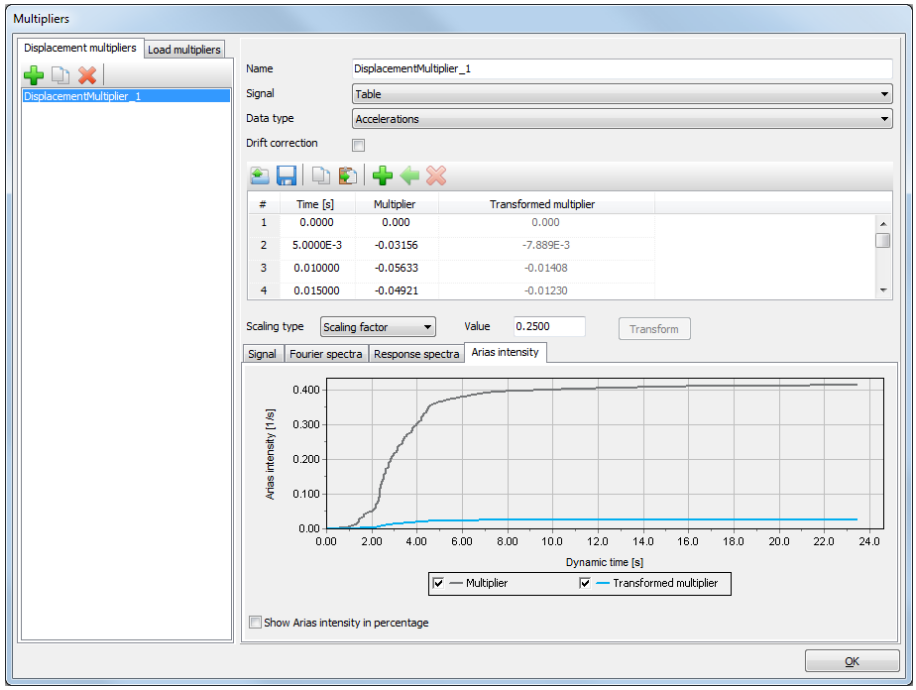


Figure 56: Arias intensity for input accelerograms

5.6.5 *Added mass*

Note:

This feature is provided as a Technology Preview.

If a project contains a large body of water, for instance a reservoir or a lake, this body of water may also be subjected to the dynamic load. The movement of the body of water due to the dynamic load may then by itself cause a hydrodynamic load on the soil surface. A good example of this is the behaviour of a dam under a seismic load, where the earthquake will cause the water in the reservoir to move. As a consequence, the water pressure of the reservoir acting on the dam will no longer be hydrostatic, but hydrodynamic as shown in Figure 57 (on page 95).

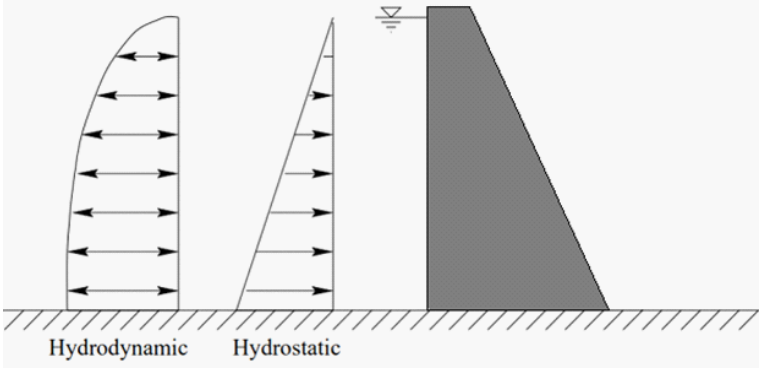


Figure 57: Hydrodynamic and hydrostatic pressures acting on the dam during a seismic event

There are two possibilities in PLAXIS to model a body of water that can be subjected to seismic loads:

1. Model the body of water using finite elements and assign a constitutive (soil) model representing the water. In PLAXIS a special Fluid model is available to do so.
2. Model the effect of the body of water using a special *Added mass* boundary condition on the boundary between soil and water. The *Added mass* boundary condition used in PLAXIS is the so-called Zangar formulation, which is an improvement of the well-known Westergaard added mass formulation.

The *Added mass* boundary condition can be found under the *Line* option in the side bar (see Figure 58 (on page 96)). By selecting the option an *Added mass* boundary can be drawn where needed. The boundary allows for the use of added mass on either side of the line where it is applied. In order to distinguish on which of the line the mass is applied one side of the line is indicated as the "negative" side and the other side as the "positive" side similar to the naming as used for interface elements.

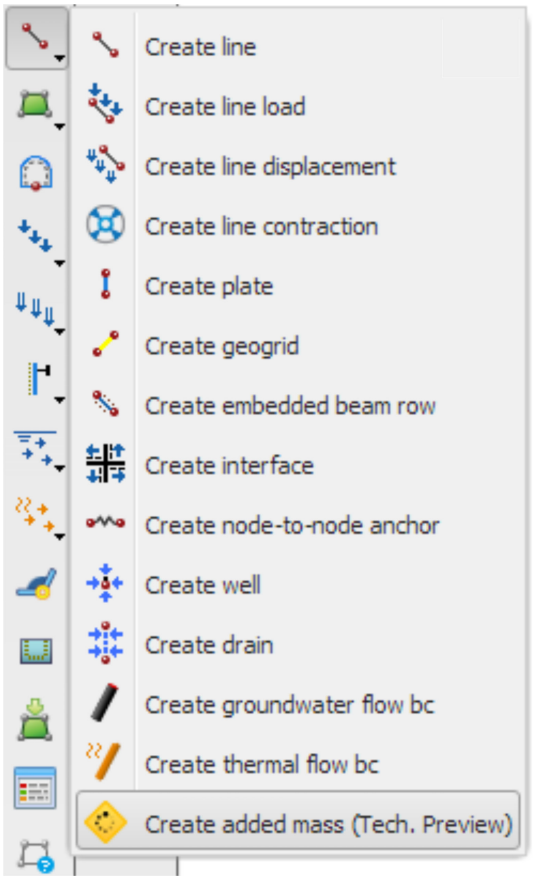


Figure 58: Create added mass feature

Both sides of the added mass conditions can be activated and deactivated independently. However, the added mass is only taken into account if no soil is present on the activated side of the boundary condition. Additionally, the bottom level of the water body must be specified. This bottom level is called the Water Base Level (WBL) and it is the y-coordinate of the lower level of the water to be taken into account for added mass. So in short, all water in between the WBL and the phreatic level is taken into account as added mass. Typically the WBL is chosen as the bottom of the reservoir or lake.

The figure below shows the simple geometry of a dam with the applied *Added mass* boundary on the left boundary. In the box on the right of the geometry the settings for the *Added mass* boundary are shown as can be seen in for instance the Selection explorer. The figure shows that the negative side, which is the reservoir side, is activated whereas the positive side, the soil side, is deactivated. As the bottom of the dam and thus the bottom of the lake is situated at y=10m the WBL specified is 10m.

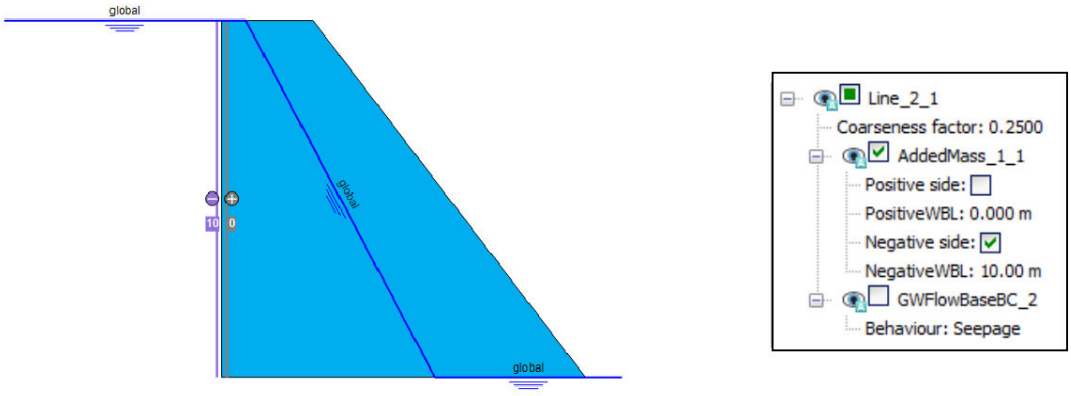


Figure 59: Application of added mass boundary - geometry and settings

5.7 Structural elements

The creation of a structure is similar to the creation of the corresponding geometric entities. To create structures:

- Select the button corresponding to the structure from the menu appearing when the *Create structure* button is selected.
- Define the structure in the same way the corresponding geometric entity is defined.

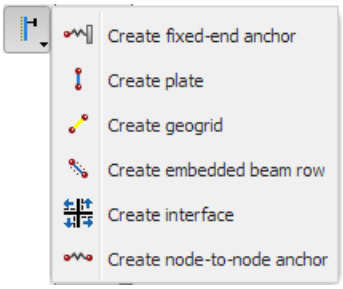


Figure 60: Create structure menu

Note that the assignment of structures to existing geometric entities is possible by right-clicking the geometric entity either in the drawing area or in the *Object explorers*, pointing on *Create* and selecting the corresponding option in the appearing menu. Assignment of materials to structures is described in Assigning material data sets to geometry components (on page 289).

Structures can be activated, de-activated or the assigned material data sets can be modified in calculation phases.

5.7.1 *Fixed-end anchors*

A fixed-end anchor is a point element that is attached to a structure at one side and fixed 'to the world' at the other side. Fixed-end anchors can be used to simulate piles in a simplified way, i.e. without taking into account pile-soil interaction. Alternatively, fixed-end anchors can be used to simulate anchors or props to support retaining walls. A fixed-end anchor is visualised as a rotated T (⊣). The length of the plotted T is arbitrary and does not have any particular physical meaning.

The creation of a fixed-end anchor is similar to the creation of a geometry point (Create point (on page 71)). By default, a fixed-end anchor is pointing in the positive x-direction, i.e. the angle in the xy-plane is zero. The user may enter the fixed-end anchor direction by specifying the individual *x*- and *y*-components. The length defined by the individual components is defined as the equivalent length of the fixed-end anchor.

The *Equivalent length* parameter of a fixed-end anchor is defined as the distance between the anchor connection point and the fictitious point in the longitudinal direction of the anchor where the displacement is assumed to be zero. For example: In the case of a symmetric excavation that is supported by a propped retaining wall, a fixed-end anchor may be used to stimulate the prop. In this case, the equivalent length of the fixed-end anchor is half the excavation width. The direction and equivalent length of a fixed-end anchor may be changed in the *Object explorers*. Changing the equivalent length will change the individual direction components while retaining the anchor direction.

Only one fixed-end anchor can be created at a single geometry point, which means that this point can only be supported in one direction. In the case that more support directions in a single point are required, the following options exist:

- If the point is to be supported in both horizontal and vertical direction, a fixed-end anchor (spring support) can be used in one direction whereas a point fixity can be applied at the same point to fix the other direction (while the anchor direction is set free). Disadvantage of this solution is that the anchor features (spring support, pre-stressing, resulting force, etc.) are not available for a point fixity.
- If the point is to be supported in two arbitrary directions, a node-to-node anchor may be used instead of a second fixed-end anchor, provided that the node-to-node anchor is at the other node connected to a model boundary that is fixed in the anchor direction. This may not necessarily lead to the right equivalent anchor length. Hence, the anchor stiffness *EA* needs to be adjusted such that the ratio *EA/L* gives the required value.

  Example (see Figure 61 (on page 99)): The basement corner is to be supported by spring supports in x- and y-direction, considering fixed-end anchors, both with an equivalent length of *2*m. Since only one fixed-end anchor can be used for a single geometry point, the vertical support is modelled by means of a node-to-node anchor that is connected to the bottom of the model (which is fully fixed). However, the length of the node-to-node anchor is *4*m in this case, which is twice as large as the desired equivalent length of *2*m. That means that a new anchor material data set needs to be created for the node-to-node anchor in which the anchor stiffness EA is increased by a factor *2* in order to have the same *EA/L* as for the fixed-end anchor.

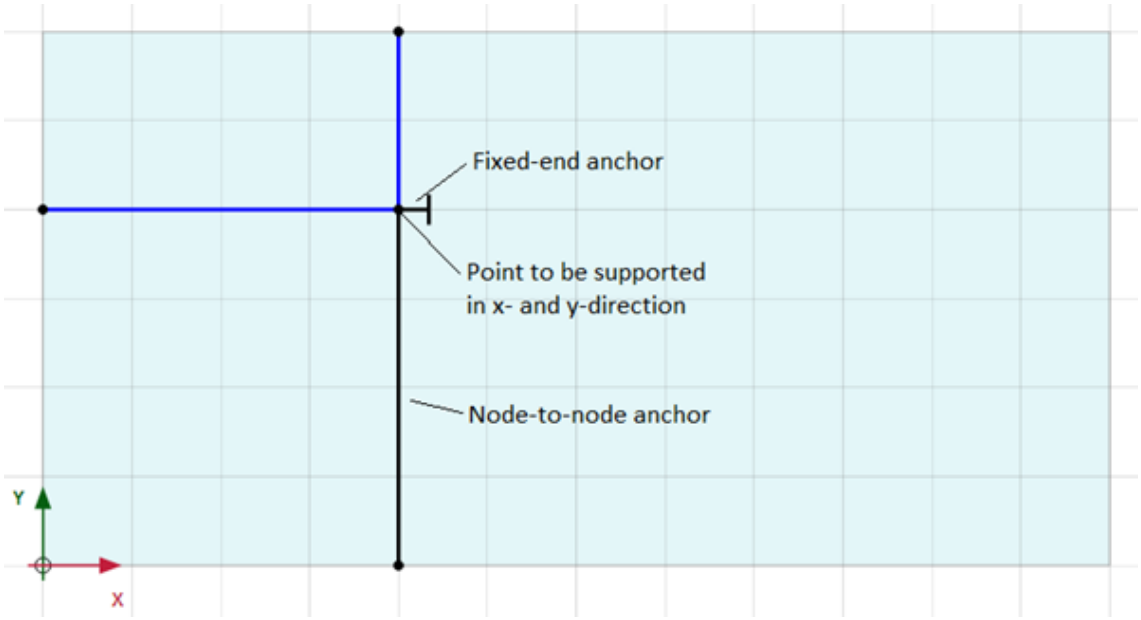


Figure 61: Representation of connection of fixed-end anchor and node-to-node anchor

Anchor material properties

The material properties of fixed-end anchors are contained in *Anchors* material data sets (Material data sets for anchors (on page 285)) and can be conveniently assigned using drag-and-drop, the *Object explorers* or the right hand mouse button menu in the drawing area (Assigning material data sets to geometry components (on page 289)).

5.7.2 *Node-to-node anchors*

A node-to-node anchor is a two-node elastic spring element with a constant spring stiffness (axial stiffness). This element can be subjected to tensile forces (for anchors) as well as compressive forces (for struts). Both the tensile force and the compressive force can be limited to allow for the simulation of anchor or strut failure. Optionally, a residual maximum force can be specified.

The creation of a node-to-node anchor is similar to the creation of a geometry line (Create line (on page 71)).

Since a node-to-node anchor is a line-based entity rather than a point-based entity, different node-to-node anchors can be connected to the same geometry point, provided that they do not fully coincide.

Anchor material properties

The material properties of node-to-node anchors are contained in *Anchors* material data sets and can be conveniently assigned using drag-and-drop, using the *Object explorers* or the right hand mouse button menu in the drawing area (Assigning material data sets to geometry components (on page 289)).

Note: A node-to-node anchor cannot be assigned to a line to which other features are assigned.

5.7.3 Embedded Beam Rows

Since the stress state and deformation pattern around piles, rock bolts and ground anchors is fully three-dimensional, it is impossible to model such structures realistically in a 2D model. Hence, the 2D embedded beam element is only a simplified approach to deal with a row of piles, rock bolts or ground anchors in the out-of-plane direction in a 2D plane strain model.

The idea behind the 2D embedded beam is that the pile (or rock bolt or grout body of a ground anchor), represented by a Mindlin beam element, is not 'in' the 2D mesh, but superimposed 'on' the mesh, while the underlying element mesh of the soil or rock is continuous (Figure 62 (on page 100), after Sluis [2012] (on page 532)). A special out-of-plane interface connects the beam with the underlying soil/rock elements. The beam is supposed to represent the movement of an out-of-plane row of individual piles or rock bolts or grout bodies, whereas displacements of the soil or rock elements are supposed to represent the 'average' soil/rock displacement in the out-of-plane direction. The interface stiffness should be chosen such that it accounts for the difference between the (average) soil/rock displacement and the pile/rock bolt/grout body displacement while transferring loads from the structures onto the soil/rock and vice versa. This requires at least the out-of-plane spacing of the structures to be taken into account in relation to their diameter. In the case of rock bolts, the interface should also account for the 'smearing' of the grout material connecting the rock bolt to the surrounding rock.

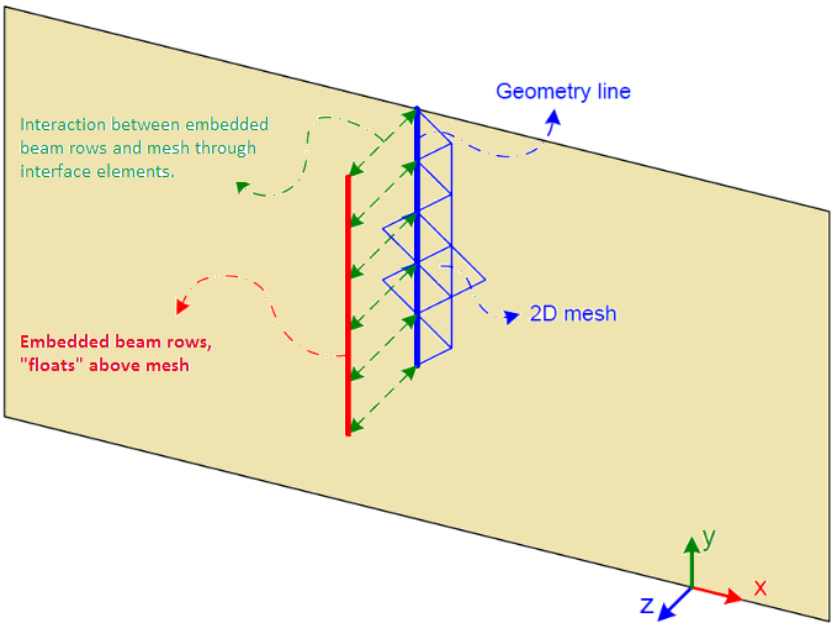


Figure 62: Schematic representation of embedded beam rows after Sluis (2012)

An embedded beam row can be utilised to model a row of long slender structural members used to transmit loads to the surrounding soil or rock. The *Embedded beam rows* feature is available only for *Plane strain* models. The information required for an embedded beam row consists of the properties of a single pile or rock bolt or grout body and their spacing in the out-of-plane direction.

Note:

The embedded beams are suitable for pile types that cause a limited disturbance of the surrounding soil during installation. This may include some types of bored piles (e.g. displacement screw piles), but not most of the technologies for replacement piles or displacement piles. The installation effect has a significant influence on the soil stress ratio (k_0) and the pile skin resistance (T_{skin}). In such a case, it is necessary to properly evaluate these factors in order to simulate the pile installation effect.

The creation of embedded beam rows in the geometry model is similar to the creation of a line (Create line (on page 71)). Note that one row in out-of-plane direction is indicated by one line in the model. When creating embedded beam rows, the corresponding geometry line is created simultaneously. Hence it is not necessary to create first a line at the position of an embedded beam row. It is also not necessary to create interface elements around the embedded beam rows, since special interface elements are automatically created with the embedded beam elements.

The behaviour options available for *Embedded beam row* are:

Pile	The *Embedded beam row* is set to *Pile*.
Rock bolt	The *Embedded beam row* is set to *Rock bolt*.
Grout body	The *Embedded beam row* is set to *Grout body*.

Depending on the *Embedded beam row* behaviour, the connection point may be indicated as:

Top/Bottom	In the case the *Embedded beam row* is a *Pile*, the connection point from the drop-down menu can be specified in the *Selection explorer* as *Top* or *Bottom*. *Top* refers to the point on the embedded beam row that has the highest y-coordinate in the model. In the rare case of an exactly horizontal embedded beam row defined as a *Pile*, *Top* refers to the point on the embedded beam that has the lowest x-coordinate in the model.
First/Second	In the case the *Embedded beam row* is a *Rock bolt*, the connection point from the drop- down menu can be specified in the *Selection explorer* as *First* or *Second*. *First* refers to the first point of the corresponding geometry line, as listed in the *Selection explorer* when the embedded beam row is selected (typically the point on a tunnel contour), whereas second refers to the second point of the corresponding geometry line(typically the point somewhere in the surrounding rock).

For the first and second types of embedded beam rows, the connection options available are:

Rigid	The displacement and rotation at the beam top are both coupled with the displacement and rotation of the element in which the beam connection point is located, provided that this element has rotational degrees of freedom. This option only applies if the beam connection point coincides with plates elements.
Hinged	The displacement at the connection point of the beam is directly coupled with the displacement of the element in which the connection point is located, which means that they undergo exactly the same displacement, but not necessarily in the same rotation.

Free	The connection point of the beam is not directly coupled with the soil element in which the beam top is located, but the interaction through the interface elements is still present. When a node-to-node anchor ends at the same geometry point, this structural node is automatically connected to the embedded beam row connection point, but it will be separate from the corresponding soil node.

Note: When embedded beams embedded beam rows are located in a polygon cluster with linear elastic material behaviour, the specified values of the shaft resistance and spacing are ignored. The reason for this is that the linear elastic material is not considered to be soil, but part of the structure. The connection between the beam and the structure is supposed to be rigid to avoid, for example, punching of piles through a concrete deck.

- To ensure a rigid behaviour between an embedded beam row and a polygon cluster structure, the beam could be elongated into the structure for a certain length. To avoid mesh-dependent results for the internal forces of the embedded beam row (linked to the elongation length), the user could insert an elastic plate at the polygon cluster side where the connection with the embedded beam row takes place. The beam can be now linked to the plate without elongating it in the cluster structure.
- When an embedded beam rows and a structure are both active and share the same geometry point, the node created at the connection point of the embedded beam row is by default rigidly connected to the structure node. However, if the structure is not active, the embedded beam node is by default connected (hinged) to the soil node at that location.
- Note that when an interface is available, the embedded beam row is NOT connected to the interface but to the structure or soil node at that location.
- It is possible to 'chain' embedded beam rows, which also enables connection to other structural elements (such as a node-to-node anchor) in the connection point.

When an anchor is present in the model at the position of the embedded beam connection point, the user can select the option *Grout body* for the behaviour of embedded beam rows. At the position of an embedded beam row connection point, the anchor point will be automatically connected to the embedded beam row connection point and not to the corresponding node in the soil element. The connection type is intrinsically set to *Free* (i.e. free from the surrounding soil polygon).

The material properties of embedded beam rows are contained in material data sets (General properties (on page 275)). Embedded beam rows can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

Embedded beam elements

Embedded beams in the 2D finite element model are composed of line elements with three degrees of freedom per node: two translational degrees of freedom (u_x, u_y) and one rotational degrees of freedom (rotation in the *x*-*y* plane: φ_z). These elements are similar as plate elements. When 6-node soil elements are employed then each embedded beam element is defined by three nodes whereas 5-node embedded beam elements are used together with the 15-node soil elements (Figure 65 (on page 105)). The elements are based on Mindlin's beam theory (Bathe, 1982 (on page 530)). This theory allows for deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied. Figure 65 (on page 105) shows a single 3-node and 5-node embedded beam element with an indication of the nodes and stress points.

Bending moments and axial forces are evaluated from the stresses at the stress points. A 3-node embedded beam element contains two pairs of Gaussian stress points whereas a 5-node embedded beam element contains four pairs of stress points. Within each pair, stress points are located at a distance $\frac{1}{6}\sqrt{3}d_{eq}$ above and below the embedded beam centre-line.

The interaction between the pile or rock bolt and the surrounding soil or rock may involve a skin resistance as well as a foot resistance. Therefore, special out-of-plane interface elements (line-to-line interface along the shaft and point-to-point interface at the base) are used to connect the beam elements to the surrounding soil or rock elements. The interface elements involve springs in the longitudinal and transverse direction and a slider in the longitudinal direction.

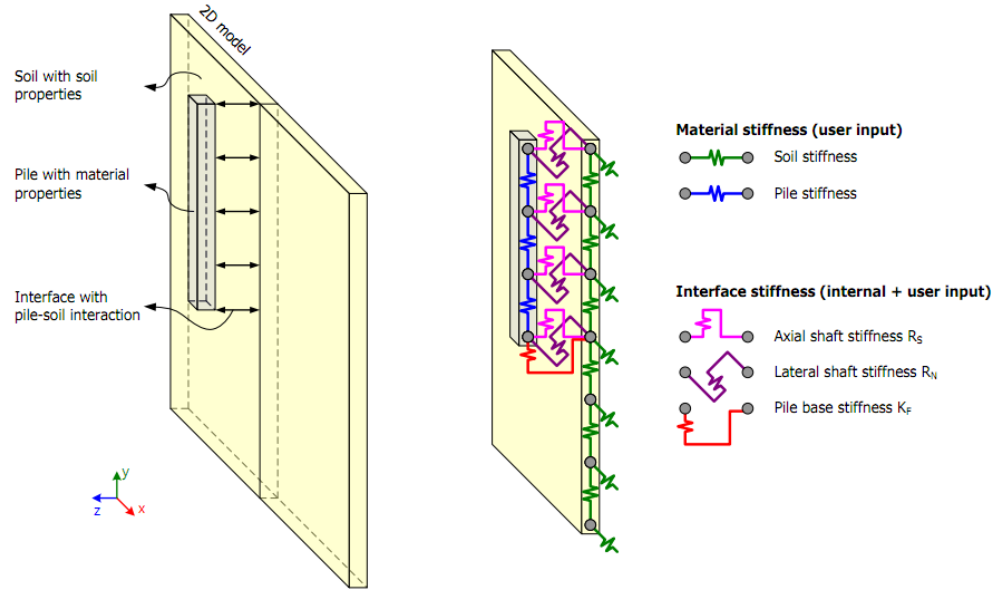


Figure 63: Embedded beam interaction with soil, after Sluis (2012)

Structural forces are evaluated at the embedded beam element integration points and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program. Details about the embedded beam element formulations are given in the Material Models Manual

Embedded beam row properties

The material properties of embedded beam rows are contained in *Embedded beam rows* material data sets (General properties (on page 275)). In the material dataset geometric features of the pile/rock bolt, material properties, spacing of the piles/rock bolts in the out-of-plane direction, skin friction and bearing capacity of the pile/rock bolt as well as the interface stiffness factor can be defined.

5.7.4 *Plates*

Plates which are actually beam elements, are structural objects used to model slender structures in the ground with a significant bending stiffness and an axial stiffness. Plates can be used to simulate the influence of walls, plates, shells or linings extending in z-direction. Examples of geotechnical structures involving plates are shown in Figure 64 (on page 104).

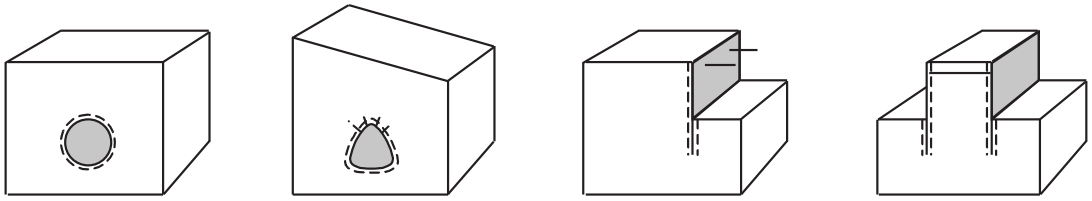


Figure 64: Application in which plates, anchors and interfaces are used

Plates can be selected by clicking on the corresponding button in the side toolbar. The creation of plates in the geometry model is similar to the creation of geometry lines (Create line (on page 71)). When creating plates, the corresponding geometry lines are created simultaneously. Hence, it is not necessary to create first a geometry line at the position of a plate.

The material properties of plates are contained in material data sets (Modelling undrained behaviour (on page 255)). The most important parameters are the bending stiffness *EI* and the axial stiffness *EA*.

From these two parameters an equivalent plate thickness deq is calculated from the equation:

$$d_{eq} = \sqrt{\frac{12EI}{EA}}$$

Plates can be activated or de-activated in calculation phases using Staged construction as Loading input.

Note: The connection between two connecting plates can be specified using the Create connection feature (Connections (on page 114))

Plate elements

Plates in the 2D finite element model are composed of beam elements (line elements) with three degrees of freedom per node: two translational degrees of freedom (u_x , u_y) and one rotational degrees of freedom (rotation in the *x-y* plane: φz). When 6-node soil elements are employed then each beam element is defined by three nodes whereas 5-node beam elements are used together with the 15-node soil elements (Figure 65 (on page 105)). The beam elements are based on Mindlin's beam theory (Bathe, 1982 (on page 530)). This theory allows for beam deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied. Beam elements can become plastic if a prescribed maximum bending moment or maximum axial force is reached.

Bending moments and axial forces are evaluated from the stresses at the stress points and extrapolated to the element nodes. A 3-node plate element contains two pairs of Gaussian stress points whereas a 5-node plate element contains four pairs of stress points. Within each pair, stress points are located at a distance $\frac{1}{6}\sqrt{3}d_{eq}$ above and below the plate centre-line.

Figure 65 (on page 105) shows a single 3-node and 5-node plate element with an indication of the nodes and stress points.

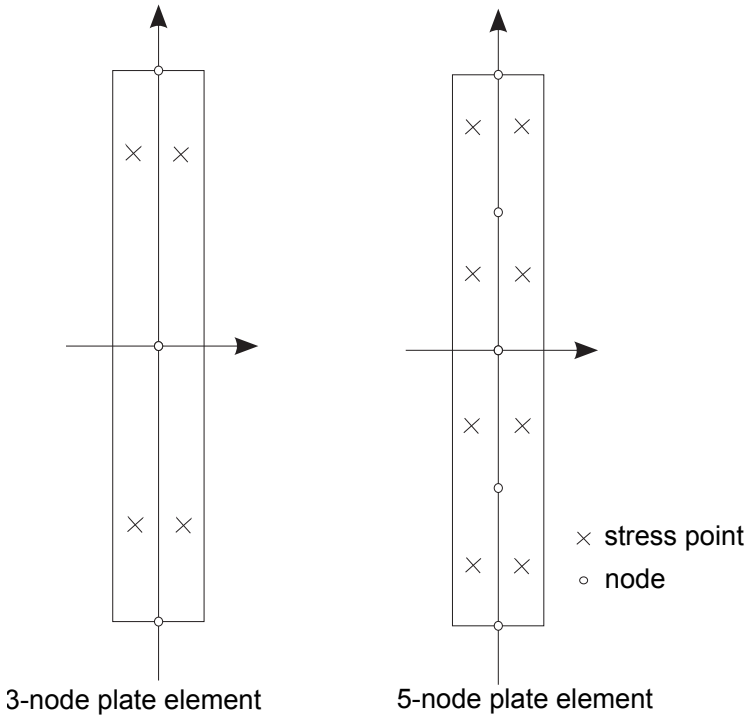


Figure 65: Position of nodes and stress points in plate elements

It is important to note that a change in the ratio *EI/EA* will change the equivalent thickness d_{eq} and thus the distance separating the stress points. If this is done when existing forces are present in the plate element, it would change the distribution of bending moments, which is unacceptable. For this reason, if material properties of a plate are changed during an analysis (for example in the framework of Staged Construction) it should be noted that the ratio *EI / EA* must remain unchanged.

Plate properties

The material properties of plates are contained in *Plates* material data sets (General properties (on page 265)) and can be conveniently assigned using drag-and-drop, the *Object explorers* or the right hand mouse button menu in the drawing area (Assigning material data sets to geometry components (on page 289)).

Structural forces are evaluated at the plate element integration points (Scientific Manual) and extrapolated to the element nodes. These forces can be viewed graphically and tabulated in the Output program. Details about the material properties are given in the Material Models Manual - - *Structural behaviour - 2D Plates*.

5.7.5 *Geogrids*

Geogrids are slender structures with an axial stiffness but with no bending stiffness. Geogrids can only sustain tensile forces and no compression. These objects are generally used to model soil reinforcements. Examples of geotechnical structures involving geotextiles are presented in Figure 66 (on page 106).

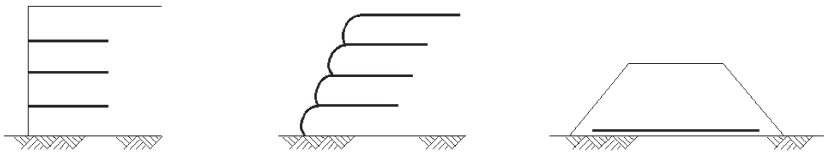


Figure 66: Applications in which geogrids are used

In a geometry model geogrids without assigned material properties appear as 'light yellow lines', whereas geogrids with assigned properties appear in their material colour. When creating geogrids, corresponding geometry lines are created simultaneously. The only material property of a geogrid is an elastic axial stiffness *EA*, which can be specified in the material data base (General properties (on page 270)). Geogrids can be erased by selecting them in the geometry and pressing the *Delete* key. Geogrids can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

Geogrid elements

Geogrids are composed of geogrid elements (line elements) with two translational degrees of freedom in each node (u_x, u_y). When 15-node soil elements are employed then each geogrid element is defined by five nodes whereas 3-node geogrid elements are used in combination with 6-node soil elements. Axial forces are evaluated at the Newton-Cotes stress points. These stress points coincide with the nodes. The locations of the nodes and stress points in geogrid elements are indicated in Figure 67 (on page 106).

Figure 67: Position of nodes and stress points in geogrid elements

Geogrid elements

The basic material parameter is the axial stiffness *EA*. In addition the tension force can be limited to allow for the simulation of tension failure. Axial forces can be viewed graphically and tabulated in the Output program. Details about the material properties are given in General properties (on page 270) and Reference Manual.

When using geogrid elements in PLAXIS 2D for modelling of true geogrids, note that most such applications require the use of updated mesh calculations to take membrane effects into account. Most applications also require the use of anisotropic properties to avoid unrealistic shear deformations.

5.7.6 *Interfaces*

Interfaces are joint elements to be added to plates or geogrids to allow for a proper modelling of soil-structure interaction. Interfaces may be used to simulate, for example, the thin zone of intensely shearing material at the contact between a plate and the surrounding soil. Interfaces can be created next to plate or geogrid elements or between two soil polygons.

An interface can be created using the *Create interface* option available in the menu displayed when the *Create structure* button in the side toolbar is clicked. The creation of an interface is similar to the creation of a line (Create line (on page 71)). In this case a line with an interface assigned to it will be created. When the geometric

entity (line) is already available in the model it is advised to assign an interface to it without recreating the geometry in order to prevent the model from being unnecessarily large and unwieldy. Assignment of features to existing geometric entities is described in Advanced geometric modelling options (on page 73).

The interface appears as a line at the side of the geometry line the interaction with the soil takes place. The side at which the interface will appear is also indicated by the arrow in the cursor pointing in the direction of drawing. To place an interface at the other side, it should be drawn in the opposite direction. Note that interfaces can be placed at both sides of a geometry line. This enables a full interaction between structural objects (walls, plates, geogrids, etc.) and the surrounding soil. To be able to distinguish between the two possible interfaces along a geometry line, the interfaces are indicated by a plus-sign (+) or a minus sign (-).

Note: The sign of an interface is only used to enable distinguishing interfaces at either side of a line, but it does not affect its behaviour.

Interfaces can be activated or de-activated in calculation phases using *Staged construction* as *Loading input*.

[ULT] Note that interfaces should be created at the boundaries of the model if *Free-field* or *Compliant base* boundary conditions are to be used in a *Dynamic* calculation. To create interfaces at the boundaries of the model click the corresponding button in the side toolbar. The interfaces are automatically created.

Interface properties

The interfaces created in the model are listed in the *Interfaces* subtree in the *Object explorers*. Properties such as *Material mode*, permeability condition and *Virtual thickness factor*, available for each interface in the *Object explorers* can be assigned to an interface in the model (Figure 68 (on page 107)).

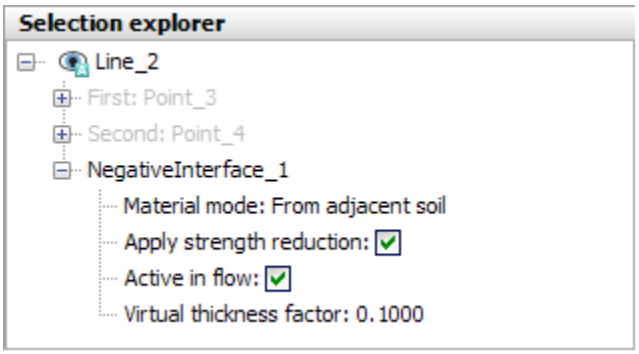


Figure 68: Interfaces in the Selection Explorer

Material mode

The material properties of an interface can be assigned according to the mode specified in the selected interface subtree in the Object explorers. As an example, consider a typical application of interfaces such as the interaction between a diaphragm wall and the soil, which is intermediate between smooth and fully rough. The options available are:

From adjacent soil	The roughness of the interaction is modelled by choosing a suitable value for the strength reduction factor (R_{inter}) specified in the **Interface** tabsheet of the material set assigned to the surrounding material. Optionally, a residual strength ($R_{inter,residual}$) can be specified if the interface strength is reached. This factor relates the interface strength (wall friction and adhesion) to the soil strength (friction angle and cohesion). Note that by default the *From adjacent soil* option is selected for the *Material mode* parameter. For detailed information on the material properties of interfaces see Interfaces tabsheet (on page 242).
Custom	A material dataset can be assigned directly to an interface when the *Custom* option is selected. Note that the strength reduction factor of the selected material is by default set to one.

Permeability

By default structural elements such as *Plates* and *Geogrids* are fully permeable. A certain permeability can be introduced by assigning an interface geometric entity to the soil besides the structural element.

In principle the permeability types of interfaces (*Cross permeability* and *Drainage conductivity* explained in Groundwater flow through interfaces (on page 247)) are defined in the corresponding material data set (See Interfaces tabsheet (on page 242)> Groundwater). However, notice that by selecting the interface geometric entity, in the **Selection Explorer** a checkbox is available to make the interface either *Active in flow* or *Inactive in flow* (see Figure 69 (on page 108)). This has consequences for the actual interface permeability being used in a selected calculation phase (Table 4 (on page 109)).

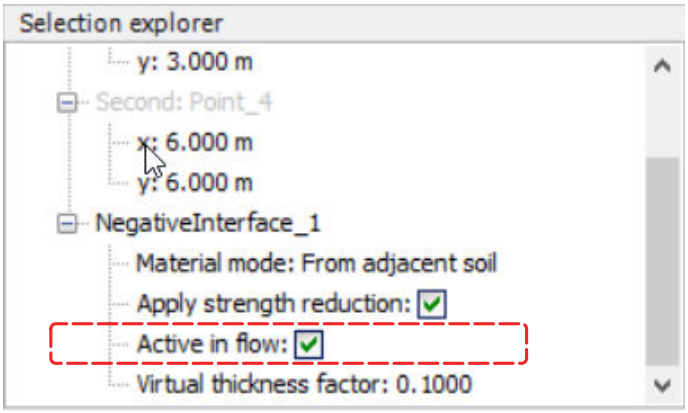


Figure 69: Active in flow option for permeability in phases

Note: For more information about permeability on interfaces please visit Groundwater flow through interfaces (on page 247) or Bentley Communities.

Table 4: Resulting settings for permeabilities in *Interfaces*

Setting in material data set			Setting in the Selection Explorer		Remarks
Permeability type	**Hydraulic conductivity direction**	**Drainage type**	**Active in flow**	**Inactive in flow**	
Cross permeability	Through the interface	Impermeable	Infinite cross resistance (or zero permeability)	No input value - Infinite cross permeability (or zero resistance)	No output of flow q_n available.
		Semi-permeable - Input value: Hydraulic resistance	Depending on hydraulic resistance assigned	Infinite cross permeability (or zero resistance)	Output of flow q_n available when active.
		Fully permeable	Infinite cross permeability (or zero resistance)	Infinite cross permeability (or zero resistance)	No output of flow q_n available.
Drainage conductivity	Longitudinal to the interface (acting as a drain)	Permeable - Input value: Drainage conductivity	Depending on drainage conductivity assigned	Zero parallel flow	Output of discharge Q_1 available when active.

Note: In *Flow only* calculations no structural element needs to be assigned to the geometric entity as deformations will not be taken into account. Thus, it is sufficient to use only interfaces to block the flow. Note that the end points of an interface are always permeable.

Virtual interface thickness

Each interface has assigned to it a 'virtual thickness' which is an imaginary dimension used to define the material properties of the interface. The higher the virtual thickness is, the more elastic deformations are generated. In general, interface elements are supposed to generate very little elastic deformations and therefore the virtual thickness should be small. On the other hand, if the virtual thickness is too small, numerical ill-conditioning may occur. The virtual thickness is calculated as the *Virtual thickness factor* times the global element size. The global element size is determined by the global coarseness setting for the mesh generation (Global settings (on page 292)). The default value of the *Virtual thickness factor* is 0.1. This value can be changed in the *Object explorers*. In general, be careful when changing the default factor. However, if interface elements are subjected to very large normal stresses, it may be required to reduce the *Virtual thickness factor*. Further details of the significance of the virtual thickness are given in Interfaces tabsheet (on page 242).

Interface elements

Interfaces are composed of interface elements. Figure 70 (on page 110) shows how interface elements are connected to soil elements. When using 15-node elements, the corresponding interface elements are defined by five pairs of nodes, whereas for 6-node soil elements the corresponding interface elements are defined by three pairs of nodes. In the figure, the interface elements are shown to have a finite thickness, but in the finite element formulation the coordinates of each node pair are identical, which means that the element has a zero thickness.

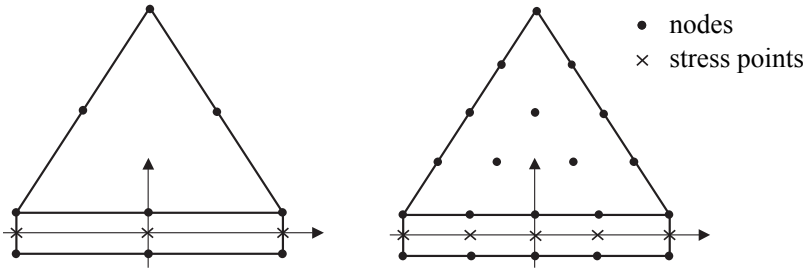


Figure 70: Distribution of nodes and stress points in interface elements and their connection to soil elements

The stiffness matrix for interface elements is obtained by means of Newton Cotes integration. The position of the Newton Cotes stress points coincides with the node pairs. Hence, five stress points are used for a 10-node interface elements whereas three stress points are used for a 6-node interface element.

At interface ends interface element node pairs are 'degenerated ' to single nodes. Also when structural elements are connected perpendicular to each other (for example, a plate is connected to another plate), interface element node pairs are locally 'degenerated' to single nodes to avoid a disconnection between both structural elements.

Interfaces around corner points

Figure 71 (on page 110) and Figure 72 (on page 110) show that problems of soil-structure interaction may involve points that require special attention. Corners in stiff structures and an abrupt change in boundary condition may lead to high peaks in the stresses and strains. Volume elements are not capable of reproducing these sharp peaks and will, as a result, produce non-physical stress oscillations. This problem can be solved by making use of interface elements as shown in Figure 72 (on page 110).

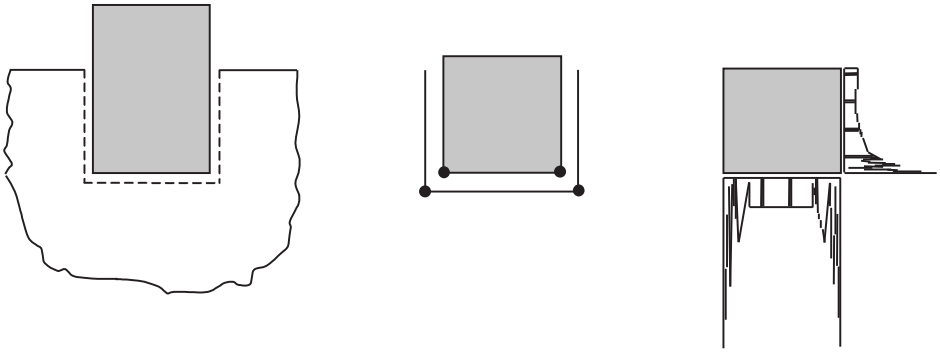


Figure 71: Inflexible corner point, causing poor quality stress results

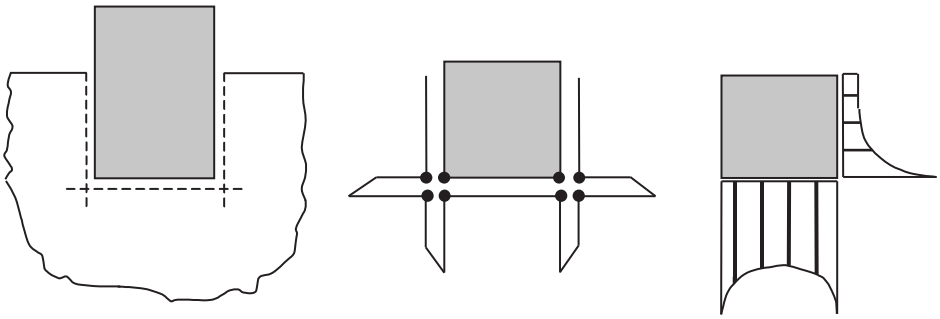


Figure 72: Flexible corner point with improved stress results

Figure 72 (on page 110) shows that the problem of stress oscillation may be prevented by specifying additional interface elements inside the soil body. These elements will enhance the flexibility of the finite element mesh and will then prevent non-physical stress results. However, these elements should not introduce an unrealistic weakness in the soil. Therefore special attention should be made to the properties of these interface elements (Interfaces tabsheet (on page 242)).

At interface ends, interface element node pairs are 'degenerated' to single nodes. When an interface exists on only one side of a structure, the interface end node is collapsed to the corresponding structure node. As a result, there exists only one node at that location. When interfaces are assigned on both sides of a structure the end nodes of the interfaces are collapsed, however the collapsed node is not the corresponding structure node. As a result, there exist two nodes at that specific location, the structure node and the node shared by both interfaces. The interactions between the structure and the surrounding soil at that node is controlled by a combined effect of both interfaces. Additional theoretical details on this special use of interface elements is provided by Goodman, Taylor & Brekke (1968) (on page 531) and Langen & Vermeer (1991) (on page 532) .

5.7.7 Discontinuities

Rock mass consists of a rock material matrix and fractures of different types and scales that represent breaks in the mechanical continuum. These fractures commonly known as *Discontinuities* describe any separation in the rock mass having zero or low tensile strength. In this way, *Discontinuities* are the collective term for the majority of joints, weak bedding planes, weak schistocity planes, weakness zones, shear zones and faults found in the rock mass (ISRM, 1978 (on page 531) and Priest, 1993 (on page 531)).

Discontinuities can be modelled using discrete elements when the discontinuities play a major role in the overall rock mass behaviour and thus the assumption of purely equivalent continuum approach cannot be assumed.

The simulation of *Discontinuities* as discrete elements can be performed using Interfaces (on page 106). However, to address more efficiently this simulation the **Create discontinuity** feature is introduced inside the **Structures mode**.

Discontinuity element

Discontinuities are composed of discontinuity elements that are embedded in the rock mass(see Figure 73 (on page 112)a)) whcih is different from soil-structure interaction that represents a disturbance at the separation between 2 different materials. Similar to interfaces, discontinuities are defined by five pairs of nodes when using 15-node elements, and three pairs of nodes when using 6-node soil elements (Figure 73 (on page 112)b)). Discontinuity elements are generally used to simulate rock joints, fractures or faults, while Interface elements are commonly used for soil structure interaction.

Note that in Figure 73 (on page 112) b) discontinuities are represented as if they have a finite thickness, however in the finite element formulation the coordinates of each node pair are identical, which means that the element has a zero thickness.

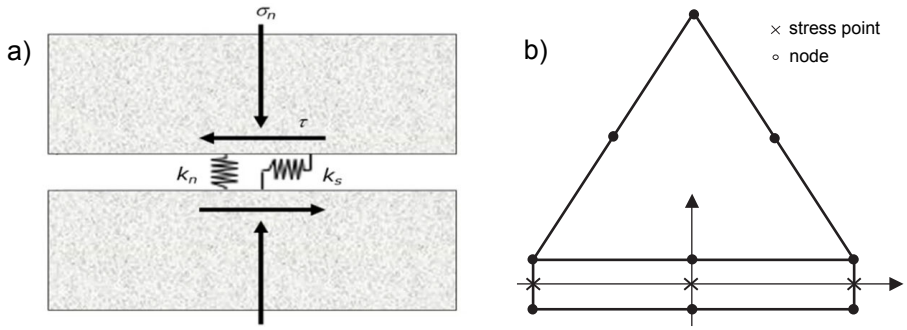


Figure 73: Discontinuity element. a) Concept of discontinuity implementation b) Discontinuity element with 6-node distribution

Discontinuity elements also have the characteristic that they work as independent features acting as separate cluster with their own material and water condition.

The use of discontinuity elements have the following restrictions:

- Other structures, loads, displacements and groundwater boundary conditions are not allowed in the discontinuity.
- Discontinuities cannot be activated on an active boundary (e.g discontinuity at the boundary of an excavation).

Creating a Discontinuity

A discontinuity can be created in the Input program inside the **Structures mode** using one of the following approaches:

- On the side toolbar go to **Create line** > **Create discontinuity**.
- On the side toolbar go to **Create Structure** > **Create discontinuity**.

Once the**Create discontinuity** option is selected, a line can be drawn on the rock mass (previously defined in the **Soil mode**).

Note:

Alternatively, in PLAXIS 2D a discontinuity can be created by drawing a line in the rock mass, then on the new element **Right-click** and from the popup menu select **Create** > **Discontinuity.**

Once drawn, the discontinuities are displayed using orange lines as shown in Figure 74 (on page 113).

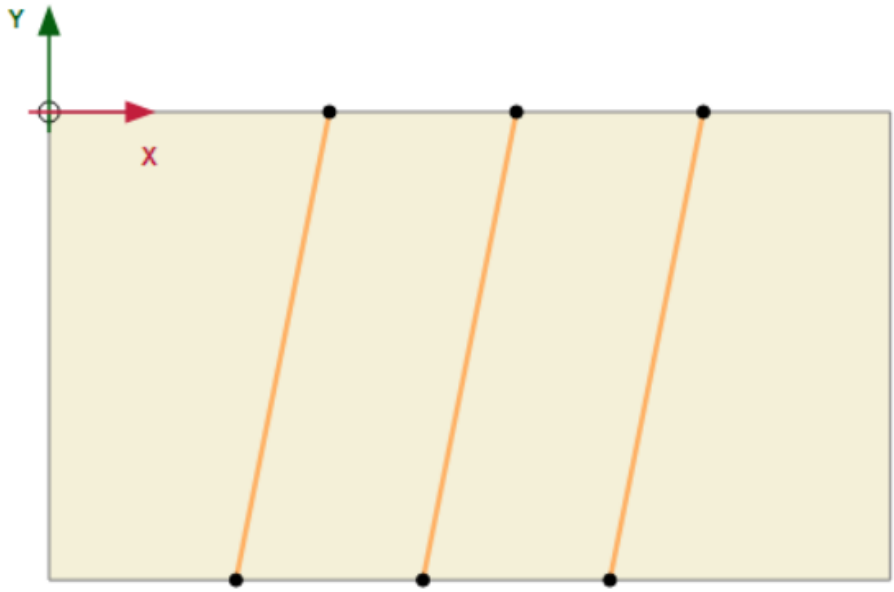


Figure 74: Discontinuity elements in a geometry

Discontinuity properties

By selecting a discontinuity in the model, its properties appear in the **Selection explorer** > **Discontinuities**. Properties such as the *Material* model, *Apply strength reduction* and *Active in flow* can be adjusted (Figure 75 (on page 113)).

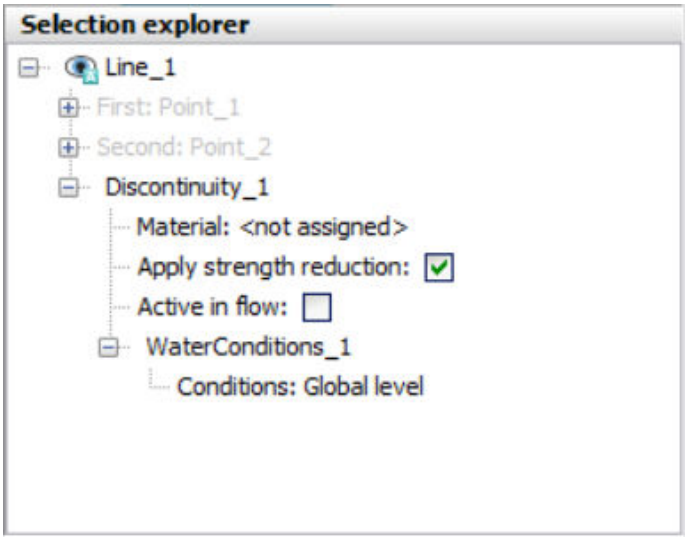


Figure 75: Discontinuity properties in the Selection explorer

1. **Material:**

   The material properties of a discontinuity involve both the strength and the deformability parameters of the rock fracture. When this option is selected a window opens to create the discontinuity material and to define its properties. Once the material has been created, it is automatically assigned to the selected discontinuity elements.

Note: For more information in the creation of Discontinuity Materials and Property definition please see Material data sets for discontinuities (on page 259)

2. **Apply strength reduction:**

   Enables the reduction of the strength parameters of the discontinuity element in a Safety Analysis.
3. **Active in flow:**

   Discontinuities always have infinite cross permeability, except when the material **Drainage type** is non-porous. When *Active in Flow* option is deselected, longitudinal flow (flow along the discontinuity) is blocked (zero permeability) whereas when this option is selected, the longitudinal flow is governed by the *Transmissivity* parameter (see Groundwater properties (on page 262)).

5.7.8 *Connections*

When two plates are connected, by default they share all the degrees of freedom (rotations and translation degrees of freedom) in the connected node, which implies that the connection is rigid. The connection between two plates can be customised by explicitly defining a *Connection*.

A connection consists of :

Reference part (or the 'parent object')	The object to which another object (a 'child object') is connected.
Custom part (or the 'child object')	The object connected to the 'parent object'.

To define a connection:

- Click the *Create connection* button in the side toolbar.
- Specify the *Custom part* by clicking on it in the drawing area.
- Specify the *Reference part* by clicking on it in the drawing area. The connection will be indicated by a yellow circle located on the *Custom part* next to the intersection point (Figure 76 (on page 115)).

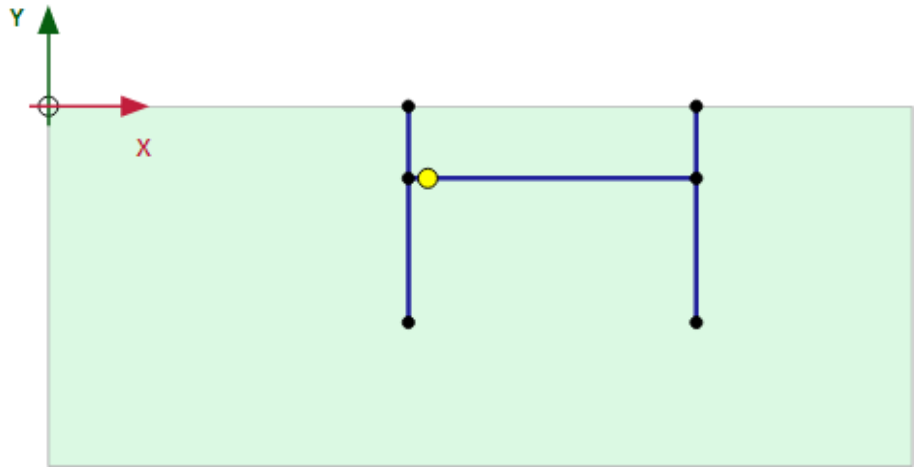


Figure 76: Representation of connections in the model

Note that the a connection can be created only if one of the end points of the custom part lies on the reference part. A connection is active when both parts are active in a calculation phase.

The properties of a connection can be specified in the *Object explorers*. The options available for the rotation are shown in (Figure 77 (on page 115)). If the *Fixed* option is selected, the connection is rigid (as if no connection was defined). If the *Free* option is selected, the connection is a hinge. If the *Elastic spring* option is selected, the rotation stiffness of the spring should be specified. If the *Elastoplastic spring* option is selected the maximum bending moment ($|M_y|$) should be specified besides the stiffness.

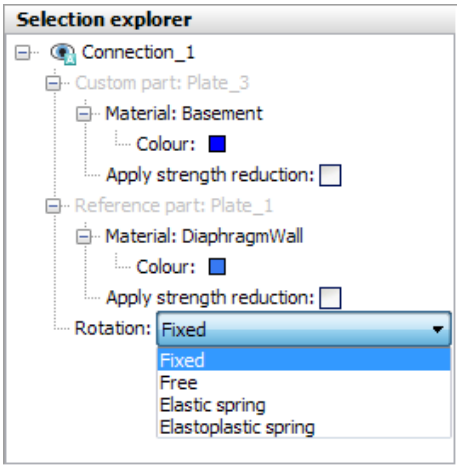


Figure 77: Properties of a connection in the Selection explorer

5.8 Tunnels [GSE]

The *Create tunnel* in the side (vertical) toolbar can be used to create arbitrary shaped tunnels which are to be included in the geometry model. A tunnel shape consists of a tunnel cross section, composed of arcs and lines,

optionally supplied with features such as lining (plates), interfaces, loads, prescribed displacements, contractions, groundwater flow boundary conditions, etc.

The tunnels available in the model are listed in the *Tunnels* subtree in the *Model explorer*. The geometry and the properties of a tunnel can be defined in the **Tunnel explorer** window.

Note:

- Note that it is not possible to undo/redo actions carried out to define a tunnel cross-section when the **Tunnel designer** window is closed.
- The **Tunnel designer** window can be reopened by right-clicking on one of the available tunnels in the *Object explorers* and selecting the *Edit* option in the appearing menu.

Warning: The Tunnels feature in PLAXIS 2D licence allows users to create a tunnel geometry . However, to generate the tunnel in the model a GSE entitlement needs to be active.

5.8.1 *Tunnel designer*

Once the *Create tunnel* button is clicked in the side toolbar, the user needs to select the tunnel insertion point in the geometry. After that, the **Tunnel designer** window appears.

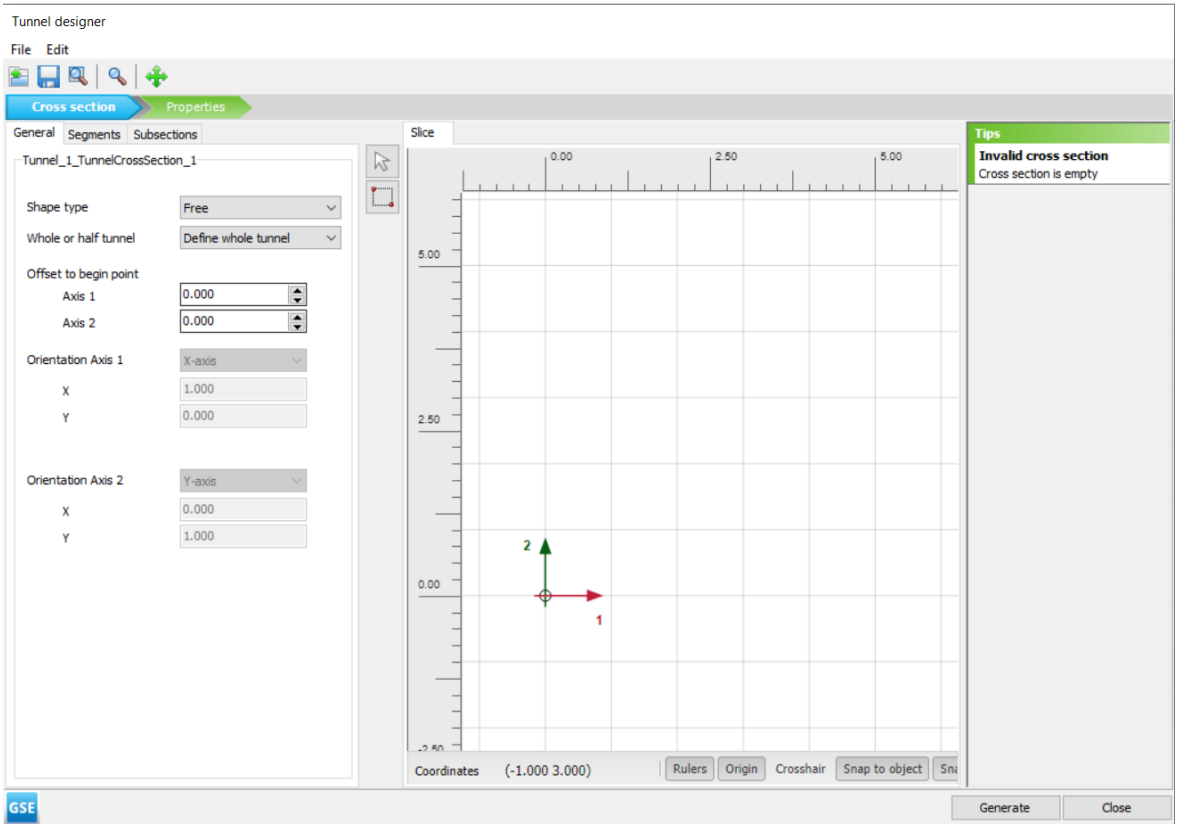


*Figure 78: The **General** tabsheet of the **Cross-section** mode*

The **Tunnel designer** contains the following items:

Tunnel menu	It contains the *File* menu that enables loading predefined tunnels as well as saving newly created ones.
Main toolbar	It contains buttons that enable loading predefined tunnels, saving new tunnels and arranging the view in the display area.
Mode bar	It contains two separate modes where the cross-section is defined and properties are assigned to its components.
Display area	It displays the defined geometry and it indicates the assigned properties. The slice is automatically updated after any change done in the modes in the left side. The supporting tools under the display area can be used to simplify the definitions and modification of the slice.
Feedback panel	It provides feedback on the quality of the defined cross section.

The design of a tunnel cross section is done in two separate modes, *Cross-section* and *Properties*. The geometric shape of the tunnel cross-sections can be defined in the **Cross-section** mode, whereas in the **Properties** mode features can be assigned to existing cross-section elements.

5.8.2 *Definition of tunnel cross section geometry*

The tunnel cross section is a collection of lines and arcs. Each line and arc composing the cross-section is defined as a segment.

1. ***General***

   The general shape settings of a tunnel are defined in the **General** tabsheet.

   Shape: The available types for the shape of a tunnel are:

Free	Allows to create a tunnel cross section composed of lines and arc segments.
Circular	Allows to create a circular tunnel.
Template	Allows to split tunnels into separate tunnels .
Whole or half tunnel	Allows to create a whole tunnel or a half one.

   Layout of the tunnel in the model: The general properties of a tunnel in the model are defined using the options in the **General** tabsheet.

   - The coordinates of the insertion point can be defined in the *Object explorers*. By default the coordinates of the location clicked in the model before the **Tunnel designer** window is displayed are shown. As the coordinates are changed, the tunnel will be relocated.
   - By default the insertion point is the starting point of the first segment of the tunnel cross section. The *Offset to begin point* group can be used to specify offset distances from the orientation axis when the

insertion point (origin of the orientation axis) and the starting point of the first segment of the cross section do not coincide.

- The *Orientation axis 1* and the *Orientation axis 2* define the orientation of the tunnel in the model.

2. ***Definition of segments***

The segments of a cross section can be defined in the **Segments** tabsheet of the **Tunnel designer** window (Figure 79 (on page 118)).

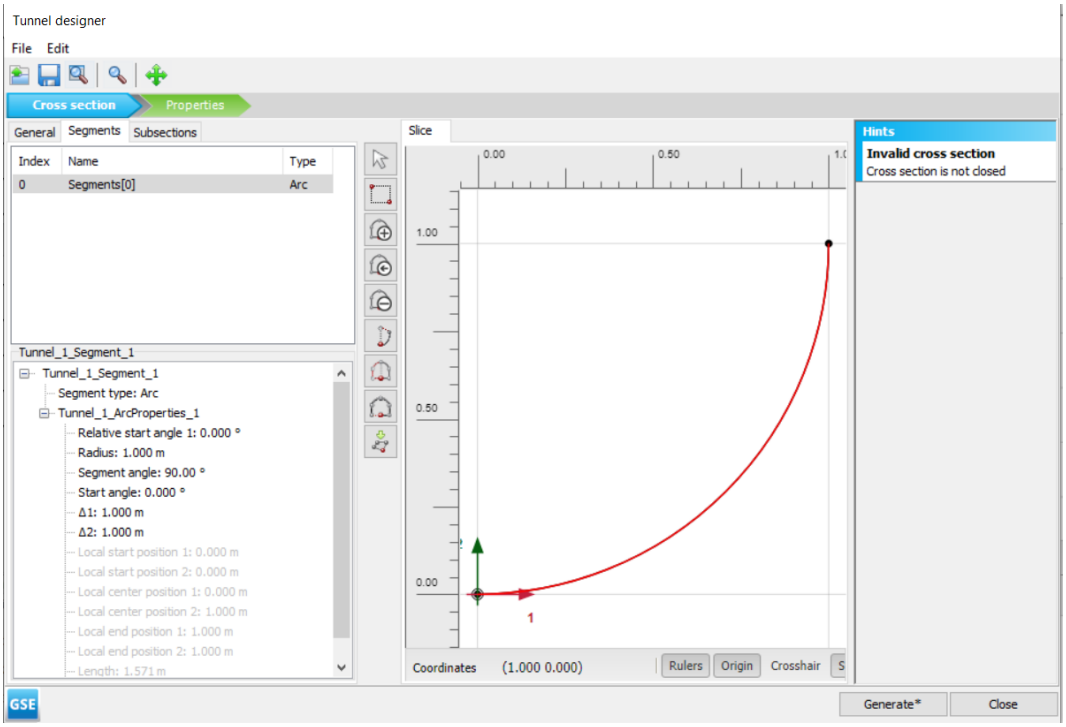


*Figure 79: The **Segments** tabsheet of the **Cross-section** mode*

A new segment is added to the last segment in a tunnel cross-section by clicking the *Add segment* button available in the side toolbar. The last point of the last segment is the first point of the newly added segment. The newly created segment is displayed in the segment list Figure 79 (on page 118).

Note: A segment can be selected by clicking it either in the list or in the display area. A selected segment is highlighted in the list and in the display area.

A new segment is inserted before a selected segment in a tunnel cross-section when the *Insert segment* button is clicked. The last point of the newly inserted segment becomes the first point of the previously selected segment. The indices of the segments are updated according to their position in the list.

A selected segment in a cross-section is deleted when the *Delete segment* button is clicked. The indices of the segments are updated according to their position in the list.

In the **Subsections** tabsheet, the points and segments can be right-clicked and the suitable option to create a subsection turns on in the side bar. This feature allows to create a new subsection starting from the end point and angle of the selected elements.

When a point is selected, it generates an horizontal straight line. When a point is selected, the option *Add subsection from point* is also available right-clicking in the *Slice* drawing area.

When a point and a segment are selected, it generates a straight line with a start angle equal to the tangent of the selected segment in the selected point. When a point and a segment are selected, the option *Add directed subsection from point* is also available right-clicking in the *Slice* drawing area.

When two points are selected, it generates a straight line connecting the chosen points. When two points are selected, the option *Add subsection between points* is also available right-clicking in the *Slice* drawing area.

Editable segments properties

Segment type	The options available are *Line* and *Arc*. For a *Line* segment the length of the segment should be specified. For an *Arc* segment the radius and the segment angle should be specified (Figure 80 (on page 120)).
Relative start angle	The value of the angle between the start tangent of the next segment and the end tangent of the previous segment (Figure 81 (on page 120)).
Radius	Arc only. The value of the radius for an arc segment.
Length	Line only. The value of the line segment length.
Segment angle	The value of the arc length expressed in degrees.
Start angle	The value of the angle between the start tangent of the segment and the global tangent.
Δ1	The horizontal increment of the line formed by the start and end point of the segment. Δ1 is in local polycurve space (defined by the orientation axes), not global model space.
Δ2	The vertical increment of the line of formed by the start and end point of the segment. Δ2 is in local polycurve space (defined by the orientation axes), not global model space.

The rest of parameters are read-only and cannot be edited.

Note: When modelling arcs using derived parameters Δ1 and Δ2, there will often be two solutions for the same absolute *x* or *y* value. In those situations, PLAXIS will always take the shortest segment. To design those ambiguous cases correctly, use the remaining parameters.

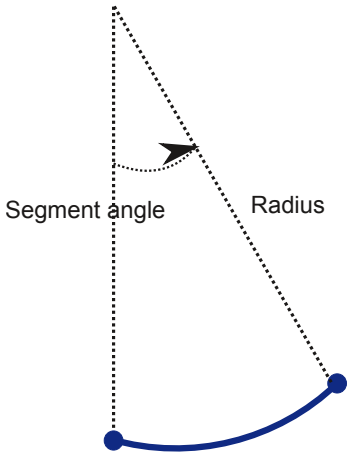


Figure 80: Properties of Arc segment

When specifying a negative radius for an *Arc* segment, the arc will be mirrored in the line described by the tangent vector. Note that when the *Circular* shape option is selected, the created cross-section consists only of one *Arc* segment with a *Segment angle* value of 360° or 180°, depending on whether the whole tunnel or only half of it is being defined.

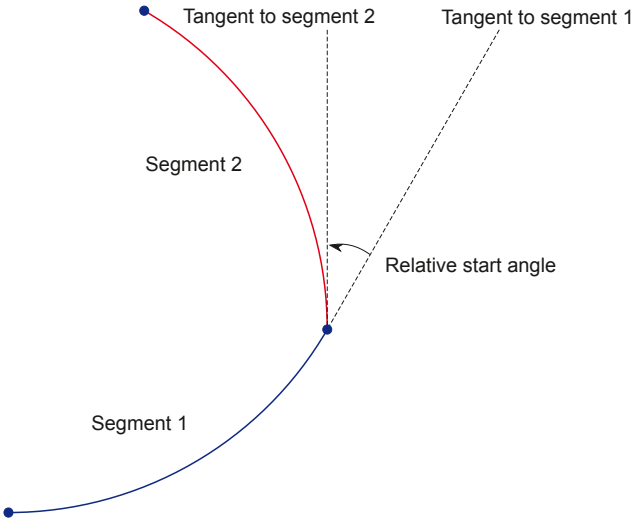


Figure 81: Description of Relative start angle

- To extend the cross-section to the symmetry axis click the corresponding button in the toolbar. A new segment is added connecting the last point of the cross-section to the axis of symmetry.
- To close the cross-section by mirroring the existing segments, click the *Symmetric close* button in the toolbar. This is only possible when one half is completed up to the axis of symmetry.
- To close the cross-section click the corresponding button in the toolbar. A new segment is created connecting the last point of the cross-section to the first one.

3. ***Definition of tunnel sub-clusters***

   The subdivision of a tunnel cross-section into different subsections can be subdivided in the **Subsections** tabsheet. In this way the various subsections for a NATM tunnel or a sequential excavation tunnelling process

can be defined (e.g. top heading, bench, invert, side drift). Essentially the geometric properties of the lines and arcs as well as the buttons in the side toolbar are the same as in the **Segments** tabsheet. There is no closure requirement for the geometry defined in the **Subsections** tabsheet. A sub-section may conveniently start from an existing geometry point, but sub-sections may also be defined independently.

To intersect the defined geometry:

- Select the geometric entities to be intersected.
- Click the *Intersect* button.
- Delete the redundant parts after the intersection.

To define thick tunnel lining:

- Click the *Generate thick lining* button in the side toolbar in the **Subsections** tabsheet. The **Generate thick lining** window pops up.
- Specify the thickness of the lining in the appearing window. A positive value will give an extra line outside of the current geometry contour. The reverse is true for a negative thickness value. Note that when the tunnel is included in the geometry model, a subcluster will be created for the thick lining. A proper material should be assigned to it, as it is done for the soil clusters.

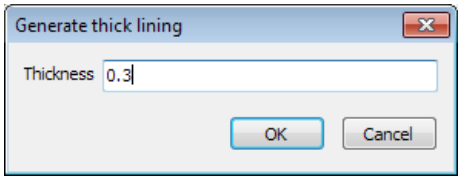


*Figure 82: The **Generate thick lining** window*

5.8.3 *Import tunnel geometry*

Existing tunnel geometry can be imported in the **Segments** tabsheet in the **Cross section** mode of the **Tunnel designer** window. The program supports *.dxf, *.step, *.stp and *.brep files. The geometry to be imported must meet the following criteria:

- It must contain a polycurve (consisting of lines and arcs), or circle, defining the outer boundary of the tunnel (called the segment path). This may consist of segments of both line and arc types. The components of the outer boundary will be listed as segments.
- The geometry objects should be defined parallel to the XY plane.

Note: Entities of incompatible types are omitted from the import.

On selecting the file to be imported, the imported cross section geometry appears in the **Import tunnel cross section preview** window (Figure 83 (on page 122)).

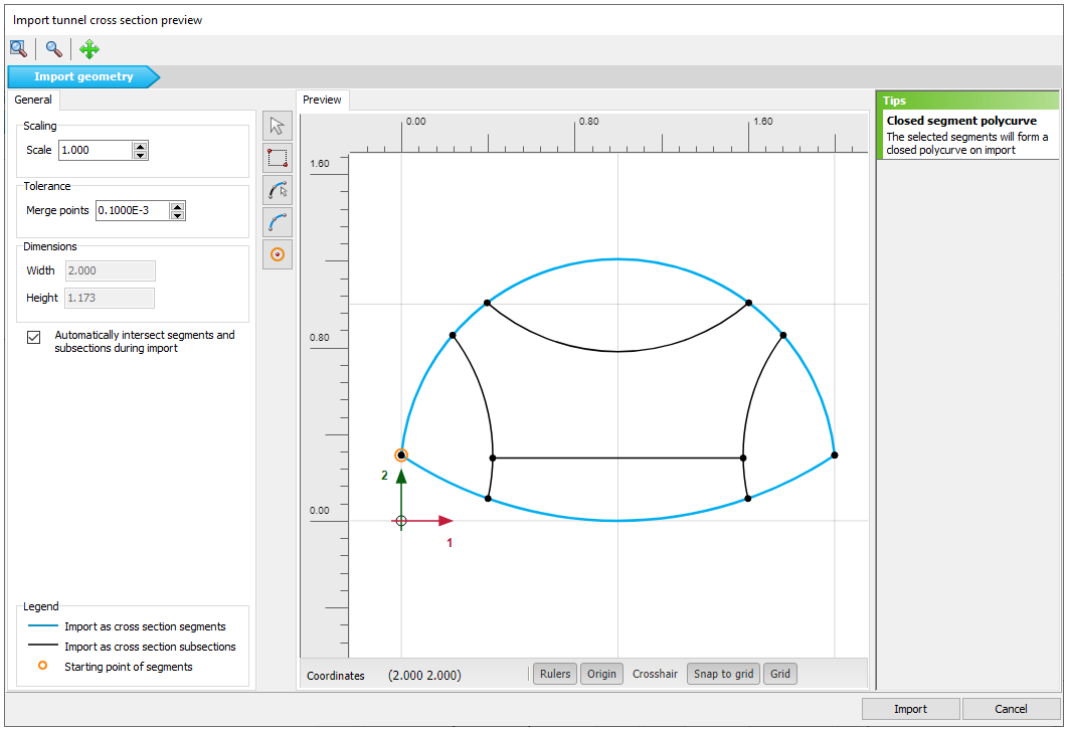


Figure 83: Import tunnel cross section preview

In the imported geometry, possible segments and subsections are automatically detected and indicated by blue and black lines respectively.

The side toolbar has the following buttons:

- The **Select** button can be used to select any segments and subsections in the preview pane.

  Note: To delete a segment or subsection, use the **Select** button to select the desired segment(s) or subsection(s), right click anywhere on the Preview pane , and click on the delete option that appears. Alternatively, the delete key on a keyboard can also be used.

- **Add or remove path segment** button can be used to add or remove individual parts to the main segment of the cross section.
- **Detect path** button can be used to select the longest path of the segments.
- **Select path start point** can be used to indicate the start point of the segments path.

General Tab sheet has the following options:

- **Merge points tolerance**:

  The segments are merged based on Merge points tolerance that can be specified in the General tab sheet panel. Gaps are eliminated based on simple geometrical translation of lines/arcs. The last line/arc is modified if it is necessary to close the selected segment path.

- **Dimensions**:

Dimensions shows the width and the height of the bounding box of the imported model. Note that the absolute coordinates of the bounding box are not relevant in the tunnel designer import.

- **Automatically intersect segments and subsections during import**:

  If the Automatically intersect segments and subsections during import checkbox is checked (which is the Default setting), the imported segments/subsections will be automatically intersected during import into the tunnel designer.

5.8.4 *Properties*

Features can be assigned to the tunnel segments and subsections in the **Properties mode** by right-clicking the segment and selecting the corresponding option in the appearing menu.

The features available in the Slice panel are:

Create line load	To assign a line load to the selected segment (or subsection).
Create line contraction	To assign a line contraction to the selected segment (or subsection).
Create line prescribed displacement	To assign a line prescribed displacement to the selected segment (or subsection).
Create plate	To model a tunnel shield or a shotcrete lining using the *Plate* feature.
Create geogrid	To model a temporary support or a shotcrete lining using the *Geogrid* feature.
Interfaces	To model the interaction of the tunnel lining with the surrounding soil (rock) in the selected segment.
Groundwater flow boundary conditions [ADV]	To assign boundary conditions for flow to the selected segment (or subsection).
Thermal flow boundary conditions [ADV]	To assign boundary conditions for thermal to the selected segment (or subsection).
Create reinforcement	To create reinforcement for the selected segment (or subsection).

Load options for tunnel segments

Loads can be added to individual tunnel segments in the *Properties* tabsheet of the tunnel designer. To do so, right-click the corresponding tunnel segment in the right drawing area and select *Create line load* from the appearing menu.

The information required to define a load assigned to a segment in the tunnel consists of load distribution, load value and the point of reference (in case of linear loads).

The distribution options for loads are:

Uniform	To create a uniform distributed load.
Linear	A linearly varying distributed load is created by defining the components and the magnitude of the force at the start and at the end points of the tunnel segment (or subsection). The start and end points are determined by their order of definition.
Perpendicular	To create a uniformly distributed load perpendicular to the selected segment (or subsection). The value of the load (σ_n) needs to be specified.
Perpendicular, vertical increment	To create a load perpendicular to the selected segment varying with depth by defining the components and the magnitude of the load at the reference point with respect to the global system of axes (*y*-axis) and its increment with depth. The values of the reference load ($\sigma_{n,ref}$) and the load increment ($\sigma_{n,inc}$) as well as the reference vertical coordinate need to be specified.

Reinforcements

Reinforcements can be added to tunnels as *Rock bolts* or *Umbrella arches*. They are modelled by means of embedded beamrows. They can be added to tunnels in the *Properties* tabsheet of the tunnel designer. This can be done by following these steps:

- Click on the *Reinforcements* tab of the *Properties* mode.
- In the drawing area, (multi) select the required segments and click on the *Define reinforcement* on the left panel to add a reinforcement. Alternatively, right-click on the selected segments and select the *Create reinforcement* option (Figure 84 (on page 125)). By default, the option is set to *Rock bolt*. In the lower left panel, the created reinforcement can be viewed and the properties can be assigned.
- For more information about material properties of rock bolts refer General properties (on page 275).
- It is possible to delete a reinforcement, by clicking on the *Delete* button in the drawing area.

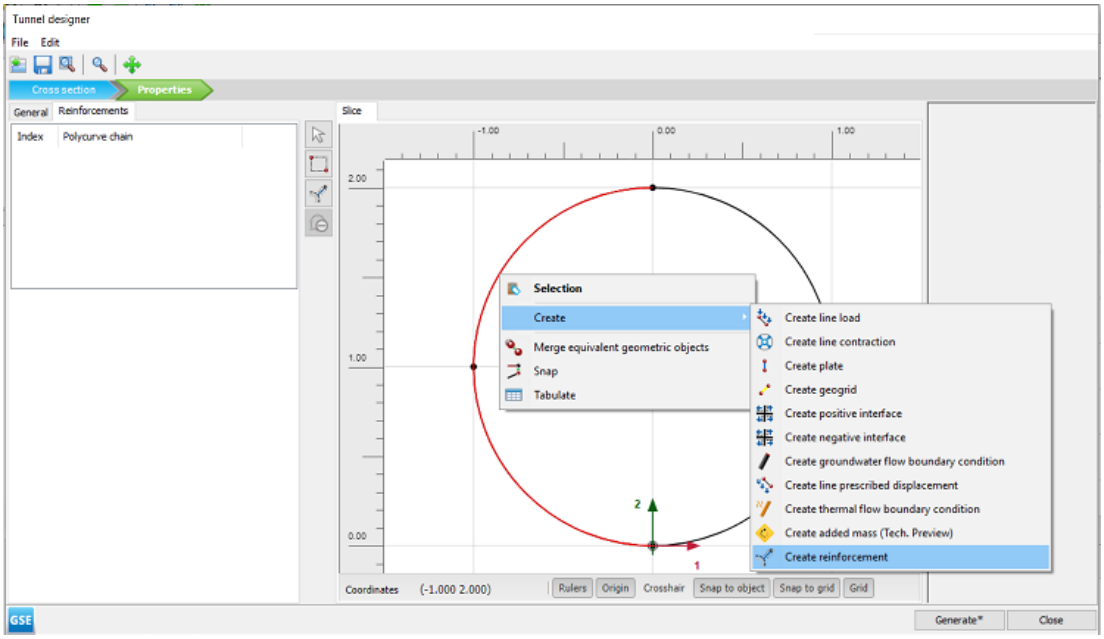


*Figure 84: Creation of **Reinforcements** in the Properties tabsheet*

Rock bolts are drawn in a counter clockwise direction from the starting point at the beginning of the segment (or subsection). It is possible to reverse this direction, by clicking on the check box *Reversed* in lower left panel. Note that the *Offset* and *Spacing* are also reversed accordingly.

Input parameters required when the configuration is set to *Rock bolts* are *Material*, *Apply strength reduction* and the following as described below:

Length of rock bolt	Length of the rock bolt perpendicular to the tunnel contour.
Offset from segments	Each rock bolt can have an offset from the tunnel contour. This is particularly useful when the tunnel has a thick lining.
Distribution	Rock bolts can be added in both breadth and depth of a tunnel trajectory. The distribution of these rock bolts is possible by choosing *Number*, *Spacing* or *Custom* from a drop down menu. When the option *Custom* is selected, both spacing and offset parameters can be specified.
Number	Number of rock bolts in breadth or depth of a tunnel trajectory.
Offset	Distance from the first tunnel point to the first rock bolt along the tunnel contour.

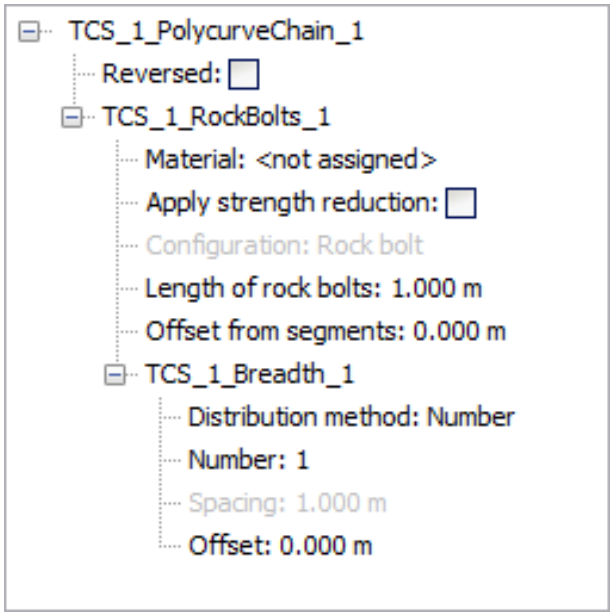


Figure 85: Rock bolt parameters

Note: Rock bolts for tunnels may also be added using the *Embedded beam row* feature and *Polar array* in Input. However, it is suggested to create it in the *Tunnel designer* for more accuracy.

5.9 Design approaches [ADV]

PLAXIS 2D has a facility to deal with partial factors for loads and model parameters (Brinkgreve & Post, 2013 (on page 530)). This facility is called '*Design approaches*' and enables PLAXIS 2D to be used for design calculations in the framework of the Eurocode, LRFD or other design methods based on partial factors.

The main idea is that a project is first analysed for a Serviceability Limit State (SLS) situation, without using *Design approaches*. The input values of the loads and the model parameters are supposed to be representative or characteristic values. The result would be a cautious estimate of deformations, stresses and structural forces.

If satisfactory results have been obtained for the serviceability state, one may consider using *Design approaches* to deal with Ultimate Limit State (ULS) design. In order to perform design calculations, new phases need to be defined in addition to the serviceability state calculations. There are two main schemes to perform design calculations in relation to serviceability calculations (Bauduin, Vos & Simpson, 2000 (on page 530)).

Scheme 1:

$$
\begin{array}{lcl}
0.\ Initial\ phase & & \\
\downarrow & & \\
1.\ Phase\ 1(SLS) & \rightarrow & 4.Phase\ 4(ULS) \\
\downarrow & & \\
2.\ Phase\ 2(SLS) & \rightarrow & 5.Phase\ 5(ULS) \\
\downarrow & & \\
3.\ Phase\ 3(SLS) & \rightarrow & 6.Phase\ 6(ULS)
\end{array}
$$

In this scheme, the design calculations (ULS) are performed for each serviceability state calculation separately. This means that Phase 4 starts from Phase 1, Phase 5 starts from Phase 2, etc. Note that in this case a partial factor on a stiffness parameter is only used to calculate additional displacements as a result of stress redistribution due to the factored (higher) loads and the factored (reduced) strength parameters.

Scheme 2:

$$
\begin{array}{lcl}
0.\ Initial\ phase & \rightarrow & 4.\ Phase\ 4(ULS) \\
\downarrow & & \downarrow \\
1.\ Phase\ 1(SLS) & & 5.\ Phase\ 5(ULS) \\
\downarrow & & \downarrow \\
2.\ Phase\ 2(SLS) & & 6.\ Phase\ 6(ULS) \\
\downarrow & & \\
3.\ Phase\ 3(SLS) & &
\end{array}
$$

In this scheme, the design calculations (ULS) start from the initial situation and are performed subsequently. This means that Phase 4 starts from the Initial phase, Phase 5 starts from Phase 4, etc.

It is the responsibility of the geotechnical engineer to consider all the different conditions that effect the design. Engineering judgment plays a vital role in the determination of different combinations to be considered in the design.

5.9.1 *Definition of design approaches*

Different design approaches, i.e. coherent sets of partial factors, can be defined for loads and model parameters according to the applicable design methods (for example: Eurocode 7 - DA 3). After defining such a coherent set of partial factors by the user, it may then be exported (stored) under an appropriate name in a global data base, after which it can be imported and re-used in other projects. Hence, once complete sets of partial factors have been defined and stored, it is a relatively small effort to make design calculations in addition to serviceability calculations.

The design approaches are global attributes and are listed in the corresponding subtree under the *Attributes library* in the *Model explorer*.

- To activate the **Design approaches** window (Figure 86 (on page 128)), right-click the corresponding subtree under the *Attributes library* in the *Model explorer*. Note that the *Design approaches* option available in the *Soil*, *Structures*, *Water conditions* and *Staged construction* modes.

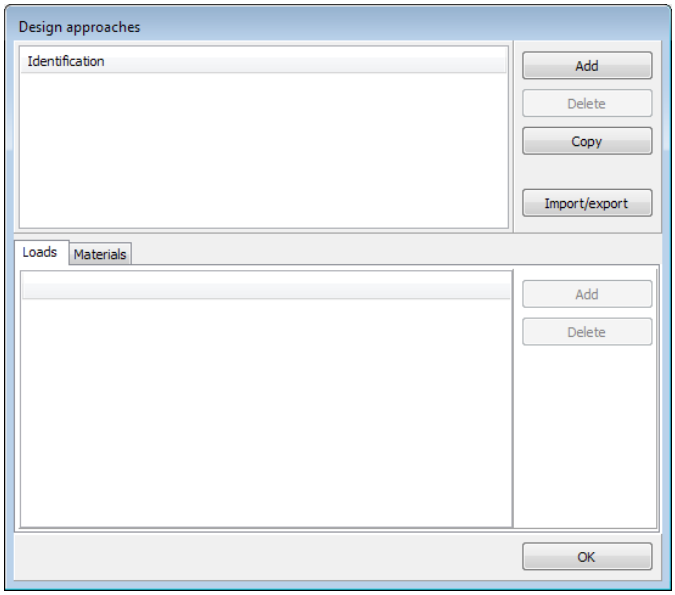


*Figure 86: The **Design approaches** window*

The **Design approaches** window consists of two parts. In the *Identification* part a list of the defined design approaches as well as options to *Add*, *Delete*, *Copy* and *Import/Export* design approaches are available.

- To add a new design approach click the corresponding option. A new design approach will be added in the list. Note that the name of a design approach can be changed by clicking it in the list and typing the new name.
- To delete a design approach from the list select the design in the list and click the corresponding button.
- To create a copy of a predefined design approach select the design approach in the list and click the *Copy* button. The name of a design approach can be changed by clicking it in the list and typing the new name.
- To import design approaches from other projects or from the global repository of design approaches. Click the corresponding button under the list. As a result, the **Import/Export to global repository** window appears (Figure 87 (on page 128)). The global repository is a database of the design approaches contained in other projects, simplifying their reuse. The address to the location of the global repository is given under the list of the global design approaches. A different repository can be selected by clicking the *Select* button and selecting the new repository. Global design approaches can be removed from the repository by selecting them first in the list and by clicking *Delete* button.

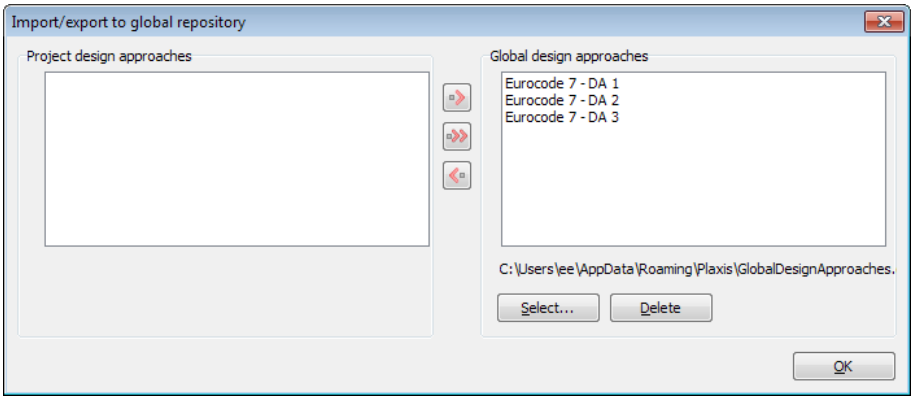


*Figure 87: **Import design approach** window*

5.9.2 *Definition of partial factors for loads*

Loads to be considered in the design approaches for a geotechnical project are:

- Distributed loads
- Point loads
- Prescribed displacements

Different partial factors may apply to different loads or groups of loads. This can be arranged by assigning *Labels* to individual loads or groups of loads. Considering partial factors for loads, distinction can be made between different load cases as shown in Figure 88 (on page 129) .

To change a label, double click it and type the new label. Up to 10 labels and their corresponding partial factors can be defined. The partial factors for loads are used as a multiplication factor to the reference values of the loads.

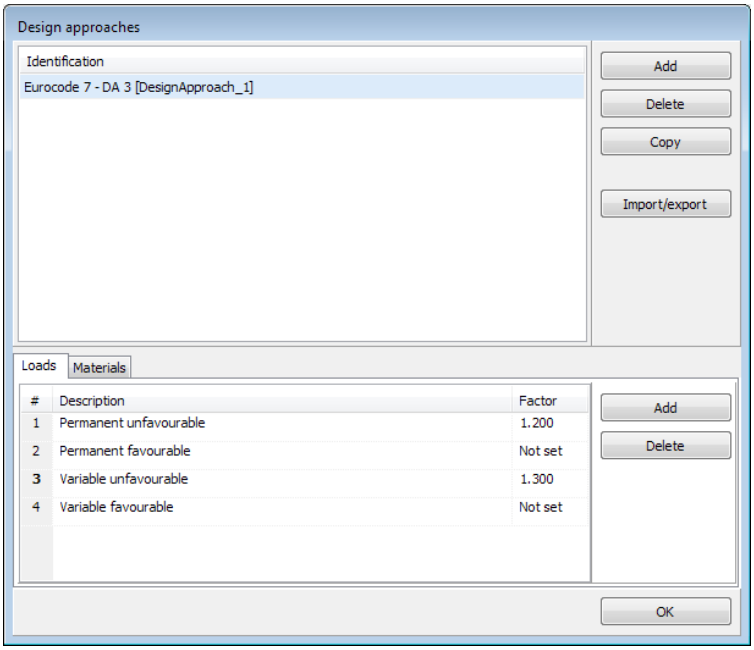


Figure 88: ***Partial factors for loads*** *tabsheet*

Note: The partial factors for loads are defined such that the design value of a parameter is the reference value multiplied with the partial factor.

Labels can be assigned to loads in the corresponding subtree in the *Model explorer* in the **Phase definition modes**, as part of the definition of a calculation phase. However, the partial factors corresponding to the load labels are only applied when the calculation is performed according to the selected design approach (see Calculation using design approaches [ADV] (on page 503)); otherwise reference values of loads are used.

5.9.3 *Definition of partial factors for materials*

The partial factors for materials for a selected design approach in the list can be defined in the corresponding tabsheet in the lower part of the **Design approaches** window. A list of all the available partial factors is displayed below.

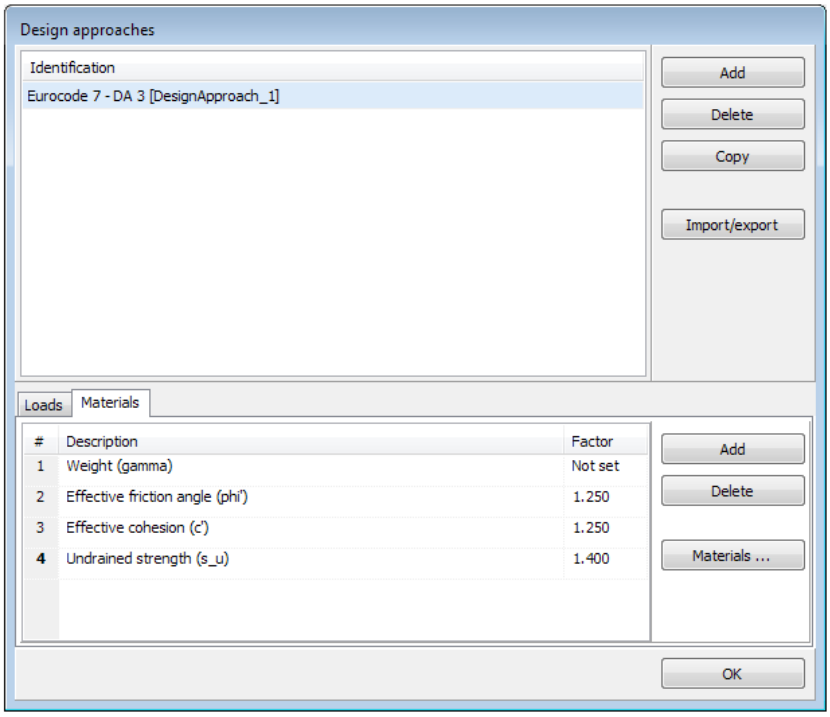


Figure 89: Partial factors for material tabsheet

Note:

- The partial factors for materials are defined such that the design value of a parameter is the reference value divided by the partial factor. This also applies to the partial factor on the unit weight. If the partial factor is intended to 'increase' the weight, it should be less than unity (1.0).
- In the case of a partial factor on the friction angle φ or the dilatancy angle ψ, the partial factor is applied to $tan(\varphi)$ and $tan(\psi)$ respectively.
- In addition to partial factors for soil model parameters, partial factors for structural model parameters may be defined as well.

To assign the specified factor to the material parameters:

- Click the *Materials* button. The **Material sets** window pops up.
- Open the material data set. Note that the view has changed. In the current view it is possible to assign factors to different soil parameters, as well as to see the effect of these factors on the soil parameters.
- Click the *Mechanical* tab. In the **Mechanical** tabsheet select the corresponding labels for the material parameters (i.e. c'_{ref} and φ' - see Figure 90 (on page 131)).

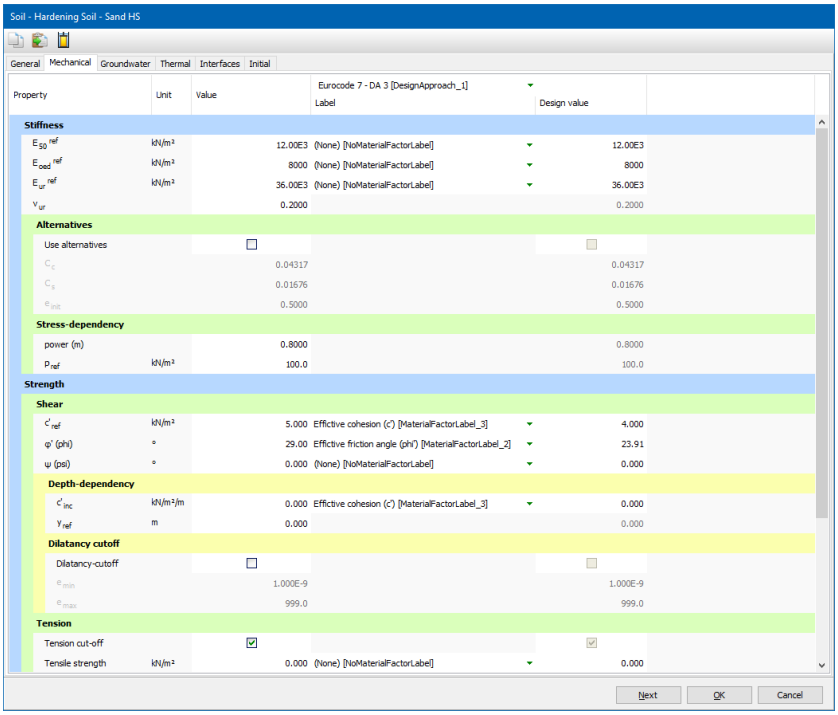


Figure 90: Assignment of partial factors to material parameters

Note: *Design approaches* are not supported for NGI-ADP model, UDCAM-S model, Sekiguchi-Ohta model and User-defined Soil Model

To complete the definition of a design calculation it is necessary to proceed to the **Stage definition modes** where the design approach to be used for a specific phase is selected and the loads in the model are labelled (Calculation using design approaches [ADV] (on page 503)). Make sure that the required load labels with corresponding partial factors are properly assigned to the external loads in the model.

When using *Design approaches* in combination with advanced soil models, these models will continue to behave as advanced models, maintaining all their features, such as stress-dependent stiffness behaviour and hardening effects. This is different than when using a *Safety* analysis with advanced models (Safety calculation (phi/c reduction) (on page 307)), since in the latter case advanced models loose their advanced features and basically switch to Mohr-Coulomb. When comparing a *Safety* analysis to a target value of *ΣMsf* with a *Design approaches* analysis using the same partial factor for *c* and *tan(φ)*, it should be realised that the results could be different because of this reason.

5.10 Hydraulic conditions [ADV]

As an alternative to a direct generation of pore pressures based on the water conditions in the boreholes and the soil clusters, the pore pressure distribution may be calculated on the basis of a groundwater flow calculation or a fully coupled flow-deformation analysis. This requires the definition of groundwater flow boundary conditions, which are termed *Hydraulic conditions*. *Hydraulic conditions* may also include special conditions that can be

imposed to the model to control the pore pressure at certain locations in the case of groundwater flow calculations or fully coupled flow-deformation analysis.

The creation of a hydraulic condition is similar to the creation of a geometric entity. The options available in the menu displayed as the *Create hydraulic conditions* button is selected in the side toolbar (Figure 91 (on page 132)), provide a faster definition of conditions.

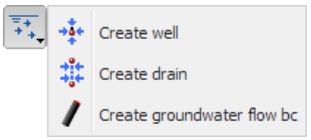


Figure 91: The Create hydraulic conditions menu

Instead of creating the geometric entity and then assigning a hydraulic condition to it, the whole process is completed in one step. To assign hydraulic conditions to existing geometry entities:

- Right-click the geometric entity either in the drawing area or in the *Object explorers*.
- In the appearing menu point to *Create* and select the hydraulic condition by clicking on the corresponding option.

Note that the assignment of hydraulic conditions is also possible by right-clicking the geometric entity either in the drawing area or in the *Object explorers* and selecting the corresponding option in the appearing menu.

Although the input values of hydraulic conditions are specified in the geometry model, the activation, deactivation or change of values may be considered in the framework of **Staged construction** (Changing geometry configuration (on page 359)).

In addition to the specific boundary conditions as considered here, global model conditions for the model as a whole in terms of 'open' and 'closed' boundaries as well as precipitation conditions can be defined for each calculation phase in the *Model conditions* subtree in the *Model explorer* (Model conditions (on page 364)). Note that specific hydraulic conditions have priority over global model conditions.

Note: For transient groundwater flow and fully coupled flow-deformation analysis, hydraulic conditions can be defined as a function of time using so-called *Flow functions* (Flow functions [ULT] (on page 141)) which are available with an Ultimate licence.

5.10.1 *Well*

In order to define a well, the *Create well* option should be selected from the menu appearing as the *Create hydraulic conditions* button is clicked. The creation of a well in the geometry model is similar to the creation of a line (Create line (on page 71)). Wells are used to prescribe lines inside the geometry model where a specific flux (discharge) is extracted from or infiltrated into the soil. The properties of a well can be modified in the *Selection explorer* (Figure 92 (on page 133)).

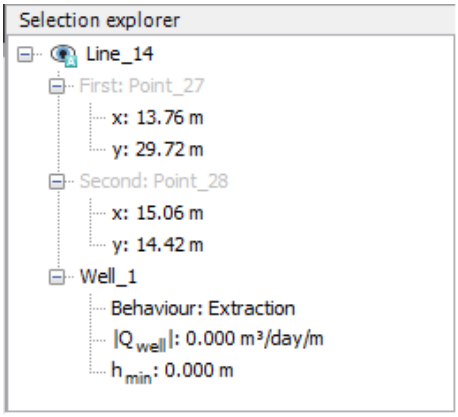


Figure 92: Well definition in the Selection explorer

The required properties of a well are:

Behaviour	The behaviour of the well needs to be specified. The available options are *Extraction* (taking water from the ground) and *Infiltration* (bringing water into the ground).
$\|Q_{well}\|$	The discharge of the well in the unit of volume per unit of time, per unit of width in the out-of-plane direction.
h_{min}	The minimum possible head in the well. When the groundwater head reduces below the h_{min} level no further extraction will occur. As a habit, h_{min} may be set equal to the bottom level of the well in the ground.

This option is only relevant for groundwater flow calculations and fully coupled flow-deformation analysis. Wells can be activated or de-activated in calculation phases.

Note:

- When a well intersects multiple soil layers the prescribed flux for each soil layer is a function of the saturated permeability and the intersected depth.
- Note that the saturated permeability is specified in the **Groundwater** tabsheet of the material dataset.

5.10.2 *Drain*

In order to define a drain, the *Create drain* option should be selected from the menu appearing as the *Create hydraulic conditions* button is clicked. The creation of a drain in the geometry model is similar to the creation of a geometry line (Create line (on page 71)). Drains are used to prescribe lines inside the geometry model where (excess) pore pressures are reduced.

Together with the creation of a drain, the input of the properties of the drain is required. This option is only relevant for consolidation analysis, groundwater flow calculations or fully coupled analysis. In such calculations, the pore pressure in all nodes of the drain is reduced such that it is equivalent to the given head. If the drain is a *Normal* drain, pore pressures lower than the equivalent to the given head are not affected by the drain. If the

drain is a *Vacuum* drain, the *Head* parameter can be used to define an 'underpressure' in the ground, equivalent to the process of vacuum consolidation. Drains can be activated or de-activated in calculation phases. The properties of a drain can be modified in the *Object explorers* (Figure 93 (on page 134)).

The input required for a drain depends on the option selected in the *Behaviour* drop-down menu. The options available are:

Normal	When the *Normal* option is selected for the drain behaviour, the input of a groundwater head (h) is required. This option is only relevant for groundwater flow calculations or fully coupled analysis. In such calculations, the pore pressure in all nodes of the drain is reduced such that it is equivalent to the given head. Pore pressures lower than the equivalent to the given head are not affected by the drain. In consolidation analysis, drains reduce the excess pore pressure to zero and the specified head is ignored.
Vacuum	When the *Vacuum* option is selected for the drain behaviour, the input of a groundwater head (h) is required. The *Head* should be used to define the 'underpressure' in the ground due to the process of vacuum consolidation. This option is only relevant for consolidation analysis, groundwater flow calculations or fully coupled analysis. In such calculations, the pore pressure in all nodes of the drain is reduced such that it is equivalent to the given head. Note that in contrast to *Normal* drains, pore pressures lower than the equivalent to the given head ARE affected by the *Vacuum* drain.

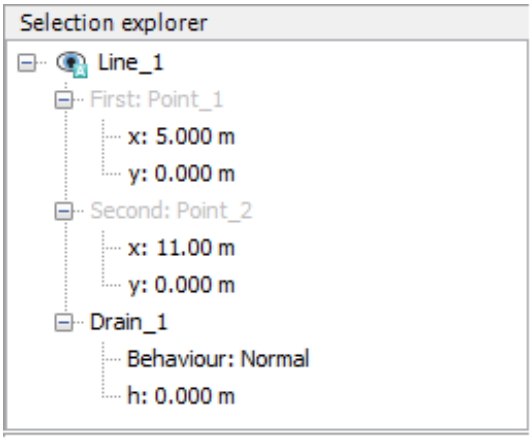


Figure 93: Normal drain definition in the Selection explorer

Drains can be activated or de-activated in calculation phases. The properties of a drain can be modified in the *Object explorers*.

5.10.3 *Groundwater flow boundary conditions*

In order to define a flow boundary condition, the *Create groundwater flow bc* option should be selected from the menu appearing as the *Create hydraulic conditions* button is clicked. The creation of a flow boundary condition in the geometry model is similar to the creation of a line (Create line (on page 71)).

The behaviour of the flow boundary conditions can be specified by selecting the corresponding option in the *Selection explorer* (Figure 94 (on page 135)).

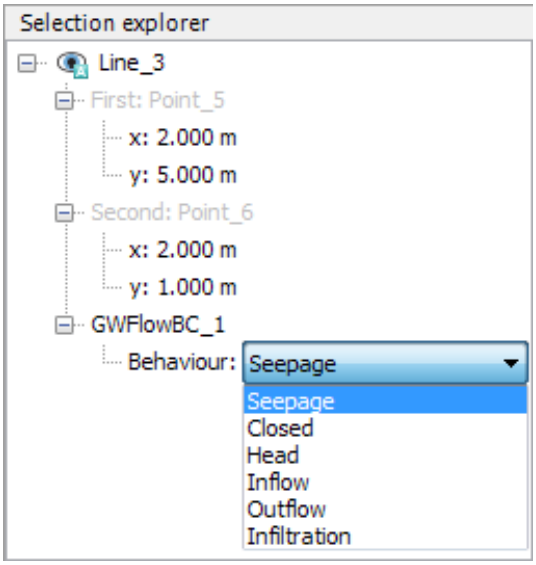


Figure 94: Options for the behaviour of Groundwater flow boundary conditions

Note:

- The boundary conditions at the extremities of the model can be conveniently specified in the *Model conditions* subtree in the *Model explorer*. By default the bottom of the model is set to *Closed*, preventing flow across it, whereas the three remaining boundaries are set to *Open* (*Seepage*).
- In a calculation phase the hydraulic conditions defined using the *Flow boundary conditions* feature always prevail over the *Model conditions*. As an example, if a flow boundary conditions is placed on the top boundary and precipitation is specified in *Model conditions* only the behaviour specified for the flow boundary condition will be considered, ignoring the precipitation.
- In a fully coupled analysis the water conditions assigned in any internal geometry are taken into account at the calculation step 0 (i.e beginning of the calculation). In the subsequent calculation steps, PLAXIS ignores the specified water conditions settings and performs the flow calculation based on the outcome of the previous step.
- The *Closed* condition is always applicable only on the external boundaries of the model. If a *Closed* condition is applied on an internal geometry, PLAXIS 2D ignores it.

Seepage

A *Seepage* boundary is a boundary where water can flow in or out freely. It is generally used at the ground surface above the phreatic level or above the external water level (in the case of an inclined line, such as a dam or a dike).

If a flow boundary is set to *Seepage* and is completely above the (external) water level, then the seepage condition applies to this boundary. This means that water inside the geometry may flow freely out of this boundary.

If a boundary is *Seepage* and completely below the (external) water level, the free boundary condition automatically turns into a groundwater head condition. In that case the magnitude of the groundwater head in each boundary node is determined by the vertical distance between the boundary node and the water level.

The pore pressure is zero where a (external) water level crosses a geometry boundary line. The part of the geometry above the transition line is treated as a boundary above the water level, whereas the part of the geometry below the transition line is treated as a boundary below the water level. Hence, different conditions can apply to such a geometry boundary . This is possible because, in general, a geometryboundary consists of many nodes and the actual information on boundary conditions as used by the calculation program is contained in the boundary nodes rather than in geometry lines.

Note: Note that a boundary that is explicitly set to *Seepage* will NOT automatically turn into an *Infiltration* boundary when precipitation is specified.

Closed

When the *Closed* option is assigned to a boundary, no flow can occur across it. Flow, in this case, refers to groundwater flow (in groundwater flow calculations and fully coupled flow-deformation analysis) as well as dissipation of excess pore pressure (in consolidation calculations).

Note: The *Closed* condition is always applicable only on the external boundaries of the model. In case it's required an impermeable line inside the soil (e.g. the cut-off of a dam), an appropriate interface element should be inserted.

Head

In addition to the automatic setting of hydraulic conditions based on the general phreatic level in *Model conditions* (Flow boundary conditions [ADV] (on page 356)), a prescribed groundwater head may be entered manually to the *Groundwater flow boundary conditions* defined by the user in the model. If a groundwater head is prescribed at an outer geometry boundary, external water pressures will be generated for that boundary. The deformation analysis program will treat external water pressures as traction loads and they are taken into account together with the soil weight and the pore pressures. Figure 95 (on page 137) shows the options available for the definition of head in *Object explorers*.

The options available for variation of head along the boundary are:

Uniform	A uniform value of the head (h_{ref}) is specified along the boundary.
Vertical increment	A vertical increment of the head along the boundary is specified. The parameters required to specify a vertical increment variation are: • h_{ref}: The reference head value at the specified reference level. • $h_{inc,y}$: Increment value specifying the change of the head with depth. • y_{ref}: The reference value for which the reference head value is specified. If this value is lower than the top of the boundary, the head value below this level is changed according to the $h_{inc,y}$ value (h = h_{ref} +y_{ref} - y $h_{inc,y}$).
Horizontal increment	A horizontal increment of the head along the boundary is specified. The parameters required to specify a horizontal increment variation are: • h_{ref}: The reference head value at the specified reference level. • $h_{inc,x}$: Increment value specifying the change of the head with depth. • x_{ref}: The reference value for which the reference head value is specified. If this value is lower than the leftmost point of the boundary, the head value right from this level is changed according to the $h_{inc,x}$ value.

Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $h_{ref,start}$: The start value for the boundary condition. • $h_{ref,end}$: The end value for the boundary condition.

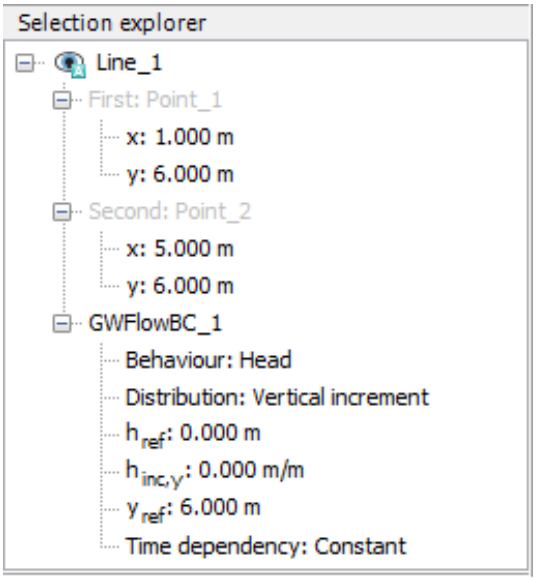


Figure 95: Head definition in the Selection explorer

The options available for *Time dependency* of *Head* assigned to a line flow boundary condition are:

Constant	The head in the boundary does not change with time.
Time dependent [ULT]	The head in the boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The flow function representing the time dependency can be selected from the drop-down menu. Information on *Flow functions* is available in Flow functions [ULT] (on page 141).

Inflow

Inflow to the model through a boundary can be specified by selecting the corresponding *Behaviour* option in the *Selection explorer*. Figure 96 (on page 138) shows the options available for the definition of inflow.

The options available for the distribution of the inflow discharge along the boundary are:

Uniform	A uniform value of the discharge ($\|q_{ref}\|$) is specified along the boundary.

Vertical increment	A vertical increment of the discharge along the boundary is specified. The parameters required to specify a vertical increment variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level in the unit of volume per unit of time per unit of width in the out-of-plane direction. • $q_{inc,y}$: Increment value specifying the rate of variation of discharge along the boundary. • y_{ref}: The reference value for which the reference discharge value is specified. If this value is lower than the top of the boundary, the discharge value below this level is changed according to the $/q_{inc,y}/$ value.
Horizontal increment	A horizontal increment of the head along the boundary is specified. The parameters required to specify a horizontal increment variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level in the unit of volume per unit of time per unit of width in the out-of-plane direction. • $q_{inc,x}$: Increment value specifying the rate of variation of discharge along the boundary. • x_{ref}: The reference value for which the reference discharge value is specified. If this value is lower than the leftmost point of the boundary, the discharge value right from this level is changed according to the $/q_{inc,y}/$ value.
Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $q_{ref,start}$: The start value for the boundary condition. • $q_{ref,end}$: The end value for the boundary condition.

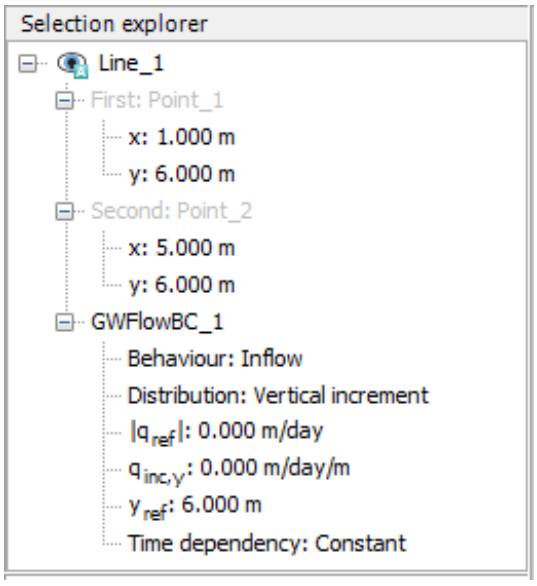


Figure 96: Inflow definition in the Selection explorer

The options available for *Time dependency* of the discharge are:

Constant	The discharge does not change with time.
Time dependent [ULT]	The discharge in the boundary varies with time. varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The flow function representing the time dependency can be selected from the drop-down menu. Information on *Flow functions* is available in Flow functions [ULT] (on page 141).

Outflow

Outflow from the model through a boundary can be specified by selecting the corresponding option for *Behaviour* in the *Selection explorer* Figure 97 (on page 140).

The options available for the distribution of the outflow discharge along the boundary are:

Uniform	A uniform value of the discharge is specified along the boundary. The value is assigned to the $\lvert q_{ref} \rvert$ parameter.
Vertical increment	A vertical increment of the discharge along the boundary is specified. The parameters required to specify a vertical increment variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level in the unit of volume per unit of time per unit of width in the out-of-plane direction. • $q_{inc,y}$: Increment value specifying the rate of variation of discharge along the boundary. • y_{ref}: The reference value for which the reference discharge value is specified. If this value is lower than the top of the boundary, the discharge value below this level is changed according to the $/q_{inc,y}/$ value.
Horizontal increment	A horizontal increment of the head along the boundary is specified. The parameters required to specify a horizontal increment variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level in the unit of volume per unit of time per unit of width in the out-of-plane direction. • $q_{inc,x}$: Increment value specifying the rate of variation of discharge along the boundary. • x_{ref}: The reference value for which the reference discharge value is specified. If this value is lower than the leftmost point of the boundary, the discharge value right from this level is changed according to the $/q_{inc,y}/$ value.
Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $q_{ref,start}$: The start value for the boundary condition. • $q_{ref,end}$: The end value for the boundary condition.

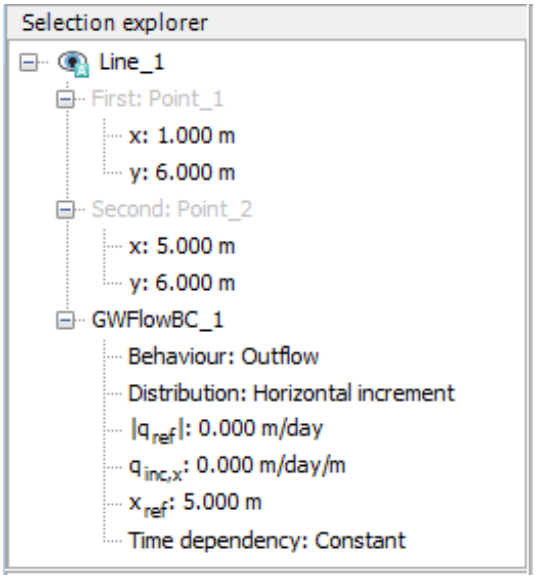


Figure 97: Outflow definition in the Selection explorer

The options available for variation of discharge with time are (*Time dependency*):

Constant	The discharge does not change with time.
Time dependent [ULT]	The discharge in the boundary varies with time. varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The flow function representing the time dependency can be selected from the drop-down menu. Information on *Flow functions* is available in Flow functions [ULT] (on page 141).

Infiltration

Apart from the automatic generation of infiltration boundary conditions from precipitation (Flow boundary conditions [ADV] (on page 356)), infiltration conditions may also be specified manually for geometry boundaries above the water level. Infiltration boundary condition is a conditional inflow. The parameters required for an infiltration boundary are:

q	Recharge (infiltration), specified in the unit of length per unit of time. Negative values can be used to model evapotranspiration (evaporation + transpiration).
ψ_{max}	Maximum pore pressure head, relative to the elevation of the boundary, specified in the unit of length (default 0.1 length units).
ψ_{min}	Minimum pore pressure head, relative to the elevation of the boundary, specified in the unit of length (default -1.0 length units).

The options available for variation of discharge with time (*Time dependency*) are:

Constant	The head in the boundary does not change with time

Time dependent [ULT]	The discharge in the boundary varies with time. varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The flow function representing the time dependency can be selected from the drop-down menu. Information on *Flow functions* is available in Flow functions [ULT] (on page 141).

Note: Note that a negative value of infiltration represents *Evaporation*.

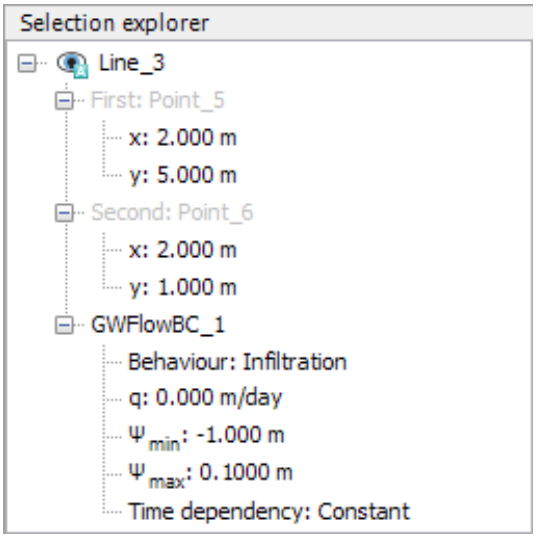


Figure 98: Infiltration definition in the Selection explorer

5.11 Flow functions [ULT]

Flow functions describing the variation with time of quantities such as *Head* and *Discharge* can be defined in the corresponding subtree under the *Attributes library* in the *Model explorer*(see Figure 99 (on page 141)).

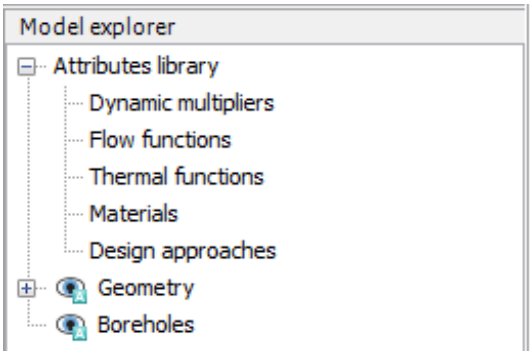


Figure 99: Flow functions subtree in Model explorer

Flow functions can be assigned to *Water levels* or *Groundwater flow boundary conditions*. Note that time-dependent conditions can be defined for all behaviour options except for *Seepage* and *Closed*.

Note: Flow functions cannot be assigned to non-horizontal water levels.

Functions can be defined in the corresponding window that pops up when the *Flow functions* subtree is right-clicked and the *Edit* option is selected in the appearing menu. The window consists of two tabsheets, i.e **Head functions** and **Discharge functions**, where the functions applicable to head and discharge can be defined respectively as shown in Figure 100 (on page 142).

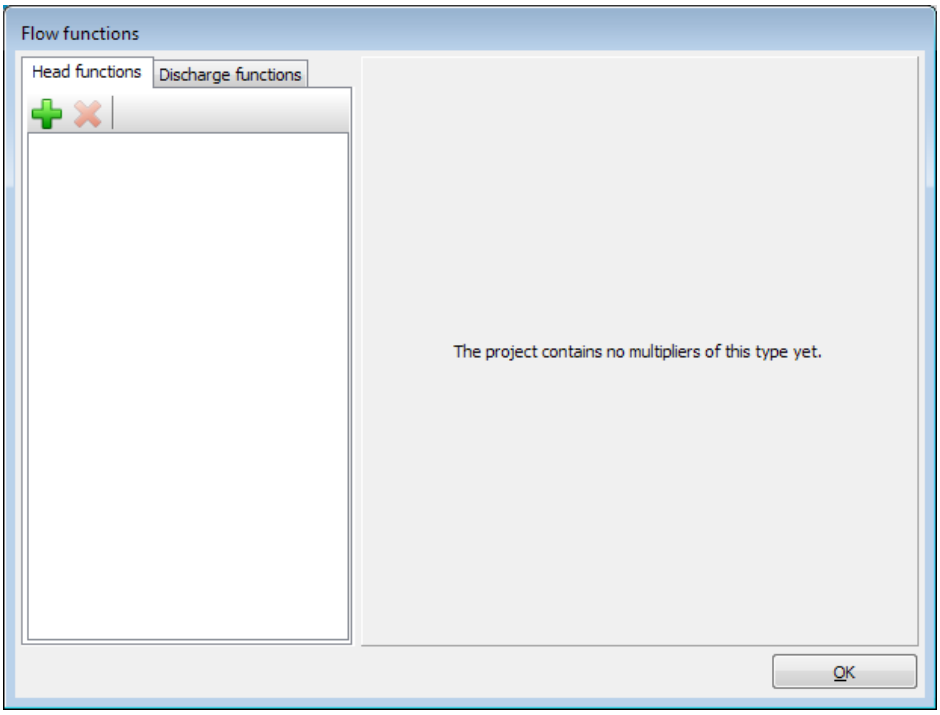


Figure 100: ***Flow functions*** *window*

The buttons available under the function tabs can be used to add new functions or to delete the function selected in the list. When a new function is added, the options to define it are displayed.

Name	The name of the flow function can be defined.
Signal	The type of the signal can be assigned, The options available are *Linear*, *Harmonic* and *Table*.

The options available in the *Signal* drop-down menu are described in the following sections.

5.11.1 Harmonic

This option can be used when a condition varies harmonically in time. The harmonic variation of the water level is generally described as:

$$y(t) = y_0 + A\sin(\omega_0 t + \phi_0) \quad with \quad \omega_0 = 2\pi / T \qquad Eq.\ [3]$$

in which A is the amplitude (in unit of length), T is the wave period (in unit of time) and φ_0 is the initial phase angle. y_0 is the centre of the harmonic variation, not necessarily equal to the water level at the end of the previous phase (y'_0). In fact the later is equal to:

$$y'_0 = y_0 + A\sin(\omega_0) \qquad Eq.\ [4]$$

Figure 101 (on page 143) shows the **Flow functions** window where a Harmonic signal is defined and plotted.

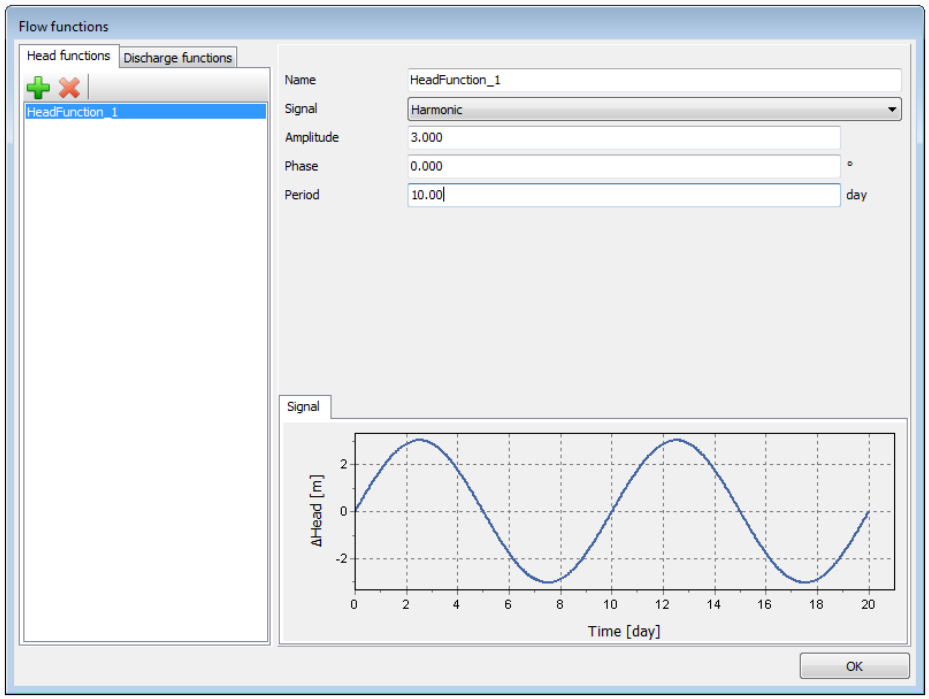


*Figure 101: Definition and display of a Harmonic signal in the **Flow functions** window*

5.11.2 Signal from table

Besides a harmonic signal there is also the possibility to define a signal by specifying the values in the table that appears when the corresponding option is selected in the *Signal* drop-down menu. The table consists of *Time* and *Head* columns. The buttons in the toolbar can be used to modify the table.

- Click the *Add row* button in the toolbar to add a new row in the table.
- Click the *Insert* button to insert a new row before the selected row in the table.
- Click the *Delete* button to delete the selected row in the table.

The values can be defined by clicking the cell in the table and by typing the value.

Importing data for signals

Besides defining the signal in the table, there is also the possibility to read data from a file with a digitised signal using the *Open* button in the toolbar.

Note: PLAXIS 2D assumes the data file is located in the current project directory when no directory is specified.

A signal, either defined in a table or received from a file and modified, can be saved using the *Save* button in the toolbar enabling the usage of the signal in other projects or validating the effect of the modifications in the current project.

A signal, either defined in a table or received from a file and modified, can be copied using the *Copy* button in the toolbar.

Copied data from other applications (using *Ctrl+c*) can be imported by using the *Paste* button. The **Import data** window appears (Figure 102 (on page 144)). The starting row of the data to be imported can be defined in the *From row* cell. The data and the plot is displayed in the **Flow functions** window after pressing *OK*.

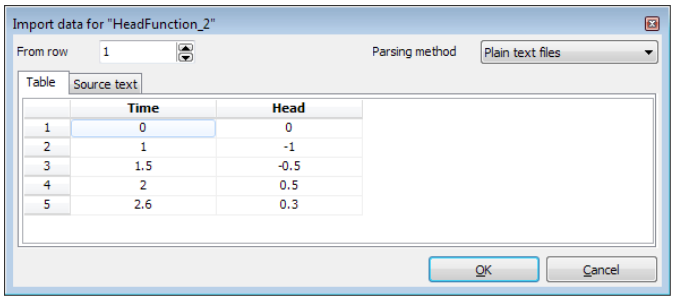


Figure 102: ***Import data*** *window for flow functions*

Clicking the *Open .txt file* button on the right hand side of the window will open the **Open** window where the file can be selected. The file must be an ASCII file that can be created with any text editor. For every line a pair of values (actual time and corresponding water level value) must be defined, leaving at least one space between them. Note that PLAXIS 2D only supports the English notation of decimal numbers using a dot. The resulting graph of the input data is shown in the **Graph** tabsheet of the **Time dependent condition** window.

5.11.3 Linear

This option can be used to describe the increase or decrease of a condition linearly in time. For a linear variation of groundwater head, the input of the following parameters are required:

$\Delta Head$	This parameter, specified in unit of length, represents the increase or decrease of the water level in the time interval for the current calculation phase. Hence, together with the time interval this parameter determines the rate of the water level increase or decrease.
t	This parameter represents the time interval for the calculation phase, expressed in units of time. Note that the time interval specified for a function does not effect the duration of a phase as defined in the **Phases** window.

For a linear variation of discharge infiltration, inflow or outflow the input of the following parameters are required for the discharge in the corresponding tabsheet:

$\Delta Discharge$	This parameter, specified in unit of volume per unit of time, per unit of width in the out-of-plane direction represents the increase or decrease of the specific discharge in the time interval of the current calculation phase.

Note: The time value in flow functions always refers to the global time in the whole list of calculation phases rather than to the time interval of an individual phase. This means that in a series of consecutive flow calculations each phase will only use its corresponding part of the flow function.

5.11.4 Flow functions in consecutive phases

A flow function can be assigned as an attribute to *Groundwater flow boundaries* or to *Water levels*. PLAXIS 2D ensures the continuity of the flow functions in consecutive phases when the corresponding options are selected in the **Phase definition** modes.

5.11.5 Function continuity for boundaries

Besides the options described in Groundwater flow boundary conditions (on page 134), two new options are added for boundary conditions in the **Phase definition** mode (Figure 103 (on page 145)). These options are:

Constant value from previous phase	The condition of the boundary in the current phase is a continuation of the state reached at the end of the previous phase and it will not change during the calculation of the phase.
Maintain function from the previous phase	The condition of the boundary in the current phase is equal to the state reached in the previous phase. PLAXIS 2D enables continuity of the state at the boundary according to the flow function assigned to the boundary in the previous phase. The accumulated time from the previous phases is used as an offset to enter the flow function for the current phase.

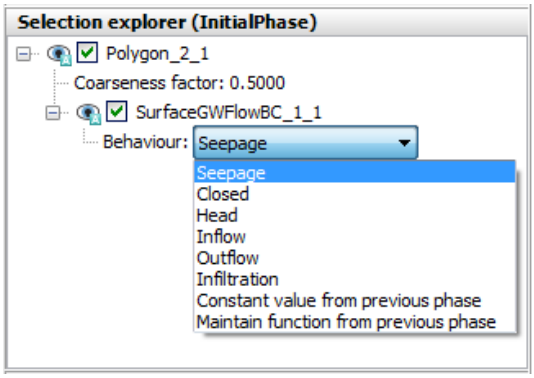


*Figure 103: Options for boundary condition behaviour in **Phase definition** modes*

5.11.6 Function continuity for water levels

The continuity of the flow functions assigned to water levels in consecutive phases can be maintained by using the options available in the pop up menu displayed when the water level is right-clicked. These options are:

Create the reached water level	When this option is selected a new water level is created at the position reached at the end of the previous phase. The location of the water level will be constant in the current phase.
Create the reached water level and continue	When this option is selected a new water level is created at the position reached at the end of the previous phase. The location of the water level in the current phase will change from that point on according to the flow function assigned to the original water level in the previous phase. PLAXIS 2D ensures the continuity of the flow function, whereby the accumulated time from previous phases is used as an offset to enter the flow function for the current phase.

5.12 Thermal conditions [ADV]

In case temperature (or actually a change in temperature) is of influence on the behaviour of soils or structures (for example thermal expansion or soil freezing), PLAXIS 2D offers the possibility to take thermal effects into account. The calculation of a temperature distribution in the ground is based on thermal calculations (Temperature calculation [ADV] (on page 324)). Since groundwater flow plays an essential role in the transport of heat in the ground, thermal calculations are generally coupled with groundwater flow calculations (thermo-hydraulic (TH) coupling). Moreover, results of coupled steady-state TH calculations may be used to analyse the effects of (changes of) temperature on stress and deformation (semi-coupled TH-M analysis). More generally, PLAXIS 2D allows for fully coupled transient thermo-hydro-mechanical (THM) calculations of problems in which the time-dependent effect of changes of temperature on stress, deformation and groundwater flow are to be taken into account simultaneously.

Thermal calculations require a set of thermal (boundary) conditions to be specified, which can be imposed on the model to define the temperature distribution in the geometry. *Thermal conditions* may involve cluster-related thermal conditions as well as line-based thermal flow boundary conditions (Thermal flow boundary conditions [ADV] (on page 147)).

The following cluster-related thermal conditions may be defined in terms of *Temperature conditions*:

None	This option shall be used if the temperature distribution in the selected cluster must be obtained from the global temperature distribution (using a reference temperature and an earth gradient) as defined under *Thermal Flow* in the *Model conditions* subtree of the *Model explorer*.
User-defined	The temperature in the cluster is directly defined on the basis of a reference temperature (T_{ref}) at a reference level (y_{ref}), and an increment per unit of depth (T_{inc}, y).

In addition to the above *Temperature conditions*, *energy conditions* may be defined for clusters that act as heat sources. In this case the *Energy conditions* parameter under *Thermal conditions* must be set to *Source* (i.e. a cluser producing energy) or *Sink* (i.e. a cluster absorbing energy). This allows the heat total flux, Q, to be specified in the unit of power per unit of volume. Optionally, the source can be made time-dependent by setting the *Time dependency* parameter to *Time-dependent*. The latter requires a *Heat total flux* function to be assigned, defined as a *Thermal function* in the *Attributes* library of the *Model explorer* (Thermal functions [ULT] (on page 154)).

Although the input values of thermal conditions are specified in the geometry model, the activation, deactivation or change of values may be considered in the framework of *Staged construction* (Changing geometry configuration (on page 359)).

Note:

- Cluster-related thermal conditions are only taken into account if the *Thermal* calculation type in the **Phases** window is set to *Steady state thermal flow* or *Transient thermal flow*.
- For calculations that use ***Steady state thermal flow*** an **Advanced entitlement** [ADV] is **required**, for ***Transient thermal or fully coupled*** *transient thermo-hydro-mechanical (THM) calculations* an **Ultimate entitlement** [ULT] is **necessary**.

5.12.1 *Thermal flow boundary conditions* [ADV]

In addition to groundwater flow boundary conditions (which are also required for thermal calculations, because of TH coupling), the creation of thermal flow boundary conditions is similar to the creation of a geometry line. Note that the assignment of thermal conditions is also possible by right-clicking an existing geometry line either in the drawing area or in the *Object explorers* and selecting the corresponding option in the appearing menu.

For transient thermal flow and fully coupled THM analysis, thermal conditions can be defined as a function of time by setting the *Time-dependency* parameter to *Time-dependent* and assigning a thermal function as defined under *Thermal function* in the *Attributes library* of the *Model explorer* (Thermal functions [ULT] (on page 154)).

For line-based thermal flow boundary conditions, the following options are available:

Closed

When the *Closed* option is assigned to a boundary, no heat flux can occur across it. In other words, the boundary is perfectly insulated. This boundary condition is similar to the 'natural' boundary condition. This means that if no thermal boundary condition is applied at a model boundary, or the thermal boundary condition is inactive, then the boundary is a closed thermal boundary. A closed thermal boundary is only taken into account at external model boundaries or at 'internal' boundaries between active and inactive clusters.

Temperature

A prescribed *Temperature* distribution may be entered for the thermal flow boundary in the unit of temperature. The options available for variation of temperature along the boundary are:

Uniform	A uniform value of the temperature (T_{ref}) is specified along the boundary.
Vertical increment	A vertical increment of the temperature along the boundary is specified. The parameters required to specify a vertical increment variation are: • T_{ref}: The reference temperature value at the specified reference level. • $T_{inc,y}$: Increment value specifying the rate of variation of temperature along the boundary. • y_{ref}: The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the temperature value below this level is changed according to $T(y) = T_{ref} + (y\text{-}y_{ref})\ T_{inc,y}$.

Horizontal increment	A horizontal increment of the temperature along the boundary is specified. The parameters required to specify a horizontal increment variation are: • T_{ref}: The reference temperature value at the specified reference level. • $T_{inc,x}$:Increment value specifying the rate of variation of temperature along the boundary. • x_{ref}:The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the temperature value below this level is changed according to $T(x)= T_{ref} + (x-x_{ref})\ T_{inc,x}$.
Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $T_{ref,start}$: The start value for the boundary condition. • $T_{ref,end}$: The end value for the boundary condition.

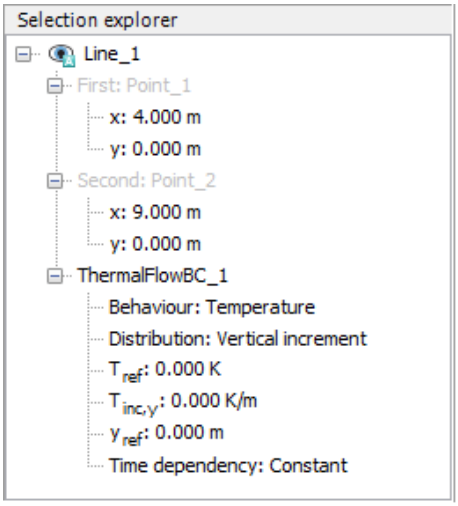


Figure 104: Temperature definition in the Selection explorer

The options available for *Time dependency* of *Temperature* assigned to a thermal flow boundary condition are:

Constant	The temperature in the thermal boundary does not change with time.
Time dependent [ULT]	The temperature in the thermal boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The thermal flow function representing the time dependency can be selected from the drop-down menu.

Inflow

A prescribed heat flux into the model may be entered in the unit of power per unit of length per unit of width in the out-of-plane direction. An *Inflow* boundary may be used as a heater (line source), adding energy to the model.

The options available are:

Uniform	A uniform value of the heat flux ($\|q_{ref}\|$) is specified along the boundary.
Vertical increment	A vertical increment of the heat flux along the boundary is specified. The parameters required to specify a vertical increment variation are: • $\|q_{ref}\|$: The reference heat flux value at the specified reference level in the unit of power per unit of time per unit of width in the out-of-plane direction. • $q_{inc,y}$: Increment value specifying the rate of variation of discharge along the boundary. • y_{ref}: The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the heat flux value below this level is changed according to $q(y) = q_{ref} + (y\text{-}y_{ref})\ q_{inc,y}$. Note that negative values of $q(y)$ are set to zero.
Horizontal increment	A horizontal increment of the heat flux along the boundary is specified. The parameters required to specify a horizontal increment variation are: • $\|q_{ref}\|$: The reference heat flux value at the specified reference level in the unit of power per unit of time per unit of width in the out-of-plane direction. • $q_{inc,x}$: Increment value specifying the rate of variation of heat flux along the boundary. • x_{ref}: The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the heat flux value below this level is changed according to $q(x) = q_{ref} + (x\text{-}x_{ref})\ q_{inc,x}$. Note that negative values of $q(x)$ are set to zero.
Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $q_{ref,start}$: The heat flux start value for the boundary condition. • $q_{ref,end}$: The heat flux end value for the boundary condition.

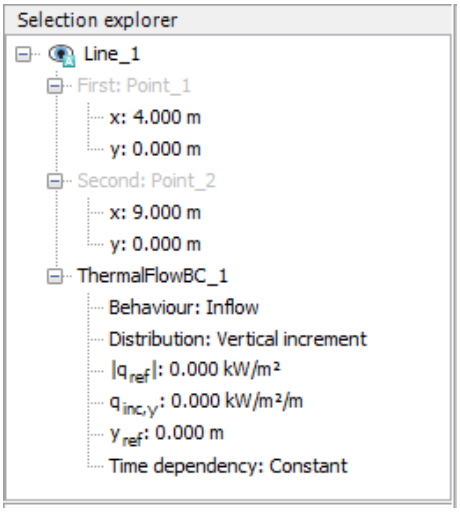


Figure 105: Inflow definition in the Selection explorer

Two options are available for the *Time dependency* of the heat flux:

Constant	The temperature in the thermal boundary does not change with time.
Time dependent [ULT]	The temperature in the thermal boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The thermal flow function representing the time dependency can be selected from the drop-down menu.

Outflow

A prescribed heat flux out of the model may be entered in the unit of power per unit of length per unit of width in the out-of-plane direction. An *Outflow* boundary may be used as a cooler (line sink), absorbing energy from the model. The options available are:

Uniform	A uniform value of the heat flux ($\|q_{ref}\|$) is specified along the boundary.
Vertical increment	A vertical increment of the heat flux along the boundary is specified. The parameters required to specify a vertical increment variation are: • $\|q_{ref}\|$: The reference heat flux value at the specified reference level in the unit of power per unit of time per unit of width in the out-of-plane direction. • $q_{inc,y}$: Increment value specifying the rate of variation of discharge along the boundary. • y_{ref}: The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the heat flux value below this level is changed according to $q(y) = q_{ref} + (y-y_{ref})\ q_{inc,y}$. Note that negative values of *q(y)* are set to zero.
Horizontal increment	A horizontal increment of the heat flux along the boundary is specified. The parameters required to specify a horizontal increment variation are: • $\|q_{ref}\|$: The reference heat flux value at the specified reference level in the unit of power per unit of time per unit of width in the out-of-plane direction. • $q_{inc,x}$: Increment value specifying the rate of variation of heat flux along the boundary. • x_{ref}: The reference value for which the reference temperature value is specified. If this value is lower than the top of the boundary, the heat flux value below this level is changed according to $q(x) = q_{ref} + (x-x_{ref})\ q_{inc,x}$. Note that negative values of *q(x)* are set to zero.
Start/end values	A specific start and end value is given along the boundary. The parameters required to specify such a variation are: • $q_{ref,start}$: The heat flux start value for the boundary condition. • $q_{ref,end}$: The heat flux end value for the boundary condition.

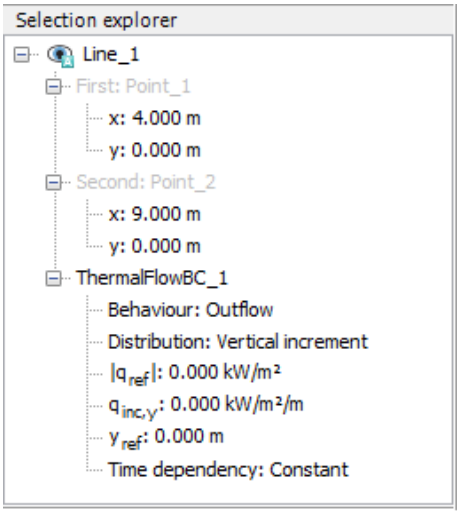


Figure 106: Outflow definition in the Selection explorer

Two options are available for the *Time dependency* of the heat flux:

Constant	The outflow in the thermal boundary does not change with time.
Time dependent [ULT]	The outflow in the thermal boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The thermal flow function representing the time dependency can be selected from the drop-down menu.

Convection

A *Convection* thermal boundary is a convective boundary at which a medium (denoted as a 'fluid') with a certain temperature, T_{fluid}, is present, whilst the medium transfers its temperature on to that boundary (i.e. the adjacent material).

Examples where *Convection* boundaries may be applied are:

- At the ground surface, the air temperature influences the temperature of the ground, but the ground surface temperature is not necessarily equal to the air temperature. This situation may be conveniently modelled using the *Climate* condition in the *Model conditions* tree in the *Model explorer*, but the actual boundary condition at the ground surface in this case is a *Convection* boundary condition.
- Another example is a freeze pipe or a borehole heat exchanger in which a refrigerant fluid transfers its temperature on to the surrounding soil. In such cases the 'boundary'is inside the model. Hence, a *Convection* boundary may be applied at a 'real' geometry boundary as well as at a geometry line inside the model.

In addition to the temperature of the medium, T_{fluid} (expressed in the unit of temperature), a heat transfer coefficient (the inverse of thermal resistance) is required. The transfer coefficient is entered in the unit of power per unit of length per unit of width in the out-of-plane direction per unit of temperature. The higher the transfer coefficient, the more heat is transferred from the medium at the boundary on to that boundary. If the transfer coefficient is zero, no heat is transferred on to the boundary, which means that the boundary condition is inactive. If the transfer coefficient would be infinite, the *Convection* boundary condition would be equivalent to a *Temperature* boundary condition.

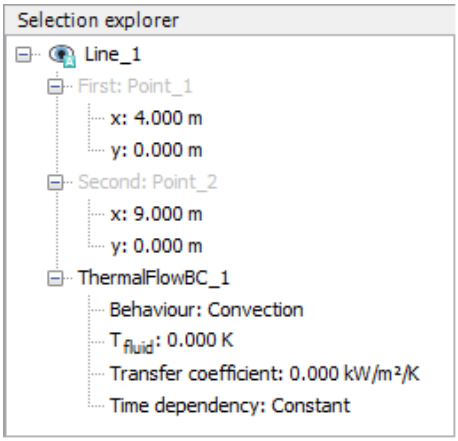


Figure 107: Convection definition in the Selection explorer

Two options are available for the *Time dependency* of T_{fluid}:

Constant	The convection in the thermal boundary does not change with time.
Time dependent [ULT]	The convection in the thermal boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The thermal flow function representing the time dependency can be selected from the drop-down menu.

Thermosyphons

Thermosyphons boundary condition, like *Convection* boundary condition, requires a heat transfer coefficient and the temperature of the medium, T_{fluid} (which is the air temperature for *Thermosyphons*). Unlike *Convection* boundary condition, *Thermosyphons* stops working and becomes a closed boundary when:

- The air temperature is higher than a limit $T_{fluid,max}$.
- The difference between the air temperature and ground temperature is smaller than the threshold value ΔT.

Thermosyphons may be applied to model the irreversible heat exchange between ground and thermosyphon systems at their boundary. Thermosyphon systems are passive refrigeration devices helping to maintain frozen ground states (permafrost) to support buildings, pipelines, railroad embankments and highways. They are charged with a working fluid circulating due to the natural convection. In the winter, the ground temperature is higher than the air temperature, so the fluid at the base of thermosyphons is heated, vaporises and moves upward to a radiator. There, the fluid is cooled down by the colder air, condenses and moves downward. This loop transfers heat from the ground to the air as long as appropriate temperature difference prevails; otherwise, the system stops working. Therefore, the cycling ceases in the summer and the permafrost is preserved.

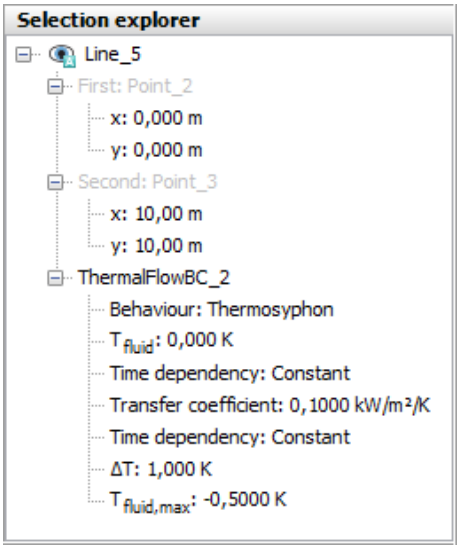


Figure 108: Thermosyphons definition in the Selection explorer

Two options are available for the *Time dependency* of $T_{fluid,max}$ and the heat transfer coefficient:

Constant	The convection in the thermal boundary does not change with time.
Time dependent [ULT]	The convection in the thermal boundary varies with time. This option is only available for transient flow and fully coupled flow-deformation analysis. The thermal flow function representing the time dependency can be selected from the drop-down menu.

Note: The thermal flow boundary conditions are visualised in the *Flow conditions* mode. Before starting a thermal calculation it is good practice to visually check the applied conditions by selecting the *Flow conditions* mode.

Constant value from previous phase

Although this option is always available, it may be selected particularly in the case that the previous calculation phase (parent phase) was a transient heat flow calculation or a fully coupled THM calculation, and the condition at the boundary to be considered was defined using a *Thermal function*. When selected, the heat flow boundary condition is defined as a constant (time independent) condition based on the conditions at the *end of the parent phase* (considering the assigned *Thermal function* and the time interval of the parent phase). If the previous phase was time independent (time interval of 0) or the boundary condition was constant, it will simply adopt the boundary condition from the previous phase. If the boundary condition was inactive in the previous phase, it remains inactive in the current phase, which means that the boundary is closed (zero flux across the boundary).

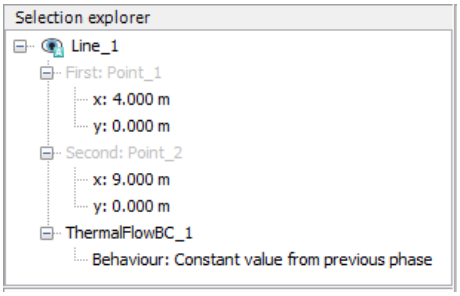


Figure 109: Constant value from previous phase definition in the Selection explorer

Maintain function from previous phase

Although this option is always available, it may be selected particularly in the case that the previous calculation phase (parent phase) was a transient heat flow calculation or a fully coupled THM calculation, and the condition at the boundary to be considered was defined using a *Thermal function*. When selected, the heat flow boundary condition is defined as a time-dependent condition, following the *Thermal function* (*Temperature* function, *Heat flux* function, or *Heat total flux* function) as defined in the parent phase. In this case, the *Thermal function* is entered at the start time of the current phase (= reached end time of parent phase).

If the previous phase was time independent (time interval of 0) or the boundary condition was constant, it will simply adopt the boundary condition from the previous phase. If the boundary condition was inactive in the previous phase, it remains inactive in the current phase, which means that the boundary is closed (zero flux across the boundary).

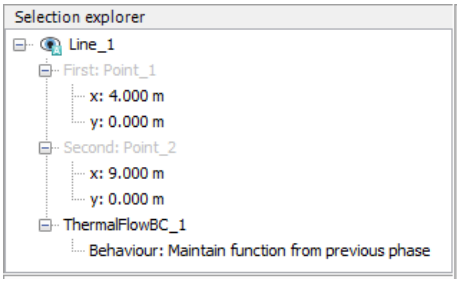


Figure 110: Maintain function from previous phase definition in the Selection explorer

5.13 Thermal functions [ULT]

Thermal functions describing the variation with time of quantities such as *Temperature* and *Head flux* can be defined in the corresponding subtree under the *Attributes library* in the *Model explorer* (Figure 111 (on page 155)).

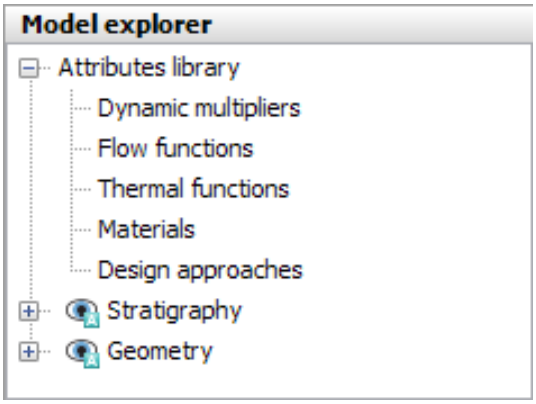


Figure 111: Thermal functions subtree in Model explorer

Thermal functions can be assigned to *Thermal flow boundary conditions* or *Soil clusters*. Note that time-dependent conditions can be defined for *Temperature*, *Inflow*, *Outflow* and *Convection* . In the latter case, the time-dependency is applied on the medium that causes the thermal convection (fluid, air) (Thermal flow boundary conditions [ADV] (on page 147)). Functions can be defined in the corresponding window that pops up when the *Thermal functions* subtree is right-clicked and the *Edit* option is selected in the appearing menu. The window consists of three tabsheets: i.e. *Temperature functions*, *Heat flux functions* and *Heat total flux functions*. Here, the functions applicable to temperature, heat flux and total heat flux can be defined, respectively (Figure 112 (on page 155)).

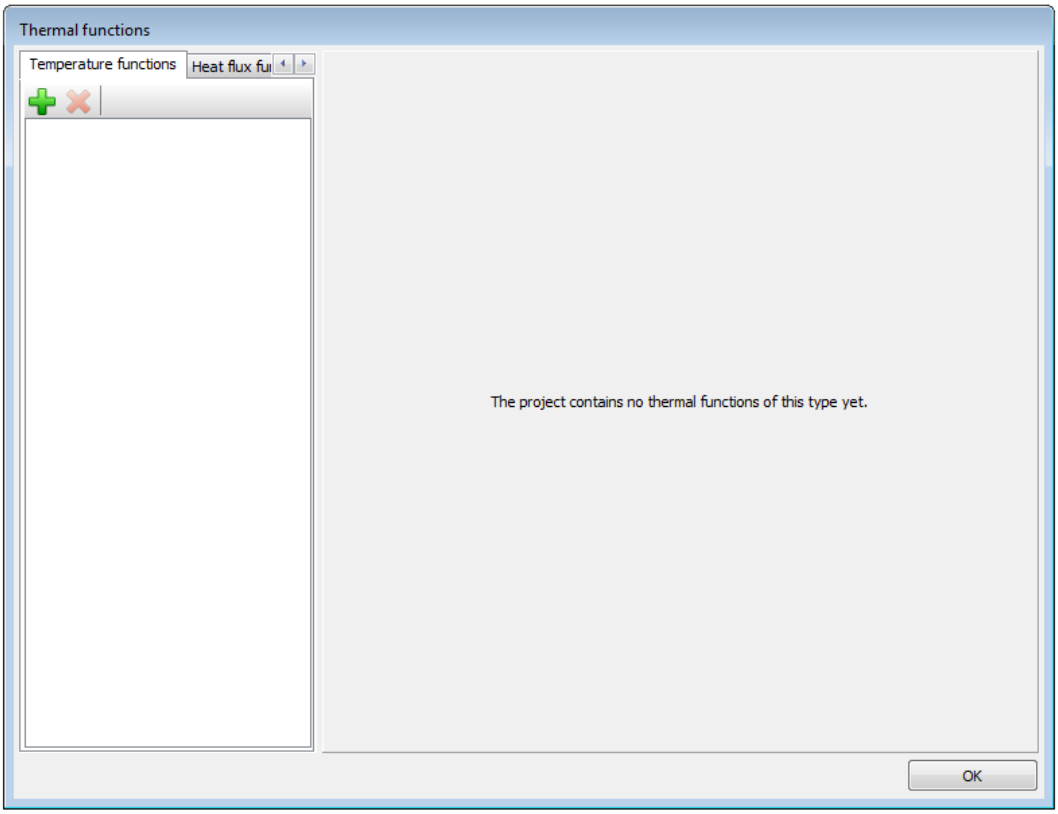


Figure 112: ***Thermal functions*** *window*

A *Temperature* function can be assigned to thermal flow boundary conditions as well as to the air temperature as defined in the *Climate* model condition under the *Model conditions* subtree. Temperature is defined in the unit of Temperature.

A *Heat flux* function can be assigned to line-based thermal flow boundary conditions. Heat flux is defined in the unit of power per unit of length (along the boundary line) per unit of width in the out-of-plane direction. A *Heat total flux* function can be assigned to soil clusters in which the energy condition under *Thermal* condition is set to *Source* or *Sink*. Heat total flux is defined in the unit of power per unit of volume. The definition of *Thermal functions* is similar as the definition of *Flow functions* (Flow functions [ULT] (on page 141)).

5.13.1 Harmonic

This option can be used when a condition varies harmonically in time. The harmonic variation of the water level is generally described as:

$$y(t) = y_0 + A\sin\left(\omega_0 t + \phi_0\right) \quad with \quad \omega_0 = 2\pi / T \qquad Eq.\ [5]$$

in which A is the amplitude (in unit of length), T is the wave period (in unit of time) and φ_0 is the initial phase angle. y_0 is the centre of the harmonic variation, not necessarily equal to the water level at the end of the previous phase (y'_0). In fact the later is equal to:

$$y'_0 = y_0 + A\sin\left(\omega_0\right) \qquad Eq.\ [6]$$

Figure 113 (on page 156) shows the **Thermal functions** window where a Harmonic signal is defined and plotted.

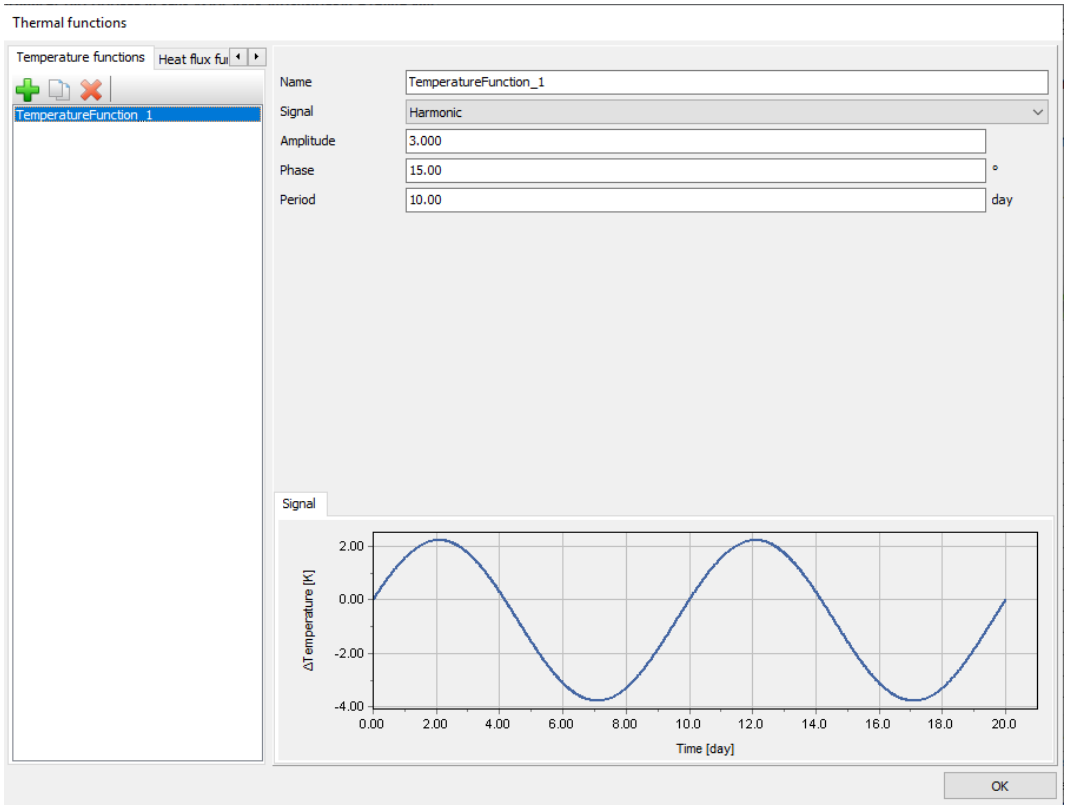


*Figure 113: Definition and display of a Harmonic signal in the **Thermal functions** window*

Besides a harmonic signal there is also the possibility to define a signal by specifying the values in the table that appears when the corresponding option is selected in the *Signal* drop-down menu. The table consists of *Time* and *Temperature* columns. The buttons in the toolbar can be used to modify the table.

- Click the *Add row* button in the toolbar to add a new row in the table.
- Click the *Insert* button to insert a new row before the selected row in the table.
- Click the *Delete* button to delete the selected row in the table.

The values can be defined by clicking the cell in the table and by typing the value.

5.13.2 Importing data for signals

Besides defining the signal in the table, there is also the possibility to read data from a file with a digitised signal using the *Open* button in the toolbar.

Note: PLAXIS 2D assumes the data file is located in the current project directory when no directory is specified.

A signal, either defined in a table or received from a file and modified, can be saved using the *Save* button in the toolbar enabling the usage of the signal in other projects or validating the effect of the modifications in the current project.

A signal, either defined in a table or received from a file and modified, can be copied using the *Copy* button in the toolbar.

Copied data from other applications (using *Ctrl+c*) can be imported by using the *Paste* button. The **Import data** window appears (Figure 114 (on page 158)). The starting row of the data to be imported can be defined in the *From row* cell. The data and the plot is displayed in the **Thermal functions** window after pressing *OK*.

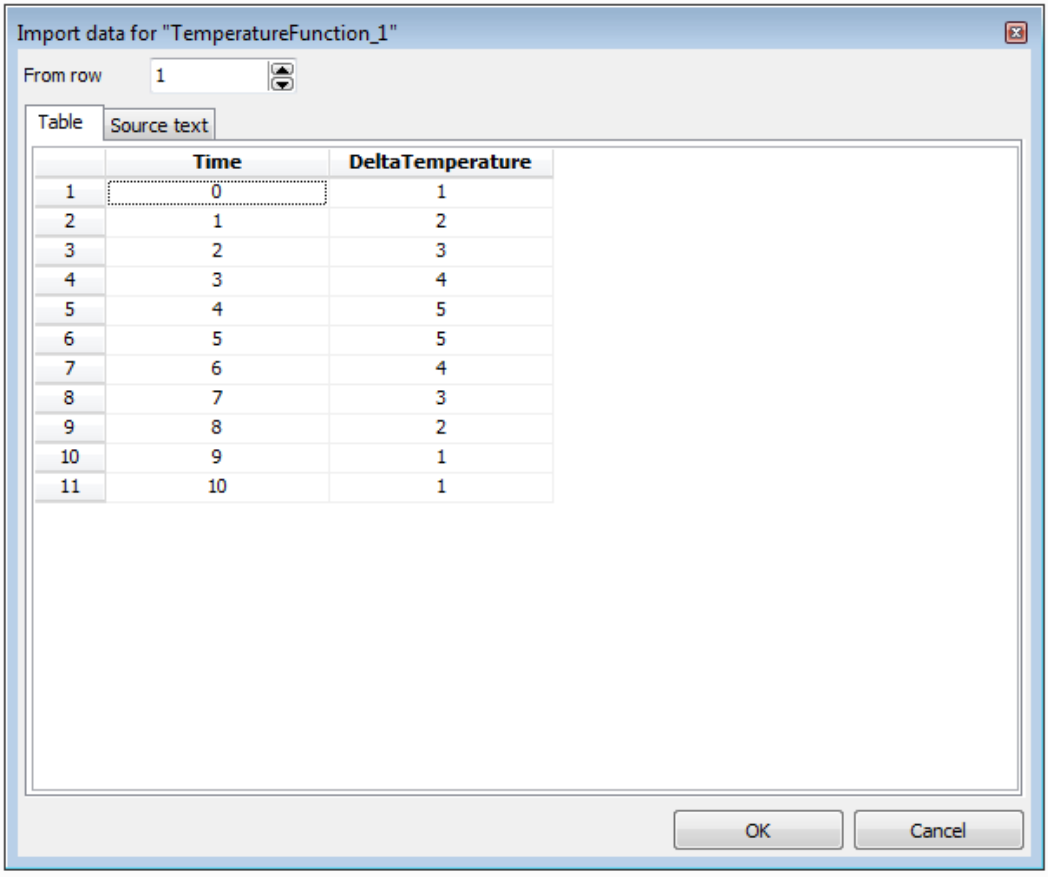


Figure 114: ***Import data*** *window for Thermal functions*

Clicking the *Open .txt file* button on the right hand side of the window will open the **Open** window where the file can be selected. The file must be an ASCII file that can be created with any text editor. For every line a pair of values (actual time and corresponding water level value) must be defined, leaving at least one space between them. Note that PLAXIS 2D only supports the English notation of decimal numbers using a dot. The resulting graph of the input data is shown in the **Graph** tabsheet of the **Time dependent condition** window.

5.13.3 Linear

This option can be used to describe the increase or decrease of a condition linearly in time. For a linear variation of groundwater Temperature, the input of the following parameters are required:

Δ*Temperature*	This parameter, specified in unit of temperature, represents the increase or decrease of the temperature in the time interval for the current calculation phase. Hence, together with the time interval this parameter determines the rate of the temperature increase or decrease.
Time	This parameter represents the time interval for the calculation phase, expressed in unit of time. Note that the time interval specified for a function does not effect the duration of a phase as defined in the **Phases** window.

For a linear variation of inflow or outflow the input of the following parameters are required for the heat flux and total heat flux in the corresponding tabsheet:

Δ*HeatFlux*	This parameter, specified in unit of power per unit of area. It represents the increase or decrease of the flux in the time.
Δ*HeatTotalFlux*	This parameter, specified in unit of power per unit of volume. It represents the increase or decrease of the flux with time.

Note: The time value in thermal functions always refers to the local time interval of each phase. This means that the time in these functions is always reset to zero at the beginning of each phase which is using them.

6 Material properties and material database

In PLAXIS 2D, soil properties and material properties of structures are stored in material data sets. There are six different types of material sets grouped as data sets for soil and interfaces, discontinuities, plates, geogrids, embedded beams and anchors. All data sets are stored in the material database. From the database, the data sets can be assigned to the soil clusters or to the corresponding structural objects in the geometry model.

The material database can be opened by selecting the option *Show materials* in the *Soil* menu of the **Soil** mode or in the *Structures* menu of the **Structures** mode. Another way to activate the material database is by clicking on the *Show materials* button available in the **Soil**, **Structures** and **Staged construction** modes. As a result, the **Material sets** window appears showing the contents of the project material database. The window can be expanded to show the global database by clicking the *Show global* button in the upper part of the window. The **Material sets** window displaying the material defined in the current project and the ones available in a selected global database is shown in Figure 115 (on page 160).

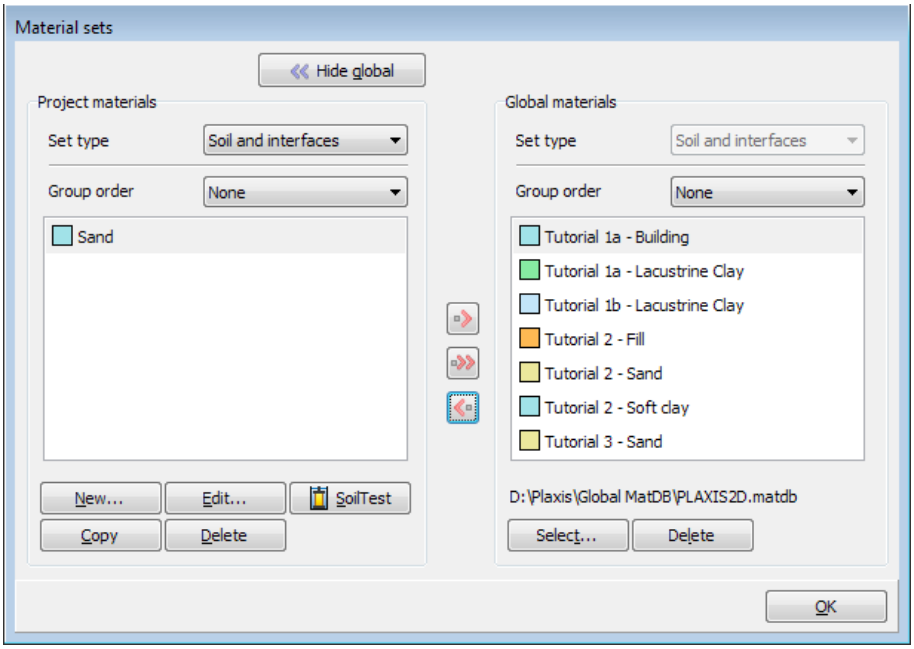


*Figure 115: **Material sets** window showing the project and the global database*

The database of a new project is empty. The global database can be used to store material data sets in a global folder and to exchange data sets between different projects.

At both sides of the window (**Project materials** and **Global materials**) there are two drop-down menus and a tree view. The *Set type* can be selected from the drop-down menu on the left hand side. The *Set type* parameter

determines which type of material data set is displayed in the tree view (*Soil and interfaces*, *Discontinuities*, *Plates*, *Geogrids*,)*Embedded beams*, *Anchors*).

The data sets in the tree view are identified by a user-defined name. The data sets for *Soil and interfaces* can be ordered in groups according to the material model, the material type or the name of the data set by selecting this order in the *Group order* drop-down menu. The *None* option can be used to discard the group ordering.

The small buttons between the two tree views can be used to copy individual data sets from the project database to the selected global database or vice versa.

- To copy the selected project material set to the global database.
- To copy all the project material sets of the specified type to the global database.
- To copy the selected global material set to the project database.

The location of the selected global database is shown below its tree view. The buttons below the tree view of the global database enable actions in the global database.

Select	To select an existing global database.
Delete	To delete a selected material data set from the selected global database.

By default, the global database for soil and interface data contains the data sets of all the tutorials and it is contained in the file 'Soil.matXdb'. This file is compatible with other PLAXIS 2D database files for soil and interfaces and is stored in the installation folder of PLAXIS 2D. Material data sets for structural elements will be contained in separate files. Similarly, the global databases for Discontinuities, Plates, Geogrids, Embedded beams and Anchors are contained in the files 'Discontinuity.matXdb', 'Plate.matXdb', 'PlateMat2D.matdb', 'Geogrid.matXdb', 'EmbeddedBeam.matXdb' and 'Anchor.matXdb' respectively.

Note:

A new global database can be created by clicking the *Select* button, defining the name of the new global database and clicking *Open*.

The project data base can be managed using the buttons below the tree view of the project database.

New	To create a new data set in the project. As a result, a new window appears in which the material properties or model parameters can be entered. The first item to be entered is always the *Identification*, which is the user-defined name of the data set. After completing a data set, it will appear in the tree view, indicated by its name as defined by the *Identification*.
Edit	To modify the selected data set in the project material database.
SoilTest	To perform standard soil lab tests. A separate window will open where several basic soil tests can be simulated and the behaviour of the selected soil material model with the given material parameters can be checked (Simulation of soil lab tests [GSE] (on page 482)).
Copy	To create a copy of a selected data set in the project material database.
Delete	To delete a selected material data set from the project material database.

Note:

In a **Material dataset** window (Material properties and material database (on page 160)) it is possible to go through the options with arrows and/or *Enter*. When a property such as *Material model* or *Drainage type*, is reached, the drop-down menu can be activated by pressing the *Space* key. Arrows and /or letters can be used to make a selection, that is finalised by pressing *Enter*.

6.1 Modelling soil and interface behaviour

The material properties and model parameters for soil clusters are entered in material data sets Figure 116 (on page 163). The properties in the data sets are divided into six tabsheets: **General**, **Mechanical**, **Groundwater**, **Thermal**, **Interfaces** and **Initial**.

6.1.1 *General tabsheet*

The **General** tabsheet contains the type of soil model, the drainage type and the general soil properties such as unit weights. Several data sets may be created to distinguish between different soil layers. A user may specify any identification title for a data set in the **General** tabsheet of the **Soil** window. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.

For easy recognition in the model, a colour is given to a certain data set. This colour also appears in the database tree view. PLAXIS 2D selects a unique default colour for a data set, but this colour may be changed by the user. Changing the colour can be done by clicking on the colour box in the **General** tabsheet.

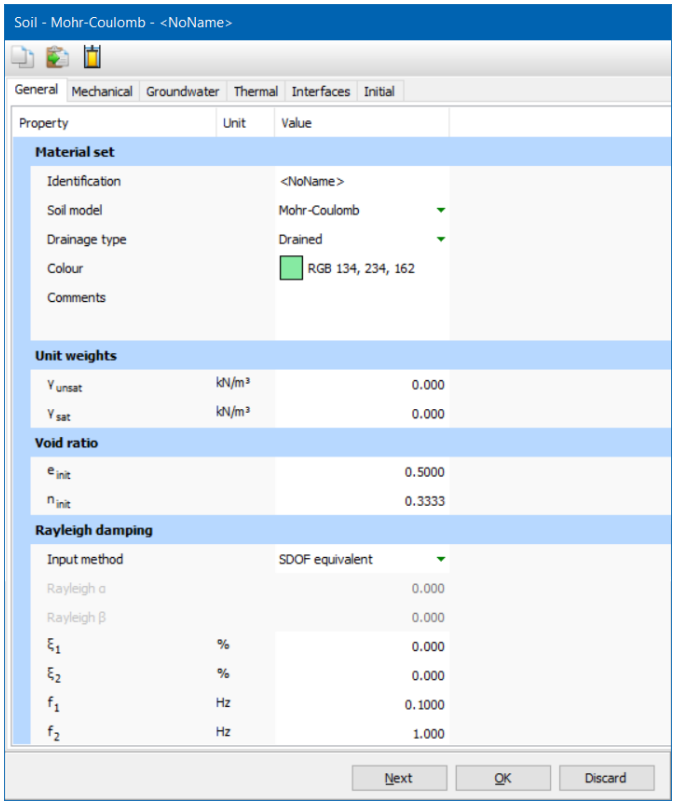


*Figure 116: **General** tabsheet of the **Soil** window*

Material model

Soil and rock tend to behave in a highly non-linear way under load. This non-linear stress-strain behaviour can be modelled at several levels of sophistication. Clearly, the number of model parameters increases with the level of sophistication. PLAXIS 2D supports different models to simulate the behaviour of soil and other continua. The models and their parameters are described in detail in the Material Models Manual. A short discussion of the available models is given below:

Linear elastic model: This model represents Hooke's law of isotropic linear elasticity. The linear elastic model is too limited for the simulation of soil behaviour. It is primarily used for stiff structures in the soil.

Mohr-Coulomb model (MC): This well-known linear elastic perfectly-plastic model is used as a first approximation of soil behaviour in general. It is recommended to use this model for a first analysis of the problem considered. A constant average stiffness is estimated for the soil layer. Due to this constant stiffness, computations tend to be relatively fast and a first estimate of deformations can be obtained.

Hardening Soil model (HS): This is an advanced model for the simulation of soil behaviour. The Hardening Soil model is an elastoplastic type of hyperbolic model, formulated in the framework of shear hardening plasticity. Moreover, the model involves compression hardening to simulate irreversible compaction of soil under primary compression. This second-order model can be used to simulate the behaviour of sands and gravel as well as softer types of soil such as clays and silts.

Hardening Soil model with small-strain stiffness (HSsmall): This is an elastoplastic type of hyperbolic model, similar to the Hardening Soil model. Moreover, this model incorporates strain dependent stiffness moduli, simulating the different reaction of soils from small strains (for example vibrations with strain levels below 10^{-5}) to large strains (engineering strain levels above 10^{-3}).

Modified Cam-Clay model (MCC): This well-known critical state model can be used to simulate the behaviour of normally consolidated soft soils. The model assumes a logarithmic relationship between the void ratio and the mean effective stress.

Hoek-Brown model (HB): This well-known elastic perfectly-plastic model is used to simulate the isotropic behaviour of rock. A constant stiffness is used for the rock mass. Shear failure and tension failure are described by a non-linear strength curve.

Jointed Rock model (JR): This is an anisotropic elastic-perfectly plastic model where plastic shearing can only occur in a limited number of shearing directions. This model can be used to simulate the anisotropic behaviour of stratified or jointed rock.

NGI-ADP model (NGI-ADP): The NGI-ADP model may be used for capacity, deformation and soil-structure interaction analysis involving undrained loading of clay-type materials. Distinct anisotropic shear strengths may be defined for different stress paths.

[ADV] **Soft soil model (SS):** This is a Cam-Clay type model that can be used to simulate the behaviour of soft soils like normally consolidated clays and peat. The model performs best in situations of primary compression.

[ADV] **Soft Soil Creep model (SSC):** This is a second order model formulated in the framework of viscoplasticity. The model can be used to simulate the time-dependent behaviour of soft soils like normally consolidated clays and peat. The model includes logarithmic primary and secondary compression.

[ADV] **UDCAM-S :** The UDCAM-S model is a derived NGI-ADP model to deal with undrained soil behaviour and degradation of the strength and stiffness in cyclic loading of clay or very low permeable silty soils. It implements a pre-processing procedure called *Cyclic accumulation and optimisation tool* (Cyclic accumulation and optimisation tool [ADV] (on page 209)) to obtain the degraded parameter set based on the type of analysis.

[ADV] **Sekiguchi-Ohta model (Inviscid):** The Sekiguchi-Ohta model (Inviscid) is a Cam-Clay type effective stress model for time-independent behaviour of clay-type soils.

[ADV] **Sekiguchi-Ohta model (Viscid):** The Sekiguchi-Ohta model (Viscid) is a Cam-Clay type effective stress model for time-dependent behaviour (creep) behaviour of clay-type soils.

[ADV] **Concrete model:** The Concrete modelis an advanced elastoplastic model for concrete and shotcrete structures. It simulates the time-dependent strength and stiffness of concrete, strain hardening-softening in compression and tension as well as creep and shrinkage. The failure criterion involves a Mohr-Coulomb yield surface for deviatoric loading, which is combined with a Rankine yield surface in the tensile regime.

[ULT] **UBC3D-PLM :** This is an advanced model for the simulation of liquefaction behaviour in dynamics applications. The model includes accumulation of irreversible strains during cyclic loading. In combination with undrained behaviour it accumulates pore pressures, which may eventually lead to liquefaction.

[ADV]/[ULT]+[GSE] **User-defined Soil Models (UDSM):** With this option it is possible to use other constitutive models than the standard PLAXIS 2D models. For a detailed description of this facility, reference is made to the Material Models Manual. Links to existing UDSM as well as all models are available on the PLAXIS Knowledge Base.

Drainage type

In principle, all model parameters in PLAXIS 2D are meant to represent the effective soil response, i.e. the relationship between the stresses and the strains associated with the soil skeleton. An important feature of soil is the presence of pore water. Pore pressures significantly influence the (time-dependent) soil response. PLAXIS 2D offers several options to enable incorporation of the water-skeleton interaction in the soil response. The most advanced option is a *Fully coupled flow-deformation* analysis. However, in many cases it is sufficient to analyse either the long-term (drained) response or the short-term (undrained) response without considering the time-dependent development of pore pressures. In the latter case (undrained), excess pore pressures are

generated as a result of stress changes (loading or unloading). The dissipation of these excess pore pressures with time can be analysed in a *Consolidation* calculation.

The simplified water-skeleton interaction, as considered in a *Plastic* calculation, a *Safety* analysis or a *Dynamics* analysis, is defined by the *Drainage type* parameter. PLAXIS 2D offers a choice of different types of drainage:

Drained behaviour: Using this setting no excess pore pressures are generated. This is clearly the case for dry soils and also for full drainage due to a high permeability (sands) and/or a low rate of loading. This option may also be used to simulate long-term soil behaviour without the need to model the precise history of undrained loading and consolidation.

Undrained behaviour: This setting is used for saturated soils in cases where pore water cannot freely flow through the soil skeleton. Flow of pore water can sometimes be neglected due to a low permeability (clays) and/or a high rate of loading. All clusters that are specified as undrained will indeed behave undrained, even if the cluster or a part of the cluster is located above the phreatic level.

Distinction is made between three different methods of modelling undrained soil behaviour. Method A is an undrained effective stress analysis with effective stiffness as well as effective strength parameters. This method will give a prediction of the pore pressures and the analysis can be followed by a consolidation analysis. The undrained shear strength (s_u) is a consequence of the model rather than an input parameter. It is recommended to check this shear strength with known data. To consider this type of analysis, the *Undrained A* option should be selected in the *Drainage type* drop-down menu.

Method B is an undrained effective stress analysis with effective stiffness parameters and undrained strength parameters. The undrained shear strength s_u is an input parameter. This method will give a prediction of pore pressures. However, when followed by a consolidation analysis, the undrained shear strength (s_u) is not updated, since this is an input parameter. To consider this type of analysis, the *Undrained B* option should be selected in the *Drainage type* drop-down menu.

Method C is an undrained total stress analysis with all parameters undrained. This method will not give a prediction of pore pressures. Therefore it is not useful to perform a consolidation analysis. The undrained shear strength (s_u) is an input parameter. To consider this type of analysis, the *Undrained C* option should be selected in the *Drainage type* drop-down menu.

More information about modelling undrained behaviour can be found in Modelling undrained behaviour (on page 255) and the Material Models Manual.

Non-porous behaviour: Using this setting neither initial nor excess pore pressures will be taken into account in clusters of this type. Applications may be found in the modelling of concrete or structural behaviour. *Non-porous* behaviour is often used in combination with the *Linear elastic* model. The input of a saturated weight is not relevant for non-porous materials or intact rock.

In a consolidation analysis or a fully coupled flow-deformation analysis it is the permeability parameter in the **Flow** tabsheet that determines the drainage capacity of a layer rather than the drainage type. Still, the drainage type has influence on the applied compressibility of water in a consolidation analysis or a fully coupled flow-deformation analysis. For more information see Appendix C.

Note:

The *Drainage type* setting is only considered in a *Plastic* calculation, a *Safety* analysis or a *Dynamics* analysis. When a *Consolidation* analysis, a *Fully coupled flow-deformation* analysis or a *Dynamics with Consolidation* analysis is performed, the *Drainage type* is ignored and the soil response is determined by the *Permeability* of the material.

Unsaturated and saturated weight (γ_{unsat} and γ_{sat})

The unsaturated and the saturated weights, entered as a force per unit volume, refer to the total unit weight of the soil skeleton including the fluid in the pores. The unsaturated weight γ_{unsat} applies to all material above the phreatic level and the saturated unit weight γ_{sat} applies to all material below the phreatic level, where the phreatic level itself is generally defined as the level where the steady-state pore pressure is zero ($p_{steady} = 0$). Only in the case of a fully coupled flow-deformation analysis, the phreatic level is defined as the level where the current pore water pressure is zero ($p_{water} = 0$). This means that during a fully coupled flow-deformation analysis the position of the phreatic level and hence the material weight can change.

For non-porous material only the unsaturated weight is relevant, which is just the total unit weight. For porous soils the unsaturated weight is obviously smaller than the saturated weight. For sands, for example, the saturated weight is generally around 20 kN/m^3 whereas the unsaturated weight can be significantly lower, depending on the degree of saturation.

Note that soils in practical situations are never completely dry. Hence, it is advisable not to enter the fully dry unit weight for γ_{unsat}. For example, clays above the phreatic level may be almost fully saturated due to capillary action. Other zones above the phreatic level may be partially saturated. PLAXIS 2D can deal with partially saturated soil behaviour above the phreatic level. However, the unit weight of soil is always defined here by γ_{unsat}, irrespective of the degree-of-saturation.

Weights are activated by means of *Gravity loading* or *K_0 procedure* in the *Calculation* mode, which is always the first calculation phase (Initial phase) (see Initial stress generation (on page 300)).

Void ratio and porosity

The void ratio, *e*, is related to the porosity, *n* (*e = n / (1-n)*). This quantity is used in some special options. The initial value e_{init} is the value in the initial situation. The actual void ratio is calculated in each calculation step from the initial value and the volumetric strain $\Delta\varepsilon_v$. These parameters are used to calculate the change of permeability when input is given for the c_k value (in the **Flow** tabsheet).

Rayleigh damping [ULT]

Material damping in dynamics calculations is caused by the viscous properties of soil, friction and the development of irreversible strains. All plasticity models in PLAXIS 2D can generate irreversible (plastic) strains, and may cause material damping. However, this damping is generally not enough to model the damping characteristics of real soils. For example, most soil models show pure elastic behaviour upon unloading and reloading which does not lead to damping at all. Some models in PLAXIS 2D include viscous behaviour, for instance the Soft Soil Creep model. Using this model in dynamics calculations may lead to viscous damping, but also the Soft Soil Creep model hardly shows any creep strain in load / reload cycles. There are also models in PLAXIS 2D that include hysteretic behaviour in loading / reload cycles, like the Hardening Soil model with small-strain stiffness (Material Models Manual - *The Hardening Soil model with small-strain*). When using this model, the amount of damping that is obtained depends on the amplitude of the strain cycles. Considering very small vibrations, even the Hardening Soil model with small-strain stiffness does not show material damping as well as numerical damping, whereas real soils still show a bit of viscous damping. Hence, additional damping is needed to model realistic damping characteristics of soils in dynamics calculations. This can be done by means of Rayleigh damping.

Rayleigh damping is a numerical feature in which a damping matrix *C* is composed by adding a portion of the mass matrix *M* and a portion of the stiffness matrix *K*:

$$[C] = \alpha[M] + \beta[K] \qquad Eq.\ [7]$$

The parameters α and β are the Rayleigh coefficients and can be specified in the corresponding cells in the **General** tabsheet of the **Soil** window shown in Figure 117.

α is the parameter that determines the influence of mass in the damping of the system. The higher α is, the more the lower frequencies are damped. β is the parameter that determines the influence of stiffness in the damping of the system. The higher β is, the more the higher frequencies are damped. In PLAXIS 2D, these parameters can be specified for each material data set for soil and interfaces as well as for material data sets for plates. In this way, the (viscous) damping characteristics can be specified for each individual material in the finite element model.

Despite the considerable amount of research work in the field of dynamics, little has been achieved yet for the development of a commonly accepted procedure for damping parameter identification. Instead, for engineering purposes, some measures are made to account for material damping. A commonly used engineering parameter is the damping ratio ξ. The damping ratio is defined as $\xi = 1$ for critical damping, i.e. exactly the amount of damping needed to let a single degree-of-freedom system that is released from an initial excitation u_0, smoothly stop without rebouncing.

Considering Rayleigh damping, a relationship can be established between the damping ratio ξ and the Rayleigh damping parameters α and β:

$$\alpha + \beta\omega^2 = 2\omega\xi \quad and \quad \omega = 2\pi f \qquad Eq.\ [8]$$

where ω is the angular frequency in rad/s and f is the frequency in Hz (1/s).

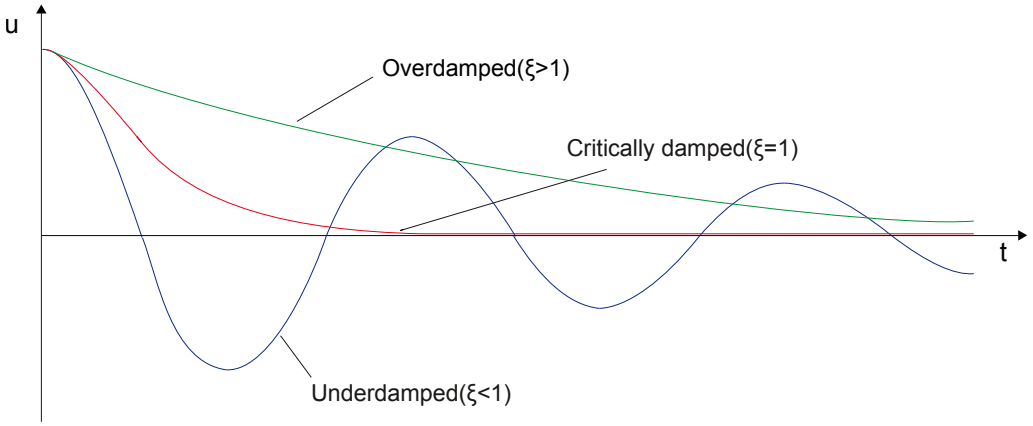


Figure 117: Role of damping ratio ξ in free vibration of a single degree-of-freedom system

Solving this equation for two different target frequencies and corresponding target damping ratios gives the required Rayleigh damping coefficients:

$$\alpha = 2\omega_1\omega_2\frac{\omega_1\xi_2 - \omega_2\xi_1}{{\omega_1}^2 - {\omega_2}^2} \quad and \quad \beta = 2\frac{\omega_1\xi_1 - \omega_2\xi_2}{\omega_1^2 - \omega_2^2} \qquad Eq.\ [9]$$

For example, when it is desired to have a target damping of *8%* at the target frequencies *f = 1.5* Hz and *8.0* Hz, the corresponding Rayleigh damping ratios are $\alpha = 1.2698$ and $\beta = 0.002681$. From Figure 118 (on page 168) it can be seen that within the range of frequencies as defined by the target frequencies the damping is less than the target damping, whereas outside this range the damping is more than the target damping.

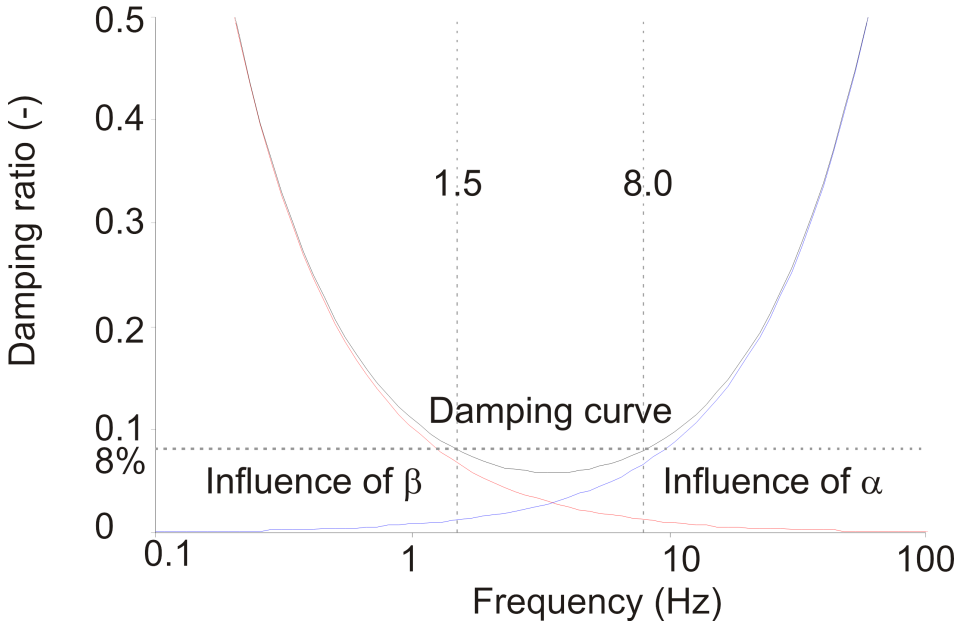


Figure 118: Rayleigh damping parameter influence

In order to calibrate the frequencies corresponding to Target 1 and Target 2, different procedures can be found in literature. In particular, Hudson, Idriss & Beirkae (1994) (on page 531) and Hashash & Park (2002) (on page 531) suggest to select the first target frequency as the first natural frequency of the soil deposit f_1, while the second target frequency is the closest odd integer larger than the ratio f_p/f_1, i.e. the predominant frequency of the input motion (that can be determined from the input Fourier spectrum) over the natural frequency of the soil. The natural frequency of the soil deposit of thickness H is related to its geometry and stiffness according to the following equation:

$$f_1 = \frac{V_s}{4H} \qquad Eq.\ [10]$$

where v_s is the shear wave velocity in the soil deposit, that is a function of the shear stiffness modulus G.

Amorosi, Boldini & Ellia (2010) (on page 530) suggest to consider the frequency interval characterised by the highest energy content that can be evaluated by plotting the Fourier spectrum at different depths of the soil deposit and the amplification function between the surface and the base level. It has been demonstrated that this procedure overcomes the errors that can occur with the previous procedure for increasing values of the ratio f_p/f_1 and of the soil deposit thickness.

More generally, the two frequencies are identified through an iterative procedure.

It is suggested *not* to use the simplified Rayleigh formulation, i.e. the small strain viscous damping effects are assumed to be proportional only to the stiffness of the soil deposit:

$$[C] = \beta[K] \quad \text{(not recommended)} \qquad Eq.\ [11]$$

where $\beta = 2\,\xi/\omega_1$ and ω_1 is the angular frequency of the first natural mode of the soil column. It has been noticed that the simplified Rayleigh formulation may lead to an underestimation of the site response, especially when the natural frequency of the soil deposit and the predominant frequency of the input motion are far from each other, resulting in overestimation of damping in the high-frequency range.

When the Rayleigh damping **Input method** is set to *SDOF equivalent*, the damping parameters (α and β) will be automatically calculated by the program when the target damping ratio (ξ) and the target frequencies (f) are

specified. The panel displayed in the **General** tabsheet when one of the cells corresponding to the damping parameters is clicked. Figure 119 (on page 169) shows the damping ratio as a function of the frequency.

Alternatively, if the **Input method** is set to *Direct* the damping parameters α and β can be directly specified by the user.

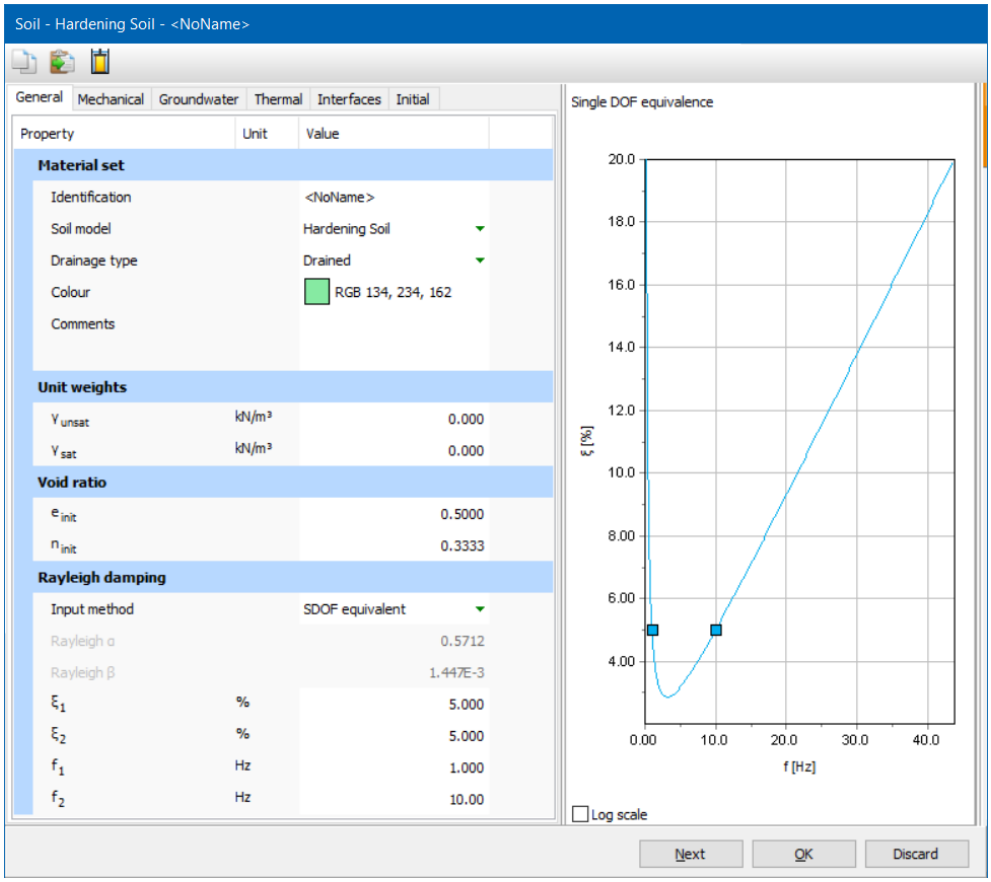


Figure 119: Input of ξ and f

6.1.2 *Mechanical tabsheet*

The **Mechanical** tabsheet contains the stiffness and strength parameters of the selected soil model. These parameters depend on the selected soil model as well as on the selected drainage type.

Linear Elastic model (LE)

The **Mechanical** tabsheet for the Linear Elastic model(drained behaviour) is shown in Figure 120 (on page 170).

The model involves two elastic stiffness parameters, namely the effective Young's modulus E' and the effective Poisson's ratio γ'.

E'_{ref}	Effective Young's modulus	[kN/m^2]

ν'	Effective Poisson's ratio	[-]

During the input for the Linear Elastic model the values of the shear modulus G and the oedometer modulus E_{oed} are presented as auxiliary parameters (alternatives).

G_{ref}	Shear modulus, where $G = E'/(2(1 + \nu'))$	[kN/m^2]
E_{oed}	Oedometer modulus, where $E_{oed} = E'(1-\nu')/((1+\nu')(1 - 2\nu'))$	[kN/m^2]

Note:

- Optional drainage types when the Linear Elastic model is selected are: *Drained*, *Undrained A*, *Undrained C*, and *Non-porous*.
- In the case of *Undrained A* or *Non-porous* drainage types, the same parameters are used as for drained behaviour.
- In the case of *Undrained C* drainage type, an undrained Young's modulus (E_u) and undrained Poisson's ratio (ν_u) are used.

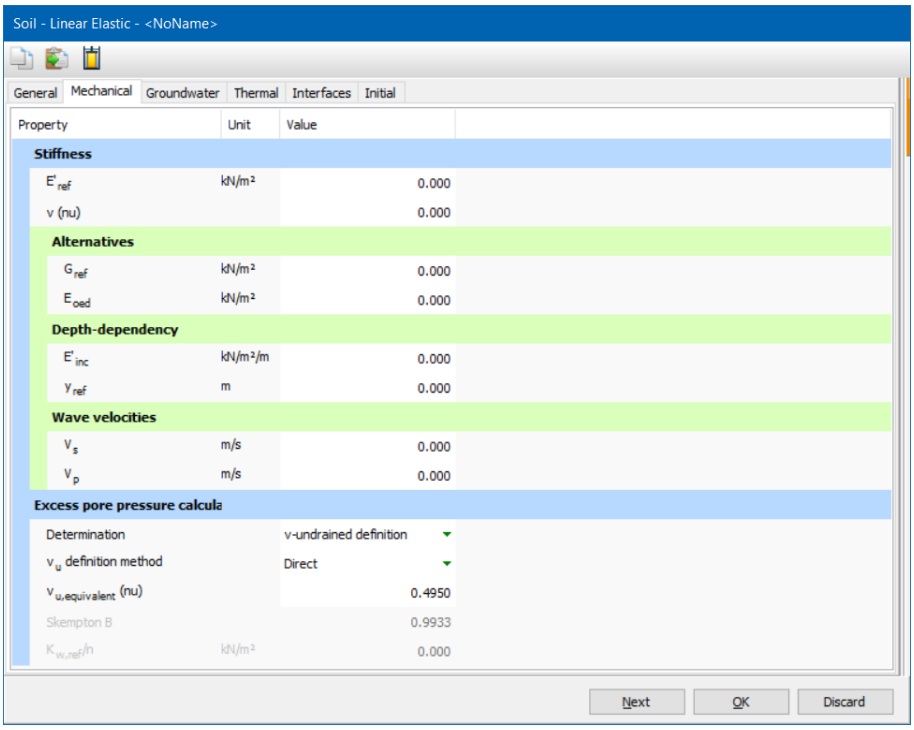


Figure 120: ***Mechanical*** *tabsheet for the Linear Elastic model(drained behaviour)*

Note that the alternatives are influenced by the input values of E' and ν'. Entering a particular value for one of the alternatives G_{ref} or E_{oed} results in a change of the Young's modulus E'_{ref}.

It is possible for the Linear Elastic model to specify a stiffness that varies linearly with depth. Therefore, the increment of stiffness per unit of depth, E'_{inc}, can be defined. Together with the input of E'_{inc} the input of y_{ref} becomes relevant. For any y-coordinate above y_{ref} the stiffness is equal to E'_{ref}. For any y-coordinate below y_{ref} the stiffness is given by:

$$E'(y) = E'_{ref} + (y_{ref} - y)E'_{inc}; \quad y < y_{ref} \qquad Eq.\ [12]$$

The Linear Elastic model is usually inappropriate to model the highly non-linear behaviour of soil, but it is of interest to simulate structural behaviour, such as thick concrete walls or plates, for which strength properties are usually very high compared with those of soil. For these applications, the Linear Elastic model will often be selected together with *Non-porous* type of material behaviour in order to exclude pore pressures from these structural elements.

Note:

When embedded beams penetrate a polygon cluster with linear elastic material behaviour, the specified value of the shaft resistance is ignored. The reason for this is that the linear elastic material is not supposed to be soil, but part of the structure. The connection between the pile and the structure is supposed to be rigid to avoid, for example, punching of piles through a concrete deck.

[ULT] As a second alternative for the stiffness parameters of the soil, the velocities of wave propagation in soil can be defined in the **Mechanical** tabsheet of the **Soil** window when the Dynamics module of the program is available. These velocities are:

V_s	Shear wave velocity, where $V_s = \sqrt{G_0 / \rho}$	[m/s]
V_p	Compression wave velocity, where $V_p = \sqrt{E / \rho}$	[m/s]

where $\rho = \gamma / g$.

As the wave velocities are influenced by the input values of E' and ν', entering a particular value for one of the wave velocities results in a change of the Young's modulus.

Note:

- Velocities of wave propagation in soil can be defined only for models with stress independent stiffness.

Mohr-Coulomb model (MC)

The linear-elastic perfectly-plastic model with Mohr-Coulomb failure contour (in short the Mohr-Coulomb model) requires a total of five parameters (two stiffness parameters and three strength parameters), which are generally familiar to most geotechnical engineers and which can be obtained from basic tests on soil samples.

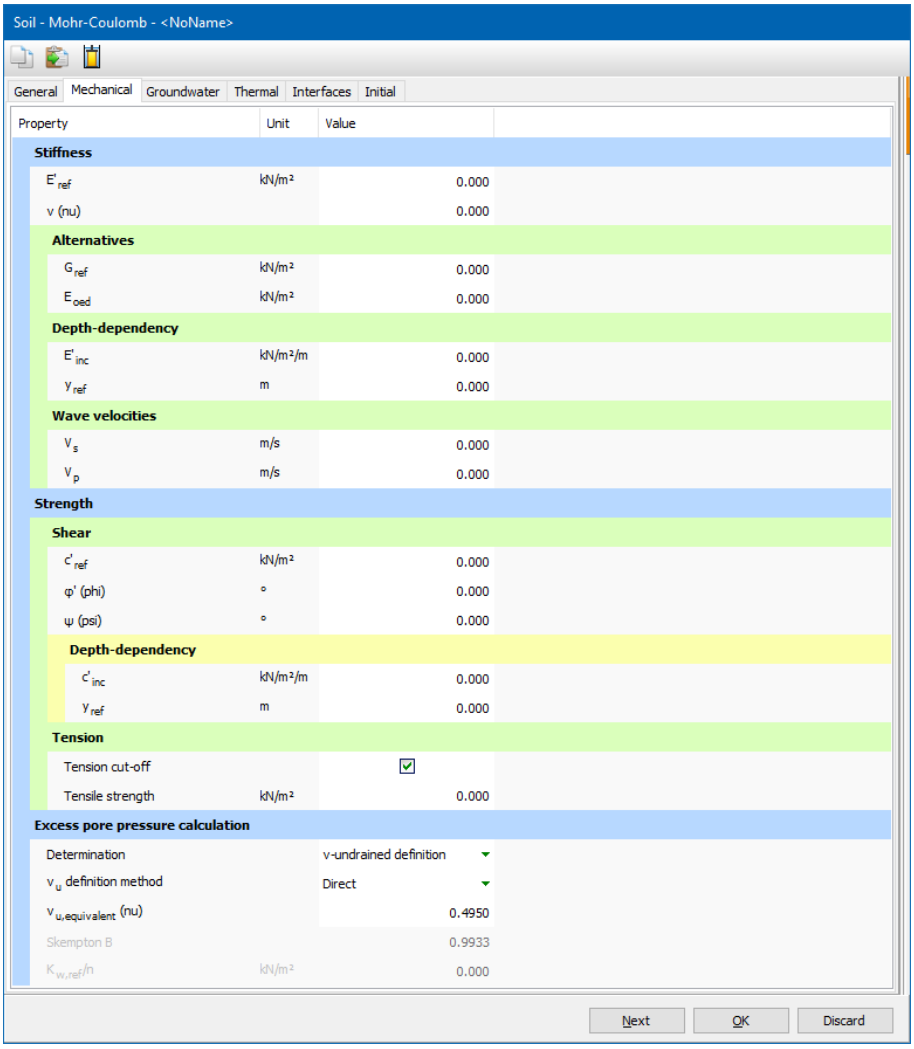


*Figure 121: **Mechanical** tabsheet for the Mohr-Coulomb model (drained behaviour)*

The stiffness parameters of the Mohr-Coulomb model (drained behaviour) are:

E'_{ref}	Effective Young's modulus	[kN/m^2]
ν	Effective Poisson's ratio	[-]

Note:

- Optional drainage types when Mohr-Coulomb model is selected are: *Drained*, *Undrained A*, *Undrained B*, *Undrained C*, and *Non-porous*.
- In the case of *Undrained A* or *Non-porous* drainage types, the same parameters are used as for drained behaviour.
- In the case of *Undrained B* drainage type, $\varphi = \varphi_u = 0$, $\psi = 0$ and the undrained shear strength s_u is used instead of the effective cohesion (c').

- In the case of *Undrained C* drainage type all parameters are undrained. i.e. E_u, ν_u and s_u as undrained Young's modulus, undrained Poisson's ratio and undrained shear strength respectively, and $\varphi = \psi = 0$.

Instead of using the Young's modulus as a stiffness parameter, alternative stiffness parameters can be entered. These parameters, the relations and their standard units are listed below:

G_{ref}	Shear modulus, where $G_{ref} = E'_{ref}/(2(1 + \nu))$	[kN/m^2]
E_{oed}	Oedometer modulus, where $E_{oed} = E'_{ref}(1-\nu)/((1+\nu)(1 - 2\nu))$	[kN/m^2]

Note that the alternatives are influenced by the input values of E'_{ref} and ν. Entering a particular value for one of the alternatives G_{ref} or E_{oed} results in a change of the Young's modulus E'_{ref}.

Stiffness varying with depth can be defined in Mohr-Coulomb model by entering a value for E'_{inc} which is the increment of stiffness per unit of depth. Together with the input of E'_{inc} the input of y_{ref} becomes relevant. For any y-coordinate above y_{ref}} the stiffness is equal to E'_{ref}. For any y-coordinate below y_{ref} the stiffness is given by:

$$E'(y) = E'_{ref} + (y_{ref} - y)E'_{inc}; \quad y < y_{ref} \qquad \textit{Eq. [13]}$$

The strength parameters for the Mohr-Coulomb model are:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]

A cohesion varying with depth can be defined in Mohr-Coulomb model by entering a value for c'_{inc} which is the increment of effective cohesion per unit of depth. Together with the input of c'_{inc} the input of y_{ref} becomes relevant. For any y-coordinate above y_{ref} the cohesion is equal to c'_{ref}. For any y-coordinate below y_{ref} the cohesion is given by:

$$c'(y) = c'_{ref} + (y_{ref} - y)c'_{inc}; \quad y < y_{ref} \qquad \textit{Eq. [14]}$$

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2D analysis by selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

[ULT] As a second alternative for the stiffness parameters, the velocities of wave propagation in soil can be defined in the **Mechanical** tabsheet of the **Soil** window. These velocities are:

V_s	Shear wave velocity, where $V_s = \sqrt{G / \rho}$	[m/s]
V_p	Compression wave velocity, where $V_P = \sqrt{E / \rho}$	[m/s]

where $\rho = \nu / g$.

The wave velocities are influenced by the input values of E' and ν', hence entering a particular value for one of the wave velocities results in a change of the Young's modulus.

Note:

- Velocities of wave propagation in soil can be defined only for models with stress independent stiffness.

Hoek-Brown model (HB)

The **Mechanical** tabsheet for the Hoek-Brown model is shown in Figure 122 (on page 174). The *Hoek-Brown* parameters can be obtained from the tools available in the right-hand panel.

Note:

- Optional drainage types when Hoek-Brown model is selected are: *Drained* and *Non-porous*.
- The same parameters are used for both *Non-porous* drainage type and for drained behaviour.

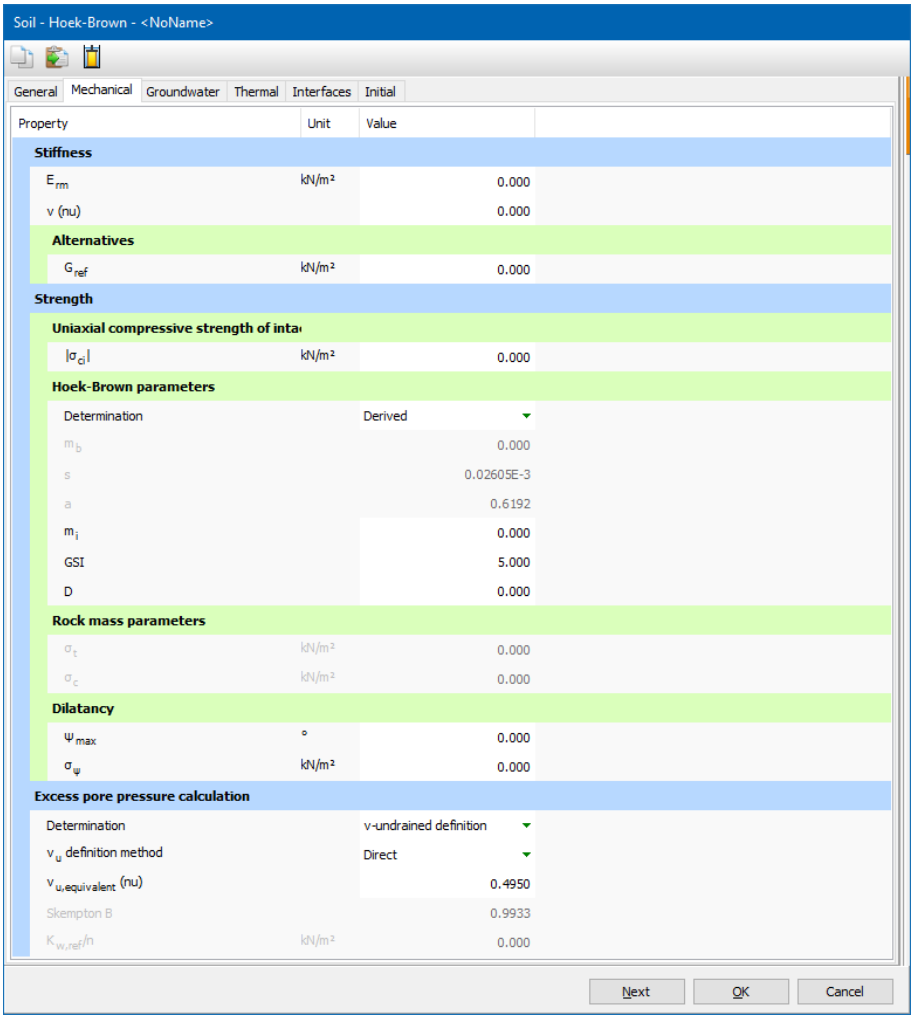


*Figure 122: **Mechanical** tabsheet for the Hoek-Brown model (drained behaviour)*

The stiffness parameters of the Hoek-Brown model are:

E_{rm}	Rock mass Young's modulus	[kN/m^2]
ν	Poisson's ratio	[-]
G_{ref}	Rock mass shear modulus, $G_{ref} = E_{rm}/(2(1+\nu))$	[kN/m^2]

Where G_{ref} is an alternative input for the rock mass Young's modulus E_{rm}.

The strength parameters have been divided in 4 subcategories: the *Uniaxial compressive strength*, the *Hoek-Brown parameters*, the *Rock mass parameters* and the *Dilatancy*.

$\|\sigma_{ci}\|$	Uni-axial compressive strength of the intact rock (>0)	[kN/m^2]

For the Hoek-Brown and Rock mass parameters the user has the choice how to enter them by setting the **Determination** option. When this option is set to *Derived*, the user is asked to specify these well-known engineering parameters:

m_i	Material constant for the intact rock	[-]
GSI	Geological Strength Index	[-]
D	Disturbance factor which depends on the degree of disturbance to which the rock mass has been subjected	[-]

When any of these parameters is selected, a side panel opens to supports the selection of the parameters.

However, when **Determination** is set to *Direct* it becomes possible to specify the Hoek-Brown parameters directly:

m_b	Reduced intact rock parameter	[-]
s	Hoek-Brown material parameter	[-]
a	Hoek-Brown power parameter	[-]
σ_t	Tensile strength of the rock mass	[kN/m^2]
σ_c	Compressive strength of the rock mass	[kN/m^2]

Finally, the dilatancy can be specified through the following parameters:

ψ_{max}	Dilatancy at zero stress level	[°]
σ_ψ	Stress level at which dilatancy is fully suppressed	[kN/m^2]

Jointed Rock model (JR)

The **Mechanical** tabsheet for the Jointed Rock model is shown in Figure 123 (on page 176).

Note:

- Optional drainage types when Jointed Rock model is selected are: *Drained* and *Non-porous*.
- The same parameters are use for both thef *Non-porous* drainage type and drained behaviour.

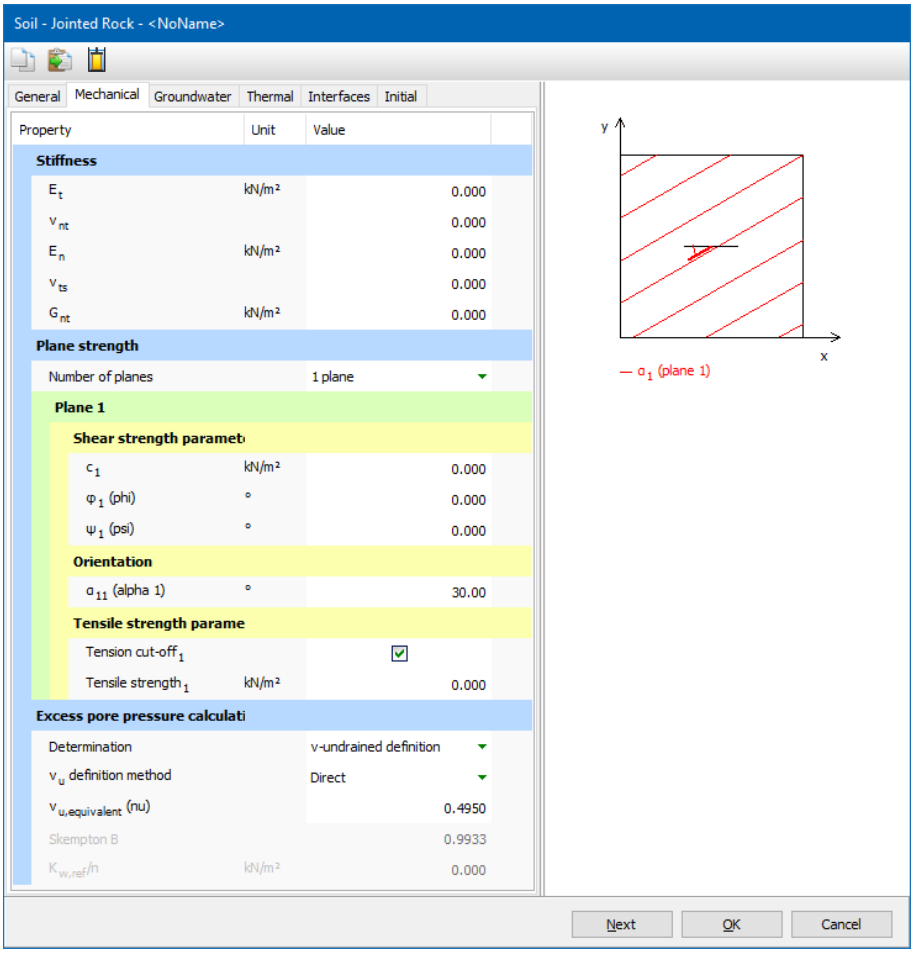


*Figure 123: **Mechanical** tabsheet for the Jointed Rock model*

Parameters for stiffness:

E_t	Young's modulus for rock as a continuum, i.e. in 'Plane 1' direction	[kN/m^2]
ν_{nt}	Poisson's ratio for rock as a continuum, i.e. in 'Plane 1' direction	[-]

Anisotropic elastic parameters 'Plane 1' direction (e.g. stratification direction):

E_n	Young's modulus perpendicular to 'Plane 1' direction	[kN/m^2]
ν_{ts}	Poisson's ratio perpendicular to 'Plane 1' direction	[-]
G_{nt}	Shear modulus perpendicular to 'Plane 1' direction	[kN/m^2]

Parameters for strength:

Strength parameters in joint directions (Plane i=1, 2, 3):

c_i	Cohesion	[kN/m^2]
φ_i	Friction angle	[°]
ψ_i	Dilatancy angle	[°]
$\sigma_{t,i}$	Tensile strength	[kN/m^2]

Definition of joint directions (Plane i=1, 2, 3):

Number of planes	Number of joint directions (*1≤n≤3*)	[-]
$\alpha_{1,i}$	Dip angle (visualized in the side panel)	[°]
$\alpha_{2,i}$	Dip direction	[°]

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2D analysis by selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

Hardening Soil model (HS)

The **Mechanical** tabsheet for the Hardening Soil modelis shown in Figure 124 (on page 178).

Note:

- Optional drainage types when Hardening Soil model is selected are: *Drained*, *Undrained A*, and *Undrained B*.
- In the case of *Undrained A* drainage type, the same parameters are used as for drained behaviour.
- In the case of *Undrained B* drainage type, $\varphi = \varphi_u = 0$, $\psi = 0$ and the undrained shear strength s_u is used instead of the effective cohesion (c').

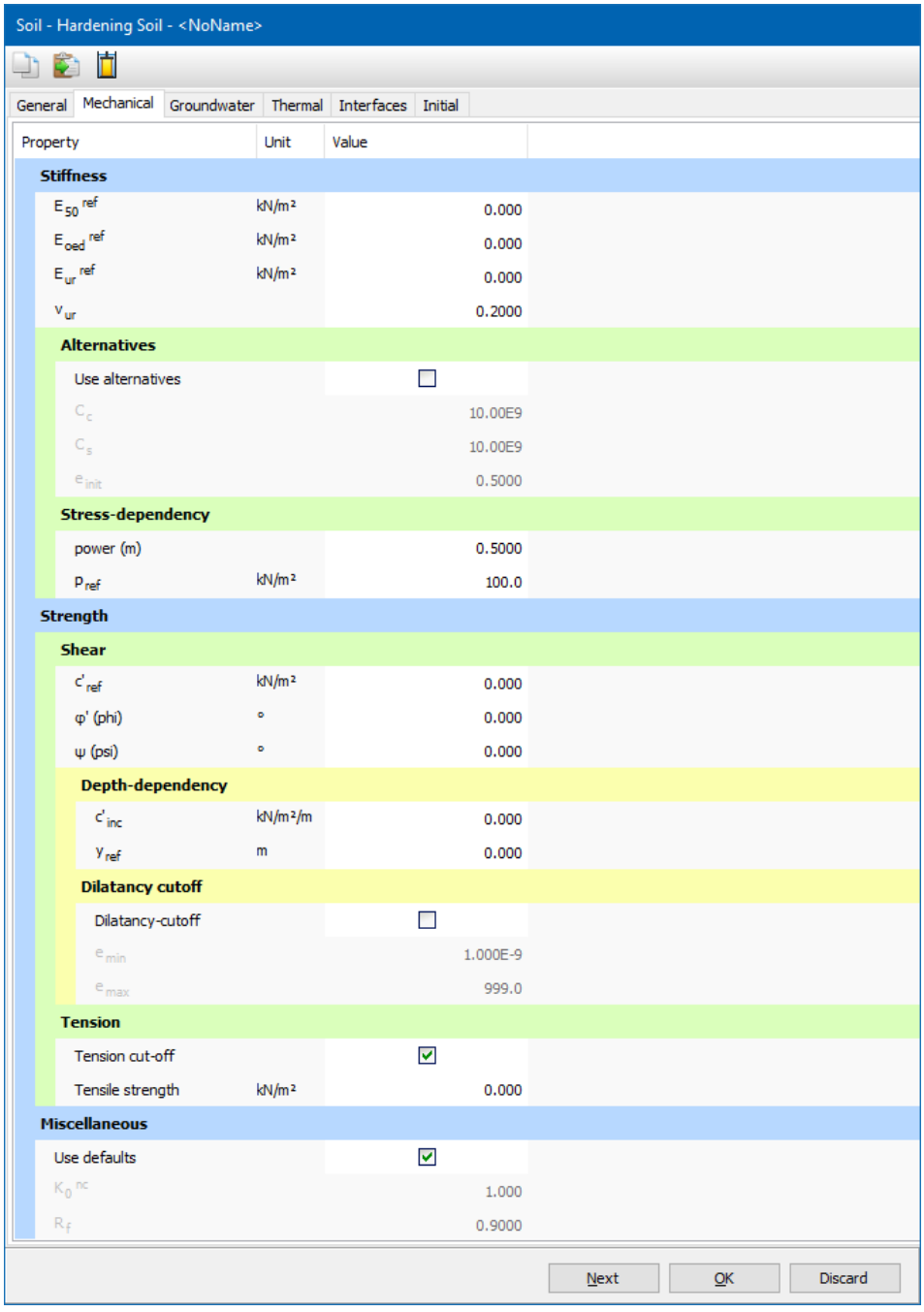


Figure 124: ***Mechanical*** *tabsheet for the Hardening Soil model(drained behaviour)*

The stiffness parameters of the Hardening Soil model are:

E_{50}^{ref}	Secant stiffness in standard drained triaxial test	[kN/m^2]
E_{oed}^{ref}	Tangent stiffness for primary oedometer loading	[kN/m^2]
E_{ur}^{ref}	Unloading / reloading stiffness (default $E_{ur}^{ref} = 3E_{50}^{ref}$)	[kN/m^2]

ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.2)	[-]

Instead of entering the basic parameters for soil stiffness, alternative parameters can be entered. These parameters are listed below:

C_C	Compression index	[-]
C_s	Swelling index or reloading index	[-]
e_{init}	Initial void ratio	[-]

In addition, stress dependent stiffness can be defined through a power law with the following parameters:

m	Power for stress-level dependency of stiffness	[-]
p^{ref}	Reference stress for stiffnesses (default p^{ref} = 100 kN/m^2)	[kN/m^2]

The strength parameters of the Hardening Soil model coincide with those of the Mohr-Coulomb model:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]

In addition, a depth dependent cohesion can be defined:

c'_{inc}	As in the Mohr-Coulomb model (default c_{inc} = 0)	[kN/m^3]
y_{ref}	Reference level	[m]

Dilatancy is generally dependent on the density of the soil: a loose soil shows less dilatancy than a dense soil. In a shearing test it can be observed that if a dense material is sheared the effective dilatancy angle reduces with the increase of volume strain, hence with reducing void ratio. On the other hand, in case of negative dilatancy, the soil can only continue to reduce volume until the maximum density, hence the minimum void ratio, is reached. PLAXIS 2D does not offer the possibility to take into account full void ratio dependent dilatancy, but it does have the option to only calculate dilatancy if the void ratio is between certain limits while the dilatancy is considered zero outside these limit values of the void ratio.

Dilatancy cut-off	To be selected when dilatancy should only be considered within the void ratio range between e_{min} and e_{max}.	[-]
e_{min}	Minimum void ratio	[-]
e_{max}	Maximum void ratio	[-]

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2D analysis by

selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

Tension cut-off	To be selected when tension cut-off should be considered	[-]
Tensile strength	The allowable tensile strength	[kN/m^2]

Finally, there are some *Miscellaneous* parameters to be defined that are not directly related to either stiffness or strength. It is recommended to leave these values to their default values:

K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - \sin(\varphi)$)	[-]
R_f	Failure ratio q_f / q_a (default $R_f = 0.9$)	[-]

Hardening Soil model with small-strain stiffness (HSsmall)

Compared to the standard HS model, the Hardening Soil model with small-strain stiffness requires two additional stiffness parameters as input: $\gamma_{0.7}$ and G_0^{ref}. The **Mechanical** tabsheet for the Hardening Soil model with small-strain stiffness is shown in Figure 125 (on page 181).

Note:

- Optional drainage types when Hardening Soil model with small-strain stiffness is selected are: *Drained*, *Undrained A*, and *Undrained B*.
- In the case of *Undrained A* drainage type, the same parameters are used as for drained behaviour.
- In the case of *Undrained B* drainage type, $\varphi = \varphi_u = 0$, $\psi = 0$ and the undrained shear strength s_u is used instead of the effective cohesion (c').

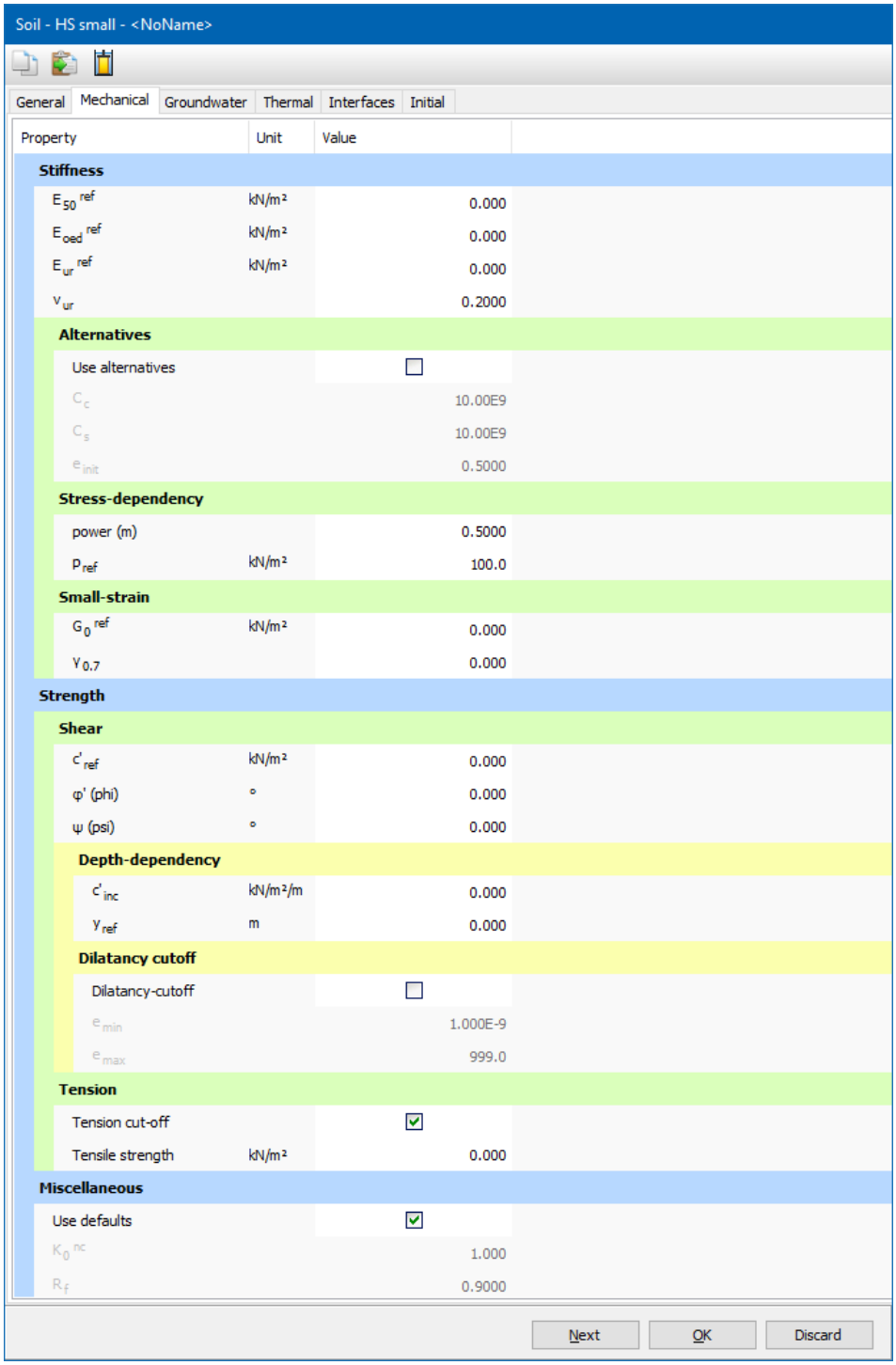


*Figure 125: **Mechanical** tabsheet for the Hardening Soil model with small-strain stiffness model (drained behaviour)*

All other parameters, including the alternative stiffness parameters, remain the same as in the standard Hardening Soil model. In summary, the input stiffness parameters of the Hardening Soil model with small-strain stiffness are listed below:

Parameters for stiffness:

E_{50}^{ref}	Secant stiffness in standard drained triaxial test	[kN/m^2]
E_{oed}^{ref}	Tangent stiffness for primary oedometer loading	[kN/m^2]
E_{ur}^{ref}	Unloading / reloading stiffness (default $E_{ur}^{ref} = 3E_{50}^{ref}$)	[kN/m^2]
m	Power for stress-level dependency of stiffness	[-]

Alternative parameters for stiffness:

C_C	Compression index	[-]
C_s	Swelling index or reloading index	[-]
e_{init}	Initial void ratio	[-]

Parameters for stress-dependent stiffness:

ν_{ur}	Poisson's ratio for unloading-reloading (default $\nu = 0.2$)	[-]
p^{ref}	Reference stress for stiffnesses (default $p^{ref} = 100$ kN/m^2)	[kN/m^2]

Parameters for small strain stiffness:

$\gamma_{0.7}$	Shear strain at which $G_s = 0.722\ G_0$	[-]
G_0^{ref}	Reference shear modulus at very small strains ($\varepsilon < 10^{-6}$)	[kN/m^2]

General parameters for strength:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]

Parameters for depth-dependent strength:

c'_{inc}	As in Mohr-Coulomb model (default $c_{inc} = 0$)	[kN/m^2/m]
y_{ref}	Reference level	[m]

Parameters to limit the dilatancy:

Dilatancy cut-off	To be selected if a zero dilatancy angle should be used when the void ratio falls outside the limits defined by e_{min} and e_{max}.	[-]
e_{min}	Minimum void ratio	[-]

e_{max}	Maximum void ratio	[-]

Parameters for tensile strength:

Tension cut-off	To be selected when tension cut-off is considered	[-]
Tensile strength	The allowable tensile strength	[kN/m^2]

Miscallaneous parameters:

K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - sin(\varphi)$)	[-]
R_f	Failure ratio q_f / q_a (default $R_f = 0.9$)	[-]

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2D analysis by selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

Hysteretic damping

The elastic modulus ratio is plotted as a function of the shear strain (γ) in a side pane when specifying the small-strain stiffness parameters (*Modulus reduction curve*). The Hardening Soil model shows typical hysteretic behaviour when subjected to cyclic shear loading. In dynamics calculations this leads to hysteretic damping. The damping ratio is plotted as a function of the cyclic shear strain γ^C. Details are given in Brinkgreve, Kappert & Bonnier (2007) (on page 530).

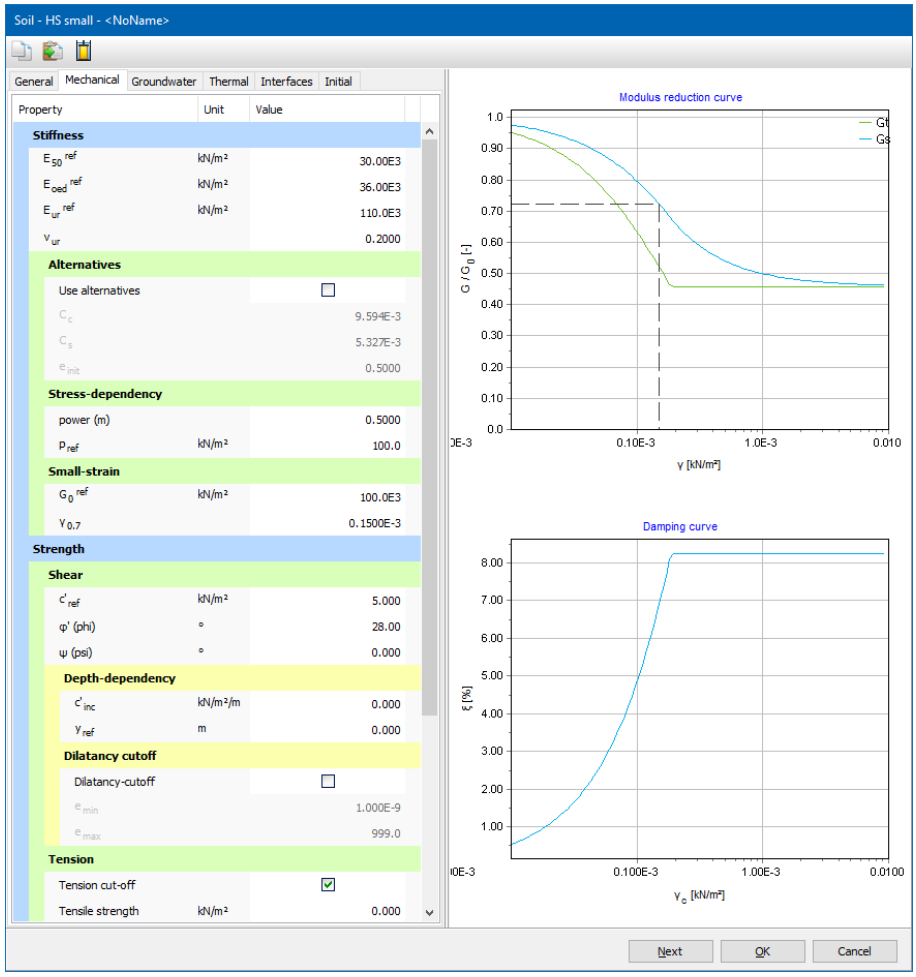


Figure 126: Effect of small strain stiffness parameters on damping

Note:

Note that the *Modulus reduction curve* and the *Damping curve* are based on fully elastic behaviour. Plastic strains as a result of hardening or local failure may lead to significant lower stiffness and higher damping.

Modified Cam-Clay model (MCC)

This is a critical state model that can be used to simulate the behaviour of normally consolidated soft soils. The model assumes a logarithmic relationship between the void ratio and the mean effective stress. The **Mechanical** tabsheet for the Modified Cam-Clay model is shown in Figure 127 (on page 185).

Note:

- Optional drainage types when Modified Cam-Clay model is selected are: *Drained* and *Undrained A*.
- In the case of *Undrained A* drainage type, the same parameters are used as for drained behaviour.

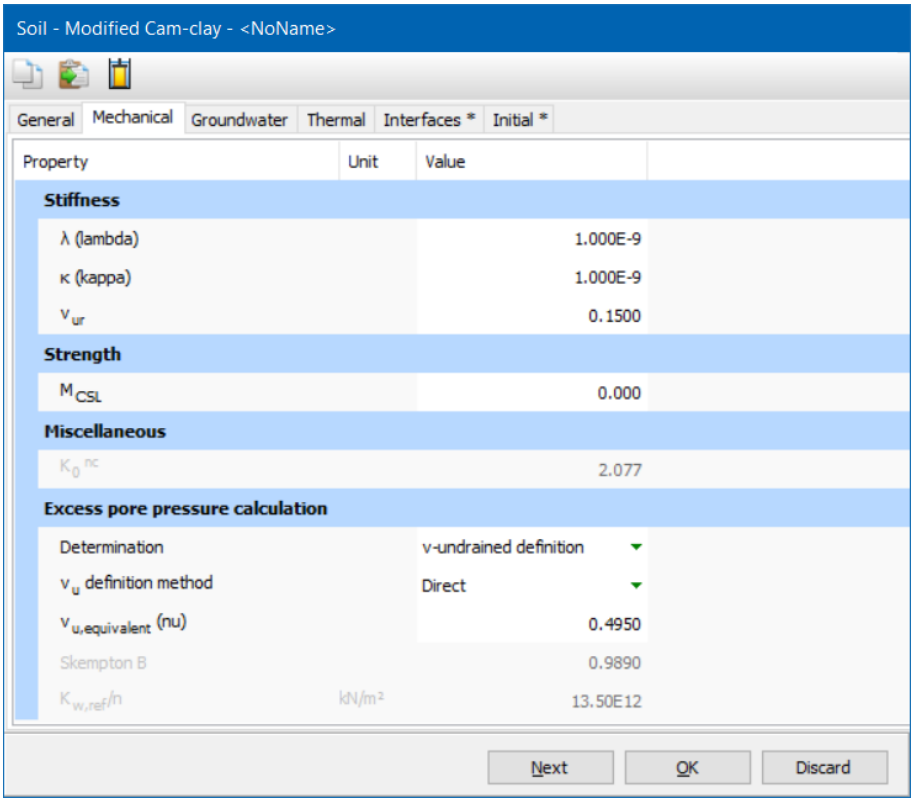


*Figure 127: **Mechanical** tabsheet for the Modified Cam-Clay model (drained behaviour)*

Parameters for stiffness:

λ	Cam-Clay compression index	[-]
κ	Cam-Clay swelling index	[-]
ν_{ur}	Poisson's ratio for unloading/reloading	[-]

Parameter for strength:

M_{CSL}	Tangent of the Critical State Line	[-]

The Modified Cam-Clay model has an implicit relation between the ratio of horizontal and vertical stresses in primary loading, K_0^{NC}, and the strength and stiffness parameters. Therefore in the *Miscellaneous* block the value of K_0^{NC} resulting from the given stiffness and strength parameters is shown, but cannot be changed directly. If independent control over the strength and stiffness parameters as well as K_0^{NC} is required, it is recommended to use the Soft Soil model instead of the Modified Cam-Clay model.

NGI-ADP model

The NGI-ADP model may be used for capacity, deformation and soil-structure interaction analysis involving undrained loading of clay-type materials. The **Mechanical** tabsheet for the NGI-ADP model is shown in Figure 128 (on page 186).

Note:

The only drainage type available when the NGI-ADP model is selected is *Undrained C*.

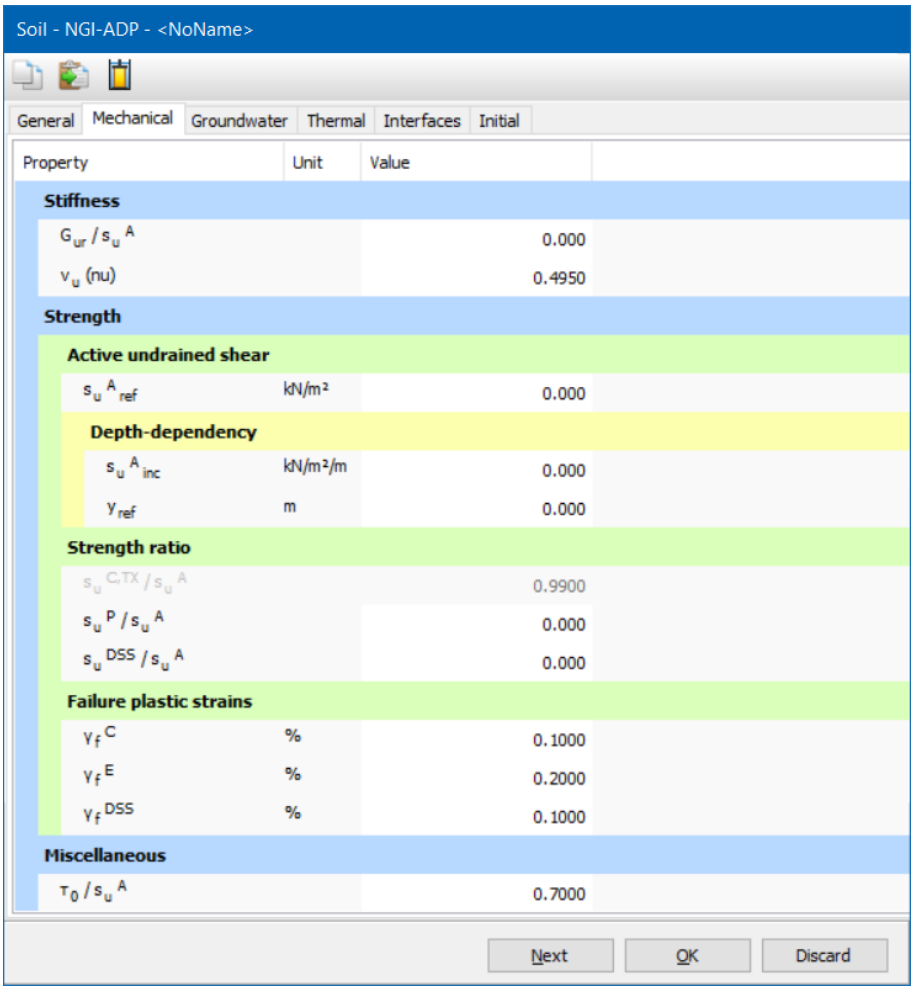


*Figure 128: **Mechanical** tabsheet for the NGI-ADP model*

Parameters for stiffness:

$G_{ur}/s_{u,inc}{}^A$	Ratio unloading/reloading shear modulus over the (plane strain) active shear strength	[-]
ν_u	Undrained Poisson's ratio	[-]

Parameters for strength

$s_u{}^{A,\,ref}$	Reference (plane strain) active shear strength	[kN/m^2]
$s_{u,inc}{}^A$	Increase of (plane strain) active shear strength with depth	[kN/m^2/m]
y_{ref}	Reference level for active, passive and DSS shear strength	[m]

$s_u^{C,TX}/s_u^A$	Ratio triaxial compressive shear strength over (plane strain) active shear strength (fixed at 0.99)	[-]
s_u^P/s_u^A	Ratio of (plane strain) passive shear strength over (plane strain) active shear strength	[-]
s_u^{DSS}/s_u^A	Ratio of direct simple shear strength over (plane strain) active shear strength	[-]
γ_f^C	Shear strain at failure in triaxial compression ($\lvert\gamma_f^C = 3/2\ \varepsilon_1^C\rvert$)	[-]
γ_f^E	Shear strain at failure in triaxial extension	[-]
γ_f^{DSS}	Shear strain at failure in direct simple shear	[-]

Miscellaneous parameters:

τ_0/s_u^A	Ratio initial mobilized shear over (plane strain) active shear strength (default = 0.7)	[-]

Soft Soil model (SS) [ADV]

The **Mechanical** tabsheet for the Soft Soil model is shown in Figure 129 (on page 188).

Note:

- Optional drainage types when Soft Soil model is selected are: *Drained* and *Undrained A*.
- For both *Drained* and *Undrained A* drainage type the same parameters are used.

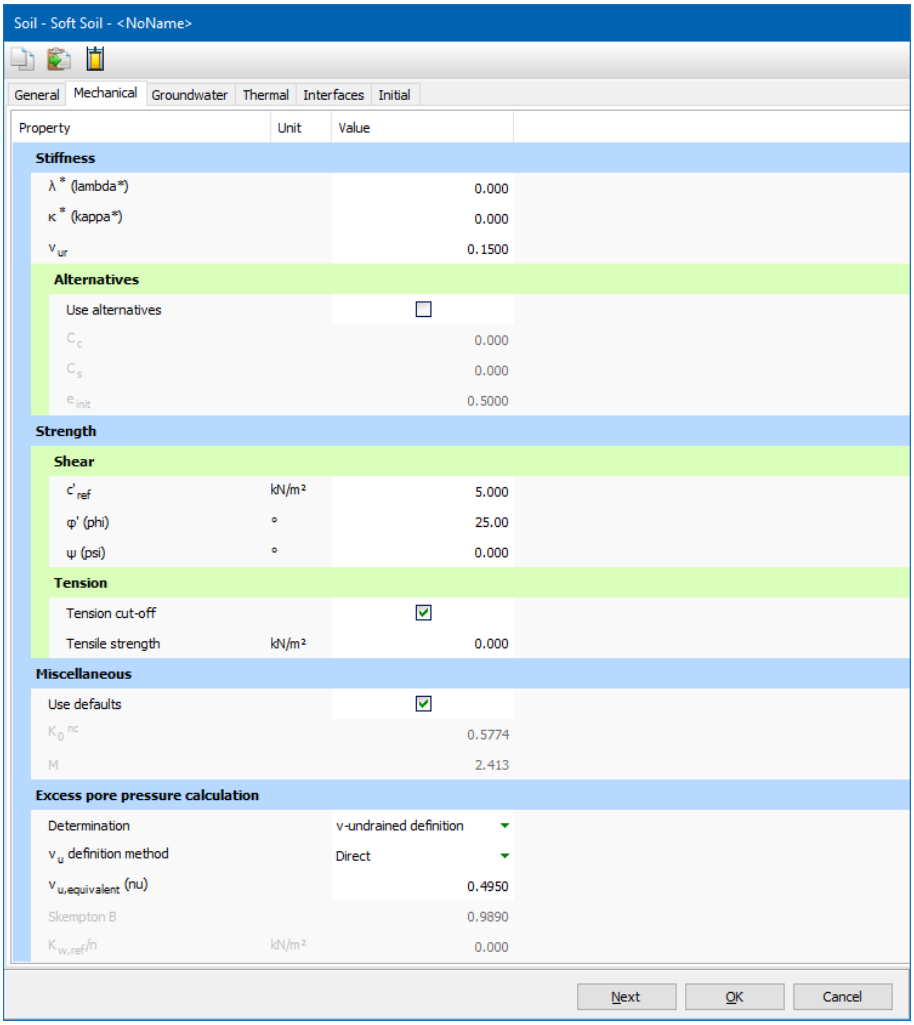


*Figure 129: **Mechanical** tabsheet for the Soft Soil model (drained behaviour)*

The parameters for stiffness are:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]
ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.15)	[-]

Alternative parameters can be used to define stiffness:

C_C	Compression index	[-]
C_s	Swelling index or reloading index	[-]
e_{init}	Initial void ratio	[-]

The parameters for strength are:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]
Tension cut-off	To be selected when tension cut-off is considered	[-]
Tensile strength	The allowable tensile strength	[kN/m^2]

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2Danalysis by selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

Miscellaneous parameters (use default settings):

K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - \sin(\varphi)$)	[-]
M	K_0^{nc}- related parameter (no direct input possible)	[-]

The parameter M is the inclination of the line passing through both the leftmost and topmost points of the ellipse forming the hardening cap of the Soft Soil model. The value of M is derived from the value of K_0^{nc} and is given here for comparison with the value M_{CSL} of for instance the Modified Cam-Clay model (MCC) (on page 184).

Soft Soil Creep model (SSC) [ADV]

The **Mechanical** tabsheet for the Soft Soil Creep model is shown in Figure 130 (on page 190).

Note:

- Optional drainage types when Soft Soil Creep model is selected are: *Drained* and *Undrained A*.
- For both *Drained* and *Undrained A* drainage type, the same parameters are used.

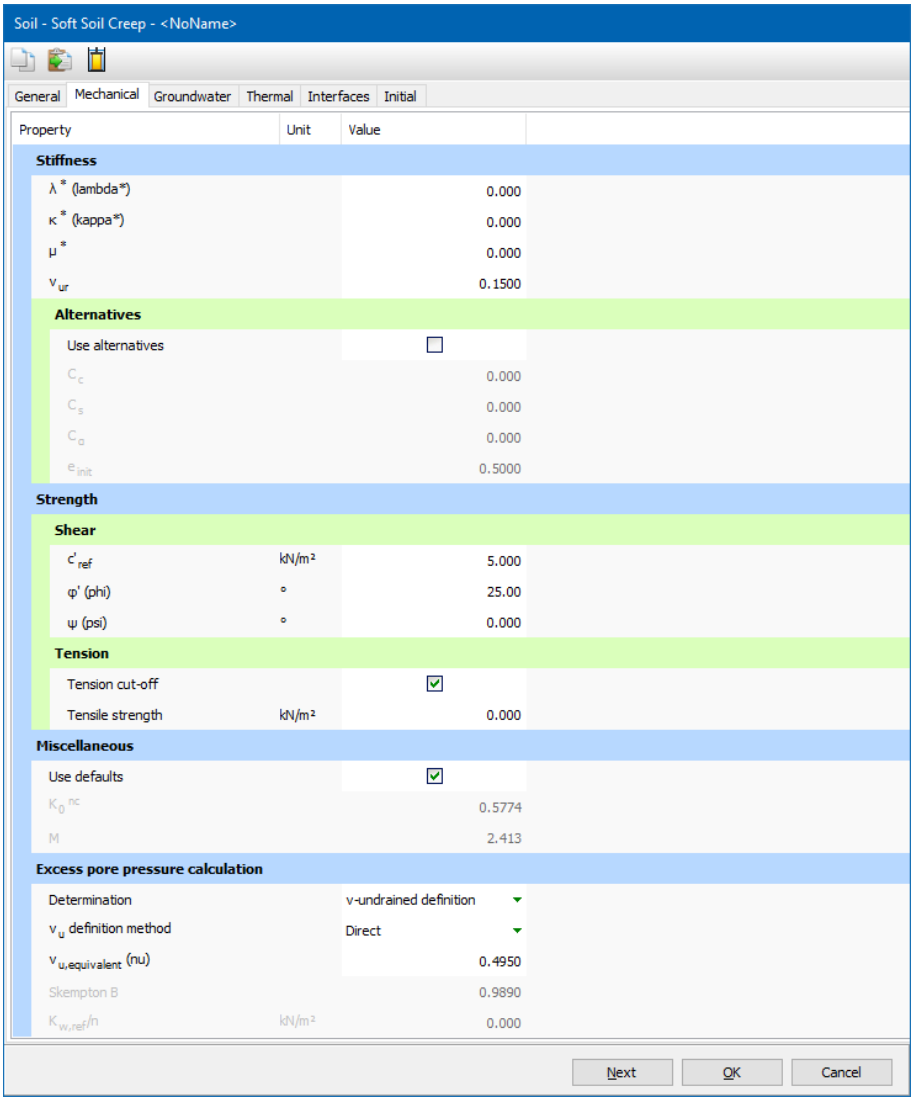


*Figure 130: **Mechanical** tabsheet for the Soft Soil Creep model (drained behaviour)*

The parameters for stiffness are:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]
μ^*	Modified creep index	[-]
ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.15)	[-]

Alternative parameters can be used to define stiffness:

C_C	Compression index	[-]

C_s	Swelling index or reloading index	[-]
C_α	Secondary compression index	[-]
e_{init}	Initial void ratio	[-]

The parameters for strength are:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]
Tension cut-off	To be selected when tension cut-off is considered	[-]
Tensile strength	The allowable tensile strength	[kN/m^2]

In some practical problems an area with tensile stresses may develop. This is allowed when the shear stress is sufficiently small. However, the soil surface near a trench in clay sometimes shows tensile cracks. This indicates that soil may also fail in tension instead of in shear. Such behaviour can be included in a PLAXIS 2D analysis by selecting the *Tension cut-off* option. When selecting the *Tension cut-off* option the allowable tensile strength may be entered. The default value of the tensile strength is zero.

Miscellaneous parameters (use default settings):

K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - \sin(\varphi)$)	[-]
M	K_0^{nc}- related parameter (no direct input possible)	[-]

The parameter M is the inclination of the line passing through both the leftmost and topmost points of the ellipse forming the hardening cap of the Soft Soil Creep model. The value of M is derived from the value of K_0^{nc} and is given here for comparison with the value M_{CSL} of for instance the Modified Cam-Clay model (MCC) (on page 184).

Sekiguchi-Ohta model (Inviscid) [ADV]

The **Mechanical** tabsheet for the Sekiguchi-Ohta model (Inviscid) is shown in Figure 131 (on page 192).

Note:

- Optional drainage types when Sekiguchi-Ohta model (Inviscid) is selected are: *Drained* and *Undrained A*.
- For both the *Drained* and *Undrained A* drainage type the same parameters are used.

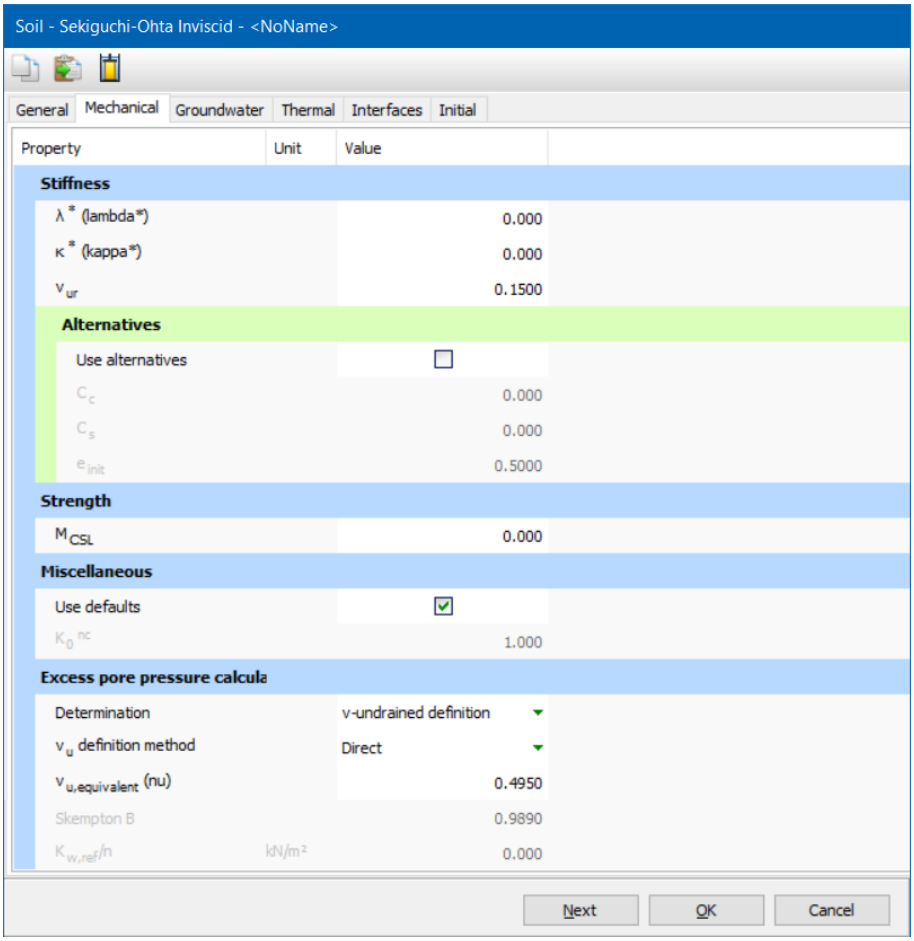


*Figure 131: **Mechanical** tabsheet for the Sekiguchi-Ohta model (drained behaviour)*

Parameters for stiffness:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]
ν_{ur}	Poisson's ratio for unloading-reloading	[-]

Alternative parameters can be used to define soil stiffnes:

C_C	Compression index	[-]
C_s	Swelling index or reloading index	[-]
e_{init}	Initial void ratio	[-]

The Sekiguchi-Ohta Inviscid model has a single strength parameter:

M_{CSL}	Tangent of critical state line	[-]

Miscellaneous parameters:

K_0^{nc}	Coefficient of lateral stress in normal consolidation	[-]

Sekiguchi-Ohta model (Viscid) [ADV]

The **Mechanical** tabsheet for the Sekiguchi-Ohta model (Viscid) is shown in Figure 132 (on page 193).

Note:

- Optional drainage types when Sekiguchi-Ohta model (Viscid) is selected are: *Drained* and *Undrained A*.
- For both the *Drained* and *Undrained A* drainage type the same parameters are used.

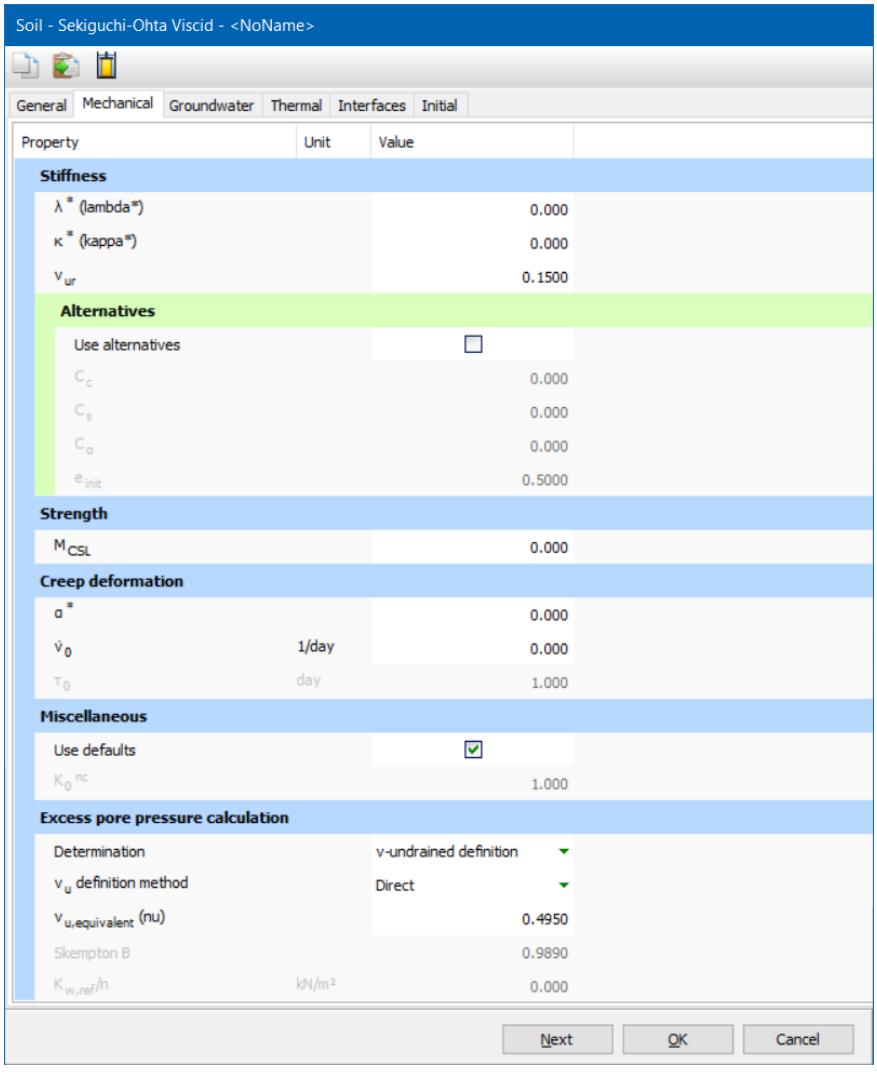


*Figure 132: **Mechanical** tabsheet for the Sekiguchi-Ohta model (Viscid) (drained behaviour)*

The stiffness parameters of the Sekiguchi-Ohta model (Viscid) are:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]
ν_{ur}	Poisson's ratio for unloading-reloading	[-]

Alternative parameters can be used to define soil stiffness:

C_C	Compression index	[-]
C_s	Swelling index or reloading index	[-]
C_{α^*}	Secondary compression index	[-]
e_{init}	Initial void ratio	[-]

The model has a single parameter for soil strength:

M_{CSL}	Tangent of the critical state line	[-]

Parameters for creep deformation:

α^*	Coefficient of secondary compression	[-]
$\dot{v}_0$	Initial volumetric strain rate	[day^{-1}]

Miscellaneous parameters (using default is recommended):

K_0^{nc}	Coefficient of lateral stress in normal consolidation	[-]

UDCAM-S model [ADV]

The UDCAM-S model can be used for capacity, deformation and soil-structure interaction analysis involving undrained loading of clay-type materials for the design of offshore structures. The **Mechanical** tabsheet for the UDCAM-S model is shown in Figure 133 (on page 195). The parameters are obtained from the *Cyclic accumulation and optimisation tool*, which can be accessed by clicking on the corresponding button in the right-hand panel.

Note:

The UDCAM-S model can only be used with the drainage type *Undrained C*.

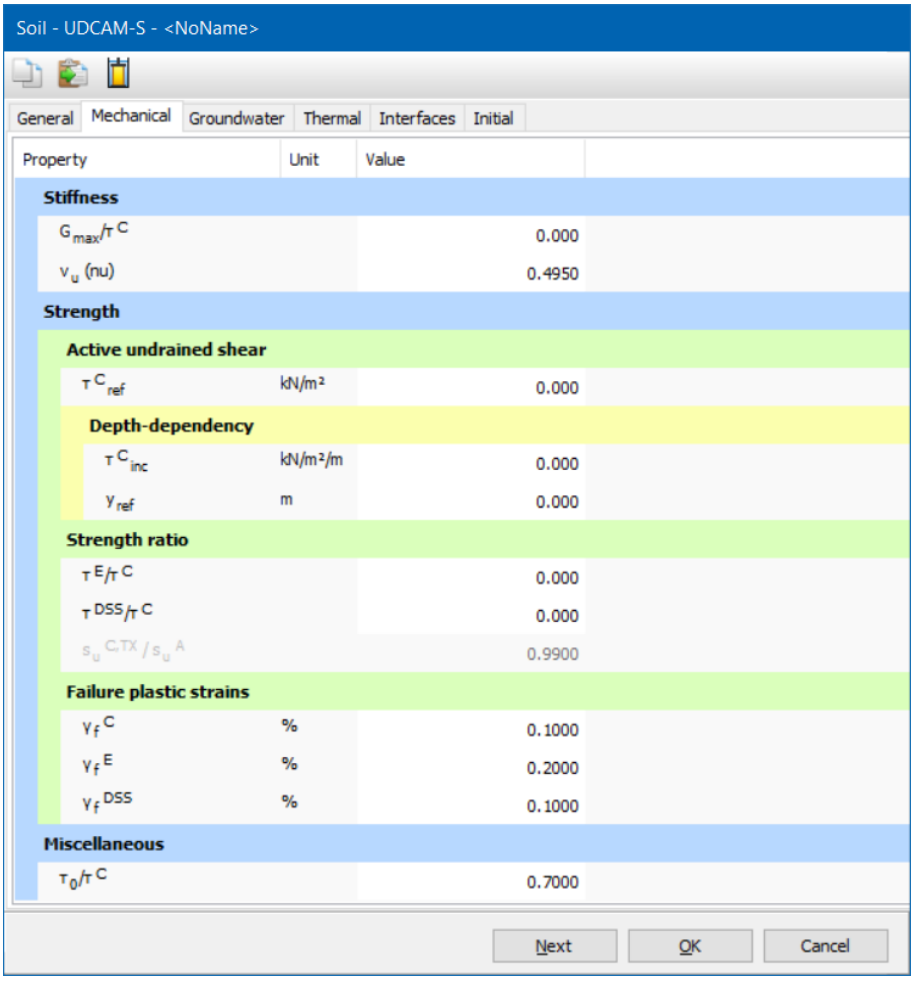


Figure 133: ***Mechanical*** *tabsheet for the UDCAM-S model*

Parameters for stiffness:

G_{max}/τ^C	Ratio unloading/reloading shear modulus over (plane strain) active shear strength	[-]
ν_u	Undrained Poisson's ratio	[-]

Parameters for strength:

τ_{ref}^C	Degraded reference cyclic shear strength in triaxial compression	[kN/m^2]
τ_{inc}^C	Increase of degraded cyclic shear strength in triaxial compression with depth	[kN/m^2/m]
y_{ref}	Reference level for depth-dependent shear strength	[m]
τ^E/τ^C	Ratio of degraded cyclic triaxial extension shear strength over degraded cyclic triaxial compression shear strength	

τ^{DSS}/τ^{C}	Ratio of degraded cyclic direct simple shear strength over degraded cyclic triaxial compression shear strength	[-]
τ_0/τ^{C}	Initial mobilisation	[-]
γ_f^{C}	Shear strain at failure in triaxial compression	[%]
γ_f^{E}	Shear strain at failure in triaxial extension	[%]
γ_f^{DSS}	Shear strain at failure in direct simple shear	[%]

Miscellaneous parameters:

τ_0/τ^{C}	Initial mobilisation	[-]

Concrete model [ADV]

The **Mechanical** tabsheet for the Concrete model are shown in Figure 134 (on page 197).

Note:

Available drainage types when Concrete model is selected are: *Drained*, in case of semi-permeable walls, and *Non-porous*, which is the general approach for concrete structural elements.

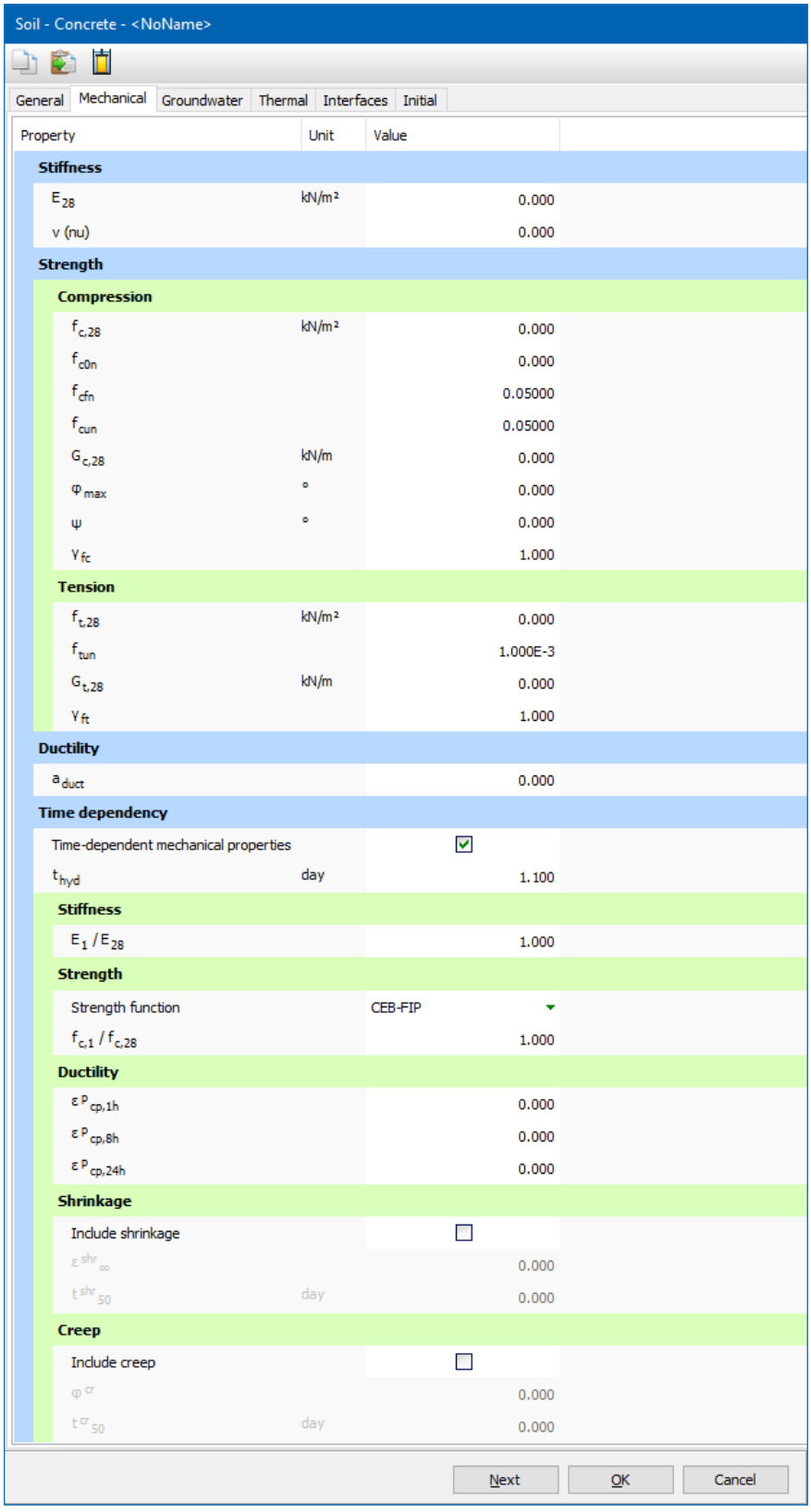


*Figure 134: **Mechanical** tabsheet for the Concrete model*

Parameters for stiffness

E_{28}	Young's modulus of cured concrete at t_{hydr}	[kN/m^2]
ν	Poisson's ratio	[-]

Parameters for strength:

Compression		
$f_{c,28}$	Uniaxial compressive strength of cured concrete at t_{hydr}	[kN/m^2]
f_{c0n}	Normalised failure strength	[-]
f_{cfn}	Normalised initially mobilised strength	[-]
f_{cun}	Normalised residual strength	[-]
$G_{c,28}$	Compressive fracture energy of cured concrete at t_{hydr}	[kN/m]
φ_{max}	Maximum friction angle	[°]
ψ	Dilatancy angle	[°]
γ_{fc}	Safety factor for compressive strength	[-]
Tension		
$f_{t,28}$	Uniaxial tensile strength of cured concrete at t_{hydr}	[kN/m^2]
f_{tun}	Ratio of residual vs. peak tensile strength	[-]
$G_{t,28}$	Tensile fracture energy of cured concrete at t_{hydr}	[kN/m]
γ_{ft}	Safety factor for tensile strength	[-]

The model has a single parameter for ductility:

a_{duct}	Increase of peak strain ε_{cp} with the increase of mean effective stress p	[-]

Parameters for time-dependent behaviour:

t_{hydr}	Time for full hydration	[day]
Stiffness		
E_1 / E_{28}	Ratio of the Young's modulus after 1 day curing over the Young's modulus at full hydration	
Strength		

Strength function	Strength development function for the first 24 hours of curing	
$f_{c,1}/f_{c,28}$	Ratio of the compressive strength after 1 day curing over the compressive strength at full hydration	[-]
Ductility		
$\varepsilon^{p}_{cp,1h}$	Uniaxial plastic failure strain at 1h (negative value)	[-]
$\varepsilon^{p}_{cp,8h}$	Uniaxial plastic failure strain at 8h (negative value)	[-]
$\varepsilon^{p}_{cp,24h}$	Uniaxial plastic failure strain at 24h (negative value)	[-]
Shrinkage		
Include shrinkage	To be selected if shrinkage should be taken into account	
$\varepsilon^{shr}_{\infty}$	Final shrinkage strain	[-]
t^{cr}_{50}	Time for 50% of shrinkage strains	[day]
Creep		
Include creep	To be selected if creep should be taken into account	
φ^{cr}	Ratio between creep and elastic strains	[-]
t^{cr}_{50}	Time for 50% of creep strains	[day]

Note:

- When simulating shotcrete the tensile strength is essential for tunnel stability. Neglecting or considering low values for tensile strength could result in unrealistic failure.
- The creep history is adjusted for the stress state at first activation of the concrete cluster, such that no creep strains are produced by initial stresses. The state variables are taken over if the previous material was also defined with the Concrete model, in which case creep will also continue. If a reset of state variables is desired, a nil step with a different material (e.g. linear elastic) is required.

User-defined soil model (UDSM) [ADV] + [GSE]

The **Mechanica** tabsheet shows two drop-down menus; the top combo box lists all the DLLs that contain valid User-defined Soil Model and the next combo box shows the models defined in the selected DLL. Each UD model has its own set of model parameters, defined in the same DLL that contains the model definition.

When an available model is chosen PLAXIS 2D will automatically read its parameter names and units from the DLL and fill the parameter table below. For a detailed description of this facility, reference is made to the Material Models Manual.

Note:

Available drainage types when user-defined soil model is selected are: *Drained*, *Undrained A* and *Non-porous*.

UBC3D-PLM model [ULT]

The **Mechanical** tabsheet for the UBC3D-PLM model is shown in Figure 135 (on page 201).

Note:

Available drainage types when UBC3D-PLM model is selected are: *Drained* and *Undrained A*.

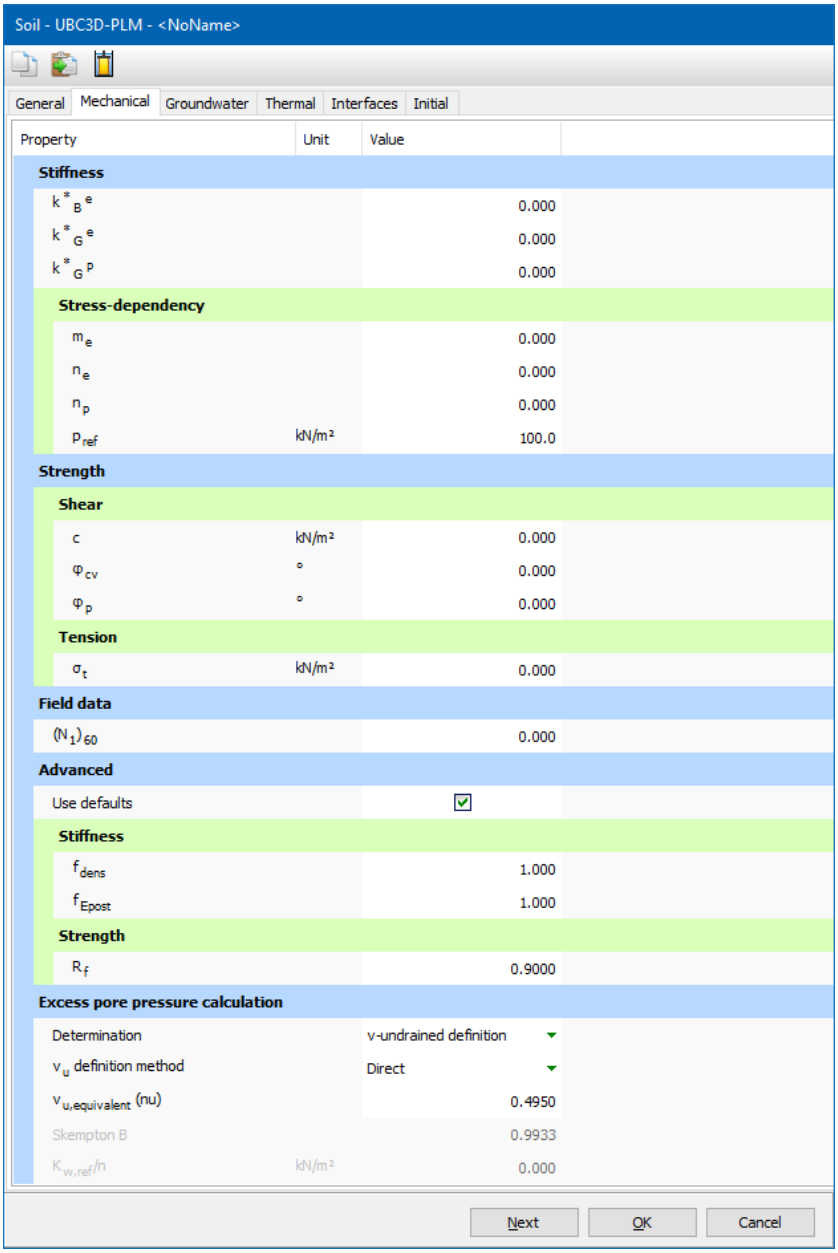


Figure 135: Mechanical tabsheet for the UBC3D-PLM model

The stiffness parameters of the UBC3D-PLM model are:

k_B^{*e}	Elastic bulk modulus factor	[-]
k_G^{*e}	Elastic shear modulus factor	[-]
k_G^{*p}	Plastic shear modulus factor	[-]

me	Power for stress dependent elastic bulk modulus	[-]
ne	Power for stress dependent elastic shear modulus	[-]
np	Power for stress dependent plastic shear modulus	[-]
p_{ref}	Reference pressure for stress dependent stiffness	[kN/m^2]

Note:

The implicit Poisson's ratio that is defined based on k^e_B and k^e_G is suitable for dynamics calculation, but it does not generate a proper initial stress state if the initial stress condition is established by *gravity loading* procedure. In such a case the user should define another material set for the stress initialization step with proper characteristics.

Parameters for strength:

c	Cohesion	[kN/m^2]
φ_{cv}	Constant volume friction angle	[°]
φ_p	Peak friction angle	[°]
σ_t	Tensile strength	[kN/m^2]

The model requires a single parameter from field data in order to model the effects of densification:

$(N_1)_{60}$	Corrected SPT blow count	[-]

Advanced parameters:

f_{dens}	Densification factor	[-]
f_{Epost}	Post-liquefaction stiffness factor	[-]
R_f	Failure ratio	[-]

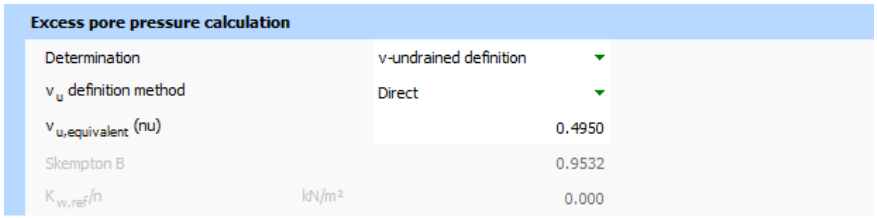
Parameters for Excess pore pressure calculation

The excess pore pressure calculation parameters available in the **Mechanical** tabsheet can be used to model the generation of excess pore pressures in a calculation that involves the *Undrained behaviour* of soils or a coupled analysis. For the calculation of excess pore pressures in an effective stress calculation, the calculation will add a bulk modulus of water to the stiffness matrix. This bulk modulus can be obtained in 2 different ways (Figure 136 (on page 203)), namely using the options *ν-undrained definition* and *Biot effective stress concept.* When the first option is chosen the user should decide whether to enter the $\nu_{u,equivalent}$ either directly or through the *Skempton B* value. Note that we refer here to the equivalent undrained Poisson's ratio as it is not directly used as parameter to calculate excess pore pressures, but merely to determine the bulk modulus of water that is then used to calculate excess pore pressures.

By default, the solid material of the soil (i.c the grains) is supposed to be incompressible ($\alpha_{Biot} = 1$) and the equivalent undrained Poisson's ratio is taken as *0.495*. This is the default behaviour. Alternatively, one can select to define the equivalent undrained Poisson's ratio through the Skempton's B-parameter and $K_{w,ref}/n$ is calculated accordingly. In the most advanced option, *Biot effective stress concept*, the compressibility of the solid material is considered and one can select α_{Biot} together with K_w after which $\nu_{u,equivalent}$, *Skempton B* and $K_{w,ref}/n$ are calculated accordingly.

α_{Biot}	Biot alpha pore pressure coefficient	[-]
$\nu_{u,equivalent}$	Equivalent undrained Poisson's ratio	[-]
Skempton-B	A parameter that determines which portion of a change in mean stress is carried by the pore water	[-]
$K_{w,ref} / n$	The corresponding reference bulk stiffness of the pore fluid	[kN/m^2]
K_w	Bulk modulus of water	[-]

More detailed information is available in the Material Models Manual.



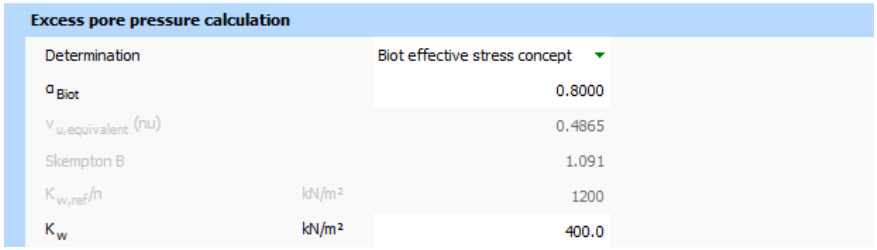


Figure 136: Parameters for the Excess pore pressure calculation

6.1.3 *Hoek-Brown pre-processing tool*

The Hoek-Brown model is the most used failure criterion for rock masses, nevertheless there are some uncertainties regarding the input parameters that require a consolidated experience (Hoek, Carranza-Torres & Corkum (2002); Hoek (2007)). For this reason, PLAXIS 2D implements in the *Mechanical* tabsheet of the Hoek-Brown model a pre-processing tool to guide the user in the determination of the rock mass strength and stiffness parameters. PLAXIS 2D sign convention is adopted, i.e. compressive stresses are considered to be negative. The pane contains the following tabsheets:

- *Analysis*: shows the Hoek-Brown failure envelope in the plane of principal effective stresses σ'_3 - σ'_1, in order to visualise the effects of changing of rock mass parameters on the failure envelope. For more information about the Hoek-Brown modeland its formulation, refer Material Models Manual.
- The second tabsheet is specific to determine the selected parameter.

Uni-axial compressive strength of intact rock $|\sigma_{ci}|$

The uni-axial compressive strength of intact rock $|\sigma_{ci}|$ can be checked based on the estimation methods generally executed in the field. Alternatively, a range can be selected and PLAXIS 2D automatically set the parameter to the middle value of the range. In case the upper bound limit of 250 MPa is selected, the parameter is set to 250 MPa.

Table 5 (on page 204) reports the range of compressive strength for each category with a qualitative description of the strength behaviour to field test.

Table 5: $|\sigma_{ci}|$ tabsheet values

Field estimate	Examples	Strength [MPa]
Only chipping is possible with a geological hammer	Chert, diabase, fresh basalt, gneiss, granite, quartzite	$\|\sigma_{ci}\| \geq 250$
Fracturing requires many blows of a geological hammer	Amphibolite, basalt, gabbro, gneiss, granodiorite, limestone, marble, rhyolite, sandstone, tuff	$100 \leq \|\sigma_{ci}\| \leq 250$
Fracturing requires more than one blow of a geological hammer	Limestone, marble, phyllite, sandstone, schist, shale	$50 \leq \|\sigma_{ci}\| \leq 100$
Fracturing is possible with a single blow from a geological hammer, but cannot be scraped or peeled with a pocket knife	Claystone, coal, concrete, schist, shale, siltstone	$25 \leq \|\sigma_{ci}\| \leq 50$
Firm blow with the point of a geological hammer leaves shallow indentation; peeling with a pocket knife is possible, but difficult	Chalk, potash, rocksalt	$5 \leq \|\sigma_{ci}\| \leq 25$
Firm blow with the point of geological hammer leads to crumbling; peeling with a pocket knife is possible	Highly weathered or altered rock	$1 \leq \|\sigma_{ci}\| \leq 5$
Thumbnail leaves indentation	Stiff fault gouge	$0.25 \leq \|\sigma_{ci}\| \leq 1$

Intact rock parameter m_i

The intact rock parameter m_i can be estimated based on the rock type.

Table 6: Values of the constant m_i for intact rock (Marinos & Hoek (2001), Wyllie & Mah (2004))

Name	Rock type	Texture	m_i	m_i \pm
Agglomerate	Igneous	Coarse	19	3

Name	Rock type	Texture	m_i	m_i \pm
Amphibolites	Metamorhpic	Medium	26	6
Andesite	Igneous	Medium	25	5
Anhydrite	Sedimentary	Fine	12	2
Basalt	Igneous	Fine	25	5
Breccia	Igneous	Medium	19	5
Breccia	Sedimentary	Coarse	19	5
Chalk	Sedimentary	Veryfine	7	2
Claystones	Sedimentary	Veryfine	4	2
Conglomerates	Sedimentary	Coarse	21	3
Crystallinelimestone	Sedimentary	Coarse	12	3
Dacite	Igneous	Fine	25	3
Diabase	Igneous	Fine	15	5
Diorite	Igneous	Medium	25	5
Dolerite	Igneous	Medium	16	5
Dolomites	Sedimentary	Veryfine	9	3
Gabbro	Igneous	Coarse	27	3
Gneiss	Metamorhpic	Fine	28	5
Granite	Igneous	Coarse	32	3
Granodiorite	Igneous	Coarse,Medium	29	3
Greywackes	Sedimentary	Fine	18	3
Gypsum	Sedimentary	Medium	8	2
Hornfels	Metamorhpic	Medium	19	4
Marble	Metamorhpic	Coarse	9	3
Marls	Sedimentary	Veryfine	7	2
Metasandstone	Metamorhpic	Medium	19	3
Mictriclimestones	Sedimentary	Fine	9	2

Name	Rock type	Texture	m_i	$m_i \pm m$
Migmatite	Metamorhpic	Coarse	29	3
Norite	Igneous	Coarse,Medium	20	5
Obsidian	Igneous	Veryfine	19	3
Peridotite	Igneous	Veryfine	25	5
Phyllites	Metamorhpic	Fine	7	3
Porphyries	Igneous	Coarse,Medium	20	5
Quartzites	Metamorhpic	Fine	20	3
Rhyolite	Igneous	Medium	25	5
Sandstones	Sedimentary	Medium	17	4
Schists	Metamorhpic	Medium	12	3
Shales	Sedimentary	Veryfine	6	2
Siltstones	Sedimentary	Fine	7	2
Slates	Metamorhpic	Veryfine	7	4
Spariticlimestones	Sedimentary	Medium	10	2
Tuff	Igneous	Fine	13	5

Geological Strength Index *GSI*

The *GSI* tabsheet allows to choose between two rock types: *General* Figure 137 (on page 207) and *Flysch* Figure 138 (on page 208). An intact rock is equivalent to *GSI* = 100, whereas a soil structure is in proximity to *GSI* = 0. Based on the selected rock type, the *GSI* can be chosen taking the structure and the surface conditions of the rock mass into account. An hint box at the bottom of the chart displays the rock characteristics for the selected *GSI*.

Note:

The *GSI* charts allows only integer values. If a higher accuracy is required, a decimal value can be manually put in the *Input* tabsheet of the material set.

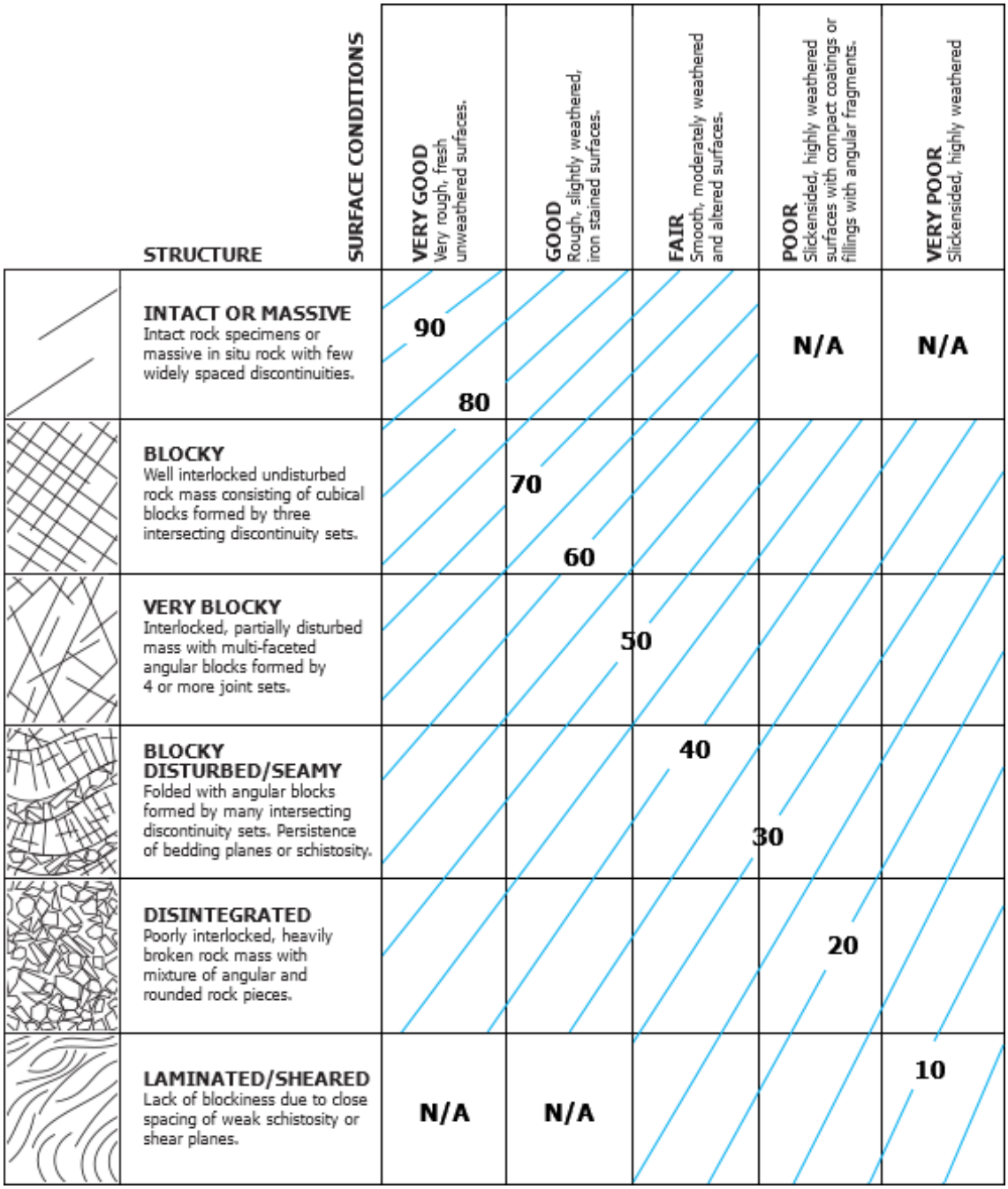


Figure 137: GSI chart for General rock type

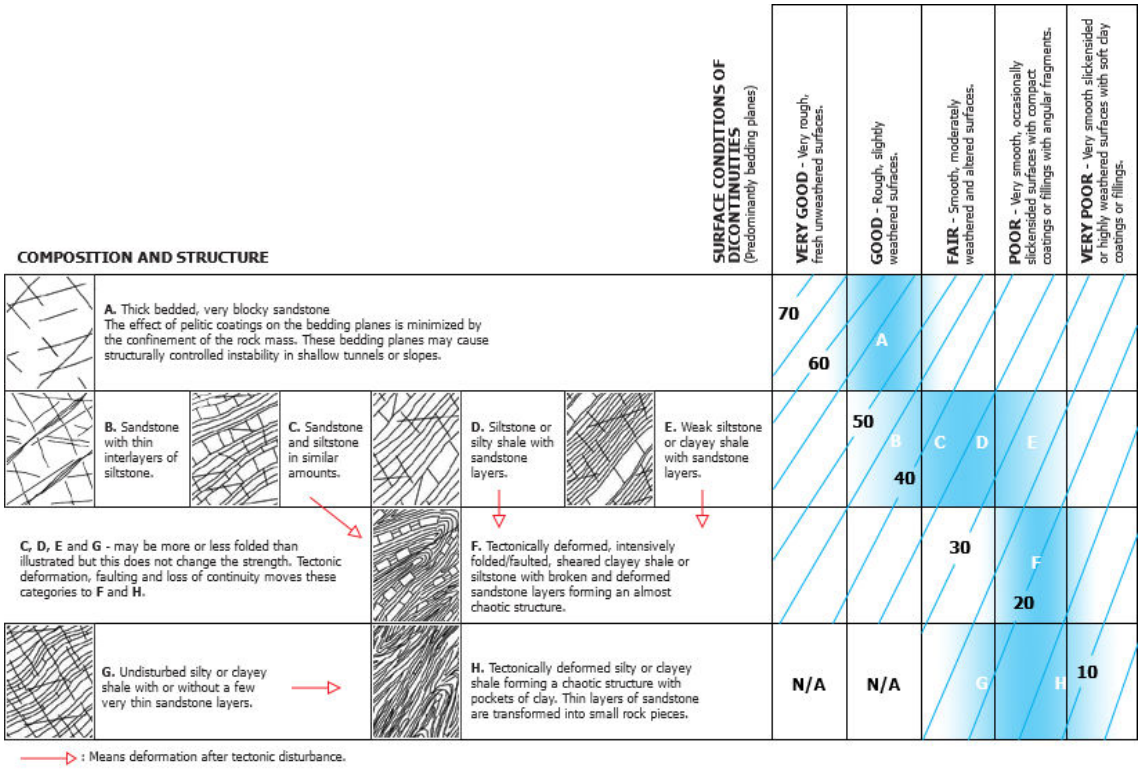


Figure 138: GSI chart for Flysch rock type

Disturbance factor (*D*)

The disturbance factor *D* depends on the amount of disturbance of the rock as a result of mechanical processes in open excavations, tunnels or mines, such as blasting, tunnel boring, machine driven or manual excavation. No disturbance is equivalent to $D = 0$, whereas severe disturbance is equivalent to $D = 1$. The *D* tabsheet allows a direct estimation of the disturbance factor for tunnel Table 7 (on page 209) and slopes Table 8 (on page 209).

In blast damaged rock, there are two main issues when applying the disturbance factor *D*:

- Choose a suitable value: a large number of factors can influence the degree of disturbance (quality of the rock mass and the excavation/blasting, loading sequence, lateral confinement produced by different radii of curvature of slopes as compared with their height, the level of strain in the failure zone, etc.). For this reason, the disturbance factor *D* is inferred on the basis of engineering experience and research Table 7 (on page 209)Table 8 (on page 209).
- Define the extent of the damaged zone: the disturbance factor *D* should only be applied to the actual zone of damaged rock in order to avoid underestimation of the strength and stability of the overall rock mass greatly. The thickness *T* of the blast damaged zone depends upon the design of the blast (Hoek & Karzulovic, 2000). In the case of bench blasting in open pit mines and civil engineering slopes, Table 8 (on page 209) contains suggested values, where *H* is the height of the slope. In the case of tunnels excavated by drill and blast methods, for very high quality controlled blasting the damage to the tunnel wall is negligible due to a well-designed blasting pattern, detonation sequence and accurate drilling control. In contrast, the lack of a good blast design and absence of any control on the drilling can result in significant damage.

Table 7: Distrubance factor *D*, guidelines for tunnels

Description	D
Tunnel excavation by TBM or blasting of excellent quality	0.0
Tunnel excavation by hand or using a mechanical process rather than blasting, in poor quality rock. There are no squeezing problems leading to floor heave, or these are mitigated using a temporary invert	0.0
Tunnel excavation by hand or using a mechanical process rather than blasting, in poor quality rock. There are unmitigated squeezing problems leading to floor heave	0.5
Tunnel excavation using blasting of very poor quality, leading to severe local damage	0.8

Table 8: Disturbance factor *D*, guidelines for slopes

Description	D
Slope created using controlled, small scale blasting of good quality	0.7
Slope created using small scale blasting of poor quality	1.0
Slope in very large open pit mine, created using mechanical excavation in softer rocks	0.7
Slope in very large open pit mine, created using heavy production blasting	1.0

Table 9: Damaged thickness *T*, guidelines for bench blasting in open pit mines and civil engineering slopes

Description	T
Large production blast, confined and with little or no control	2.0 to 2.5 H
Production blast with no control but blasting to a free face	1.0 to 1.5 H
Production blast, confined but with some control, e.g. one or more buffer rows	1.0 to 1.2 H
Production blast with some control, e.g. one or more buffer rows, and blasting to a free face	0.5 to 1.0 H
Carefully controlled production blast with a free face	0.3 to 0.5 H

6.1.4 *Cyclic accumulation and optimisation tool* [ADV]

The *Cyclic accumulation and optimisation tool* is used to determine the UDCAM-S model parameters. The model is especially suited for the foundation design of offshore structures subjected to a design storm (i.e. a combination of a wave, wind loading, and currant). The stiffness and strength of saturated soils under cyclic loading are different from the static case: they may increase due to strain rate effects or reduce due to degradation processes, as pore pressure build up and destructuration. The soil beneath structures subjected to

cyclic loading experiences different behaviour following different stress paths. For a general stress path, the behaviour may be interpolated between triaxial compression (TxC), triaxial extension (TxE) and direct simple shear (DSS) conditions.

The *Cyclic accumulation and optimisation tool* offers the possibility to check the effect of a specified input load history for a specific soil type and, through an accumulation and interpolation procedure, it provides stress-strain curves for the different characteristic stress paths. The UDCAM-S model parameters are optimised in order to match the stress-strain curves.

The tool is suitable only for clay and low permeable silts in undrained condition since the effect of drainage is not taken into account. The accumulation procedure is based on the cyclic shear strains.

Note:

In addition to the cyclic shear strain accumulation procedure, the pore pressure accumulation and average shear strain accumulation procedures can be found in literature. The reason for using different procedures is that the development of one of these parameters may be more prominent than the others, depending on soil type, static and cyclic shear stresses, etc. (Andersen, 1991 (on page 530)). The cyclic shear strain accumulation procedure has been extensively used for clays, while the pore pressure accumulation procedure has been used for sands since drainage is likely to occur during the complete load history (even though not within each cycle) in this type of soils. In principle, the pore pressure accumulation procedure could also be used for clays. In practice, however, accurate laboratory measurement of pore pressure is harder to perform in clays than in sand. Since drainage is not likely to occur during the cyclic load history in clays, it is preferable to use the cyclic strain accumulation procedure (Andersen, 2004 (on page 530)).

Starting the tool

To start the tool, the *Cyclic accumulation and optimisation tool* is clicked from the side panel in the **Mechanical** tabsheet of the UDCAM-S model . The window consists of two blue tabs and one orange tab, see Figure 139 (on page 211) for an overview.

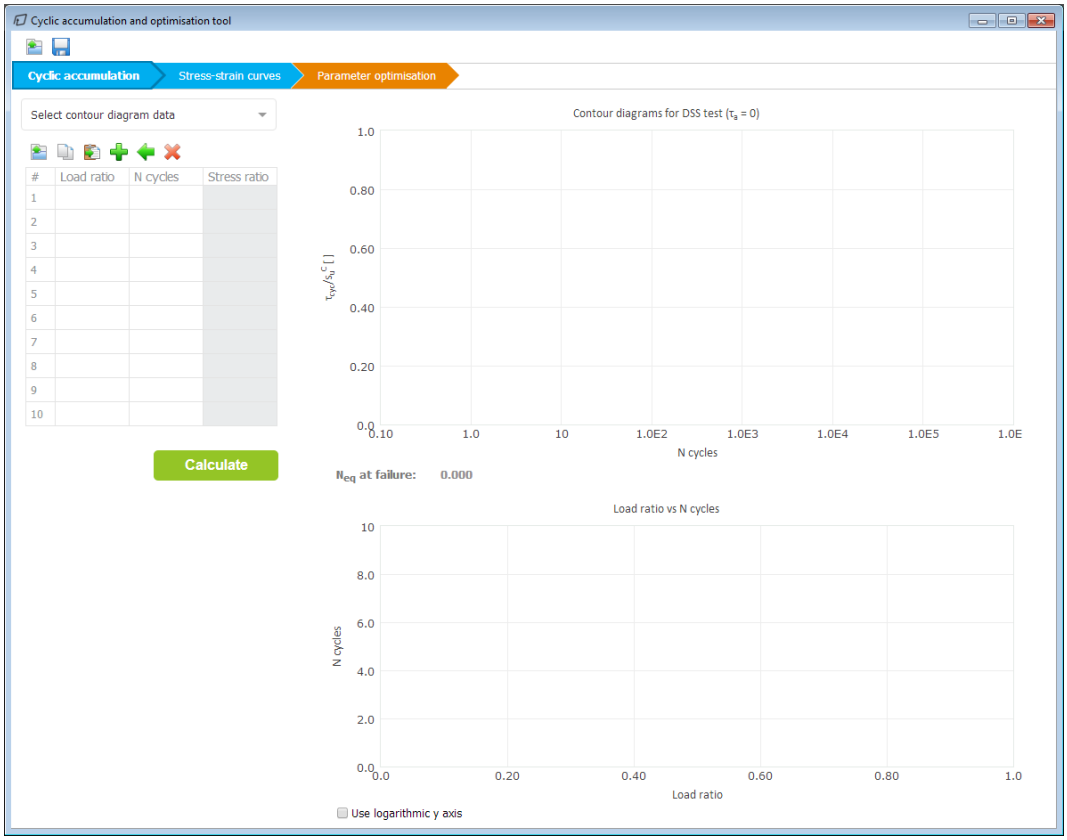


Figure 139: Cyclic accumulation and optimisation tool

The first blue tabsheet, *Cyclic accumulation*, gives information regarding the input load history and the soil type. The second blue tabsheet, *Stress-strain curves*, gives information about the stress-strain curves for different stress paths. The orange tabsheet *Parameter optimisation*, allows the user to optimise the UDCAM-S model parameters.

Above the tabsheets there are *Load* and *Save* buttons (see Figure 140 (on page 211)) to load or save the app state. Saving and loading the app state is useful when running the same or similar calculations again, after the project has been closed and reopened. Also, similar calculations in different projects can be done more easily by loading a previously saved state.

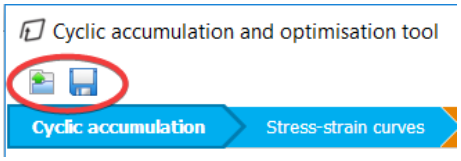


Figure 140: Load and save app state

The state is saved in the JSON file format with the .json extension. All the settings and data on the tabsheet, like table values and graph data are saved and can be loaded back into the tool.

Cyclic accumulation

In the design process of offshore structures, the design load is often a storm which can be transformed or idealised into packages or parcels. Each parcel corresponds to a number of cycles at a load constant amplitude.

The maximum cyclic load and the number of cycles at each load level are determined from a time record of loads. The counting procedure to obtain the number of cycles can be done following different methods (e.g. the rainflow-counting algorithm (Matsuishi & Endo (1968) (on page 531))) or the recently developed method, for more information see Noren - Cosgriff, Josatd & Madshus (2015) (on page 531)).

The effect of the design storm on the soil deposit can be described through the equivalent number of cycles N_{eq} associated to the last applied normalised load amplitude F/F_{max} and the soil cyclic contour diagram. The equivalent number of cycles N_{eq} is calculated at failure for capacity calculations. If it is found to be conservative, the same value may also be used in stiffness calculations.

Note:

It is assumed here that N_{eq} is representative for all the elements in the selected soil polygon. Even though this assumption may underestimate the effect of the stress redistribution and progressive failure, it has been shown that the results are in good agreement with the ones from the model tests (Andersen, Puech & Jardine, 2013 (on page 530)) and can certainly be used for stiff shallow foundations like offshore structures, while for monopiles and piles this is generally not correct.

The cyclic contours (Figure 141 (on page 212)) are a set of curves where each of them represents the locus of points characterised by the same cyclic shear strain. They can be visualized in a chart with the number of cycles on the x-axis (in a logarithmic scale) and τ_{cyc}/s_u on the y-axis. Each point of the contour is then characterised by a certain number of cycles and a normalised shear stress that induce a defined cyclic shear deformation in the soil. The contours are defined for DSS tests with the average stress τ_a equals to 0 (for Drammen clay).

Figure 141: Contour diagram for DSS test for Drammen Clay with OCR = 4

Four lines limit the contours:

- Each contour line starts with a point at N equal to 1 and an ending point at N equal to the number of cycles above which all the contours are assumed to become horizontal (i.e. τ_{cyc} is constant). This leads to two vertical boundaries, $N = 1$ and $N = N_{max}$.
- The first contour is always defined at zero stress level, i.e. the contour line at the base where τ_{cyc}/s_u is equal to zero and $\gamma_{cyc} = 0\%$.
- The last contour represents the locus of points corresponding to γ equal to the cyclic shear strain at failure (typically 15%).

Contour diagram data

The cyclic behaviour of clays depends on OCR, as well as the other index properties such as Plasticity Index, clay content, water content, etc. NGI has extensively studied the cyclic behaviour of Drammen clays, realizing contour diagrams from DSS tests. These contour data for Drammen clay with OCR equal to 1, 2, 4, 10, 20 and 40 are available in the tool and can be selected from the drop-down menu.

In the case of other clays, it is possible to upload custom contours. In general, the contours can be determined by different tests at different average shear stress levels, τ_a. For Drammen clay, N_{eq} was found to be relatively independent of τ_a and the type of test (Andersen, Kleven & Heien, 1988 (on page 530)) and the shear strain at failure was taken as 15%.

Note: When uploading custom contours, a specific data format has to be used. For more information, see Cyclic accumulation Tool - Contour File Formats [ADV] (on page 536).

By moving the mouse over any line in the contour chart, a hint box appears showing the value of γ_{cyc} that describes that contour and the corresponding N_{cycles} and τ_{cyc}/s_u values.

Note:

- The shear strain at a given degree of shear strength mobilisation, τ / s_u, increases with increasing OCR, meaning that the normalised secant modulus, G/s_u, decreases with increasing OCR for a given degree of strength mobilisation. However, the absolute value of the stiffness will increase with increasing OCR for given consolidation stress and strength mobilisation but significantly less than the increase in shear strength (Andersen, 2015 (on page 530)).
- Even though in most cases the strength of the soil under cyclic loading may be reduced compared to the static value, because of the degradation process, the cyclic shear strength can also be higher than the static shear strength (i.e. $\tau_{cy} / s_u > 1$) for a low number of cycles in some cases. The reason for this is that the clay strength is rate dependent. Since the cyclic tests are typically run with a load period of 10 seconds and the monotonic tests are brought to failure in about 2 hours, the cyclic strength may thus be higher than the static shear strength (Andersen, 2007 (on page 530)).

Load ratio (F/F_{max})

The idealised load history, described by a series of parcels, is specified in a table that consists of four columns. The first one indicates the number of the load parcel, the second one represents the load ratio value F / F_{max} for each parcel, the third one shows the corresponding number of cycles and the last one is the output of the calculation performed in this tabsheet (i.e. the stress ratio). The load ratio F/F_{max} refer to pure cyclic loading condition (horizontal force, vertical force or bending moment). The user should define which load direction is more crucial for the model and calculate consequently F/F_{max}. For capacity analysis, when F is equal to F_{max} (i.e. the load ratio is equal to 1), failure has been reached and the shear strain is equal to its failure value γ_f. The normalized shear strength τ/s_u at γ_f corresponds to its maximum value. A linear relationship between the load and the shear stress is assumed, so that:

$$\left(\frac{\tau}{s_u}\right)_{\max} = \chi \frac{F}{F_{\max}} \qquad Eq.\ [15]$$

where χ is the scaling factor automatically calculated during the strain accumulation procedure. At the end of the accumulation procedure, the maximum stress ratio τ/s_u is displayed in the corresponding column in the Figure 142 (on page 214) .

The buttons in the toolbar can be used to modify the Figure 142 (on page 214).

#	Load ratio	N cycles	Stress ratio
1	0.02000	2371	
2	0.1100	2877	
3	0.2600	1079	
4	0.4000	163	
5	0.5100	64	
6	0.6200	25	
7	0.7500	10	
8	0.8900	3	
9	1.000	1	
10			

Figure 142: Input table for load parcels of the cyclic accumulation and optimisation tool

- Click the *Add row* button in the toolbar to add a new row in the table.
- Click the *Insert* button to insert a new row before the selected row in the table.
- Click the *Delete* button to delete the selected row in the table.

The values can be defined by clicking the cell in the table and by typing the value. A maximum of 100 load parcels can be entered. The load ratio can be defined as any positive value larger than zero; its maximum value is 1. The number of cycles is a positive integer larger than zero.

As soon as a parcel is specified in the table, the graph "Load ratio vs N cycles'' at the bottom of the same tab will be updated. There is an option to choose for a logarithmic representation on the y-axis. See an example of where a series of parcels were defined in Figure 143 (on page 215).

- If the data for the load parcels exists in a file, it can be loaded by clicking the *Open file*. Any delimited tabular text file or an excel spreadsheet file can be loaded. When the selected file is opened, a window pops up where the user can choose the columns to extract the data from, see Figure 142 (on page 214). Also, the field separator (delimiter) and start row can be defined. A preview of what is going to be imported is shown.
- A load history, defined in the table, can be copied using the *Copy* button in the toolbar.

- Copied data from other applications (using *Ctrl+c*) can be imported by using the *Paste* button.

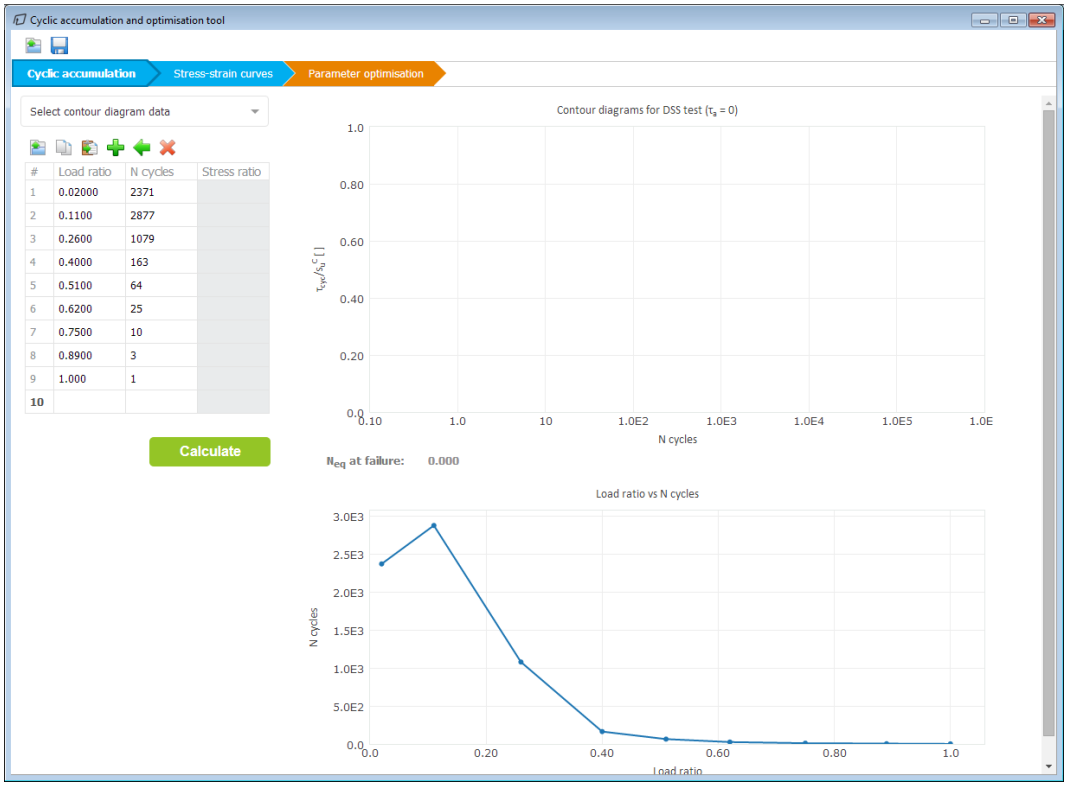


Figure 143: Automatic update of Load ratio graph when specifying parcels

Note:

If a load parcel is characterized by a number of cycles larger than the maximum value represented in the contours diagram, it is advised to decompose it into two or more load parcels with slightly different load ratios and lower number of cycles such that their sum is equal to the initial total N_{cycles}.

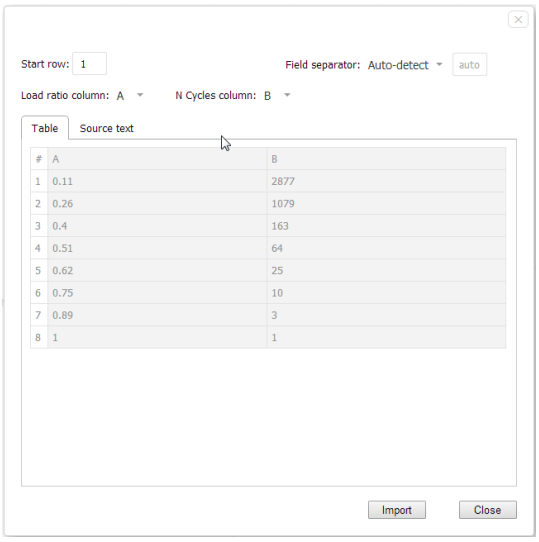


Figure 144: Import load parcels settings dialog

To start the strain accumulation procedure, click the *Calculate* button. Prior to the calculation, the tool performs some consistency checks on the input data to ensure that both the load parcel table and the contour diagram are defined.

At the end of the calculation, the box of the equivalent number of cycles N_{eq} is updated with the value resulting from the strain accumulation procedure. The last accumulation up to failure is now visible in the contour diagrams together with the locus of the end points for each accumulation (i.e. for all scaling factors used in the process).

Cyclic accumulation tab possible errors

The errors that may occur in the *Cyclic accumulation* tab are listed in Calculation warning and errors in PLAXIS (on page 602).

Stress-strain curves

The *Stress-strain curves* tabsheet allows to determine the stress-strain curves for different stress paths. The curves are determined based on contour diagrams for different laboratory test types at a given equivalent number of cycles. It is possible to choose among several Drammen clay contours (for OCR = 1, 2, 4, 10, 20 and 40) or to upload custom diagrams.

Note: When uploading custom contour diagrams, a specific data format has to be used. Please see Cyclic accumulation (on page 536) for more information

The Drammen clay contours have been built as result of a large DSS and triaxial cycling testing programme (Andersen et al, 1988 (on page 530)). Each point in the diagram is described by the average and cyclic shear stresses, τ_a and τ_{cyc}, under which the tests were run, normalised with respect to the static undrained shear strength in triaxial compression s_u^C, together with the number of cycles and the corresponding value of the average and cyclic shear strain at failure. For Drammen clay, failure is considered to be reached when either γ_a or γ_{cyc} is equal to 15%. The total maximum shear stress $\tau_{f,\,cyc}$ that can be mobilized is given by the sum of the average and the cyclic shear stress. In each diagram, the cyclic shear strength is determined for a case with a constant cyclic shear stress during the cyclic load history. In reality, during a storm, the cyclic shear stress varies from one cycle to the next one. The value of N_{eq} is therefore essential to use these diagrams. By grouping the test results for selected number of cycles, it has been established contour diagrams for N = 1, 10, 100, 1000. The

Stress-strain curves tabsheet automatically interpolates the contours for the desired N_{eq}, representative of the real cyclic load history.

Figure 145: Stress-strain curves tabsheet, with contour diagrams for Drammen clay with OCR = 4.

By default, the equivalent number of cycles is copied from the *Cyclic accumulation* tabsheet. The contour diagrams for DSS and triaxial tests are selected to be consistent with the chosen ones to perform the strain accumulation procedure. If a custom contour has been chosen, it is necessary to upload a custom contour for the strain interpolation procedure (describe format of the file). This can be done by setting N_{eq} *determination*> *Manual*. Then the option *Contour diagrams* will enable to Select contour diagrams by selecting *Custom contours* from the drop-down list.

Note:

For an advanced use of the tool, it is possible to select the *Manual* option from the N_{eq} *determination* drop-down menu. This allows to manually enter a value for N_{eq} or to modify the existing one, as well as selecting another contour diagram from the list in the corresponding drop-down menu.

If the *Soil behaviour* is *Anisotropic*, i.e. the stress-strain curves are determined both for DSS and for triaxial tests, it is generally expected that the anisotropy ratios τ^{DSS}/τ^{C} and τ^{E}/τ^{C} are different from 1. On the contrary, when choosing the *Isotropic* option in the corresponding drop-down menu, the strain interpolation is performed for the DSS test only and the anisotropy ratios are all equal to 1.

Two different scaling factors can be used to scale the y-axis of the DSS and triaxial diagrams, respectively.

The stress-strain curves are determined by specifying the ratio of the cyclic shear stress to the average shear stress for each test condition (DSS, TXC, TXE) according to the desired failure mode and loading condition. When a stress ratio different from zero is defined, the path line is automatically updated in the corresponding chart. Note that the starting point of the path line is always on the x-axis ($\tau_{cyc}/{s_u}^{C}$ and γ_{cyc} equal to zero) but it does not

always coincide with the origin of the diagram: it is represented by the intersection point between the x-axis and the contour line that corresponds to γ_a = 0%. For the DSS test, when the average strain is equal to zero also τ_a/s_u^C is equal to zero, while for the triaxial tests, the intersection generally occurs at τ_a/s_u^C different from zero. In fact, it corresponds to the initial in situ mobilization τ_0/s_u^C.

Note:

The contour diagram shows that the maximum cyclic shear strength is mobilized when large cyclic shear strains occur and when the average shear stress is small. On the contrary, the maximum average shear strength is mobilized when the average shear stress approaches the static shear strength and large average shear strains occur. For average shear stresses between zero and the static shear strength, the failure mode is a combination of average and cyclic shear strains.

The stress ratios should be defined such that $(\Delta\tau_{cyc} / \Delta\tau_a)^{DSS}$ is representative of the failure mode or, in case of difference between the application direction of the cyclic and average load (e.g. one acts mostly in the horizontal direction and the other one mainly in the vertical), it is advised to calculate τ_{cyc}/τ_a as the ratio between stresses representative for the boundary value problem. The stress ratios for the triaxial stress may be chosen in order to have strain compatibility with the DSS test, i.e. similar average and cyclic shear strains must be reached. As shown in the chart, since each stress ratio represents the inclination of the corresponding stress path line, a negative value must be entered for $(\Delta\tau_{cyc}/\Delta\tau_a)^{TXE}$ and a positive one for the others.

For establishing stress-strain curves, it is possible to choose among three load types:

- Cyclic load: the calculation follows the actual ratio between τ_{cyc} and τ_a (inclined path), but only the relationship between τ_{cyc} and γ_{cyc} is considered.
- Average load: the calculation follows the actual ratio between τ_{cyc} and τ_a (inclined path), but only the relationship between τ_a and γ_a is considered.
- Total load: the calculation considers the total strength, $(\tau_{cyc} + \tau_a)$ and the total shear strain, $(\gamma_{cyc} + \gamma_a)$.

To start generating stress-strain curves, click the *Calculate* button. Prior to the calculation, the tool performs some consistency checks on the input data to ensure that both a valid equivalent number of cycles (larger than 0) and a contour diagram have been selected.

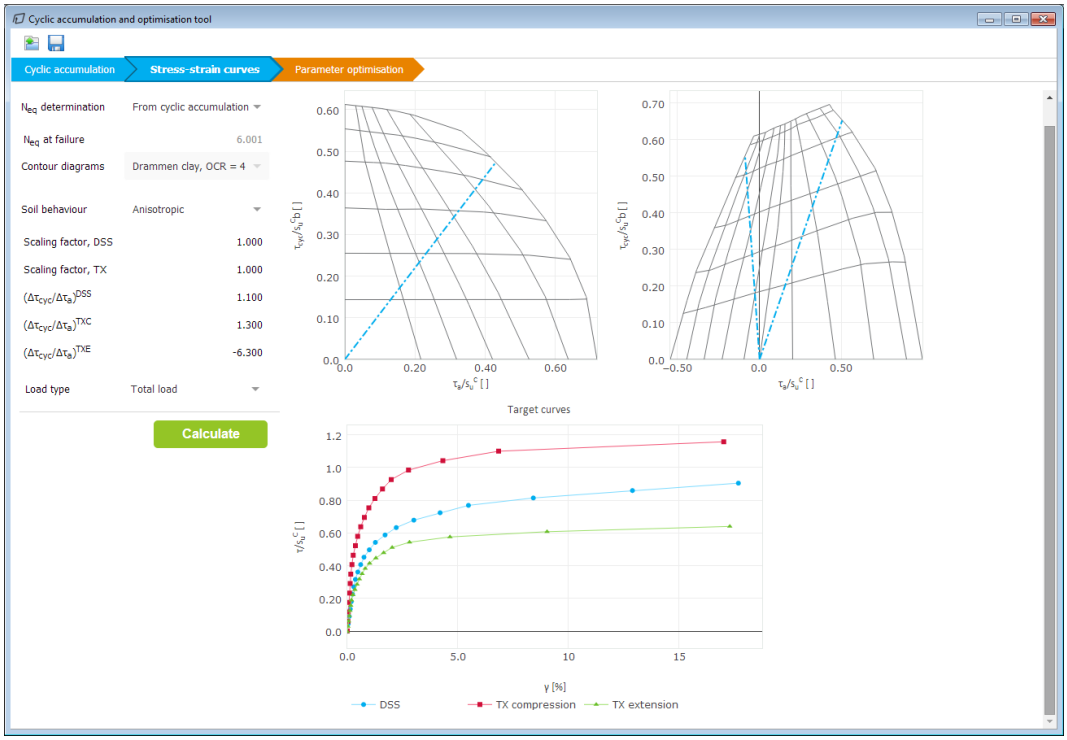


Figure 146: Stress-strain curves tabsheet displaying results after calculation

The *Target curves* chart shows the stress-strain curves for all tests (e.g. only DSS if the isotropic behaviour has been chosen) and for the selected load type. On the x-axis the shear strain γ is plotted in percentage, while the y-axis corresponds to the normalised shear strength τ / s_u^C. The curves are used as input to the third and last tabsheet *Parameter optimisation*.

Parameter optimisation

In the *Parameter optimisation* tabsheet (Figure 147 (on page 220)) the UDCAM-S model optimised parameters can be determined by simulating undrained DSS, TxC, and TxE tests on one material point. The optimisation procedure is based on the *Particle Swarm Algorithm* that allows to find the parameter values resulting in a stress-strain curve that matches the target one. The laboratory test conditions are determined based on the *Static properties* table.

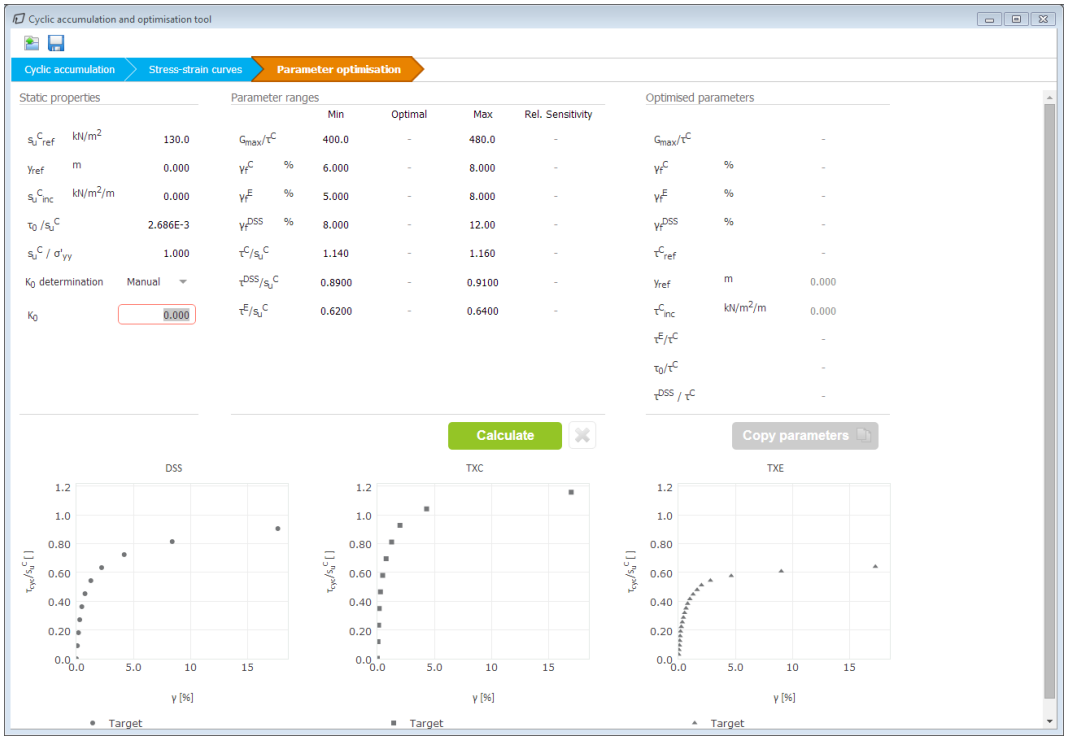


Figure 147: Parameter optimisation tabsheet

The parameters with their standard units are listed below.

Static properties:

$s_{u,ref}{}^{C}$	Reference undrained triaxial compression TxC shear strength	[kN/m^2]
y_{ref}	Reference depth	[m]
$s_{u,inc}{}^{C}$	Increase of TxC shear strength with depth	[kN/m^2/m]
$\tau_0/s_u{}^{C}$	Initial mobilization	[-]
$s_u{}^{C} / \sigma'_{yy}$	Ratio of the undrained compression shear strength over the current vertical effective stress	[-]
K_0	Lateral earth pressure coefficient at rest	[-]

The initial mobilization is determined based on the contour diagrams in the *Stress-strain curves* tabsheet: it corresponds to the value of $\tau_a / s_u{}^{C}$ at the intersection point between the x-axis (γ_{cyc} = 0%) and the contour line for γ_a equal to 0%.

The K_0 *determination* option is set by default to automatic. This means that the value of K_0 used to determine the initial stress condition of the test is determined based on the following equation:

$$K_0 = 1 - \left(2 \frac{\tau_0}{s_u^C} \frac{s_u^C}{\sigma'_{yy}} \right) \qquad \textit{Eq. [16]}$$

Note:

If in the *Stress-strain curves* tabsheet the user has selected the *Cyclic* load type, the initial mobilisation and the undrained shear strength normalised by the vertical effective stress cannot be modified and are equal to 0 and 1, respectively.

In the *Parameter ranges* table it is required to define the minimum and maximum value that should be considered for each parameter during the optimisation procedure. The ranges can be estimated based on the target charts.

Double-clicking on one of the graphs opens the selected chart in a bigger window (Figure 148 (on page 221)). This window shows the selected diagram, the table of the data points that are used to plot it as well as the tangent and the secant values of the plot. Note that the point to be taken into consideration for the calculation of the tangent and the secant values can be determined by clicking on the plot. The secant and the tangent are useful for the back-calculation of stiffness parameters from stress-strain diagrams. The corresponding secant and tangent values are indicated below the table. The graph or the data can be copied to the clipboard by selecting the corresponding option in the drop-down menu displayed when the *Copy* button is clicked. The diagram can be zoomed in or out using the mouse by first clicking and holding the left mouse button in the diagram area and then moving the mouse to a second location and releasing the mouse button. Moving the mouse from the left upper corner to the right lower corner zooms the diagram to the selected area, whereas moving the mouse from the right lower corner to the left upper corner resets the view. The zoom action can also be undone using the Zoom out option on the toolbar. The wheel button of the mouse can be used for panning: click and hold the mouse wheel down and move the diagram to the desired position.

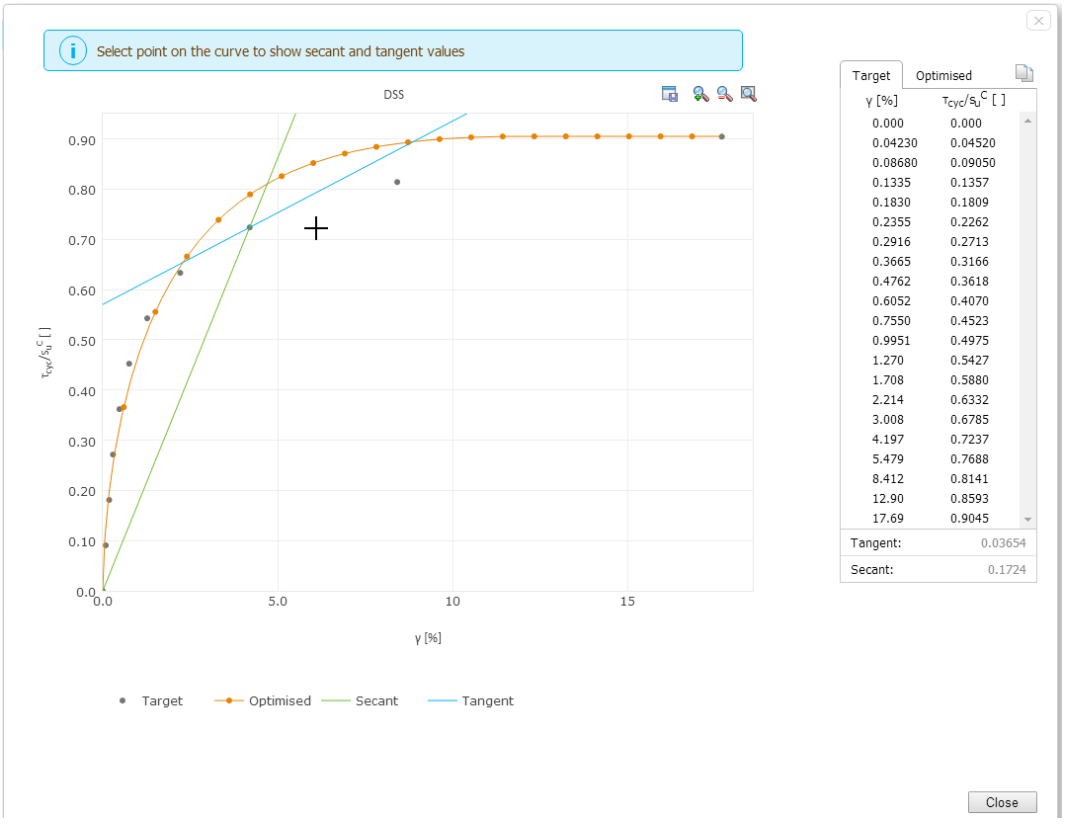


Figure 148: Chart displaying target, optimised, secant and tangent curves, and corresponding table of values.

Parameter ranges:

G_{max}/τ^C	Ratio of the initial shear modulus to the cyclic compression shear strength	[-]
γ_f^C	Shear strain at failure in triaxial compression	[%]
γ_f^E	Shear strain at failure in triaxial extension	[%]
γ_f^{DSS}	Shear strain at failure in direct simple shear	[%]
$\tau^C/{s_u}^C$	Ratio of the cyclic compression shear strength over the undrained static compression shear strength	[-]
$\tau^E/{s_u}^C$	Ratio of the cyclic extension shear strength over the undrained static compression shear strength	[-]

The initial shear modulus defines the tangent value of the stress strain curves at the initial shear stress (Anderson, 2004 (on page 530)). For high loading levels, the area where G is equal to G_{max} is very limited and therefore not governing. For most problems, $G_{max}/{s_u}^C$ has to be chosen to fit the stress-strain curve in the actual strain range of interest. In the case of isotropic behaviour, γ_f^C and γ_f^E are set equal to the optimised γ_f^{DSS}, while the ratios τ^{DSS}/τ^C and τ^E / τ^C are equal to 1.

To run the optimisation procedure, click the *Calculate* button.

The results are shown both in the *Parameter ranges* table and in the charts. The optimum values of the parameters used to obtain the best fit to the test data are shown in the *Optimal value* column of the table. If the optimum value is equal to the minimum or maximum value, it might be that the best value lies outside the specified range. The last column of the table shows the sensitivity of each parameter. A sensitivity of 100% means that the parameter has a high influence on the simulated test results, whereas a low sensitivity values means that the parameter has a low influence on the simulated test results.

The optimised curves are shown in the corresponding test graph. As explained above, each chart can be opened in a larger window by double-clicking it. Also for the optimised curve the table with the data points and the possibility to draw the secant and tangent lines are available.

The optimal values are used to determine the optimised parameters used in the UDCAM-S model and shown in the *Optimised parameters* table (Figure 147 (on page 220)).

Figure 149: Optimised Parameters for Cyclic accumulation and optimisation tool

The parameters with their standard units are listed below.

Derived parameters:

G_{max}/τ^C	Ratio of the initial shear modulus to the cyclic compression shear strength	[-]
γ_f^C	Shear strain at failure in triaxial compression	[%]
γ_f^E	Shear strain at failure in triaxial extension	[%]
γ_f^{DSS}	Shear strain at failure in direct simple shear	[%]
τ_{ref}^C	Cyclic compression shear strength	[kN/m^2]
y_{ref}	Reference depth	[m]
$\tau_{inc}{}^C$	Increase of the cyclic shear strength with depth	[kN/m^2]
τ^E/τ^C	Ratio of the cyclic extension shear strength over the cyclic compression shear strength	[-]
τ_0/τ^C	Cyclic mobilization	[-]
τ^{DSS}/τ^C	Ratio of the cyclic DSS shear strength over the cyclic compression shear strength	[-]

For more information about the parameters of the UDCAM-S model , reference is made to Material Models Manual - *UDCAM-S*.

To copy the *Optimised parameters* table to the material database, click the *Copy parameters*.

Possible errors of Parameter optimisation tab

For more information on the errors that may occur in the *Parameter optimisation* tab, refer to Table 27 (on page 610).

6.1.5 *Groundwater tabsheet*

Flow parameters are required when dealing with problems that involve flow of pore water in saturated or unsaturated soils, i.e. when using *Groundwater flow*, *Consolidation* or *Fully coupled flow-deformation* types of calculation. When considering steady-state groundwater flow or consolidation of fully saturated soil layers, only the soil's (saturated) permeability is a relevant parameter. However, when considering unconfined flow, seepage, transient (time-dependent) flow or fully coupled flow-deformation analysis, partially saturated soil behaviour becomes an issue and needs to be described in more detail. This requires, amongst other things, the selection of a so-called soil-water retention curve relating the suction (positive pore water stress) in the unsaturated zone to the degree of saturation.

PLAXIS 2D incorporates functions to describe the flow behaviour in the unsaturated zone, among which the famous Mualem-Van Genuchten functions. In order to enable an easy selection of the unsaturated flow parameters in these functions, predefined data sets are available for common types of soil. These data sets can be selected based on standardised soil classification systems.

Note:

Although the predefined data sets have been created for the convenience of the user, the users remain at all times responsible for the model parameters that they use. Note that these predefined data sets have limited accuracy.

Hydraulic data sets and models

The program provides different data sets and models to model the flow in the saturated zone in soil. The data sets available in the program are:

1. **Standard:**

   This option allows for a simplified selection of the most common soil types (*Coarse*, *Medium*, *Medium fine*, *Fine* and *Very fine* non-organic materials and *Organic* material) and is based on the Hypres topsoil classification series (Wosten, Lilly, Nemmes & Bas, 1999 (on page 532)). The only model available for this data set is *Van Genuchten* (see Material Models Manual - *Hydraulic Models*).

   When one of the soil type options is selected, the particle fractions are automatically defined and the soil type is indicated in the soil texture triangle (Figure 150 (on page 225)). The particle fractions can also be defined by clicking on the corresponding location in the soil texture triangle or by directly typing the values.

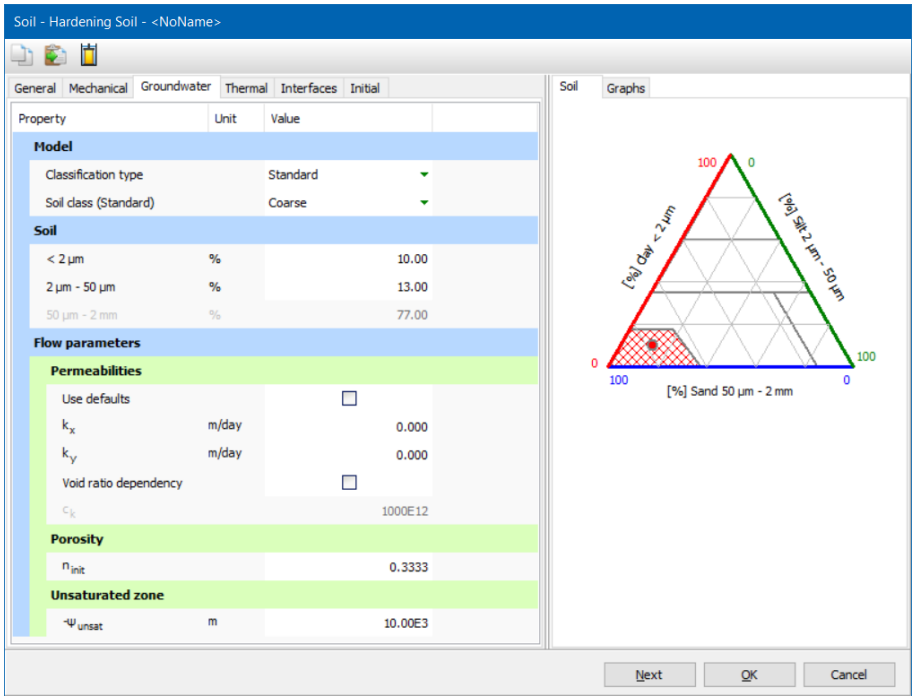


Figure 150: Groundwater parameters for Standard data set (Wösten, Lilly, Nemmes & Bas, 1999)

2. **Hypres:**

   The *Hypres* series is an international soil classification system (Wosten, Lilly, Nemmes & Bas, 1999 (on page 532)). The hydraulic models available for *Hypres* data set are the *Van Genuchten* model and the *Approximate Van Genuchten* (see Material Models Manual - *Hydraulic Models*).

   A distinction can be made between *Topsoil* and *Subsoil*. In general, soils are considered to be *subsoils*. The *Type* drop-down menu for the *Hypres* data set includes *Coarse*, *Medium*, *Medium fine*, *Fine*, *Very fine* and *Organic* soils.

   Note:

   Only soil layers that are located not more than *1* m below the ground surface are considered to be *Top* soils.

   The selected soil type and grading (particle fractions) is indicated in the soil texture triangle. As an alternative, the user can also select the type of soil by clicking one of the sections in the triangle or by manually specifying the particle fraction values (Figure 151 (on page 226)).

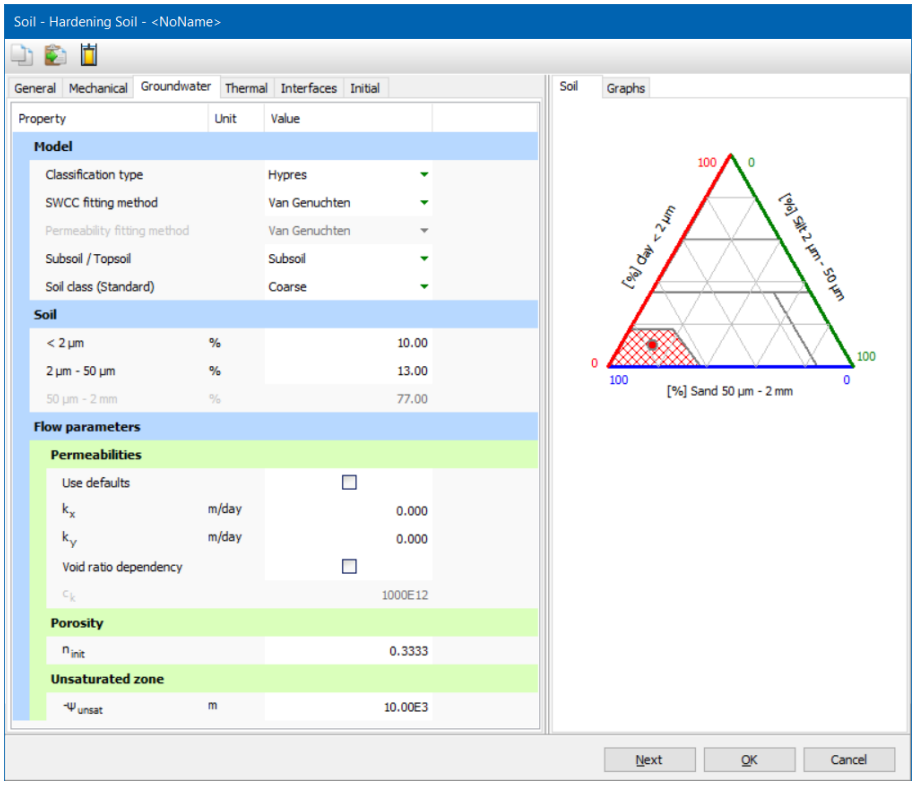


Figure 151: Groundwater parameters for Hypres data set (Wösten, Lilly, Nemmes & Bas, 1999)

The predefined parameters for both the *Van Genuchten* model as well as the *Approximate Van Genuchten* model are shown in Table 10 (on page 226) and Table 11 (on page 227).

Table 10: *Hypres* series with Van Genuchten parameters (Wösten, Lilly, Nemmes & Bas, 1999)

Soil class	θ_r (-)	θ_s (-)	k_{sat} (m/day)	g_a (1/m)	g_l (-)	g_n (-)
Topsoil						
coarse	0.025	0.403	0.600	3.83	1.2500	1.3774
medium	0.010	0.439	0.121	3.14	-2.3421	1.1804
medium fine	0.010	0.430	0.023	0.83	-0.5884	1.2539
fine	0.010	0.520	0.248	3.67	-1.9772	1.1012
very fine	0.010	0.614	0.150	2.65	2.5000	1.1033
Subsoil						
coarse	0.025	0.366	0.700	4.30	1.2500	1.5206
medium	0.010	0.392	0.108	2.49	-0.7437	1.1689

Subsoil						
medium fine	0.010	0.412	0.040	0.82	0.5000	1.2179
fine	0.010	0.481	0.085	1.98	-3.7124	1.0861
very fine	0.010	0.538	0.082	1.68	0.0001	1.073
organic	0.010	0.766	0.080	1.30	0.4000	1.2039

Table 11: *Hypres* series with Approximate Van Genuchten parameters

Soil Class	ψ_s(m)	ψ_k (m)
Topsoil		
coarse	-2.37	-1.06
medium	-4.66	-0.50
medium fine	-8.98	-1.20
fine	-7.12	-0.50
very fine	-8.31	-0.73
Subsoil		
coarse	-1.82	-1.00
medium	-5.60	-0.50
medium fine	-10.15	-1.73
fine	-11.66	-0.50
very fine	-15.06	-0.50
organic	-7.35	-0.97

3. **USDA:**

   The *USDA* series is another international soil classification system (Carsel & Parrish, 1988 (on page 531)). The hydraulic models available for *USDA* data set are the *Van Genuchten* model and the *Approximate Van Genuchten* (see Material Models Manual).

   The *Type* drop-down menu for the *USDA* date set includes *Sand*, *Loamy sand*, *Sandy loam*, *Loam*, *Silt*, *Silt loam*, *Sandy clay loam*, *Clay loam*, *Silty clay loam*, *Sandy clay*, *Silty clay* and *Clay*. The selected soil type and grading (particle fractions) are different from the *Hypres* data sets and can be visualised in the soil texture triangle. As an alternative, the user can also select the type of soil by clicking one of the sections in the triangle or by manually specifying the particle fraction values (Figure 152 (on page 228)).

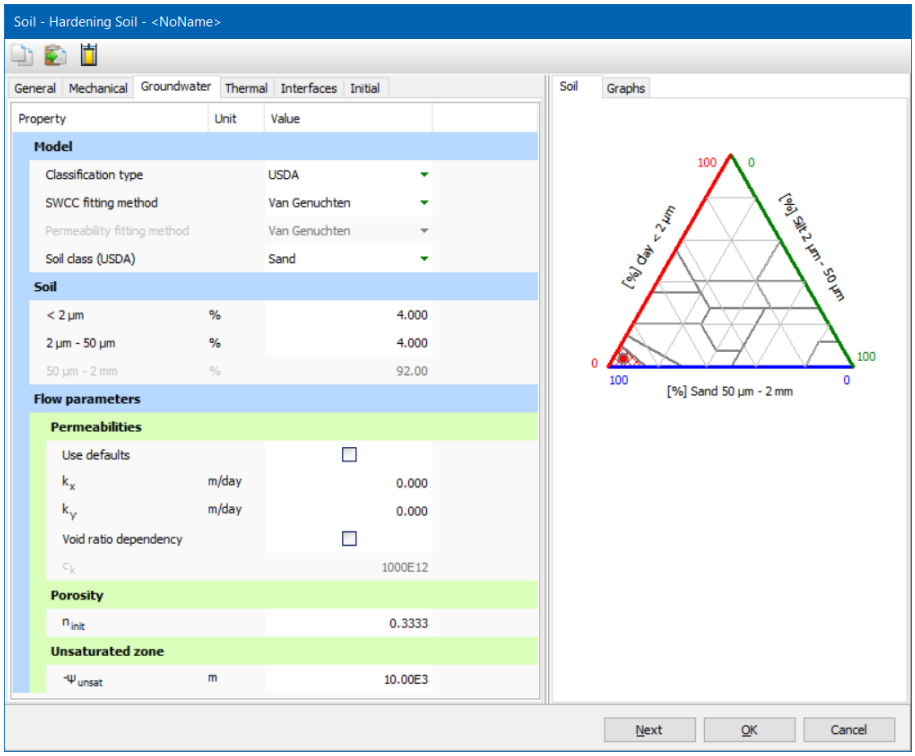


Figure 152: Groundwater parameters for USDA data set (Carsel & Parrish, 1988)

The parameters for the *Van Genuchten* and the *Approximate Van Genuchten* models are shown in Table 12 (on page 228) and Table 13 (on page 229) .

Table 12: USDA series with Van Genuchten parameters, $g_l = 0.5$ (Carsel & Parrish, 1988)

Soil texture	θ_r (-)	θ_s (-)	k_{sat} (m/day)	g_a (1/m)	g_n (-)
sand	0.045	0.43	7.13	14.5	2.68
loamy sand	0.057	0.41	3.50	12.4	2.28
sandy loam	0.065	0.41	1.06	7.5	1.89
loam	0.078	0.43	0.25	3.6	1.56
silt	0.034	0.46	0.60	1.6	1.37
silty loam	0.067	0.45	0.108	2.0	1.41
sandy clay loam	0.100	0.39	0.314	5.9	1.48
clayey loam	0.095	0.41	0.624	1.9	1.31
silty clayey loam	0.089	0.43	0.168	1.0	1.23
sandy clay	0.100	0.38	0.288	2.7	1.23

Soil texture	θ_r (-)	θ_s (-)	k_{sat} (m/day)	g_a (1/m)	g_n (-)
silty clay	0.070	0.36	0.00475	0.5	1.09
clay	0.068	0.38	0.0475	0.8	1.09

Table 13: USDA series with Approximate Van Genuchten parameters

Soil texture	ψ_s (m)	ψ_k (m)
sand	-1.01	-0.50
loamy sand	-1.04	-0.50
sandy loam	-1.20	-0.50
loam	-1.87	-0.60
silt	-4.00	-1.22
silty loam	-3.18	-1.02
sandy clay loam	-1.72	-0.50
clayey loam	-4.05	-0.95
silty clayey loam	-8.23	-1.48
sandy clay	-4.14	-0.55
silty clay	-31.95	-0.95
clay	-21.42	-0.60

Note:

In Table 10 (on page 226) to Table 12 (on page 228)the symbols θ_r and θ_s stand for the residual and the saturated water content correspondingly. Based on van Genuchten (1980) (on page 532), the effective degree of saturation is calculated as $S_{eff} = (\theta-\theta_r)/(\theta_s- \theta_r)$, in which θ is the water content.

In PLAXIS 2D, the effective degree of saturation is calculated as $S_{eff} = (S- S_r) / (S_s-S_r)$ (see Conventional and unsaturated soil behaviour in PLAXIS (on page 543)) , in which S is the degree of saturation, S_r the residual degree of saturation and S_s the saturated degree of saturation. Considering that $S = \theta / n$ and under the assumption that $S_s = \theta_s / n = 1$, S_r could be derived from Table 10 (on page 226) to Table 12 (on page 228) as $S_r = \theta_r / \theta_s$.

4. **Staring:**

The *Staring* series is a soil classification system which is mainly used in The Netherlands (Wosten, Veerman, DeGroot & Stolte, 2001 (on page 532)). The hydraulic models available for *Staring* data set are the *Van Genuchten* model and the *Approximate Van Genuchten* (see the Material Models Manual - *Hydraulic Models*).

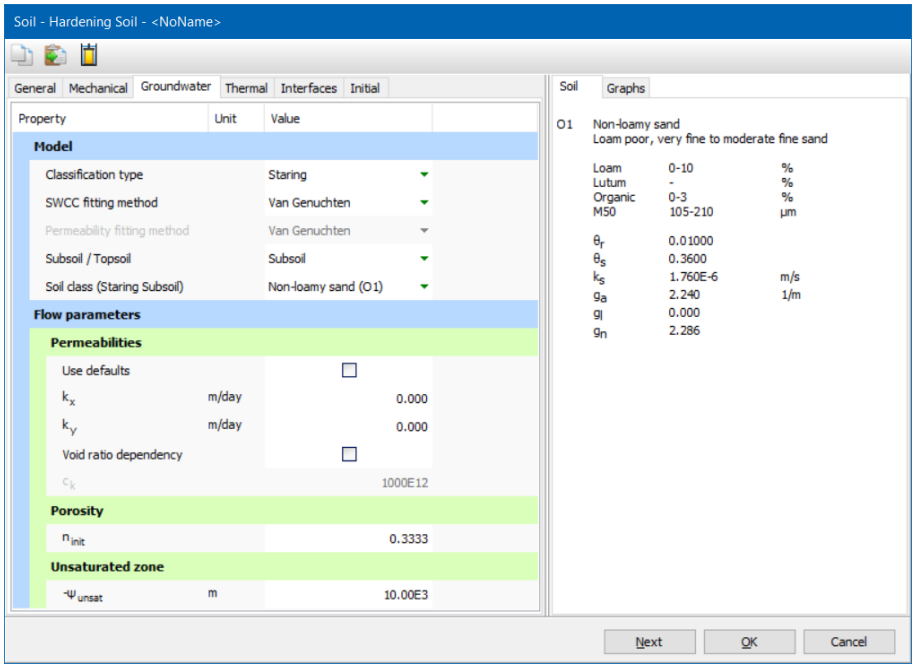


Figure 153: Groundwater parameters for Staring data set (Wösten, Veerman, DeGroot & Stolte, 2001)

A distinction can be made between *Topsoil* and *Subsoil*. In general, soils are considered to be subsoils. The *Type* drop-down menu for the *Staring* series (Figure 153 (on page 230)) contains the following subsoils (Wosten, Veerman, DeGroot & Stolte, 2001 (on page 532)):

Non-loamy sand (O1), *Loamy sand (O2)*, *Very loamy sand (O3)*, *Extremely loamy sand (O4)*, *Coarse sand (O5)*, *Boulder clay (O6)*, *River loam (O7)*, *Sandy loam (O8)*, *Silt loam (O9)*, *Clayey loam (O10)*, *Light clay (O11)*, *Heavy clay (O12)*, *Very heavy clay (O13)*, *Loam (O14)*, *Heavy loam (O15)*, *Oligotrophic peat (O16)*, *Eutrophic peat (O17)* and *Peaty layer (O18)*, and the following topsoils: *Non-loamy sand (B1)*, *Loamy sand (B2)*, *Very loamy sand (B3)*, *Extremely loamy sand (B4)*, *Coarse sand (B5)*, *Boulder clay (B6)*, *Sandy loam (B7)*, *Silt loam (B8)*, *Clayey loam (B9)*, *Light clay (B10)*, *Heavy clay (B11)*, *Very heavy clay (B12)*, *Loam (B13)*, *Heavy loam (B14)*, *Peaty sand (B15)*, *Sandy peat (B16)*, *Peaty clay (B17)* and *Clayey peat (B18)*.

The selected soil type and grading (particle fractions) are different from the *Hypres* and the *USDA* data sets. The parameters of the hydraulic model for the selected soil type are displayed in the **Soil** tab at the right side of the **Groundwater** tabsheet and are summed up in the tables below.

Note:

Only soil layers that are located not more than *1* m below the ground surface are considered to be *Topsoils*. In the tables below the notation *"B"* corresponds to *Topsoils*; and the notation *"O"* indicates *Subsoil*.

Table 14: *Staring* series with Van Genuchten parameters (Heinen, Bakker & Wösten, 2001)

Soil class	θ_r (-)	θ_s (-)	k_{sat} (m/day)	g_a (1/m)	g_l (-)	g_n (-)
Sand						
B1	0.02	0.43	0.234	2.34	0.000	1.801

Soil class	θ_r (-)	θ_s (-)	k_{sat} (m/day)	g_a (1/m)	g_l (-)	g_n (-)
Sand						
B2	0.02	0.42	0.125	2.76	-1.060	1.491
B3	0.02	0.46	0.154	1.44	-0.215	1.534
B4	0.02	0.46	0.292	1.56	0.000	1.406
B5	0.01	0.36	0.529	4.52	-0.359	1.933
B6	0.01	0.38	1.01	2.22	-1.747	1.238
O1	0.01	0.36	0.152	2.24	0.000	2.286
O2	0.02	0.38	0.127	2.13	0.168	1.951
O3	0.01	0.34	0.109	1.70	0.000	1.717
O4	0.01	0.35	0.0985	1.55	0.000	1.525
O5	0.01	0.32	0.250	5.21	0.000	2.374
O6	0.01	0.33	0.340	1.62	-1.330	1.311
O7	0.01	0.51	0.391	1.23	-2.023	1.152
Silt						
B7	0.00	0.40	0.141	1.94	-0.802	1.250
B8	0.01	0.43	0.0236	0.99	-2.244	1.288
B9	0.00	0.43	0.0154	0.65	-2.161	1.325
O8	0.00	0.47	0.0907	1.36	-0.803	1.342
O9	0.00	0.46	0.0223	0.94	-1.382	1.400
O10	0.01	0.48	0.0212	0.97	-1.879	1.257
Clay						
B10	0.01	0.43	0.007	0.64	-3.884	1.210
B11	0.01	0.59	0.0453	1.95	-5.901	1.109
B12	0.01	0.54	0.0537	2.39	-5.681	1.094
O11	0.00	0.42	0.138	1.91	-1.384	1.152

Clay						
O12	0.01	0.56	0.0102	0.95	-4.295	1.158
O13	0.01	0.57	0.0437	1.94	-5.955	1.089
Loam						
B13	0.01	0.42	0.130	0.84	-1.497	1.441
B14	0.01	0.42	0.008	0.51	0.000	1.305
O14	0.01	0.38	0.0151	0.30	-0.292	1.729
O15	0.01	0.41	0.037	0.71	0.912	1.298
Peat						
B15	0.01	0.53	0.813	2.42	-1.476	1.280
B16	0.01	0.80	0.0679	1.76	-2.259	1.293
B17	0.00	0.72	0.0446	1.80	-0.350	1.140
B18	0.00	0.77	0.0667	1.97	-1.845	1.154
O16	0.00	0.89	0.0107	1.03	-1.411	1.376
O17	0.01	0.86	0.0293	1.23	-1.592	1.276
O18	0.01	0.57	0.0345	1.38	-1.204	1.323

Table 15: *Staring* series with Approximate Van Genuchten parameters

Class	ψ_s(m)	ψ_k (m)
Sand		
B1	-1.87	-1.35
B2	-2.32	-0.79
B3	-3.37	-2.18
B4	-3.81	-2.36
B5	-1.31	-0.56
B6	-4.51	0.70

Class	ψ_s(m)	ψ_k (m)
Sand		
O1	-1.48	-1.15
O2	-1.79	-1.51
O3	-2.46	-1.93
O4	-3.22	-2.30
O5	-1.11	-0.48
O6	-4.50	-1.18
O7	-9.76	-0.98
Silt		
B7	-4.72	-1.14
B8	-7.03	-1.46
B9	-9.61	-2.33
O8	-4.78	-1.74
O9	-5.89	-2.07
O10	-7.77	-1.59
Clay		
B10	-13.06	-1.47
B11	-9.51	-0.27
B12	-9.61	-0.21
O11	-7.20	-0.76
O12	-11.55	-0.82
O13	-11.45	-0.24
Loam		
B13	-6.19	-2.25
B14	-12.82	-7.09

Loam		
B12	-9.61	-0.21
O14	-20.68	-9.37
O15	-9.30	-14.08

Peat		
B15	-3.77	-0.73
B16	-4.45	-0.82
B17	-8.03	-1.19
B18	-6.98	-0.64
O16	-5.64	-1.86
O17	-6.06	-1.39
O18	-4.93	-1.45

5. **User-defined:**

The *user-defined* option enables the user to define both saturated and unsaturated properties manually. Please note that this option requires adequate experience with unsaturated groundwater flow modelling. The hydraulic models available are:

Van Genuchten	This well-known and widely accepted model requires direct input of the residual saturation S_{res}, the saturation at $p = 0$, S_{sat} and the three fitting parameters g_n, g_a and g_l (see Material Models Manual- *Hydraulic models*).
Spline	The *Spline* function requires direct input of the capillary height ψ (in unit of length), the relative permeability K_r (-), and the degree of saturation S_r (-). Data for the *Spline* function can be entered by clicking the **Table** tab. The degree of saturation at saturated conditions S_{sat} equals the value assigned to S_r at ψ_{max} (e.g. -ψ = 0 m) and the residual degree of saturation S_{res} equals the minimum value assigned to S_r at ψ_{min} (e.g -ψ = 20 m). During the calculations, the flow calculation kernel employs 'smooth' relationships based on a spline function between K_r-ψ and S_r-ψ.
Saturated	When the *Saturated* option is selected, no extra data input is required. During the calculations, PLAXIS 2D will continuously use the saturated permeabilities for soil layers where a *Saturated* data set was assigned.

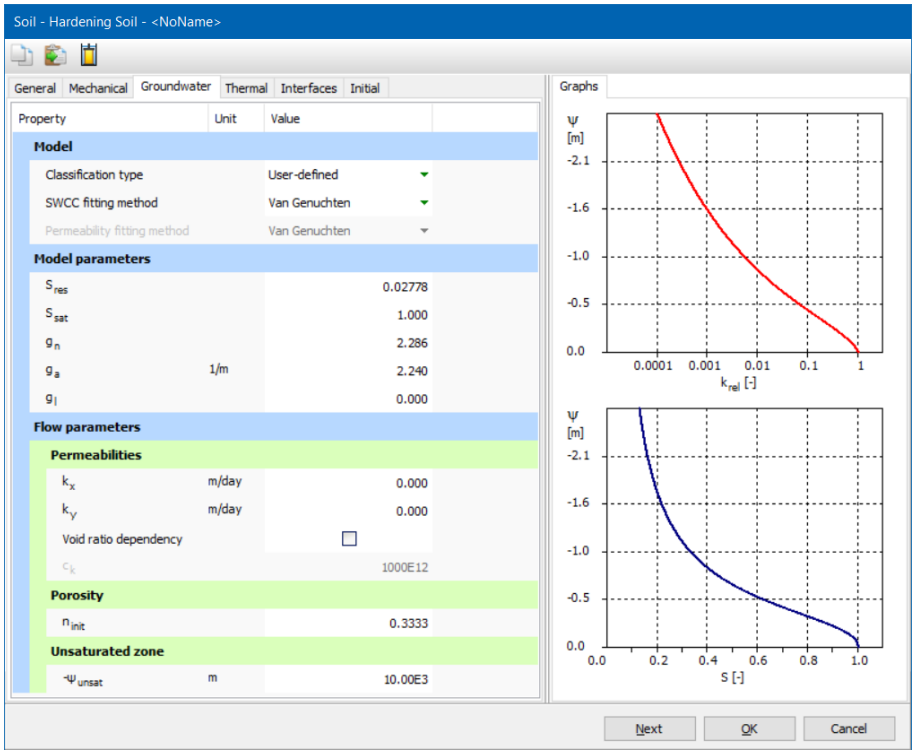


Figure 154: Groundwater parameters for user-defined data set

Saturated permeabilities (k_x and k_y)

Coefficients of permeability (hydraulic conductivity) have the dimension of velocity (unit of length per unit of time). The input of permeability parameters is required for groundwater flow and consolidation calculations. In such calculations, it is necessary to specify the coefficient of permeability for all soil materials with drainage type *Drained*, *Undrained A* and *Undrained B*, including almost impermeable layers, except for fully impervious layers with a *Non-porous* drainage type and layers using a total stress analysis (*Undrained C*). PLAXIS 2D allows for the anisotropic permeability of soils where the anisotropy directions coincide with the principal axes *x* and *y*.

Users can choose to either enter the values for the permeabilities directly, or choose default values. When opting for default values there are 2 available options:

From data set	The values of k_x and k_y are obtained from the selected data set.
From grain size distribution	The values for k_x and k_y are obtained from particle size distribution as explained above. The value for θ_{sat} is calculated internally using the equation, $\theta_{sat} = e / (1+e)$.

In real soils, the difference in permeabilities between the various layers can be quite large. However, care should be taken when very high and very low permeabilities occur simultaneously in a finite element model, as this could lead to ill-conditioning of the flow matrix. In order to obtain accurate results, the ratio between the highest and lowest permeability value in the geometry should not exceed 10^5.

Note that the input field for permeabilities are greyed out when the drainage type of the material is either *Non-porous* or *Undrained C*.

In case of a *Standard*, *Hypres* or *USDA* data set, values for the permeability can be automatically set by either selecting the *From data set* option or *From grain size distribution*.

When a predefined soil data set is chosen or the user specifies the individual particle fractions manually, assuming the soil has a *log-normal* function of particle size distribution, then the geometric mean particle diameter d_g and a geometric standard deviation σ_g can be calculated as:

$$d_g = \exp\left(m_{cl}\ln\left(d_{cl}\right) + m_{si}\ln\left(d_{si}\right) + m_{sa}\ln\left(d_{sa}\right)\right) \qquad \textit{Eq. [17]}$$

$$\sigma_g = \exp\left[\sum_{i=1}^{3} m_i \ln\left(d_i\right)^2 - \left(\sum_{i=1}^{3} m_i \ln\left(d_i\right)\right)^2\right]^{\frac{1}{2}} \qquad \textit{Eq. [18]}$$

where m_{cl}, m_{si} and m_{sa} are particle fractions for clay, silt and sand ; d_{cl}, d_{si} and d_{sa} are the particle size means for clay, silt and sand respectively (i.e. $d_{cl} = 1\mu m$, $d_{si} = 26\ \mu m$, $d_{sa} = 1.025mm$)

According to Aukenthaler, Brinkgreve & Haxaire (2016) (on page 530) The *Specific surface area (SSA)* can be approximated using the geometric mean particle diameter in *mm*:

$$SSA = 3.89 d_g^{-0.905}$$

From the above, the hydraulic conductivity of saturated soils (permeabilities) can be obtained from the soil texture and porosity of the soil using the following relation:

$$k_x = k_y = K_{sat}$$

$$K_{sat} = 4 \times 10^{-5}\left(\frac{0.5}{1-\theta_{sat}}\right)^{1.3b} \times \exp\left(-6.88 m_{cl} - 3.63 m_{si} - 0.025\right) \qquad \textit{Eq. [19]}$$

$$b = d_g^{-0.5} + 0.2\sigma_g$$

where

θ_{sat}	=	volumetric water content of saturated soil which is equal to the porosity of the soil *(n(n = e / (1 + e))*.
d_g	=	the geometric mean particle diameter.
σ_g	=	the geometric standard deviation.

Void ratio dependency

This advanced feature is to account for the change of permeability with void ratio during a consolidation analysis. This can be applied by entering a proper value for the c_k parameter and the initial void ratio e_{init}. On entering a real value, the permeability will change according to the formula:

$$\log\left(\frac{k}{k_0}\right) = \frac{\Delta e}{c_k} \qquad \textit{Eq. [20]}$$

where Δe is the change in void ratio, k is the permeability in the calculation and k_0 is the input value of the permeability in the data set (= k_x and k_y). Note that a proper input of the initial void ratio e_{init}, in the **General** tabsheet is required. It is recommended to use a changing permeability only in combination with the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model or the Soft Soil Creep model. In that case the c_k-value is generally in the order of the compression index C_c. For all other models the c_k-value should be left to its default value of 10^{15}.

Unsaturated zone (ψ_{unsat})

ψ_{unsat} (in unit of length relative to the phreatic level) sets the maximum pressure head until which the Mualem-Van Genuchten functions are used for calculation of relative permeability and degree of saturation. The negative sign indicates suction. Above the level of ψ_{unsat}, the value of K_r and S remains constant. In this way a minimum degree of saturation (S_{min}) is guaranteed (Figure 155 (on page 237)). It is used to limit the relative permeability K_r and degree of saturation for high unsaturated zones.

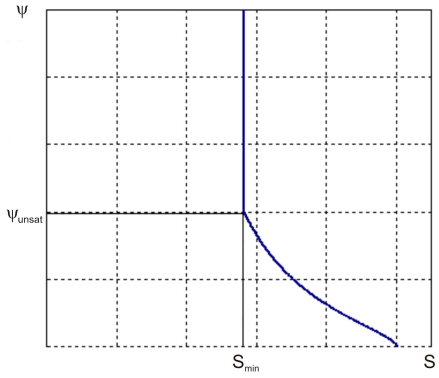


Figure 155: Relative permeability vs. Degree of saturation

By default a very large value is assigned to ψ_{unsat} ($= 10^4$). This value is only an indication that the unsaturated zone is by default unlimited.

6.1.6 *Thermal tabsheet* [ADV]

The *Thermal* tabsheet (Figure 156 (on page 237)) contains thermal parameters that are required when dealing with problems involving (a change of) temperature and the influence on the stress, deformation or groundwater flow. The meaning of these parameters is described below.

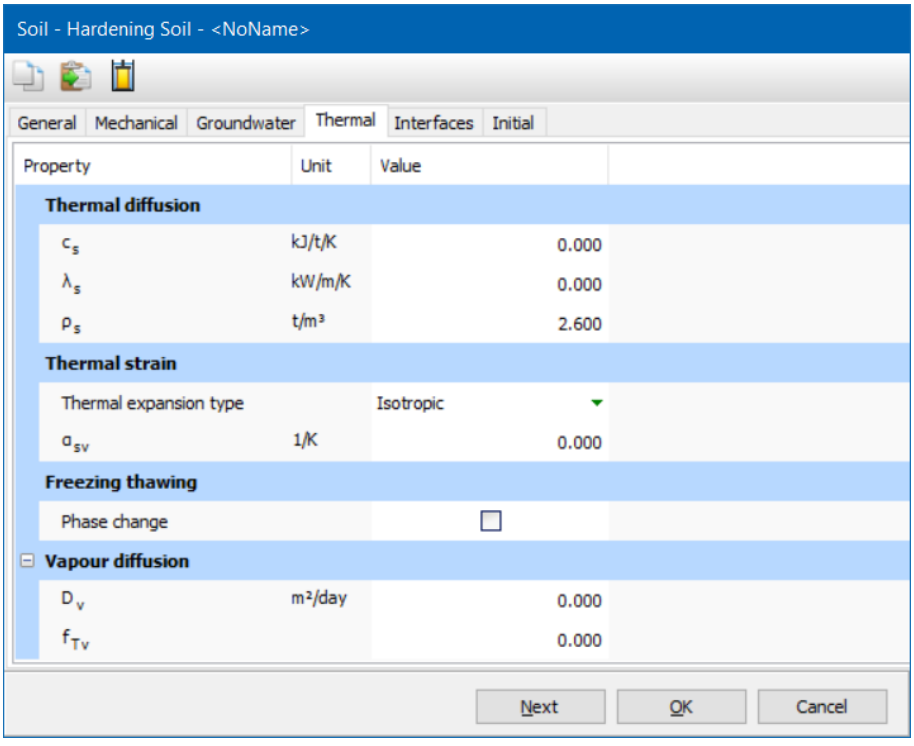


*Figure 156: **Thermal** tabsheet of the **Soil** window*

Thermal diffusion parameters

The first parameters that are required in order to do a thermal flow analysis are the parameters for the thermal flow or thermal diffusion itself.

The *Specific heat capacity* of the solid material, c_s, is a parameter that describes the amount of energy (heat) that can be stored in the solid material (i.e. the soil particles) per unit of mass. It is specified in the unit of energy per unit of mass per unit of temperature. The larger the specific heat, the more energy it takes to increase the temperature of the material.

The total heat storage in a solid material is the product of the density, ρ_s, and the specific heat capacity, c_s. Considering soil as a porous medium, the heat storage of the soil is composed of the heat storage in the soil particles and the heat storage in the pore fluid. The following situations are considered:

For non-porous material	$(\rho C)_{soil} = \rho_s C_s$
For dry material	$(\rho C)_{soil} = (1-n)\, \rho_s C_s$
For phase transition (e.g. frozen soil)	$(\rho C)_{soil} = (1-n)\, \rho_s C_s + nS\,[(1-w_u)\rho_i C_i + w_u\, \rho_w C_w] + n\,(1-S)\, \rho_v C_v$
In other cases	$(\rho C)_{soil} = (1-n)\, \rho_s C_s + n\, S\, \rho_w C_w + n\,(1-S)\, \rho_v\, C_v$

where

n	=	porosity = $[e/(1+e)]$ (where e = void ratio)
S	=	degree of saturation
w_u	=	property of ice (water in solid state)
index *i*	=	unfrozen water content
index *w*	=	property of water (water in liquid state)
index *v*	=	property of vapour (water in gas state)

Note:

Note that the properties of water in the three different phases are contained in the **Constants** tabsheet of the **Project properties** window (Project properties (on page 21)).

The *Thermal conductivity* of the solid material, λ_s, is a parameter that describes the rate of energy (heat) that can be transported in the solid material (i.e. the soil particles). It is specified in the unit of power per unit of length per unit of temperature. The larger the conductivity, the more energy is transported, resulting in a faster propagation of a change of temperature in the material.

Considering soil as a porous medium, the total heat conductivity of the soil is composed of the heat conductivity in the soil particles and the heat conductivity in the pore fluid. The following situations are considered:

For non-porous material	$\lambda_{soil} = \lambda_s$
For dry material	$\lambda_{soil} = (1-n)\, \lambda_s$
For phase transition	$\lambda_{soil} = (1-n)\, \lambda_s + n\, S\, [(1-w_u)\lambda_i + w_u\, \lambda_w] + n\,(1-S)\, \lambda_v$
In other cases	$\lambda_{soil} = (1-n)\, \lambda_s + n\, S\, \lambda_w + n\,(1-S)\, \lambda_v$

where

λ_i	=	thermal conductivity of ice (water in solid state)
λ_w	=	thermal conductivity of water (water in liquid state)
λ_v	=	thermal conductivity of vapour (water in gas state)

The *Density* of the solid material, ρ_s, is the parameter that describes the density of the soil particles, material expressed in the unit of mass per unit of volume. For example, the density of quartz is 2.65 t/m^3 The density relates to the unit weight of the material (see Project properties (on page 21)), and it contributes to the total heat storage.

Note:

Note that PLAXIS 2D does NOT check the consistency between the unit weight and the density.

Thermal strain parameters

In addition to parameters related to the thermal diffusion, a parameter describing the mechanical effects of a change of thermal conditions is required.

The *Thermal expansion coefficients*, α, describe how much the material expands (or elongates) when the temperature increases. In other words, the thermal expansion coefficient is the (change of) strain per unit of temperature. If the thermal expansion is isotropic, all coefficients are equal and a single value can be specified for isotropic expansion, but PLAXIS 2D can also handle anisotropic thermal expansion in x-, y- and z- direction through the input of 3 linear thermal expansion coefficients.

Freezing / thawing parameters

The use of the *Phase change* option may be relevant in the case of permafrost soil, ground freezing and other situations of frozen soil. In frozen soils, part of the pore water is present in solid state and part of the water may still be present in liquid state. By activating the phase change option the part of the water in liquid state as a function of temperature can be defined by means of the *Unfrozen water saturation method*.

The available options for *Unfrozen water saturation method* from the drop-down menu are:

User-defined	When this option is selected, the user should provide details of the unfrozen water content, w_u as a function of temperature (described below).
From grain size distribution	When this option is selected, the unfrozen water content is automatically calculated using the *Specific surface area (SSA)*

user-defined water saturation method: After selection of this option, a table should be created in the right-hand panel, in which the ratio of unfrozen water content over total water content, $w_u/W0$, is defined as a function of temperature.

- Click the *Add row* button in the toolbar to add a new row in the table.
- Click the *Insert* button to insert a new row before the selected row in the table.
- Click the *Delete* button to delete the selected row in the table.

The values can be defined by clicking the cell in the table and by typing the value. A plot of the unfrozen water content as a function of temperature is shown below the table (Figure 158 (on page 241)).

- Besides defining the values in the table, there is also the possibility to read data from a file using the *Open* button in the toolbar.
- The current table can be saved using the *Save* button in the toolbar enabling the usage in other projects or validating the effect of the modifications in the current project.
- The current table can be using the *Copy* button in the toolbar.
- Copied data from other applications (using *Ctrl+c*) can be imported by using the *Paste* button. The **Import data** window appears (Figure 157 (on page 240)). The starting row of the data to be imported can be defined in the *From row* cell. The data and the plot is displayed in the **Unfrozen water content** window after pressing *OK*.

Note:

PLAXIS 2D assumes the data file is located in the current project directory when no directory is specified.

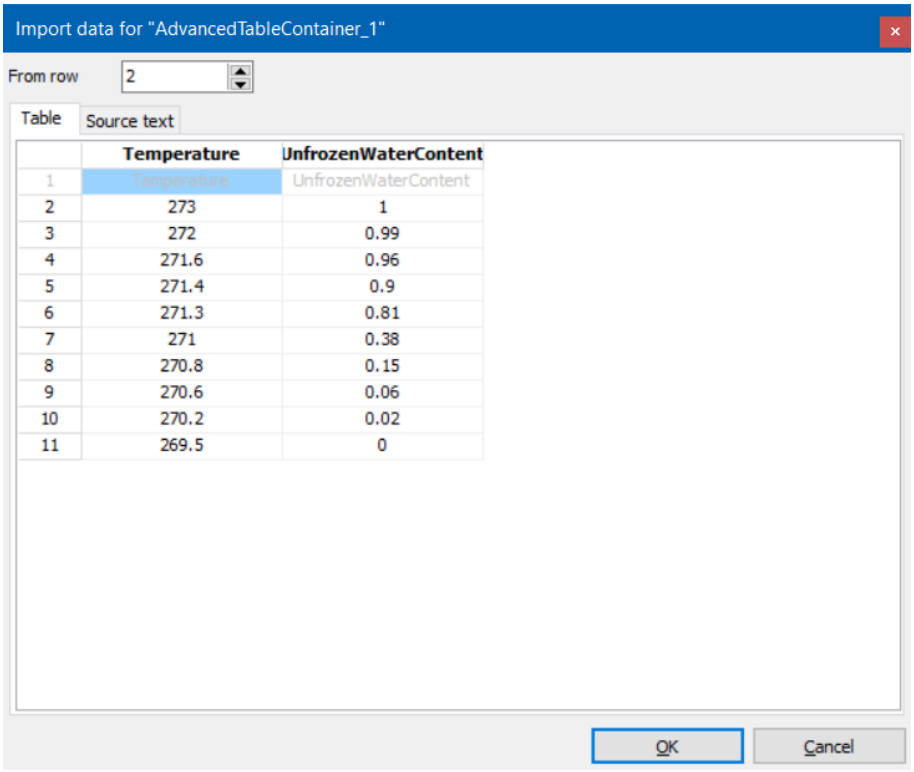


*Figure 157: **Import data** window for Unfrozen water content*

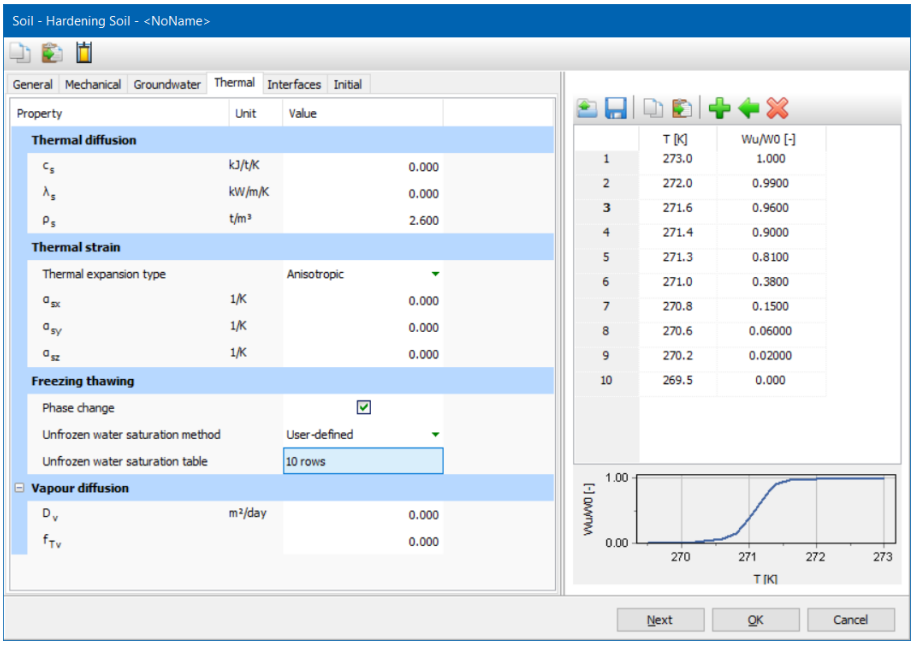


Figure 158: A plot for Unfrozen water content as a function of temperature

Clicking the *Open .txt file* button on the right hand side of the window will open the **Open** window where the file can be selected. The file must be an ASCII file that can be created with any text editor. For every line a pair of values (temperature and corresponding unfrozen water content value) must be defined, leaving at least one space between them. Note that PLAXIS 2D only supports the English notation of decimal numbers using a dot.

In thermal calculations of saturated soils, in which phase transition of the pore water (from water to ice) is considered, the unfrozen water content is used to identify the amount of water and ice in the pores. This is important since the different phases of the pore fluid have different thermal and hydraulic properties.

From grain size distribution: When this option is selected, the *Unfrozen water content* is automatically calculated using the *Specific surface area (SSA)* (refer to Eq. [17]).

$$SSA = 3.89 d_g^{-0.905} \qquad Eq.\ [21]$$

The temperature dependent unfrozen water content w_{un} can now be determined as follows (Aukenthaler, Brinkgreve & Haxaire, 2016 (on page 530)) :

$$w_{un} = \exp\left(0.2618 + 0.5519 \ln\left(SSA\right) - 1.4495 SSA^{-0.2640} \ln\left(\Delta T\right)\right) \qquad Eq.\ [22]$$

The unfrozen water saturation can now be determined as (Aukenthaler, Brinkgreve & Haxaire, 2016 (on page 530)):

$$\frac{w_{un}}{w_0} = \frac{\rho_s}{100 \rho_w e} w_{un} \qquad Eq.\ [23]$$

where ρ_w is the density of pure free water, ρ_s is the density of solids and e is the void ratio; ΔT is the temperature difference (positive value) between the bulk freezing point temperature (= 273.16K) and the temperature below the freezing point and SSA is the specific surface area.

Vapour diffusion coefficient

The *Vapour diffusion coefficient*, D_v, governs the diffusion of vapour in the material. A value of zero disables the mass flux of vapour in the material.

Thermal diffusion enhancement factor

The *Thermal diffusion enhancement factor*, f_{Tv}, influences the dependence of the temperature on the mass flux of vapour. A value of zero means that the mass flux of vapour is governed only by the variations of pore pressure.

Note:

By default, vapour is not considered, i.e. the *Vapour diffusion coefficient* and the *Thermal diffusion enhancement factor* are set to zero.

6.1.7 *Interfaces tabsheet*

The properties of interface elements are related to the soil model parameters of the surrounding soil. The required parameters to derive the interface properties are defined in the **Interfaces** tabsheet of the **Soil** window. These parameters depend on the material model selected to represent the behaviour of the surrounding soil. In case the Linear Elastic model, the Mohr-Coulomb model, the Hardening Soil model, the Hardening Soil model with small-strain stiffness, the UBC3D-PLM model, the Soft Soil model, the Soft Soil Creep model, the Jointed Rock model, the Hoek-Brown model, the NGI-ADP model or the UDCAM-S model has been selected as the *Material model*, the strength reduction factor R_{inter} is the main interface parameter (see Figure 159 (on page 243)). In case of the Modified Cam-Clay model, the interface parameters required are the effective cohesion c'_{ref}, the effective friction angle φ' and the dilatancy angle ψ. In case of the User-defined Soil Model, the tangent stiffness for primary oedometer loading $E_{oed}{}^{ref}$, the effective cohesion c'_{ref}, the effective friction angle φ', the dilatancy angle ψ and the parameters *UD-Power* and *UD-P*^{ref} are required as interface parameters. For more information on the interface parameters required for the User-defined Soil Model (see the Material Models Manual - *Input of UD model parameters via user-defined.*

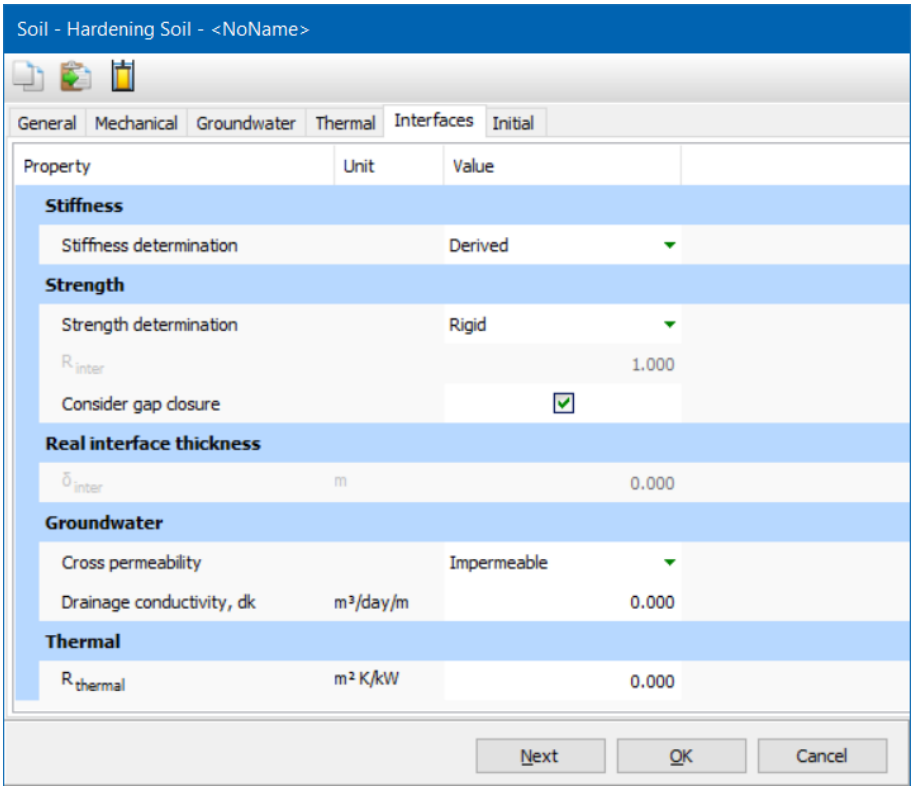


Figure 159: ***Interfaces*** *tabsheet of the* ***Soil*** *window*

Stiffness

With the option *Stiffness determination* the user can choose to either have the interface stiffness *Derived* from the soil parameters, or to have a *Direct* input of the interface stiffness.

Derived: by default, stiffness is set to *Derived*. The shear and compression moduli are related by the expressions:

$$E_{oed,i} = 2G_i \frac{1-\nu_i}{1-2\nu_i}$$
$$G_i = R_{\text{inter}}^2 G_{soil} \leq G_{soil} \qquad \textit{Eq. [24]}$$
$$\nu_i = 0.45$$

When using user-defined material model, the user inserts the interface stiffness in the form of a power law formulation with E_{oed}^{ref} along with *UD-*p^{ref} and *UD-Power*.

Direct: this option allows to directly specify the k_n and k_s values.

k_n	Elastic interface normal stiffness	kN/m^3
k_s	Elastic interface shear stiffness	kN/m^3

Strength

In case of the Linear Elastic model, the Mohr-Coulomb model, the Hardening Soil model, the Hardening Soil model with small-strain stiffness, the UBC3D-PLM model, the Soft Soil model, the Soft Soil Creep model, the Jointed Rock model, the Hoek-Brown model, the NGI-ADP model or the UDCAM-S model, the interface strength is defined by the parameter R_{inter}. The interface strength can be set using the following options:

Rigid: This option is used when the interface should not have a reduced strength with respect to the strength in the surrounding soil. For example, extended interfaces around corners of structural objects (Figure 71 (on page 110)) are not intended for soil-structure interaction and should not have reduced strength properties. The strength of these interfaces should be assigned as *Rigid* (which corresponds to R_{inter} = 1.0). As a result, the interface properties, including the dilatancy angle ψ_i, are the same as the soil properties in the data set, except for Poisson's ratio ν_i (see further).

Manual: The value of R_{inter} can be entered manually if the interface strength is set to *Manual*. In general, for real soil-structure interaction the interface is weaker and more flexible than the surrounding soil, which means that the value of R_{inter} should be less than 1. Suitable values for R_{inter} for the case of the interaction between various types of soil and structures in the soil can be found in the literature. In the absence of detailed information it may be assumed that R_{inter} is of the order of 2/3. A value of R_{inter} greater than 1 cannot be used.

When the interface is elastic then both slipping (relative movement parallel to the interface) and gapping or overlapping (i.e. relative displacements perpendicular to the interface) could be expected to occur.

The magnitudes of the interface displacements are:

$$Elastic\ gap\ displacement = \frac{\sigma_n}{k_n} = \frac{\sigma_n t_i}{E_{oed,i}} \qquad Eq.\ [25]$$

$$Elastic\ slip\ displacement = \frac{\tau}{k_s} = \frac{\tau t_i}{G_i} \qquad Eq.\ [26]$$

where G_i is the shear modulus of the interface, $E_{oed,i}$ is the one-dimensional compression modulus of the interface, t_i is the virtual thickness of the interface generated during the creation of interfaces in the geometry model (Interfaces (on page 106)), k_n is the elastic interface normal stiffness and k_s is the elastic interface shear stiffness.

Note:

Note that a reduced value of R_{inter} not only reduces the interface strength, but also the interface stiffness.

It is clear from these equations that, if the elastic parameters are set to low values, the elastic displacements may be excessively large. If the values of the elastic parameters are too large, however, this can result in numerical ill-conditioning of the stiffness matrix. The key factor in the stiffness is the virtual thickness. This value is automatically chosen such that an adequate stiffness is obtained. The user may change the virtual thickness. This can be done in the **Interface** window that appears after double clicking an interface in the geometry model (Interfaces (on page 106)).

Manual with residual strength: When the limit value of the interface strength as defined by R_{inter} is reached, the interface strength may soften down to a reduced value as defined by $R_{inter,residual}$. Definition of the $R_{inter,residual}$ is possible when the *Manual with residual strength* option is selected for the interface strength.

Interface reduction factor (R_{inter}): An elastic-plastic model is used to describe the behaviour of interfaces for the modelling of soil-structure interaction. The Coulomb criterion is used to distinguish between elastic behaviour, where small displacements can occur within the interface, and plastic interface behaviour when permanent slip may occur. For the interface to remain elastic the shear stress τ is given by:

$$| \tau | < -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [27]}$$

where σ_n is the effective normal stress.

For plastic behaviour τ is given by:

$$| \tau | = -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [28]}$$

where φ_i and c_i are the friction angle and cohesion (adhesion) of the interface. The strength properties of interfaces are linked to the strength properties of a soil layer. Each data set has an associated reduction factor for interfaces R_{inter}. The interface properties are calculated from the soil properties in the associated data set and the strength reduction factor by applying the following rules:

$$c_i = R_{\text{inter}} c_{soil}$$
$$\tan(\varphi_i) = R_{\text{inter}} \tan(\varphi_{soil}) \leq \tan(\varphi_{soil}) \qquad \textit{Eq. [29]}$$

$$\psi_i = 0 \text{ for } R_{\text{inter}} < 1, \text{ otherwise } \psi_i = \psi_{soil} \qquad \textit{Eq. [30]}$$

In the Hardening Soil model, the Hardening Soil model with small-strain stiffness and the UBC3D-PLM model, φ_{soil} is always referred to the failure-peak value of φ chosen as input parameter.

In addition to Mohr-Coulomb's shear stress criterion, the tension cut-off criterion, as described before (see Mechanical tabsheet (on page 169)), also applies to interfaces (if not deactivated):

$$\sigma_n < \sigma_{t,i} = R_{\text{inter}} \sigma_{t,soil} \qquad \textit{Eq. [31]}$$

where $\sigma_{t,soil}$ is the tensile strength of the soil.

Interface strength in undrained condition: When the drainage condition is *Undrained B* or *Undrained C*, the calculation is performed with undrained strength parameters. For the interface to remain elastic the shear stress τ is given by:

$$| \tau | < s_u \qquad \textit{Eq. [32]}$$

where s_u is the undrained shear strength.

For plastic behaviour τ is given by:

$$| \tau | = s_u \qquad \textit{Eq. [33]}$$

The interface property is calculated from the soil property in the associated data set and the strength reduction factor R_{inter} by applying the following rule:

$$s_{u,i} = R_{\text{inter}} s_{u,soil} \qquad \textit{Eq. [34]}$$

when $s_{u,i}$ is the undrained shear strength of the interface and $s_{u,soil}$ is the undrained shear strength of the soil.

The tension cut-off criterion applies as in calculation with effective parameters for strength.

In the NGI-ADP model, $s_{u,soil}$ is computed accordingly to what is reported in the Material Models Manual.

Residual interface strength ($R_{inter,\ residual}$): When the *Manual with residual strength* option is selected the parameter $R_{inter,\ residual}$ can be specified. The interface strength will reduce to the residual strength as defined by ($R_{inter,\ residual}$) and the strength properties of the soil, as soon as the interface strength is reached.

Note:

- Note that the same values of partial factors in *Design approaches* are applied to both interface strength R_{inter} and residual interface strength $R_{inter,\ residual}$.

- A reduced residual strength is not recommended to be used in *Safety* calculations.

Consider gap closure: When the interface tensile strength is reached a gap may occur between the structure and the soil. When the load is reversed, the contact between the structure and the soil needs to be restored before a compressive stress can developed. This is achieved by selecting the *Consider gap closure* option in the **Interfaces** tabsheet of the **Soil** window. If the option is NOT selected, contact stresses will immediately develop upon load reversal, which may not be realistic.

Interfaces using the Hoek & Brown model : When using the Hoek-Brown model as a continuum model to describe the behaviour of a rock section in which interface elements are used, equivalent interface strength properties φ_i, c_i and $\sigma_{t,i}$ are derived from this model. The general shear strength criterion for interfaces as well as the tensile strength criterion are still used in this case:

$$| \tau | \leq -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [35]}$$

$$\sigma_n \leq \sigma_{t,i} \qquad \textit{Eq. [36]}$$

Starting point for the calculation of the interface strength properties is the minor principal effective stress σ'_3 in the adjacent continuum element. At this value of confining stress the tangent to the Hoek-Brown contour is calculated and expressed in terms of φ and c:

$$\sin(\varphi) = \frac{\bar{f}'}{2 + \bar{f}'}$$
$$c = \frac{1 - \sin(\varphi)}{2\cos(\varphi)} \left(\bar{f} + \frac{2\sigma'_3 \sin(\varphi)}{1 - \sin(\varphi)} \right) \qquad \textit{Eq. [37]}$$

where

$$\bar{f} = \sigma_{ci} \left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s \right)^a$$
$$\bar{f}' = a m_b \left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s \right)^{a-1} \qquad \textit{Eq. [38]}$$

and a, m_b, s and σ_{ci} are the Hoek-Brown model parameters in the corresponding material data set. The interface friction angle φ'_i and adhesion c'_i as well as the interface tensile strength $\sigma_{t,i}$ are now calculated using the interface strength reduction factor R_{inter}:

$$\tan(\varphi_i) = R_{\text{inter}} \tan(\varphi)$$
$$c_i = = R_{\text{inter}} c \qquad \textit{Eq. [39]}$$
$$\sigma_{t,i} = R_{\text{inter}} \sigma_t = R_{\text{inter}} \frac{s\sigma_{ci}}{m_b}$$

For more information about the Hoek-Brown model and an explanation of its parameters, reference is made to the Material Models Manual.

Interfaces using the Modified Cam-Clay model If the Modified Cam-Clay model is selected in the **General** tabsheet to describe the behaviour of the surrounding soil, the following parameters are required to model the interface behaviour:

c_{ref}	Cohesion of the interface	[kN/m^2]
φ'_i	Internal friction angle of the interface	[°]

ψ_i	Dilatancy angle of the interface	[°]

When the interface is elastic then both slipping (relative movement parallel to the interface) and gapping or overlapping (i.e. relative displacements perpendicular to the interface) could be expected to occur.

The magnitudes of these displacements are:

$$Elastic\ gap\ displacement = \frac{\sigma_n}{k_n} = \frac{\sigma_n t_i}{E_{oed,i}} \qquad Eq.\ [40]$$

$$Elastic\ slip\ displacement = \frac{\tau}{k_s} = \frac{\tau t_i}{G_i} \qquad Eq.\ [41]$$

where G_i is the shear modulus of the interface, $E_{oed,i}$ is the one-dimensional compression modulus of the interface and t_i is the virtual thickness of the interface, generated during the creation of interfaces in the geometry model (Interfaces (on page 106)). k_n is the elastic interface normal stiffness and k_s is the elastic interface shear stiffness. The shear and compression moduli are related by the expressions:

$$E_{oed,i} = \frac{3}{\lambda}\frac{(1-\nu_i)}{(1+\nu_i)}\frac{\sigma_n}{(1+e_0)}$$

$$G_i = \frac{3(1-2\nu_i)}{2(1+\nu_i)}\frac{\sigma_n}{\lambda(1+e_0)} \qquad Eq.\ [42]$$

$$\nu_i = 0.45$$

Real interface thickness (δ_{inter})

The real interface thickness δ_{inter} is a parameter that represents the real thickness of a shear zone between a structure and the soil. The value of δ_{inter} is only of importance when interfaces are used in combination with the Hardening Soil model or the Hardening Soil model with small-strain stiffness. The real interface thickness is expressed in the unit of length and is generally of the order of a few times the average grain size. This parameter is used to calculate the change in void ratio in interfaces for the dilatancy cut-off option. The dilatancy cut-off in interfaces can be of importance, for example, to calculate the correct bearing capacity of tension piles.

Groundwater flow through interfaces

In situations involving groundwater flow, consolidation or fully coupled flow-deformation analysis, interface elements can contribute to the flow of groundwater and thereby influence the pore pressure distribution. Therefore, interface permeabilities are relevant in such situations. It is also important for the modelling of flow through a system of joints or faults in low permeable rocks when analysing groundwater flow, pore pressures and seepage for applications in rock.

Flow in interface elements may involve flow *through* the element, as well as, flow in the *longitudinal* direction of the element (see Figure 161 (on page 250)). Therefore, the following distinction is made:

1. **Cross permeability**, defined by means of the *Hydraulic resistance*, where water can flow through the interface.

   The cross permeability can be set using the following options (see Figure 160 (on page 249)):

- *Impermeable* [No user input required]: This is the default option. This option sets the interface to have a zero cross permeability (infinite cross resistance), provided the interface is *active in flow* as defined by the interface settings in the **Selection explorer** (see Table 4 (on page 109)). Note that there is no output of groundwater flow q_n available.
- *Semi-permeable*: In this case the interface has a specific non-zero hydraulic resistance, expressed in the unit of time (hydraulic delay), provided the interface is *active in flow* as defined by the interface settings in the **Selection explorer** (see Table 4 (on page 109)). The output program will show the amount of groundwater flow q_n crossing the interface.

  Hydraulic resistance (d/k): The hydraulic resistance defines the permeability through the interface (perpendicular to the interface longitudinal direction). Considering a semi-permeable wall with a thickness d and permeability (or hydraulic conductivity) k, the *hydraulic resistance* is defined by d/k, expressed in the unit of time.To determine d/k, one needs to measure the average discharge q through a wall (per unit of area) for a given head difference Δh, so $d/k = \Delta h/q$.
- *Fully permeable* [No user input required]: In this case the interface has an infinite cross permeability (zero cross resistance), irrespective whether it is *active in flow* as defined by the interface settings in the **Selection explorer** (see Table 4 (on page 109)). Note that there is no output of groundwater flow available when choosing this option.

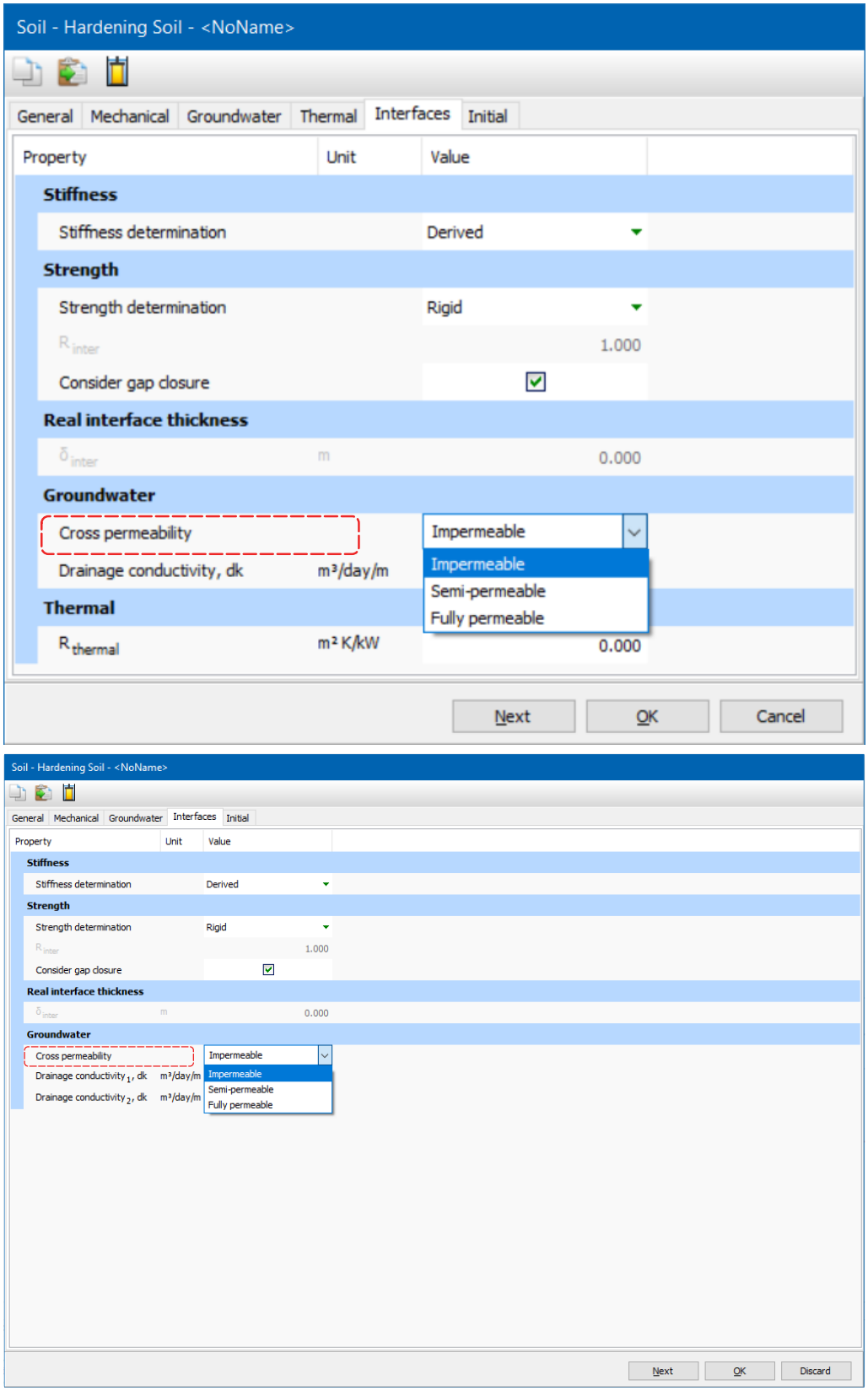


Figure 160: Cross permeability options for interfaces

2. **Drainage conductivity [dk]**, where the interface acts as an additional flow "channel". The *drainage conductivity* defines the permeability in the interface longitudinal direction parallel to the local axes (drainage capacity), provided the interface is *active in flow* as defined by the interface settings in the **Selection explorer** (see Table 4 (on page 109)). The drainage conductivity is expressed in the unit of volume (water volume that is transported in the interface longitudinal direction) per unit of time per unit of width in the out-of-plane direction. The default value is zero, which means that there is no drainage capacity in the interface.

   Considering a semi-permeable gap with a thickness *d* and permeability *k* between two impermeable media, the drainage conductivity is defined by the product of *d* and *k* (*dk*), expressed in the unit of volume per unit of time per unit of width in the out-of-plane direction. This quantity defines the total amount of water that is transported through the gap (drain) per unit of time per unit width. In order to determine *dk*, one needs to measure the total discharge *Q/b* through the gap (per unit of width in the out-of-plane direction) for a given head difference *Δh* and a given length of the gap *L*, such that *dk = (Q/b) (L/Δh)*.

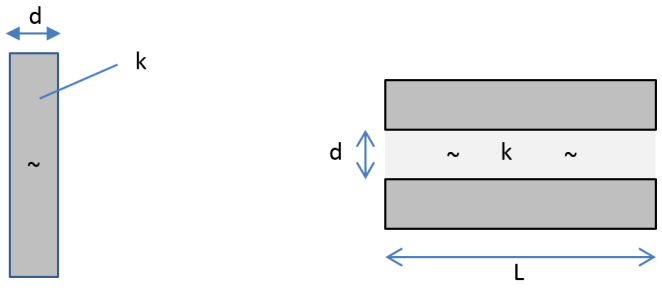


Figure 161: Cross permeability(left) and Drainage conductivity (right)

Note: Notice that, neither for the hydraulic resistance nor for the drainage conductivity, the actual thickness d and permeability k really matter.

Practical remarks: The interface elements are generally at both sides of a wall: for example, in the case of an excavation where the wall is to some extent permeable. In such a case it is suggested to assign the appropriate interface permeability to the 'outside' (soil side) of the wall, whereas the interface at the 'inside' (side that is excavated) is made fully permeable, even along the embedded part of the wall below the excavation. Similarly for a semi-permeable tunnel lining: the appropriate interface permeability should be given to the interface at the outside of the tunnel lining, whereas the interface at the inside (which will be excavated) should be fully permeable. Hence, this requires an additional material data set to be defined which should be assigned directly to the interface.

[ADV] Thermal resistance

In *Thermal* calculations, interface elements can act as insulators, separating the temperature at either side of the interface. As interfaces behave in groundwater flow calculations (where they block the flow completely, behaving as semi-permeable or do not form any resistance at all), interfaces, also, transfer some heat in thermal calculations. The *Thermal resistance*, $R_{thermal}$, is used to control the heat transfer across the interface. Thermal resistance is defined as the ratio of the interface thickness, *d*, and the thermal conductivity of the interface material, λ:

$$R_{thermal} = \frac{d}{\lambda} \qquad Eq.\ [43]$$

Thermal resistance values are generally provided by manufacturers of insulating materials that are produced as plates or blankets (foams, wools). Otherwise, the thermal resistance can easily be calculated from the above equation by considering the material's thermal conductivity and a characteristic thickness of the interface layer.

Interfaces below or around corners of structures

When interfaces are extended below or around corners of structures to avoid stress oscillations (Interfaces (on page 106)), these extended interfaces are not meant to model soil-structure interaction behaviour, but just to allow for sufficient flexibility. Hence, when using $R_{inter} < 1$ for these interface elements an unrealistic strength reduction is introduced in the ground, which may lead to unrealistic soil behaviour or even failure. Therefore it is advised to create a separate data set with $R_{inter} = 1$ and to assign this data set only to these particular interface elements. This can be assigned by dropping the appropriate data set on the individual interfaces rather than dropping it on the associated soil cluster (the interface lines should blink red; the associated soil cluster may not change colour). Alternatively, you can click the right-hand mouse button on these particular interface elements and select *Properties* and subsequently *Positive interface element* or *Negative interface element*. In the **Interface** window, select the appropriate material set in the *Material set* drop-down menu and click the *OK* button.

Interface permeability

The permeability of the interfaces can be specified by checking the corresponding check box *Active in flow* in the *Interface* subtree. Impermeable interfaces may be used to block the flow perpendicular to the interface (cross-permeability) in a consolidation analysis or a groundwater flow calculation, for example to simulate the presence of an impermeable wall. This is achieved by a full separation of the pore pressure degrees-of-freedom of the interface node pairs. On the other hand, if interfaces are present in the mesh it may be the user's intension to explicitly avoid any influence of the interface on the flow and the distribution of (excess) pore pressures, for example in interfaces around corner points of structures (Interfaces (on page 106)). In such a case the interface should be semi-permeable or fully permeable. For fully permeable interfaces the pore pressure degrees-of-freedom of the interface node pairs are fully coupled.

6.1.8 *Initial tabsheet*

The **Initial** tabsheet contains parameters to generate the initial stresses by means of the *K0 procedure* (Figure 162 (on page 252)).

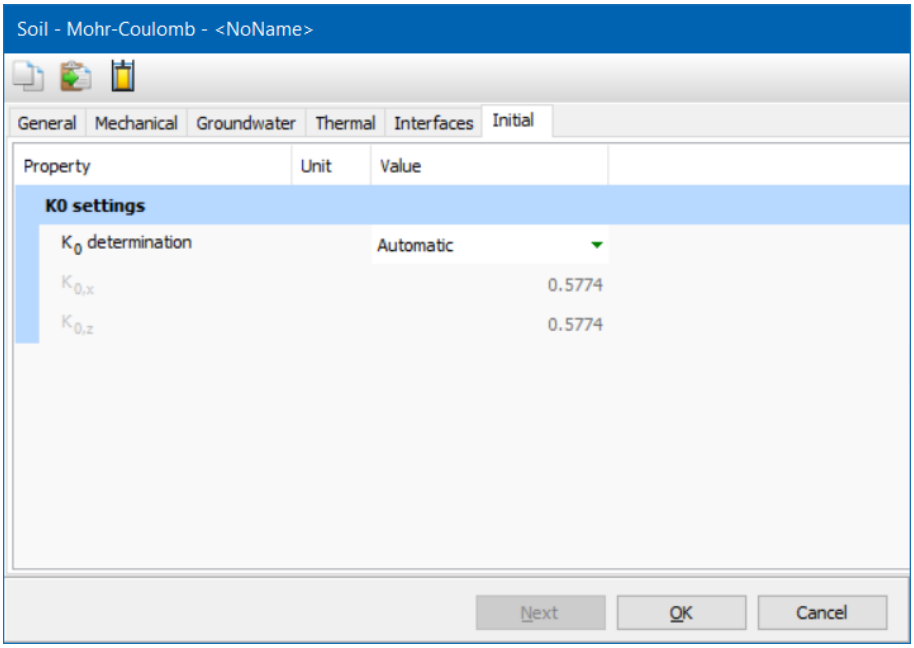


*Figure 162: **Soil** window (**Initial** tabsheet of the Mohr-Coulomb model)*

The K_0-values can be defined automatically by selecting the option *Automatic* in the *K_0 determination* drop-down box or manually by selecting the option *Manual*.

K_0-values

In general, two K_0-values can be specified, one for the *x*-direction (in-plane) and one for the *z*-direction (out-of-plane):

$$K_{0,x} = \sigma'_{xx} / \sigma'_{yy} \quad K_{0,z} = \sigma'_{zz} / \sigma'_{yy} \qquad Eq.\ [44]$$

The checkbox can be used to set the $K_{0,z}$ value equal to the $K_{0,x}$ value.

The default K_0-values are then in principle based on Jaky's formula:

$$K_0 = 1 - \sin(\varphi) \qquad Eq.\ [45]$$

For advanced models (Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model) the default value is based on the K_0^{nc} model parameter and is also influenced by the *OCR*-value and *POP*-value in the following way:

$$K_{0,x} = K_{0,z} = K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1) + \frac{K_0^{nc} POP - \frac{\nu_{ur}}{1-\nu_{ur}} POP}{\left| \sigma'^{0}_{yy} \right|} \qquad Eq.\ [46]$$

The POP-value will result in a stress-dependent K_0 value within the layers and thus the K_0-values are hidden when automatic K_0 determination is chosen (e.g. Figure 163 (on page 253)).

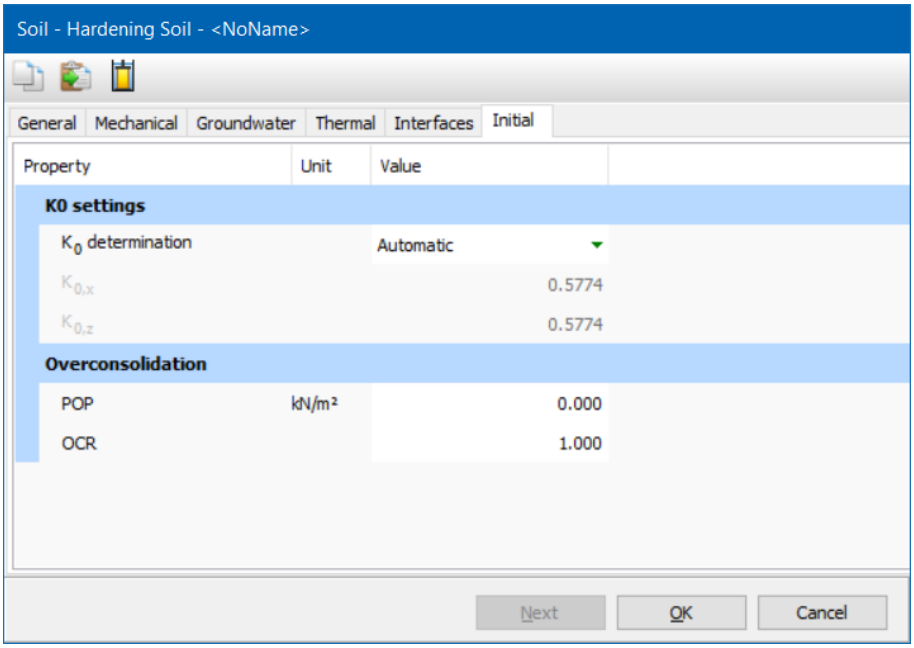


*Figure 163: **Soil** window (**Initial** tabsheet of the Hardening Soil model*

Be careful with very low or very high K_0-values, since these values might bring the initial stress in a state of failure. For a cohesionless material it can easily be shown that to avoid failure, the value of K_0 is bounded by:

$$\frac{1 - \sin(\varphi)}{1 + \sin(\varphi)} < K_0 < \frac{1 + \sin(\varphi)}{1 - \sin(\varphi)} \qquad \textit{Eq. [47]}$$

OCR and POP

When using advanced models (Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model, Sekiguchi-Ohta model) an initial preconsolidation stress has to be determined. In the engineering practice it is common to use a vertical preconsolidation stress, σ_p, but PLAXIS 2D needs an equivalent isotropic preconsolidation stress, p_p^{eq} to determine the initial position of a cap-type yield surface. If a material is overconsolidated, information is required about the overconsolidation Ratio (OCR), i.e. the ratio of the greatest effective vertical stress previously reached, σ_p (see Figure 164 (on page 254)), and the in-situ effective vertical stress, σ'^{0}_{yy}.

$$OCR = \frac{\sigma_p}{\sigma'^{0}_{yy}} \qquad \textit{Eq. [48]}$$

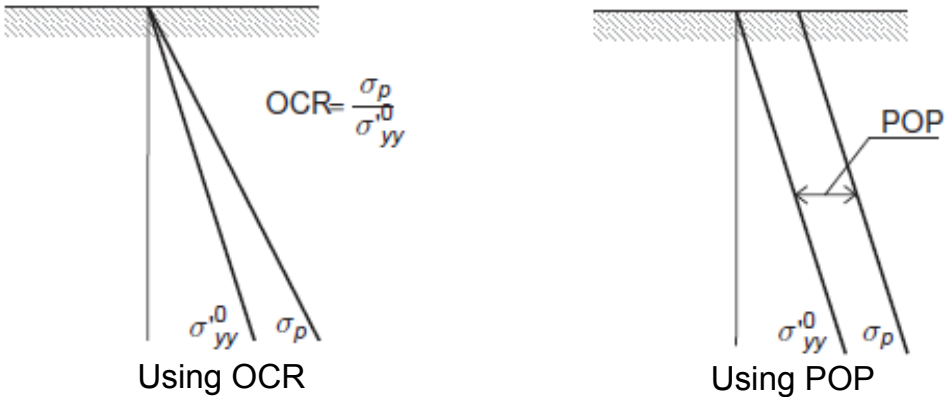


Figure 164: Illustration of vertical preconsolidation stress in relation to the in-situ vertical effective stress

It is also possible to specify the initial stress state using the Pre-Overburden Pressure (POP) as an alternative to prescribing the overconsolidation ratio. The Pre-Overburden Pressure is defined by:

$$POP = \left| \sigma_p - \sigma'^{0}_{yy} \right| \qquad Eq.\ [49]$$

These two ways of specifying the vertical preconsolidation stress are illustrated in Figure 164 (on page 254).

The preconsolidation stress σ_p is used to compute the equivalent preconsolidation pressure p_p^{eq} which determines the initial position of a cap-type yield surface in the advanced soil models. The calculation of p_p^{eq} is based on the principal stress history ($\sigma'_{1,max}$, σ'_2, σ'_3). The actual determination of p_p^{eq} depends on the constitutive model being used.

The principal stress history is initialised in the *Initial phase (K_0-procedure or Gravity loading)* based on the *Cartesian effective stress* components and the *pre-overburden pressure (POP)* or *overconsolidation ratio (OCR)* defined in the boreholes or data set. From this, Cartesian preconsolidation stress levels are calculated based on the following equations:

$$\begin{aligned} \sigma'_{xx,c} &= K_0^{nc} OCR \sigma'_{yy} \\ \sigma'_{yy,c} &= OCR \sigma'_{yy} \\ \sigma'_{zz,c} &= K_0^{nc} OCR \sigma'_{yy} \\ \sigma'_{xy,c} &= \sigma'_{xy} \end{aligned} \qquad Eq.\ [50]$$

where K_0^{nc} is the K_0-value associated with normally consolidated states of stress. K_0^{nc} is a model parameter in advanced constitutive models and estimated in simple models (K_0^{nc}*= 1-sin*φ). Models that do not have φ as input parameter use K_0^{nc}*=0.5* (φ*=30°*).

The Cartesian stress components ($\sigma'_{xx,c}$, $\sigma'_{yy,c}$, $\sigma'_{zz,c}$, $\sigma'_{xy,c}$) are transformed to principal stress components ($\sigma'_{1,max}$, σ'_2, σ'_3) and the maximum major principal stress, $\sigma'_{1,max}$, is kept as a general state parameter which is available for succeeding phases. In subsequent phases, $\sigma'_{1,max}$ is updated if the major principal stress is larger than the current one.

If, in later calculation phases, the soil behaviour is changed to an(other) advanced material model, the equivalent preconsolidation pressure p_p^{eq} is initialised according to the current (updated) principal stress history ($\sigma'_{1,max}$, σ'_2, σ'_3).

Table 16: K_0 behaviour

Material Model	K_0
Linear Elastic model	*0.5*
Mohr-Coulomb model	*1- sin φ*
Hardening Soil model	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Hardening Soil model with small-strain stiffness	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
UBC3D-PLM model	*1- sin* φ_p
Soft Soil model	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Soft Soil Creep model	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Jointed Rock model	0.5
Modified Cam-Clay model	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
NGI-ADP model	Calculated by the kernel
UDCAM-S model	Calculated by the kernel
Hoek-Brown model	0.5
Sekiguchi-Ohta model (Inviscid)	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Sekiguchi-Ohta model (Viscid)	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Concrete model	*1- sin* φ_{max}
User-defined Soil Model	0.5

6.2 Modelling undrained behaviour

In undrained conditions, no water movement takes place. As a result, excess pore pressures are built up. Undrained analysis is appropriate when:

- Permeability is low or rate of loading is high.
- Short term behaviour has to be assessed.

Different modelling schemes are possible in PLAXIS 2D to model undrained soil response in a *Plastic* calculation, a *Safety* analysis or a *Dynamic* analysis. The modelling scheme depends on the selection of the *Drainage type* parameter (General tabsheet (on page 162)). More details about these methods are in the Material Models Manual - *Preliminaries on material modelling.*

Note:

The *Drainage type* setting is only considered in a *Plastic* calculation, a *Safety* analysis or a *Dynamic* analysis. When a *Consolidation* analysis, a *Fully coupled flow-deformation* analysis or a *Dynamic with consolidation* analysis is performed, the *Drainage type* is ignored and the soil response is determined by the saturated permeability of the material that is specified in the **Groundwater** tabsheet of the material datase.

Note:

The modelling of undrained soil behaviour is even more complicated than the modelling of drained behaviour. Therefore, the user is advised to take the utmost care with the modelling of undrained soil behaviour.

Before considering the consequences of a particular selection of the drainage type parameter for undrained soil behaviour, first a general description is given of the various modelling possibilities:

1. ***Undrained effective stress analysis with effective stiffness parameters***

   A change in total mean stress in an undrained material during a *Plastic* calculation phase gives rise to excess pore pressures. PLAXIS 2D differentiates between steady-state pore pressures and excess pore pressures, the latter generated due to small volumetric strain occurring during plastic calculations and assuming a low (but non-zero) compressibility of the pore water. This enables the determination of effective stresses during undrained plastic calculations and allows undrained calculations to be performed with effective stiffness parameters. This option to model undrained material behaviour based on effective stiffness parameters is available for all material models in the PLAXIS 2D. The undrained calculations can be executed with effective stiffness parameters, with explicit distinction between effective stresses and (excess) pore pressures.

2. ***Undrained effective stress analysis with effective strength parameters***

   Undrained effective stress analysis can be used in combination with effective strength parameters φ' and c' to model the material's undrained shear strength. In this case, the development of the pore pressure plays a crucial role in providing the right effective stress path that leads to failure at a realistic value of undrained shear strength (c_u or s_u). However, note that most soil models are not capable of providing the right effective stress path in undrained loading. As a result, they will produce the wrong undrained shear strength if the material strength has been specified on the basis of effective strength parameters. Another problem is that for undrained materials effective strength parameters are usually not available from soil investigation data.

   The advantage of using effective strength parameters in undrained loading conditions is that after consolidation a qualitatively increased shear strength is obtained, although this increased shear strength could also be quantitatively wrong, for the same reason as explained before.

3. ***Undrained effective stress analysis with undrained strength parameters***

Especially for soft soils, effective strength parameters are not always available, and one has to deal with measured undrained shear strength (c_u or s_u) as obtained from undrained tests. Undrained shear strength, however, cannot easily be used to determine the effective strength parameters φ' and c'. Moreover, even if one would have proper effective strength parameters, care has to be taken as to whether these effective strength parameters will provide the correct undrained shear strength in the analysis. This is because the effective stress path that is followed in an undrained analysis may not be the same as in reality, due to the limitations of the applied soil model.

In order to enable a direct control on the shear strength, PLAXIS 2D allows for an undrained effective stress analysis with direct input of the undrained shear strength (*Undrained B*).

6.2.1 *Undrained A*

The *Drainage type Undrained A* enables modelling undrained behaviour using effective parameters for stiffness and strength. The characteristic features of method *Undrained A* are:

- The undrained calculation is performed as an effective stress analysis. Effective stiffness and effective strength parameters are used.
- Pore pressures are generated, but may be inaccurate, depending on the selected model and parameters.
- Undrained shear strength s_u is not an input parameter but an outcome of the constitutive model. The resulting mobilised shear strength must be checked against known data.
- Consolidation analysis can be performed after the undrained calculation, which affect the shear strength.

Undrained A drainage type is available for the following models: Linear Elastic model, Mohr-Coulomb model, Hardening Soil model, Hardening Soil model with small-strain stiffness, UBC3D-PLM model, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and UDSM.

6.2.2 *Undrained B*

The *Drainage type Undrained B* enables modelling undrained behaviour using effective parameters for stiffness and undrained strength parameters. The characteristic features of method *Undrained B* are:

- The undrained calculation is performed as an effective stress analysis.
- Effective stiffness parameters and undrained strength parameters are used.
- Pore pressures are generated, but may be highly inaccurate.
- Undrained shear strength s_u is an input parameter.
- Consolidation analysis should not be performed after the undrained calculation. If consolidation analysis is performed anyway, s_u must be updated.

Undrained B drainage type is available for the following models: Mohr-Coulomb model, Hardening Soil model, Hardening Soil model with small-strain stiffness, and UDCAM-S model. Note that when using *Undrained B* in the Hardening Soil model or Hardening Soil model with small-strain stiffness, the stiffness moduli in the model are no longer stress-dependent and the model exhibits no compression hardening.

6.2.3 *Undrained C*

The *Drainage type Undrained C* enables simulation of undrained behaviour using a total stress analysis with undrained parameters. In that case, stiffness is modelled using an undrained Young's modulus E_u and an undrained Poisson ratio ν_u, and strength is modelled using an undrained shear strength c_u (s_u) and $\varphi = \varphi_u = 0°$. Typically, for the undrained Poisson ratio a value close to 0.5 is selected (between 0.495 and 0.499). A value of exactly 0.5 is not possible, since this would lead to singularity of the stiffness matrix. The disadvantage of this approach is that no distinction is made between effective stresses and pore pressures. Hence, all output referring to effective stresses should now be interpreted as total stresses and all pore pressures are equal to zero. Note that a direct input of undrained shear strength does not automatically give the increase of shear strength with consolidation. The characteristic features of method *Undrained C* are:

- The undrained calculation is performed as a total stress analysis.
- Undrained stiffness parameters and undrained strength parameters are used.
- Pore pressures are not generated.
- Undrained shear strength s_u is an input parameter.
- Consolidation analysis should not be performed after the undrained calculation. If consolidation analysis is performed anyway, s_u must be updated.

Undrained C drainage type is available for the following models: Linear Elastic model, Mohr-Coulomb model, NGI-ADP model and UDCAM-S model.

Note:

For *Undrained B* and *Undrained C* an increased shear strength with depth can be modelled using the advanced parameter $s_{u,inc}$.

6.2.4 *Undrained behaviour for unsaturated soil*

As aforementioned, the build-up of pore pressures plays a crucial role in undrained analyses, especially *Undrained A* and *Undrained B*. Excess pore pressures are generated assuming very low compressibility of pore fluids. This is only valid for fully saturated soils where voids are filled by water. In unsaturated soils, the porous space is filled by both water and air. It is thus expected that the compressibility of pore fluid is much higher compared to saturated soils, and thereby a lower excess pore pressure.

On physical grounds, water in unsaturated soils co-exists with air under the form of water meniscus due to the capillary effect (surface tension). The excess pore pressure is governed by three factors: the compressibility of water, compressibility of air and surface tension (suction) effect. The latter is related to the Soil-Water retention curve (defined in Groundwater tabsheet (on page 224)).

In PLAXIS, we assume that the air pressure is always constant, namely excess air pressure and air compressibility are not considered (see Appendix Definition of stresses and related quantities (on page 543)). With this in mind, the undrained analysis only means that the water phase is undrained, while air phase is always drained. Excess pore pressure in undrained behaviour for an unsaturated zone is generated following two cases:

- If Ignore suction (on page 544) is not chosen , the suction effect is considered. Excess pore pressure is governed by an ''unsaturated bulk modulus of pore fluid'' which takes into account both the compressibility

of water and Soil-Water retention curve (see Table 22 (on page 551)). This fluid bulk modulus usually is much smaller than the water bulk modulus of water.

- If *Ignore suction* is chosen, the suction effect is neglected, and soil behaves like a saturated soil. Excess pore pressure is only governed by the compressibility of water in a similar way as saturated soil and is generally much higher than the case of considering suction. This might probably result in a stiffer undrained volume change behaviour, but a lower shear strength due to a higher pore pressure build-up. This is certainly dependent on the constitutive model used.

Note:

The behaviour of unsaturated soil is more realistic but also more complicated than saturated soil. The user is advised to take the utmost care with the modelling as well as Soil-Water retention curve input parameters.

6.3 Material data sets for discontinuities

As mentioned in Discontinuities (on page 111), the material properties for discontinuities are entered in the material data set window.

6.3.1 *General properties*

Several data sets may be created to distinguish different discontinuities. As displayed in Figure , the material data set includes the following options:

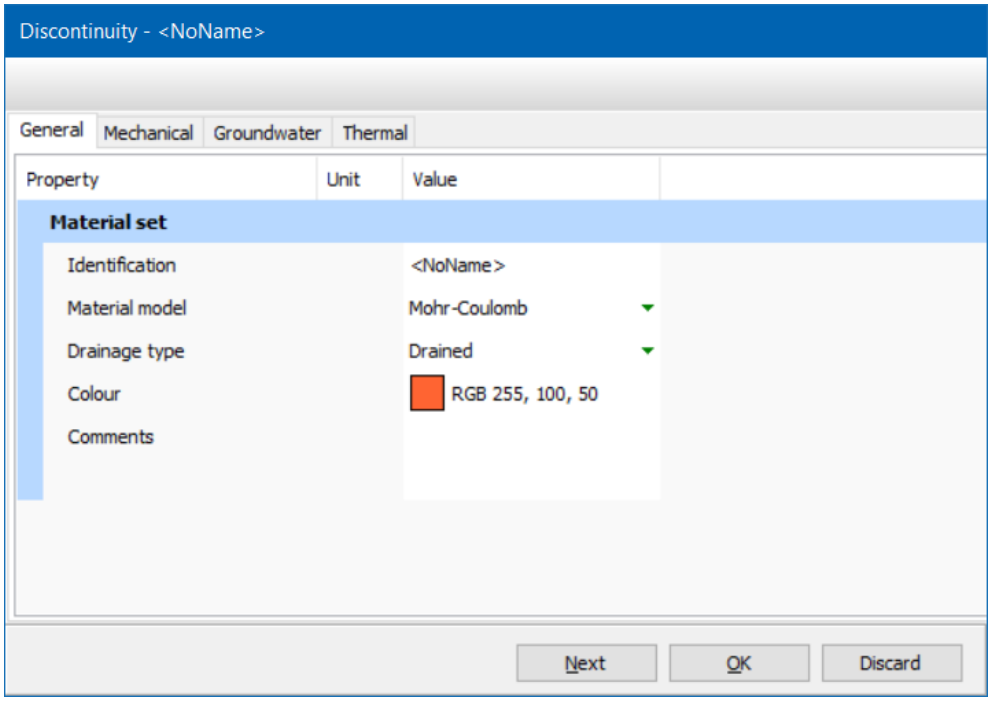


Figure 165: General tabsheet - Material properties of discontinuities

Identification	A user may specify any identification title for a discontinuity data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.
Material Model[1]	Material model to be considered in the discontinuity.
Drainage type	Two options allowed *Drained* and *Non-porous*. Discontinuities can be open or filled with certain porous materials (for instance quartz, calcite or sand) which can be considered as *Drained*. However, if the infill is considered non-porous (for instance rock grouting for impermebilisation techniques), the discontinuity element can also be considered *Non-porous*.
Colour	Colour can be used as a distinction tool in the model. The default color for a discontinuity is orange.
Comments	A user may write down comments related to the material data set.

Note:

- [1]Currently only the Mohr-Coulomb material model is available for the modelling of discontinuity elements.

6.3.2 *Mechanical properties*

The required parameters for the strength and stiffness of discontinuity are defined in the **Mechanical** tabsheet of the material properties definition as shown in Figure 166 (on page 261).

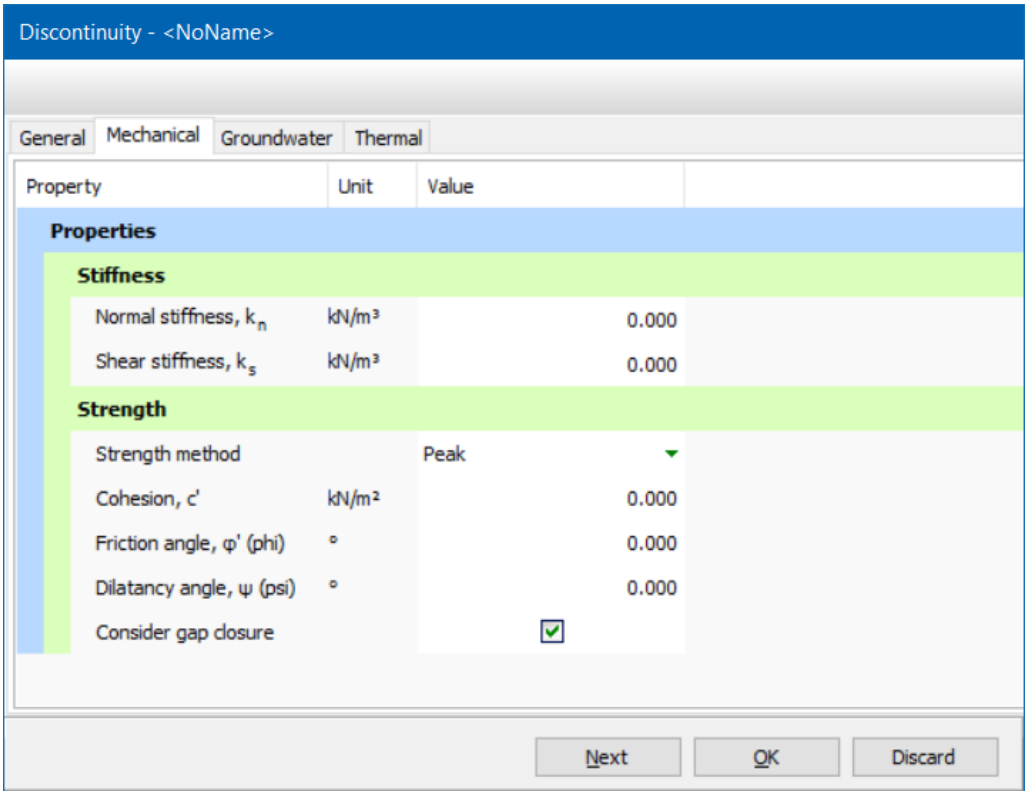


Figure 166: Mechanical properties of discontinuities with peak strength

Stiffness

Two distinct material stiffness can be assigned to discontinuities k_n and k_s.

k_n	Normal stiffness	kN/m^3
k_s	Shear stiffness	kN/m^3

Strength

Defines the strength parameters of the material in the discontinuity according the material model available (see General properties (on page 259)). Two strength methods are available: *Peak* and *Peak and Residual*.

Peak

This is the default method. The strength parameters represent the peak strength and no material softening occurs when the peak strength is reached.

c,	Effective cohesion of the material in the discontinuity	kN/m^2
φ'	Effective Internal friction angle of the material in the discontinuity	°
ψ_i	Dilatancy angle of the material in the discontinuity	°

Consider gap closure	When tensile stresses are applied to a discontinuity, the discontinuity will open, creating a gap in the element. When the load is reversed, the contact between either sides of the discontinuities needs to be restored before a compressive stress can develop	-

Peak and Residual

When this *Strength method* is selected, the RSF (Residual Strength Factor) can be specified for the discontinuity (as shown in Figure 167 (on page 262)). Once the peak strength is reached, the discontinuity material strength will reduce to its residual strength as defined by RSF, as follows:

If $RSF < 1$:

- $c_{res} = RSF \cdot c_{ini}$
- $\tan(\varphi_{res}) = RSF \cdot \tan(\varphi_{ini})$

RSF	Residual Strength Factor	-

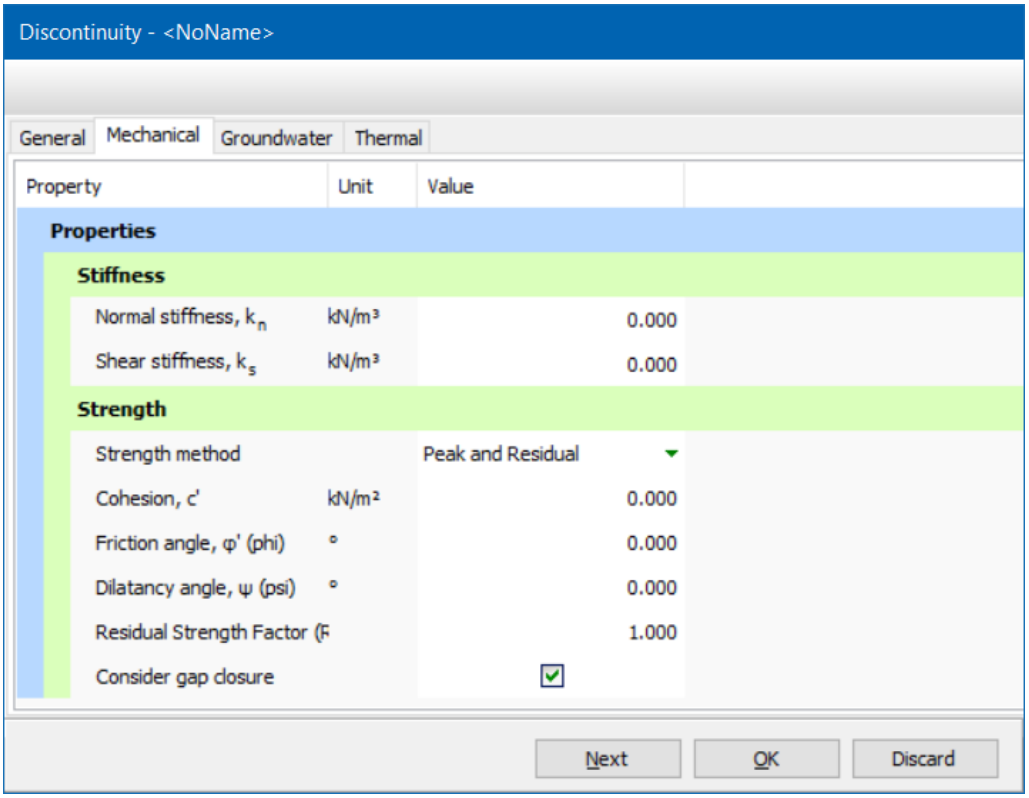


Figure 167: Mechanical properties of discontinuities with peak and residual strength

6.3.3 *Groundwater properties*

Groundwater flow parameters are required when dealing with the hydraulic conditions of discontinuities specifically when drained behaviour is addressed. For this reason, to enable the use of groundwater hydraulic parameters the *Drainage type* of material needs to be set as *Drained* (see General properties (on page 259)). When drained conditions are selected, the **Groundwater** tabsheet is displayed as in Figure 168 (on page 263).

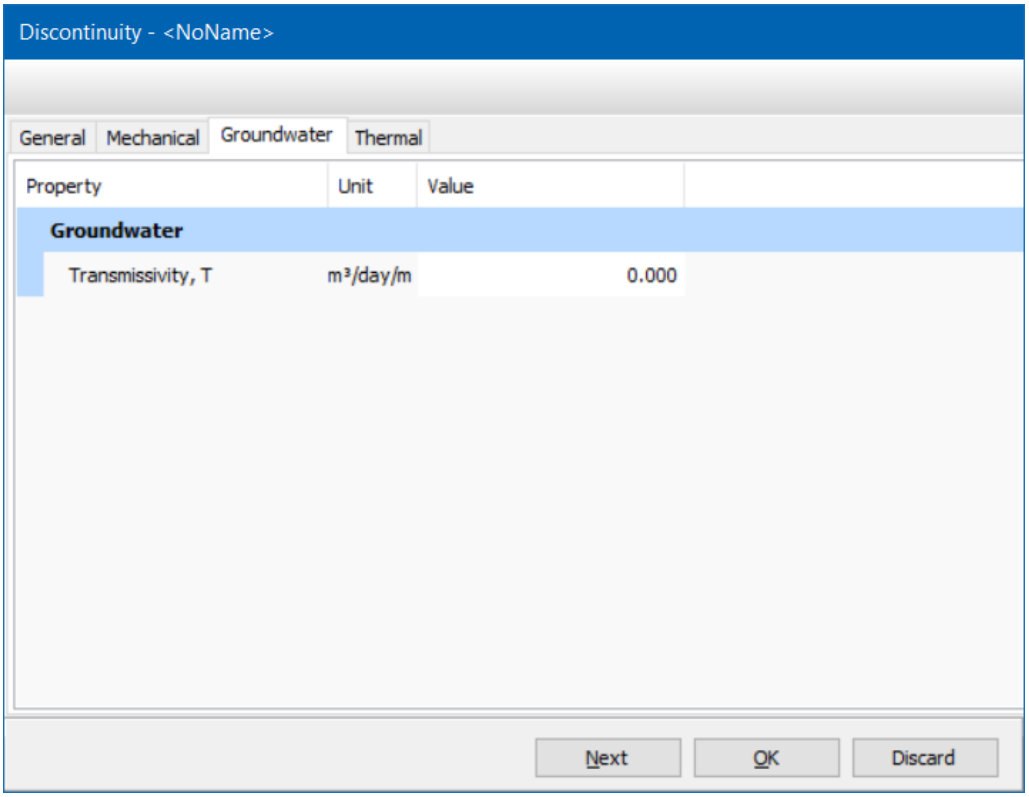


Figure 168: Groundwater properties of discontinuities

The groundwater parameter available is listed below:

T_1	Transmissivity in the local direction 1 of a discontinuity: volumetric flow rate though the discontinuity in this local direction per unit width (perpendicular to the flow direction) under unit hydraulic gradient	$m^3/day/m$

It is important to mention that under drained conditions the discontinuity behaves as Fully permeable (the discontinuity has an infinite cross permeability or zero cross resistance), and the Transmissivity represents the drainage conductivity in the longitudinal direction of the fracture. .

Note: If *Drainage type* defined in General properties (on page 259) is set as *Non-porous* the *Transmissivity* (T) is disabled.

6.3.4 *Thermal properties [ADV]*

The **Thermal** tabsheet (see Figure 169 (on page 264)) contains discontinuity properties for problems involving (a change of) temperature and the influence on stress, deformation or groundwater.

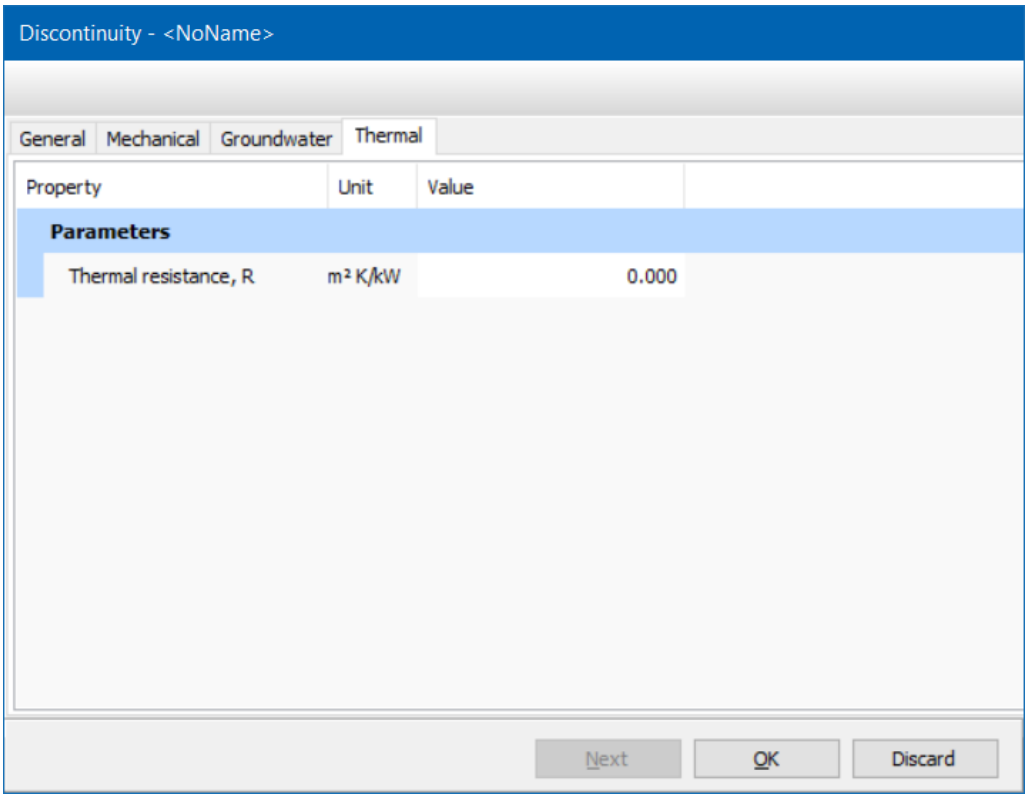


Figure 169: Thermal tabsheet - Discontinuity element

The thermal parameter available for discontinuities is listed below:

R	Thermal Resistance	m^2K/kW

The thermal resistance R is used to control the heat transfer across the discontinuity. R is defined as the ratio between the aperture of the discontinuity, d, and the thermal conductivity, λ, of the discontinuity material.

$$R = \frac{d}{\lambda} \qquad Eq.\ [51]$$

6.4 Material data sets for plates

In addition to material data sets for soil and interfaces, the material properties and model parameters for plates are also entered in separate material data sets. Plates are used to model the behaviour of slender walls, plates or thin shells. Distinction can be made between elastic, elastoplastic and elastoplastic (M-κ) behaviour. A data set for plates generally represents a certain type of plate material, and can be assigned to the corresponding (group of) plate elements in the geometry model.

6.4.1 *General properties*

In the General tabsheet properties with respect to the material behavioru, weight and also damping in a Dynamics analysis must be specifed. Figure 169 shows the *General* tabsheet of the **Plate** window.

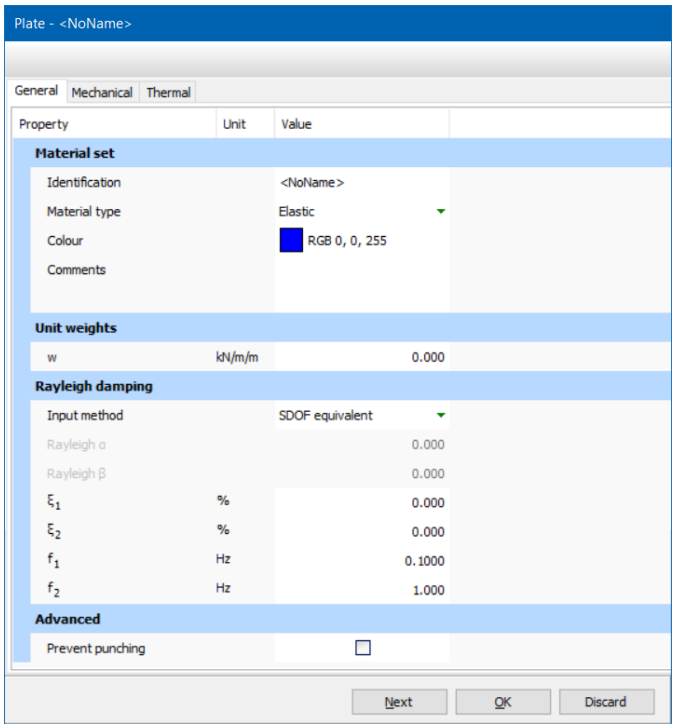


*Figure 170: General tabsheet of the **Plate** window*

Material set

The material data set is defined by:

Identification	A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.
Material type	There are three available options, describing the material type of a plate. These options are *Elastic*, *Elastoplastic* and *Elastoplastic (M-κ)*. The availability of the parameters defined at the *Mechanical* tabsheet depends on the selected material type.
Comments	A user may write down comments related to the material data set.
Colour	Colour can be used as a distinction tool in the model.

Unit weight

Additionally, the *unit weight* of the plate must be specified:

w	In a material set for plates a specific weight can be specified.	[kN/m/m]

Rayleigh damping[ULT]

For dynamic behaviour, the Rayleigh α and Rayleigh β damping parameters can be specified either directly or through Single Degree of Freedom (SDOF) equivalent parameters.

For more information on Rayleigh damping and how to specify the parameters, see Rayleigh damping [ULT] (on page 166).

Advanced

Prevent punching: In reality vertical loads on walls for example, as a result of vertical components of anchor forces are sustained by the shaft friction and the tip resistance. A certain amount of resistance is offered by the soil under the tip, depending on the thickness or the cross section area of the tip.

Slender walls are often modelled as plates. Due to the zero thickness of the plate elements vertical plates (walls) have no end bearing. The effects of end bearing can still be considered in the calculation when the *Prevent punching* option is selected in the material data set. In order to consider end bearing at the bottom of plates, a zone in the soil volume elements surrounding the bottom of the plate is identified where any kind of soil plasticity is excluded (elastic zone). The size of this zone is determined as $D = \sqrt{12EI / EA}$.

It is not the idea that this end bearing represents the true end bearing capacity of the wall. It just prevents unrealistic vertical movement of penetration of the wall into deeper layers. The value of end bearing cannot be prescribed by the user, and it is neither theoretically determined by PLAXIS 2D.

Note:

Note that sheet pile walls have very little end bearing, considering that the thickness of the steel is much less than D_{eq}, when modelling them as plates. Hence, the *Prevent punching* option shall not be used for sheet pile walls.

6.4.2 *Mechanical properties*

The mechanical properties required for plates can be grouped into stiffness properties and strength properties (in case of elastoplastic behaviour).

Stiffness properties

For elastic behaviour, several parameters should be specified as material properties. PLAXIS 2D allows for orthotropic material behaviour in plates where the in-plane stiffness and out-of-plane stiffness are different. This is most relevant for axisymmetric models when modelling sheet pile profiles (which have a low stiffness in the out-of-plane direction).

Isotropic	If selected, the in-plane stiffness and out-of-plane stiffness will be equal.	[-]
EA_1	In-plane elastic axial stiffness.	[kN/m]

EA_2	Out-of-plane elastic axial stiffness.	[kN/m]
EI	Elastic bending stiffness. Note that in the case of *Elastoplastic (M-κ)* behaviour, EI is automatically determined based on the first line segment in the M-κ diagram.	[kN/m^2/m]
ν	Poisson's ratio.	[-]

From the ratio of EI and EA_1 an equivalent thickness for an equivalent plate (d) is automatically calculated from the equation:

$$d = \sqrt{12\frac{EI}{EA_1}} \qquad Eq.\ [52]$$

For the modelling of plates, PLAXIS 2D uses the Mindlin beam theory as described in Bathe (1982) (on page 530). This means that, in addition to bending, shear deformation is taken into account. The shear stiffness of the plate is determined from:

$$Shear\ stiffness = \frac{5EA}{12(1+\nu)} = \frac{5E(d \cdot 1m)}{12(1+\nu)} \qquad Eq.\ [53]$$

This implies that the shear stiffness is determined from the assumption that the plate has a rectangular cross section. In the case of modelling a solid wall, this will give the correct shear deformation. However, in the case of steel profile elements, like sheet-pile walls, the computed shear deformation may be too large. You can check this by judging the value of d_{eq}. For steel profile elements, d_{eq} should be at least of the order of a factor 10 times smaller than the length of the plate to ensure negligible shear deformations.

More information about the behaviour and structural forces in plates can be found in the Material Models Manual - *Structural elements - 2D Plates.*

In addition to the above stiffness parameters, a Poisson's ratio, ν, is required. For thin structures with a certain profile or structures that are relatively flexible in the out-of-plane direction (like sheet-pile walls), it is advisable to set Poisson's ratio to zero. For real massive structures (like concrete walls) it is more realistic to enter a true Poisson's ratio of the order of 0.15.

Since PLAXIS 2D considers plates (extending in the out-of-plane direction) rather than beams (one-dimensional structures), the value of Poisson's ratio will influence the bending stiffness of the isotropic plate in plane strain applications as follows:

Input value of bending stiffness: EI

Observed value of bending stiffness: EI/(1 - v ^{2})

The stiffening effect of Poisson's ratio is caused by the stresses in the out-of-plane direction (σ_{zz}) and the fact that strains are prevented in this direction. Note that the Poisson's ration (ν) is assumed to be zero in anisotropic case.

Strength properties

Elastoplastic

Strength parameters are required in the case of elastoplastic behaviour:

M_p	Maximum bending moment.	[kNm/m]
$N_{p,1}$	The maximum axial force in 1-direction.	[kN/m]

$N_{p,2}$	The maximum axial force in 2-direction (anisotropic behaviour).	[kN/m]

Plasticity may be taken into account by specifying a maximum bending moment, M_p. The maximum bending moment is given in units of force times length per unit width. In addition to the maximum bending moment, the axial force is limited to N_p. The maximum axial force, N_p, is specified in units of force per unit width. When the combination of a bending moment and an axial force occur in a plate, then the actual bending moment or axial force at which plasticity occurs is lower than respectively M_p or N_p. More information is given in the Material Models Manual - *Structural elements - 2D Plates*.

The relationship between M_p and N_p is visualised in Figure 171 (on page 268). The diamond shape represents the ultimate combination of forces for which plasticity will occur. Force combinations inside the diamond will result in elastic deformations only. The Scientific Manual describes in more detail how PLAXIS 2D deals with plasticity in plates.

Bending moments and axial forces are calculated at the stress points of the plates elements (Figure 65 (on page 105)) . If M_p or N_p is exceeded, stresses are redistributed according to the theory of plasticity, so that the maxima are complied with. This will result in irreversible deformations. Output of bending moments and axial forces is given in the nodes, which requires extrapolation of the values at the stress points. Due to the position of the stress points in a beam element, it is possible that the nodal values of the bending moment may slightly exceed M_p. If the *Isotropic* option is checked the input is limited to $N_{p,1}$ where as $N_{p,1} = N_{p,2}$.

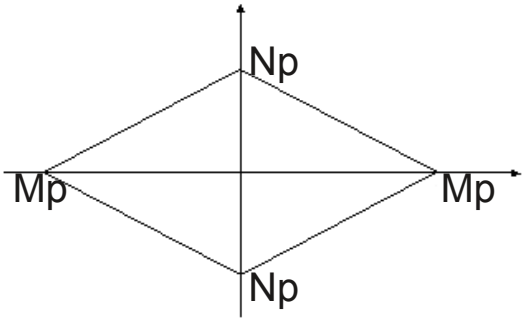


Figure 171: Combinations of maximum bending moment and axial force

Elastoplastic (M-κ)

In the case the *Material type* is set to *Elastoplastic (M-κ)*, the bending behaviour of the plate is governed by a user-defined moment-curvature (*M-κ*) diagram. Input values of the *κ* diagram can be specified in the table that appears at the right hand panel when the *M-κ* diagram input field is selected. Here, a name can be given to the *M-κ* diagram to be defined. Adding rows to the table can be done by clicking the '+' sign above the table. Only positive values can be specified for curvature *κ* (in the unit of 1/length) and moment M (in the unit of Force times length per unit of width in the out-of-plane direction), since the *M-κ* diagram is assumed to govern the plate's behaviour in both positive and negative loading directions. A graphical representation of the M-*κ* diagram is presented below the table (only positive values).

The first row in the *M-κ* diagram determines the plate's elastic bending stiffness *EI*:

$$EI = M(1) / \kappa(1) \qquad Eq.\ [54]$$

This *EI* value is included in the set of stiffness properties. Additional rows in the *M-κ* table define the plate's non-linear elastoplastic behaviour. The last M-value in the table is assumed to be the maximum bending moment, M_p, at which the plate fails. Elastic bending strains are calculated from the aforementioned *EI* value, whereas additional bending (following the defined *M-κ* diagram) is assumed to be plastic, as long as the loading is in the

same direction. Upon unloading from an elastoplastic state, the behaviour is initially elastic, but after significant unloading it may again generate plastic strains.

In the *Elastoplastic (M-κ)* plates, the axial force *N* is considered elastic and independent from the bending moment *M*. The axial force influences the stresses and displacements of the element but without affecting the bending moment. The M-*κ* diagram is assumed to be the only one governing the plate's flexural behaviour.

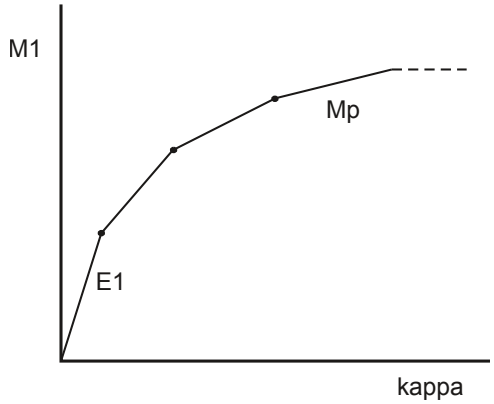


Figure 172: M-κ diagram

It is possible to change the material data set of a plate in the framework of staged construction . However, it is very important that the ratio of *EI*/*EA* is not changed, since this will introduce an out-of-balance force (see Plates (on page 103)).

6.4.3 Thermal properties [ADV]

For thermal behaviour, additional constants are specified in the *Thermal* tabsheet.

c	The specific heat capacity of the solid material. Describes the amount of energy (heat) that can be stored in the solid material per unit of mass.	[kJ/t/K]
λ	The thermal conductivity of the solid material. Describes the rate of energy (heat) that can be transported in the solid material.	[kW/m/K]
ρ	The density of the solid material. Describes the density of the solid particles.	[t/m^3]
α	The thermal expansion coefficient. Describes how much the material expands (or elongates) when the temperature increases. In other words, it is the (change of) strain per unit of temperature.	[1/K]
$A_{eff,T}$	The effective area for thermal flux.	[m^2]

6.5 Material data sets for geogrids

In addition to material data sets for soil and interfaces, the material properties and model parameters for geogrids are also entered in separate material data sets. Geogrids are flexible elastic or elastoplastic elements that represent a grid or sheet of fabric. Geogrids can only sustain tensile forces, but not compressive forces. A data set for geogrids generally represents a certain type of geogrid material, and can be assigned to the corresponding (group of) geogrid elements in the geometry model.

6.5.1 *General properties*

Several data sets may be created to distinguish between different types of geogrids or geosynthetics. Figure 173 (on page 270) shows the *General* tabsheet of the **Geogrid** window. The general properties of a geogrid are:

Identification	A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.
Comments	A user may write down comments related to the material data set.
Colour	Colour can be used as a distinction tool in the model.
Material type	There are four available options, describing the material type of a plate. These options are *Elastic*, *Elastoplastic*, *Elastoplastic (N-ε)* and *Visco-elastic (time-dependent)*. The availability of the parameters defined at the *Mechanical* tabsheet depends on the selected material type.

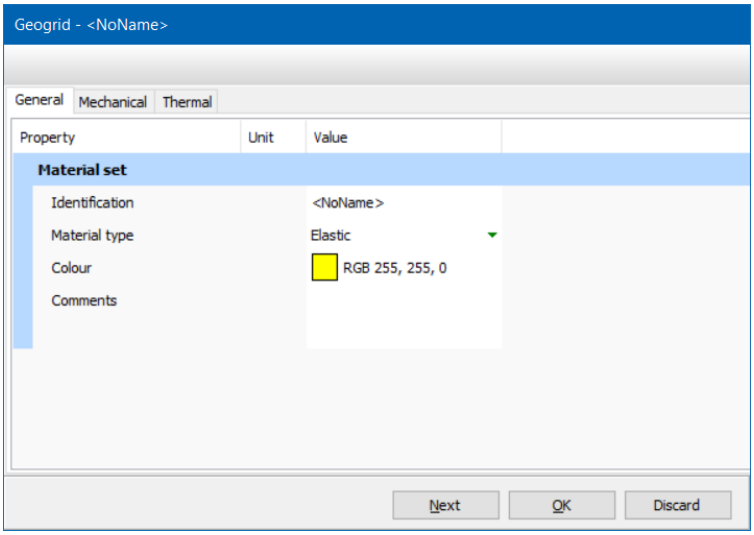


*Figure 173: **Geogrid** window*

6.5.2 *Mechanical properties*

The properties required for geogrids can be grouped into stiffness properties and strength properties in case of elastoplastic behaviour.

Isotropic

Different stiffness and strength in-plane and out-of-plane may be considered. If this is not the case, the *Isotropic* option may be selected to ensure that the stiffness in both directions is equal.

Stiffness properties

Elastic

For elastic behaviour, the axial stiffness *EA* should be specified. PLAXIS 2D allows for orthotropic material behaviour in geogrids, which is defined by the following parameters:

EA_1	The axis axial elastic stiffness in 1-direction (in-plane).	[kN/m]
EA_2	The axial elastic stiffness in 2-direction (out-of-plane, anisotropic behaviour).	[kN/m]

The axial stiffness *EA* is usually provided by the geogrid manufacturer and can be determined from diagrams in which the elongation of the geogrid is plotted against the applied force in a longitudinal direction. The axial stiffness is the ratio of the axial force *F* per unit width and the axial strain ($\Delta l/l$ where Δl is the elongation and l is the original length):

$$EA = \frac{F}{\Delta l / l} \qquad Eq.\ [55]$$

If the *Isotropic* option is checked the input is limited to EA_1 where as $EA_2 = EA_1$.

Visco-elastic (time-dependent)

The parameters are based on a visco-elastic perfectly-plastic Kelvin-Voigt model in each direction, which allows for time-dependent behaviour.

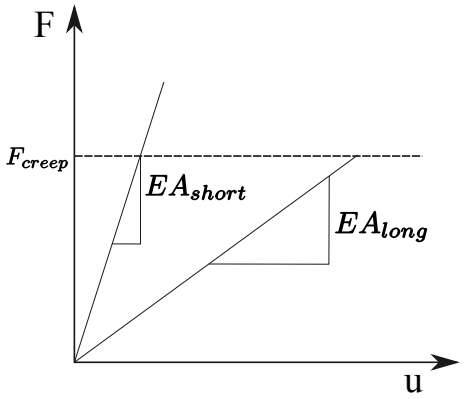


Figure 174: Force versus displacement showing EA_{short} and EA_{long}

Considering one *Kelvin-Voigt* element (see Figure 175 (on page 272)),

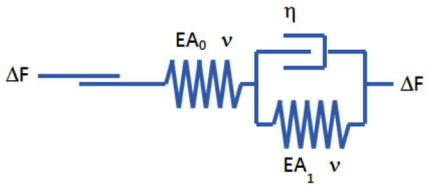


Figure 175: Kelvin-Voigt single element representation

$$EA_0 = EA_{short}$$

$$\frac{1}{EA_0} + \frac{1}{EA_1} = \frac{1}{EA_{long}}$$

$$EA_1 = \frac{1}{\left(\frac{1}{EA_{long}} - \frac{1}{EA_0}\right)} \qquad \textit{Eq. [56]}$$

$$\eta = EA_1 \times \text{Retardation time}$$

where η is viscous damping, EA_0 and EA_1 are internal stiffness used in the geogrid material model.

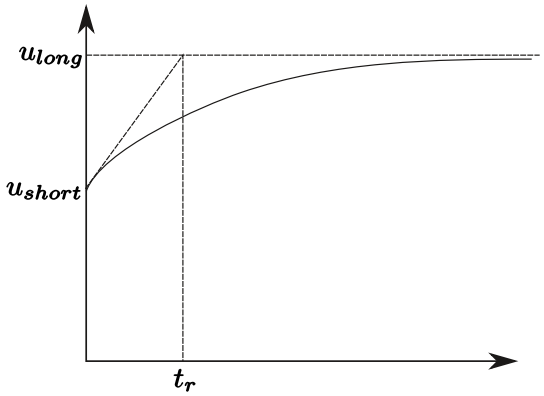


Figure 176: Displacement versus time in a Creep test

Retardation time can be obtained from a creep test in which a force F is applied, using the following equations (see Figure 176 (on page 272)):

$$t_{\text{retardation}} = \frac{\eta_1}{E_1}$$

$$EA_{short} = \frac{F}{u_{short}}$$

$$EA_{long} = \frac{1}{EA_{short}} \qquad \textit{Eq. [57]}$$

$$u_{short} = \frac{F}{EA_{short}}$$

$$u_{long} = \frac{F}{EA_{long}}$$

The short-term and long-term stiffnesses can be obtained from the same creep test by measuring the corresponding displacement u at the short and long time, respectively.

Parameters which are required for time dependent visco-elasticity are (Figure 177 (on page 273)):

$EA_{1,short}$	Elastic stiffness during initial (instantaneous) strain increment in 1-direction (in-plane).	[kN/m]
$EA_{2,short}$	Elastic stiffness during initial (instantaneous) strain increment in 2-direction (out-of-plane, anisotropic behaviour).	[kN/m]
$EA_{1,long}$	Elastic stiffness during (infinitely) long strain increment in 1-direction (in-plane).	[kN/m]
$EA_{2,long}$	Elastic stiffness during (infinitely) long strain increment in 1-direction (out-of-plane, anisotropic behaviour).	[kN/m]
Retardation time	The time where a linear extrapolation of the initial creep rate intersects the long-term displacement line.	[day]

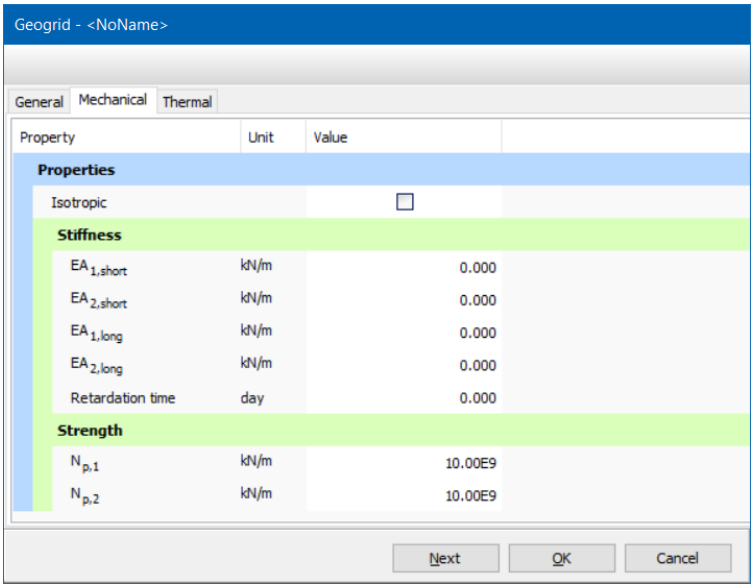


Figure 177: Geogrid window

Strength properties

Elastoplastic

Strength parameters are required in case of elastoplasticity:

$N_{p,1}$	The maximum force in 1-direction (in-plane).	[kN/m]
$N_{p,2}$	The maximum force in 2-direction (out-of-plane, anisotropic behaviour).	[kN/m]

The maximum axial tension force N_p is specified in units of force per unit width. If N_p is exceeded, stresses are redistributed according to the theory of plasticity, so that the maxima are complied with. This will result in irreversible deformations. Output of axial forces is given in the nodes, which requires extrapolation of the values at the stress points. Due to the position of the stress points in a geogrid element, it is possible that the nodal values of the axial force may slightly exceed N_p.

If the *Isotropic* option is checked the input is limited to $N_{p,1}$ where as $N_{p,2} = N_{p,1}$.

Elastoplastic (N-ε)

A non-linear *N-ε* diagram may be specified in case of elastoplasticity (*N-ε*):

N_1-ε_1 *diagram*	The *N-ε* diagram in 1-direction (in-plane).
N_2-ε_2 *diagram*	The *N-ε* diagram in 2-direction (out-of-plane, anisotropic behaviour).

The user can add both *N-ε* diagrams, by specifying values as a table or importing a text file containing values (Figure 178 (on page 274)). The axial tension force is specified in units of force per unit width. The first part of the *N-ε* diagram is used to calculate the *EA* value.

If the *Isotropic* option is checked the input is limited to N_1 *-eps*$_1$ where as N_2*-eps*$_2$ $= N_1$*-eps*$_1$.

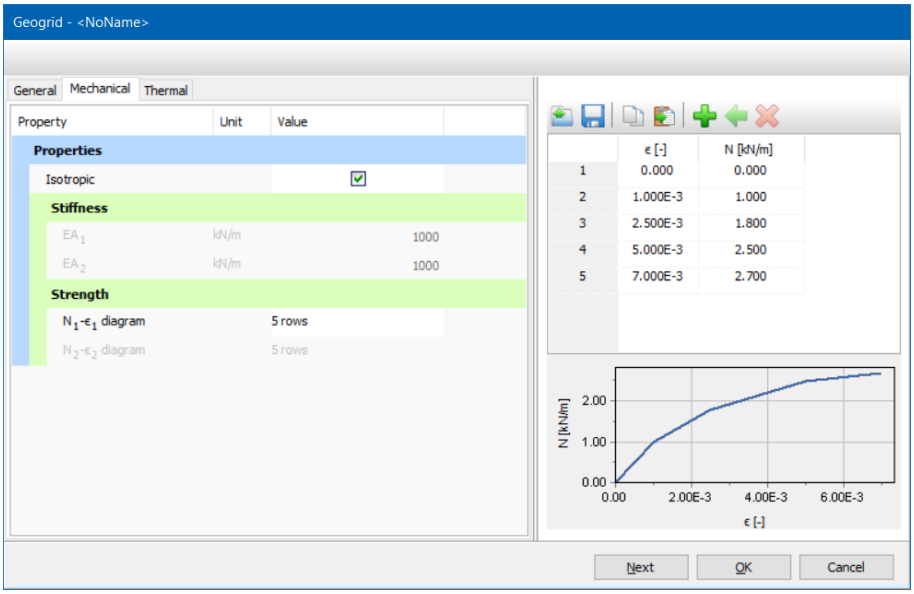


Figure 178: Geogrid window

6.5.3 *Thermal properties* [ADV]

For thermal behaviour, additional constants are specified in the *Thermal* tabsheet.

c	The specific heat capacity of the solid material. Describes the amount of energy (heat) that can be stored in the solid material per unit of mass.	[kJ/t/K]
λ	The thermal conductivity of the solid material. Describes the rate of energy (heat) that can be transported in the solid material.	[kW/m/K]
ρ	The density of the solid material. Describes the density of the solid particles.	[t/m^3]

α	The thermal expansion coefficient. Describes how much the material expands (or elongates) when the temperature increases. In other words, it is the (change of) strain per unit of temperature.	[1/K]
$A_{eff,T}$	The area in consideration for the thermal expansion.	[m^2]

Note: The creep properties of geogrids are very sensitive to the temperature. The visco-elastic perfectly-plastic model adopted for geogrids allows time-dependent behaviour but without temperature-dependency. Hence, the thermal expansion coefficient does not take the temperature effect on creep into account.

6.6 Material data sets for embedded beams

Properties and model parameters for embedded beam are entered in separate material data sets. Embedded beams can be used to model different types of slender structures that interact with the surrounding soil/rocks such as piles, ground anchors or rock bolts. A data set for embedded beams generally represents a certain type of embedded beam, including the pile/rock bolt material and geometric properties, the interaction properties with the surrounding soil or rock (bearing capacity) as well as the out-of-plane spacing of the piles/rock bolts.

Note that the embedded beam material data set does not contain so-called 'p-y curves', nor equivalent spring constants. In fact, the stiffness response of an embedded beam subjected to loading is the result of the specified embedded beam length, equivalent radius, spacing, stiffness, bearing capacity, the stiffness of the interface as well as the stiffness of the surrounding soil.

Note:

In contrast to what is common in the Finite Element Method, the bearing capacity of an embedded beam is considered to be an input parameter rather than the result of the finite element calculation. The user should realise the importance of this input parameter. Preferably, the input value of this parameter should be based on representative pile load test or pull out test data. Moreover, it is advised to perform a calibration in which the behaviour of the embedded beam is compared with the behaviour as measured from the test. When embedded beams are used in a row, the group action must be taken into account when defining their bearing capacity.

Note: Embedded beams can only be used in plane strain models and are not available in axisymmetry.

6.6.1 *General properties*

Several data sets may be created to distinguish between different types of embedded beams. Figure 179 (on page 276) shows the **Embedded beam**window.

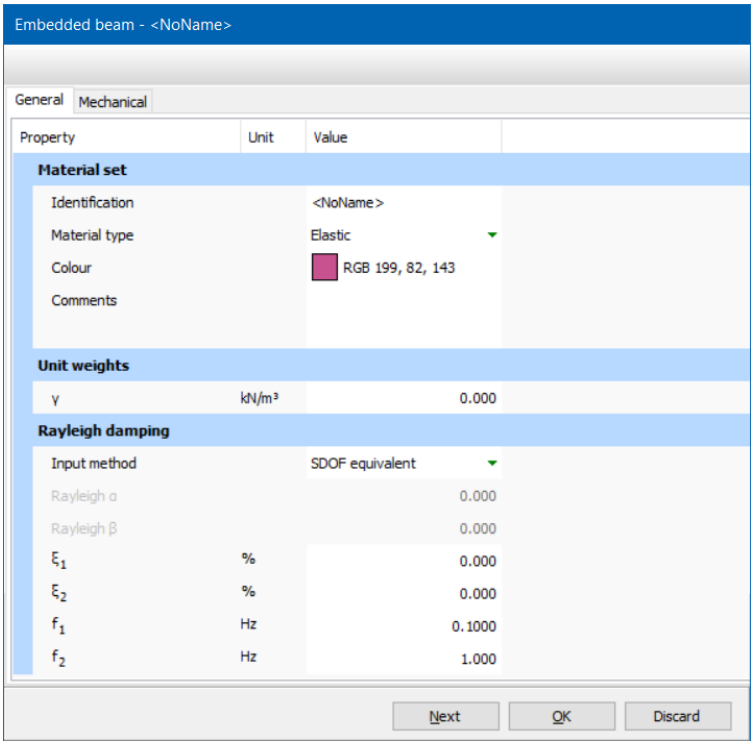


Figure 179: Embedded beam window

Material set

The material data set is defined by:

Identification	A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.
Material type	There are three available options, describing the material type for embedded beams. These options are *Elastic* , *Elastoplastic* and *Elastoplastic (M-κ)*. The availability of the parameters defined on the *Mechanical* tabsheet depends on the selected material type.
Colour	Colour can be used as a distinction tool in the model.
Comments	A user may write down comments related to the material data set.

Unit weight

Additionally, the *unit weight* of the embedded beam must be specified:

γ	The unit weight is the unit weight of the material from which the embedded beam is composed.	[kN/m^3]

Rayleigh damping [ULT]

For dynamic behaviour, the Rayleigh α and Rayleigh β damping parameters can be specified either directly or through Single Degree of Freedom (SDOF) equivalent parameters.

For more information on Rayleigh damping and how to specify the parameters, see Rayleigh damping [ULT] (on page 166).

6.6.2 *Mechanical properties*

The material properties are defined for a single beam, but the use of PLAXIS 2D implies that a row of piles or rock bolts in the out-of-plane direction is considered where properties are specified per unit width in out-of-plane direction.

Figure 180 (on page 278) shows the Mechanical tabsheet for an embedded beam in PLAXIS 2D.

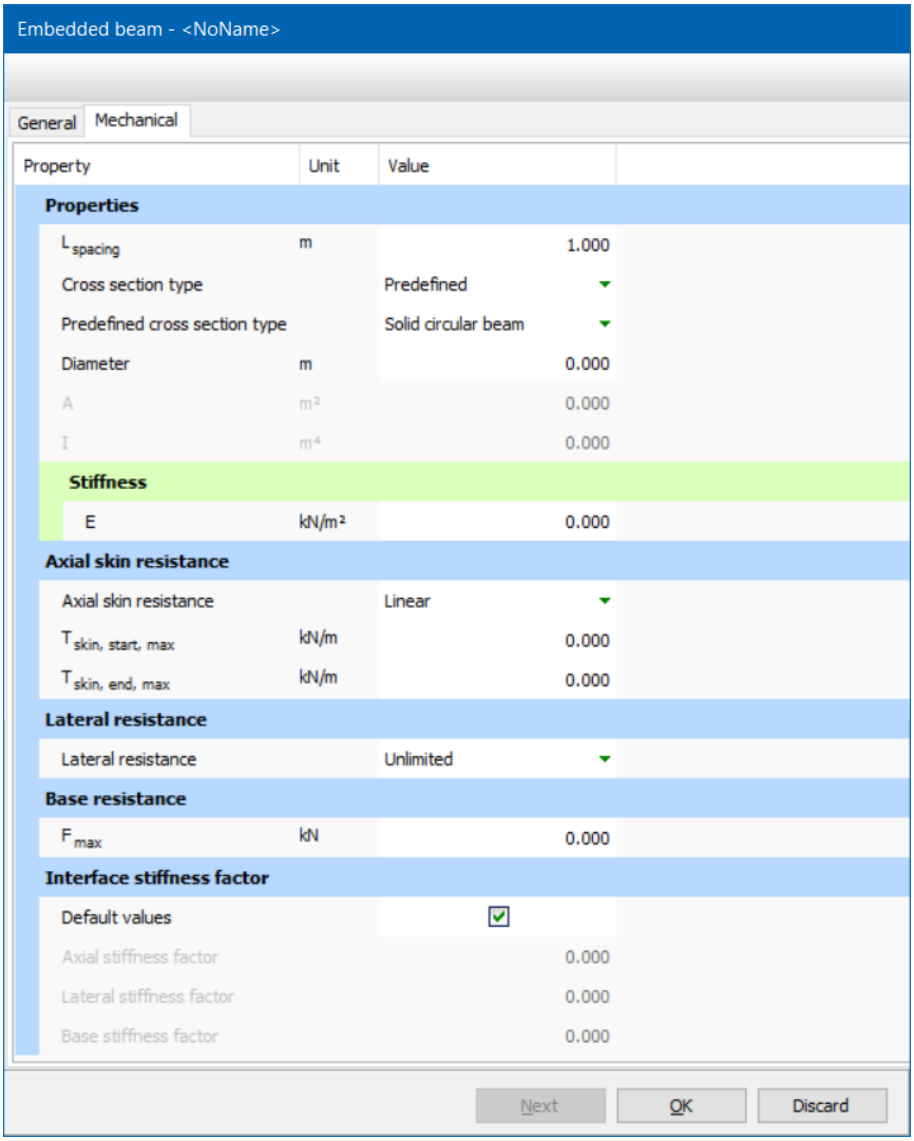


*Figure 180: Mechanical tabsheet of the **Embedded beam** window*

Note:

- In a finite element model, embedded beam rows are superimposed on a continuum and therefore ' overlap' the soil. Especially for Solid structures, to calculate accurately the total weight of soil and structures in the model, the unit weight of the soil should be subtracted from the unit weight of the embedded beam rows material.
- Please note that when reducing the unit weight, the axial forces in the embedded beam rows may not be realistic.

Cross section properties

An embedded beam requires several geometric parameters used to calculate additional properties. First of all, the out-of plane spacing $L_{spacing}$ in [m] must be defined in order to calculate all parameters per meter width out-of-plane.

Cross section type	Either a *Predefined* or a *user-defined* type can be selected.
Predefined cross section type	A list of predefined types (*Solid circular beam*, *Circular tube*, *Solid square beam*). After entering the required data, the parameters A, I_3, I_2, W_3 and W_2 are filled automatically).
Diameter	The embedded beam diameter is to be defined for *Solid circular beam* and *Circular tube* predefined embedded beam types. The embedded beam diameter determines the size of the elastic zone in the soil under the embedded beam in which plastic soil behaviour is excluded. It also influences the default values of the interface stiffness factors (Interface stiffness factor (on page 283)).
Width	The embedded beam width is to be defined for a *Solid square beam* predefined beam type. The embedded beam width is recalculated into an equivalent diameter, $D_{eq} = \sqrt{12EI / EA}$. This diameter determines the size of the elastic zone in the soil under the embedded beam in which plastic soil behaviour is excluded. It also influences the default values of the interface stiffness factors.
Thickness	The wall thickness needs to be defined for a *Circular tube* predefined beam type.

Alternatively, a user-defined type may be defined by means of the embedded beam cross section area, A, and its respective moment of inertia I:

A	The cross section area is the actual area (in the unit of length squared) perpendicular to the embedded beam axis where embedded beam material is present. For embedded beams that have a certain profile (such as steel beams), the cross section area can be found in tables that are provided by the manufacturer.	[m^2]
I	Moment of inertia against bending around the embedded beam axis.	[m^4]

$$l = \frac{1}{64}\pi D^4 \quad \text{Solid circular embedded beam}$$
$$l = \frac{1}{4}\left[\left(\frac{D}{2}\right)^4 - \left(\frac{D}{2} - t\right)^4\right] \quad \text{Circular tube} \qquad \textit{Eq. [58]}$$
$$l = \frac{1}{12}h^4 \quad \text{Solid square embedded beam}$$

where D is the embedded beam diameter, t is the wall thickness and h is the embedded beam width.}

Stiffness

An embedded beam has only a single stiffness parameter:

E	Young's modulus.	[kN/m^2]

Strength

Elastoplastic

For elastoplastic behaviour of an embedded beam, the strength is defined by the plastic limits for bending moment and axial force:

M_p	Maximum (plastic) bending moment	[kNm]
N_p	Maximum (plastic) axial force	[kN]

Elastoplastic (M-κ)

In the case the *Material type* is set to *Elastoplastic (M-κ)*, the bending behaviour of the plate is governed by a user-defined moment-curvature (*M-κ*) diagram. Input values of the *κ* diagram can be specified in the table that appears at the right hand panel when the *M-κ* diagram input field is selected.

For more information on elastoplastic behaviour see Strength properties (on page 267)

6.6.3 *Interaction properties: bearing capacity*

The interaction between the pile or rock bolt (embedded beam element) and the surrounding soil or rock (soil volume element) is modelled by means of a special interface element. An elastoplastic model is used to describe the behaviour of the interface. The elastic behaviour of the interface should account for the difference in pile/ rock bolt displacements and average soil/rock displacements in the out-of-plane direction. This depends on the out-of-plane spacing in relation to the diameter. Regarding the plastic behaviour, distinction is made in the material data set between the *Skin resistance* (in the unit of force per unit embedded beam length) and the *Base resistance* (in the unit of force). In a plane strain analysis, these values are automatically recalculated per unit of width in the out-of-plane direction.For the skin resistance, as well as the base resistance, a failure criterion is used to distinguish between elastic interface behaviour and plastic interface behaviour. For elastic behaviour relatively small displacement differences occur within the interface (i.e. between the pile/rock bolt and the average soil/rock displacement), and for plastic behaviour permanent slip may occur.

For the interface, to remain elastic, the shear force t_s at a particular point is given by:

$$| t_s | < T_{\max} \qquad Eq.\ [59]$$

where T_{max} is the equivalent local skin resistance at that point.

For plastic behaviour the shear force t_s is given by:

$$| t_s | = T_{\max} \qquad Eq.\ [60]$$

The input for the shaft resistance is defined by means of the axial skin resistance F_{skin} and the base resistance F_{max}. Using this approach the total pile bearing capacity, N_{pile}, is given by:

$$N_{pile} = F_{\max} + F_{skin} \qquad Eq.\ [61]$$

The skin resistance F_{skin} can be defined in three ways: *Linear*, *Multi-linear* and *Layer dependent*. The base resistance F_{max} can be entered directly (in the unit of force) in the embedded beam material data set window.

Note:

The base resistance is only mobilised when the pile body moves in the direction of the base (example: with a load on top).

For rock bolts, the bearing capacity may be defined in a similar way, although rock bolts generally do not have an end bearing ($F_{max} = 0$):

$$N_{\text{rockbolt}} = F_{skin} \qquad \textit{Eq. [62]}$$

In order to ensure that a realistic bearing capacity as specified can actually be reached, a zone in the soil volume elements surrounding the beam is identified where any kind of soil plasticity is excluded (elastic zone). The size of this zone is determined by the embedded beam's diameter D or equivalent diameter D_{eq}. The elastic zone makes the embedded beam almost behave like a volume element. Installation effects are not taken into account and the embedded beam-soil interaction is modelled at the centre rather than at the circumference.

The bearing capacity is automatically divided by the spacing in order to obtain the equivalent bearing capacity per unit of width in the out-of-plane direction.

In addition to displacement differences and shear forces in axial direction along the embedded beam row element, the beam can undergo transverse forces, $t_{\perp}$, due to lateral displacements. The lateral displacements can be induced by a transverse force applied at the top of the pile or as a consequence of the transverse distributed load induced by the lateral displacement field of the surrounding soil. In the first case, the overall behaviour may not show fully realistic results. In the second case, although embedded beam row elements are not meant to be used as laterally loaded piles, they show reasonable results and the overall behaviour is quite realistic. For more information about the modelling techniques for embedded structures, see the Appendix of the Material Models Manual.

The user can limit the transverse forces using the *Lateral resistance* option. This can be done in three ways: *Unlimited*, *Linear* and *Multi-linear* lateral resistance.

More details about the way the shear and transverse forces are calculated on the basis of displacement differences between the embedded beam element and the surrounding soil element are described in the Material Models Manual.

Axial skin resistance

Linear

The *Linear* option is the easiest way to enter the skin resistance and lateral resistance profile. The input is defined by means of the skin resistance and lateral resistance at the beginning of the embedded beam row, $T_{skin,start,max}$ and $T_{lat,start,max}$ (in force per unit beam length), and the skin resistance and lateral resistance at the end of the embedded beam row, $T_{skin,end,max}$ and $T_{lat,end,max}$ (in force per unit beam length). This way of defining the skin resistance and lateral resistance is mostly applicable to piles in a homogeneous soil layer. Using this approach the total pile bearing capacity, N_{pile}, is given by:

$$N_{pile} = F_{\max} + \tfrac{1}{2} L_{pile} \left(T_{skin,start,\max} + T_{skin,end,\max} \right) \qquad \textit{Eq. [63]}$$

where L_{pile} is the pile length. $T_{skin,start,max}$ and $T_{skin,end,max}$ are measured at the pile top and the bottom of the pile respectively.

For rock bolts, the bearing capacity may be defined as follows:

$$N_{\text{rockbolt}} = \tfrac{1}{2} L_{\text{rockbolt}} \left(T_{skin,start,\max} + T_{skin,end,\max} \right) \qquad \textit{Eq. [64]}$$

where $L_{rockbolt}$ is the rock bolt length. $T_{skin,start,max}$ and $T_{skin,end,max}$ are measured at the first point of the geometry line and the second one respectively.

Multi-linear

The *Multi-linear* option can be used to take into account inhomogeneous or multiple soil layers with different properties and, as a result, different resistances. The skin resistance, T_{max}, is defined in a table at different positions along the pile, L, where L is measured from the pile top (L=0) to the bottom of the pile (L= L_{pile}). Using this approach the total pile bearing capacity, N_{pile}, is given by:

$$N_{pile} = F_{\max} + \sum_{i=1}^{n-1} \frac{1}{2}(L_{i+1} - L_i)(T_i + T_{i+1}) \qquad Eq.\ [65]$$

where i is the row number in the table and n is the total number of rows.

For rock bolts, the bearing capacity is defined as follows:

$$N_{\text{rockbolt}} = \sum_{i=1}^{n-1} \frac{1}{2}(L_{i+1} - L_i)(T_i + T_{i+1}) \qquad Eq.\ [66]$$

where L is measured from the first point of the corresponding geometry line (L=0) to the second one (L= $L_{rockbolt}$).

Layer dependent

The *Layer dependent* option can be used to relate the local skin resistance to the strength properties (cohesion c and friction angle φ) and the interface strength reduction factor, R_{inter}, as defined in the material data set of the corresponding soil or rock layers (Interfaces tabsheet (on page 242)) in which the pile or rock bolt is located:

$$\begin{aligned} \tau_i &= c_i + \sigma'_n \tan\varphi_i \\ c_i &= R_{\text{inter}} c_{soil} \\ \tan\varphi_i &= R_{\text{inter}} \tan\varphi_{soil} \end{aligned} \qquad Eq.\ [67]$$

where τ_i is the local shear stress resistance of the interface, φ_i and c_i are the friction angle and the cohesion of the interface, φ_{soil} and c_{soil} are the friction angle and cohesion of the correspondent soil layer, R_{inter} is the strength reduction factor associated to the soil layer, $\sigma'_n = (\sigma'_2 + \sigma'_3) / 2$ is the normal stress. Using this approach the pile/rock bolt bearing capacity is based on the stress state in the soil, and thus unknown at the start of a calculation. The special interface in the embedded beam row behaves similar as an interface along a wall, except that it is a line interface rather than a sheet.

Note:

In case of advanced soil material models, the calculation of φ_i and c_i is as described in Interfaces tabsheet (on page 242). For user-defined Soil Models, φ_i and c_i are based on direct input specified by the user.

The skin resistance, T_i, as a force per unit of depth, is defined as:

$$T_i = 2\pi R_{eq} \tau_i \qquad Eq.\ [68]$$

where R_{eq} is the embedded beam's equivalent radius.

To avoid that the skin resistance could increase to undesired high values, an overall maximum resistance T_{max} (constant value along the pile/rock bolt in force per unit pile/rock bolt length) can be specified in the embedded beam row material data set, which acts as an overall cut-off value.

Note:

The embedded beam-soil interaction parameters in the embedded beam material data set involve only the bearing capacity (skin resistance and base resistance). Note that the material data set does NOT include the stiffness response of the pile in the soil (or *p-y* curve). The stiffness response is the result of the pile length, equivalent radius, stiffness and bearing capacity as well as the stiffness of the soil layers in which the beam is located.

Lateral resistance

Unlimited

The value of the lateral resistance in this case is unlimited.

Linear

Similar to the linear axial skin resistance the start and end value, $T_{lat,start,max}$ and $T_{lat,end,max}$ must defined.

Multi-linear

Again similar to the multi-linear axial skin resistance a table of lateral skin resistance as a functionof the distance along the beam can be specified.

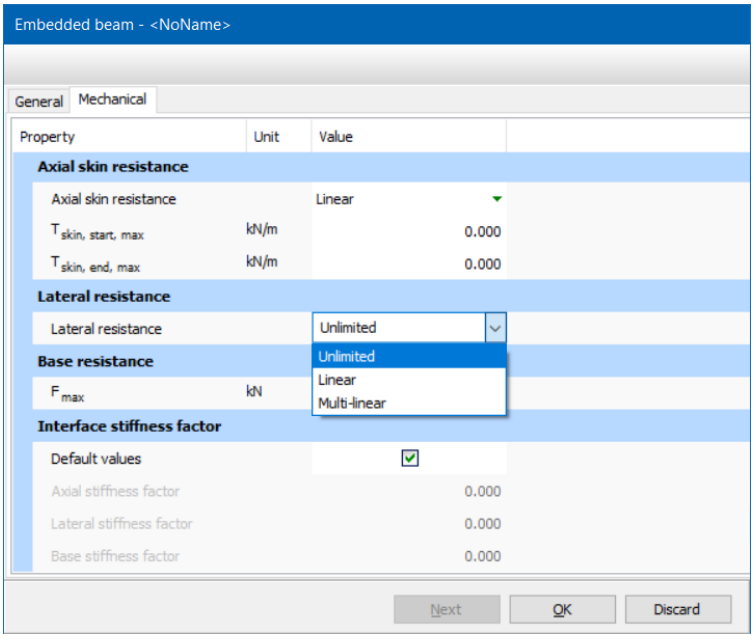


Figure 181: Available options for Lateral resistance

Base resistance

The base resistance F_{max} is the maximum foot force of the embedded beam, hence the end-bearing capacity. Note that the base of the embedded beam can only sustain a compressive force and no tensile force.

6.6.4 *Interface stiffness factor*

The interface stiffnesses are related to the shear stiffness of the surrounding soil (G_{soil}) according to:

$$R_s = ISF_{RS}\frac{G_{soil}}{L_{spacing}}$$

$$R_N = ISF_{RN}\frac{G_{soil}}{L_{spacing}} \qquad \textit{Eq. [69]}$$

$$K_F = ISF_{KF}\frac{G_{soil}R_{eq}}{L_{spacing}}$$

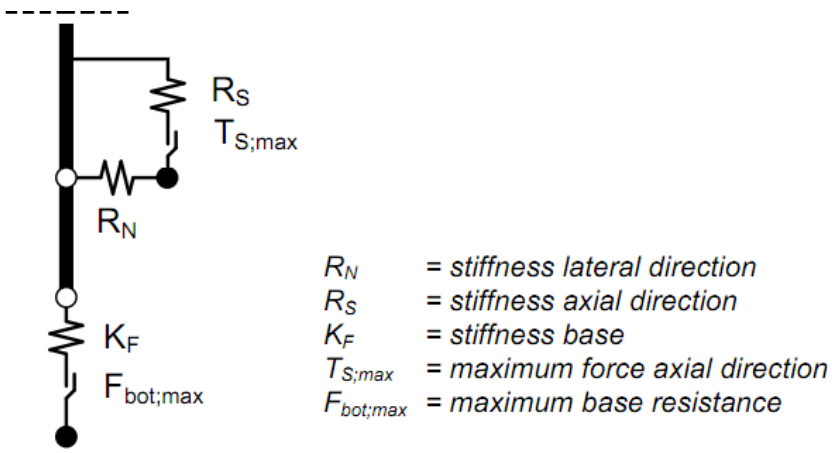


Figure 182: Modelling of soil-pile interaction

The interface stiffness factors to be defined are:

- Axial skin stiffness factor, ISF_{RS}
- Lateral stiffness factor, ISF_{RN}
- Pile base stiffness factor, ISF_{KF}

where the default values of the interface stiffness are calculated according to:

$$ISF_{RS} = 25\left(\frac{L_{spacing}}{D}\right)^{-0.75}$$

$$ISF_{RN} = 25\left(\frac{L_{spacing}}{D}\right)^{-0.75} \qquad \textit{Eq. [70]}$$

$$ISF_{KF} = 25\left(\frac{L_{spacing}}{D}\right)^{-0.75}$$

where D is *diameter* in the case of a circular pile or *width* in the case of a square pile or *equivalent width* in the case the case of a user-defined pile.

In the case of an embedded beam as a pile a realistic pile bearing capacity as specified can actually be reached, a zone in the soil volume elements surrounding the bottom of the pile is identified where any kind of soil plasticity is excluded (elastic zone). The size of this zone is determined by the embedded beam's diameter D_{eq} or equivalent radius R_{eq} (= $D_{eq}/2$) (Figure 183 (on page 285))

where

$$D_{eq} = \sqrt{\frac{12EI}{EA}} \qquad Eq.\ [71]$$

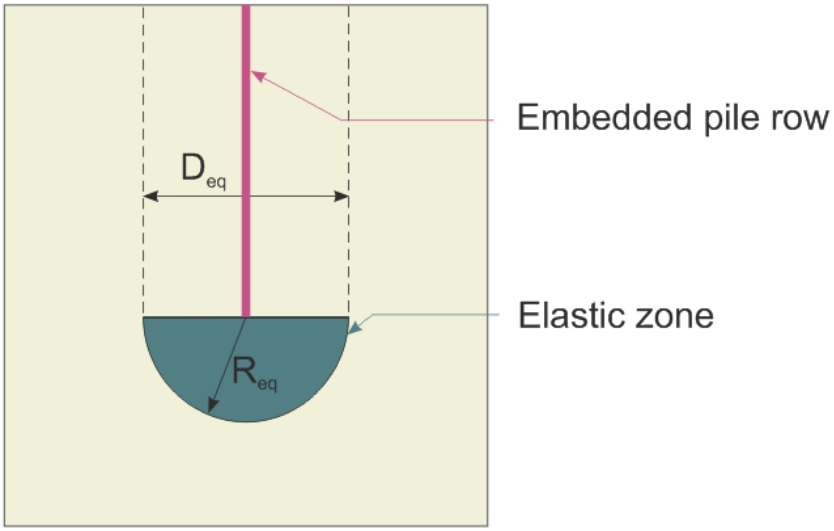


Figure 183: Plastic zone surrounding the bottom of the pile (after Sluis, 2012)

In addition to displacement differences and shear forces in axial direction along the pile, the pile can undergo transverse forces, $t_{\perp}$, due to lateral displacements. These transverse forces are not limited in the special interface element that connects the pile with the soil, but, in general, they are limited due to failure conditions in the surrounding soil itself. However, embedded beams are not meant to be used as laterally loaded piles and will therefore not show accurate failure loads when subjected to transverse forces.

Note that the default values of the interface stiffness factors are valid for bored piles which are loaded statically in the axial direction and behaviour of the surrounding soil is modelled using the Hardening Soil model with small-strain stiffness. The phreatic level is assumed to be located at the ground surface. These values should be modified if the conditions in the model are different from the ones assumed to derive the default values.

6.7 Material data sets for anchors

In addition to material data sets for soil and interfaces, the material properties and model parameters for anchors are also entered in separate material data sets. A material data set for anchors may contain the properties of node-to-node anchors as well as fixed-end anchors. In both cases the anchor is just a spring element. A data set for anchors generally represents a certain type of anchor material, and can be assigned to the corresponding (group of) anchor elements in the geometry model.

Anchors can be prestressed in a *Staged construction* calculation. In such a calculation the prestress force for a certain calculation phase can directly be given in the *Object explorer*. The prestress force is not considered to be a material property and is therefore not included in an anchor data set.

6.7.1 *General properties*

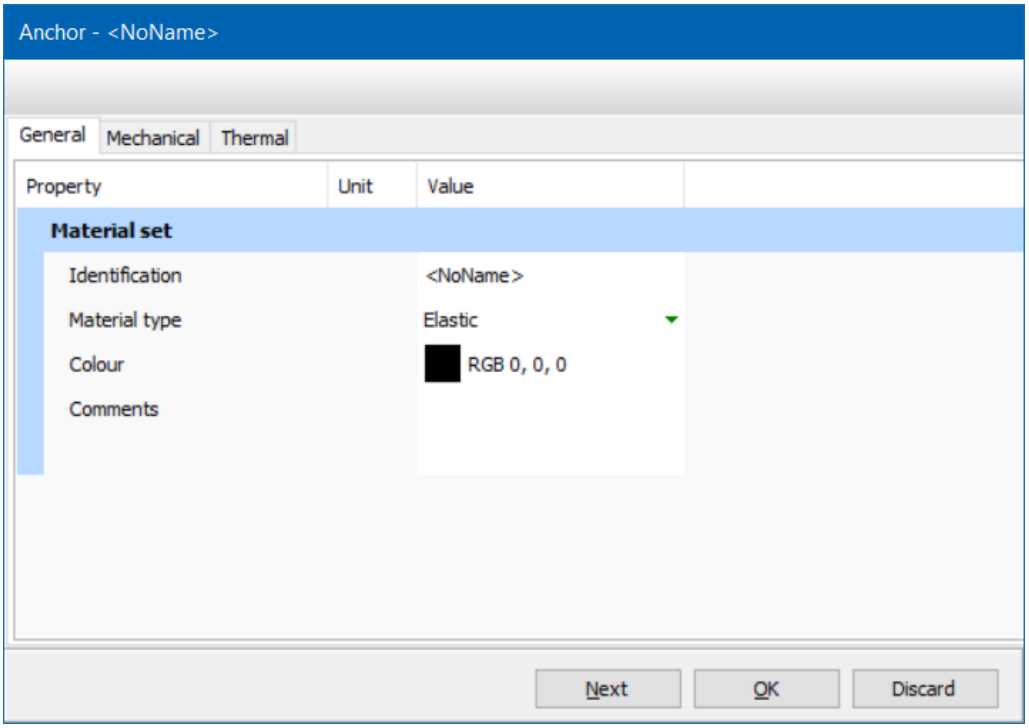


*Figure 184: **Anchor** window*

Several data sets may be created to distinguish between different types of anchors. Figure 184 (on page 286) shows the *General* tabsheet of the **Anchor** window. The material data set is defined by:

Identification	A user may specify any identification title for a data set. It is advisable to use a meaningful name since the data set will appear in the database tree view by its identification.
Material type	There are three available options, describing the material type of an anchor. These options are *Elastic*, *Elastoplastic* and *Elastoplastic with residual strength*. The availability of the parameters defined on the *Mechanical* tabsheet depends on the selected material type.
Colour	Colour can be used as a distinction tool in the model.
Comments	A user may write down comments related to the material data set.

6.7.2 *Mechanical properties*

The properties required for anchors can be grouped into stiffness properties and strength properties in case of elastoplastic behaviour.

Stiffness

To calculate an equivalent stiffness per unit width, the out-of-plane spacing, L_s, must be entered.

Note: For axisymmetric problems L_s is considered in radians and not in unit width.

An anchor requires only one stiffness parameter:

EA	Axial stiffness of a single anchor.	[kN]

Strength

Elastoplastic

If the material type is selected as *Elastoplastic*, two maximum anchor forces can be entered:

$F_{max,tens}$	Maximum tension force of a single anchor	[kN]
$F_{max,comp}$	Maximum compression force of a single anchor	[kN]

The Force-displacement diagram displaying the elastoplastic behaviour of the anchors is given in Figure 185 (on page 287).

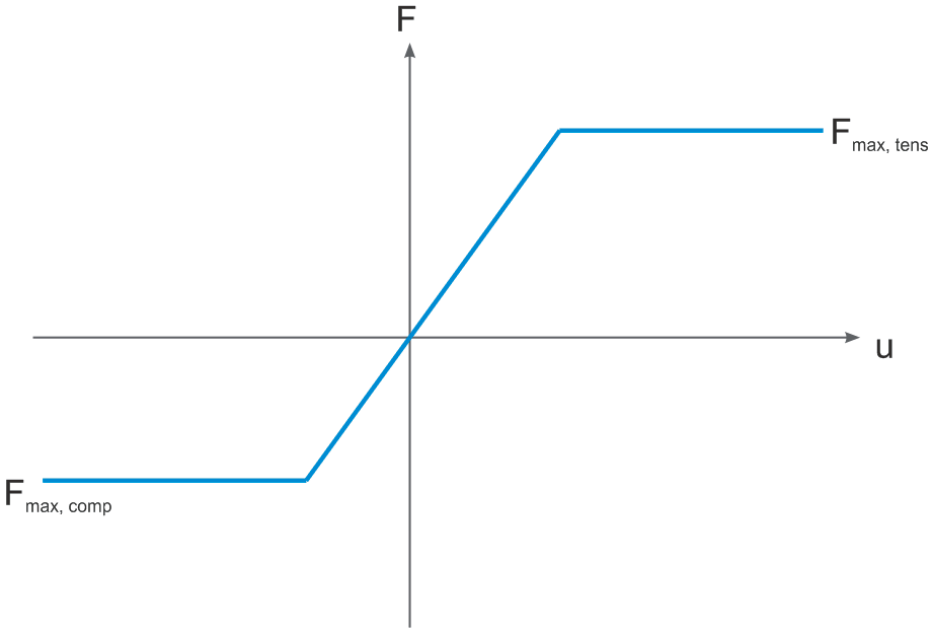


Figure 185: The force-displacement diagram displaying the elastoplastic behaviour of anchors

In the same way as the stiffness, the maximum anchor forces are divided by the out-of-plane spacing in order to obtain the proper maximum force in a plane strain analysis.

Elastoplastic with residual strength

The *Elastoplastic with residual strength* option can be used to model anchor failure or softening behaviour (e.g. buckling of struts). When this option is selected two residual anchor forces can be specified:

$F_{max,tens}$	Maximum tension force of a single anchor	[kN]
$F_{max,comp}$	Maximum compression force of a single anchor	[kN]
$F_{residual,tens}$	Residual tension force of a single anchor	[kN]
$F_{residual,comp}$	Residual compression force of a single anchor	[kN]

The Force-displacement diagram displaying the elastoplastic behaviour with residual strength of the anchors is given in Figure 185 (on page 287).

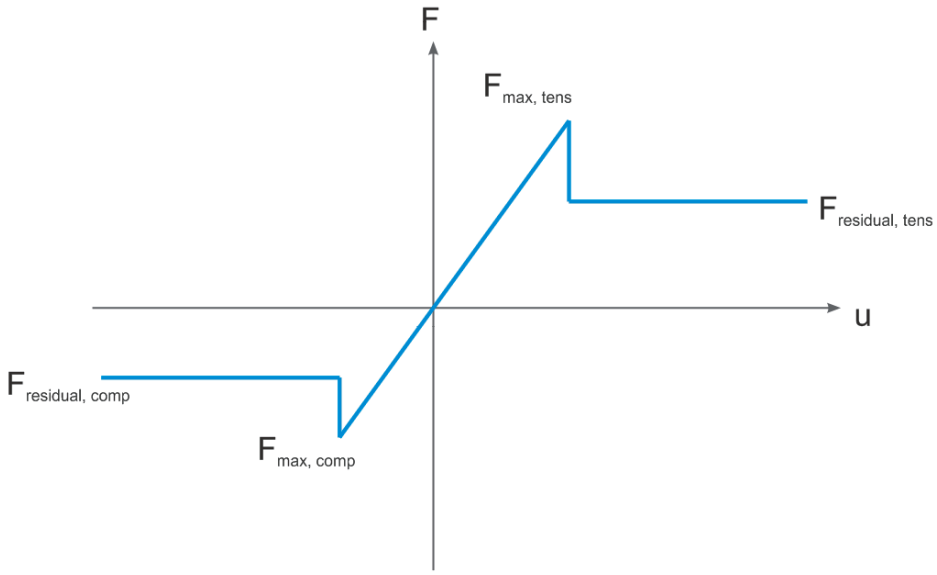


Figure 186: The force-displacement diagram displaying the elastoplastic behaviour with residual strength of the anchor

If, during a calculation, the maximum anchor force is reached, the maximum force will immediately reduce to the residual force. From that point on the anchor force will not exceed the residual force anymore. Even if the anchor force would intermediately reduce to lower values, the defined residual force will be its maximum limit.

Note: If the anchor has failed (in tension, compression or both) the residual force will be valid in the following calculation phases where the anchor is active. If the anchor is deactivated in a phase and reactivated in the next phase, the maximum anchor force will be restored, assuming that the anchor has been replaced by a new anchor.

Note: Using reduced residual strength is not recommended in *Safety* calculations.

6.7.3 *Thermal properties* [ADV]

For thermal behaviour, additional constants are specified in the *Thermal* tabsheet.

c	The specific heat capacity of the solid material. Describes the amount of energy (heat) that can be stored in the solid material per unit of mass.	[kJ/t/K]

λ	The thermal conductivity of the solid material. Describes the rate of energy (heat) that can be transported in the solid material.	[kW/m/K]
ρ	The density of the solid material.	[t/m^3]
α	The thermal expansion coefficient. Describes how much the material expands (or elongates) when the temperature increases. In other words, it is the (change of) strain per unit of temperature.	[1/K]
$A_{eff,T}$	The cross sectional area of the anchor considered for thermal flux.	[m^2]

6.8 *Assigning material data sets to geometry components*

After creating material data sets, the data sets must be assigned to the corresponding geometry components (soil layers and structures). This can be done in different ways, which are explained below. The methods described below are primarily meant to assign properties to the initial geometry. For details on the change of properties during calculations in the framework of *staged construction* (Reassigning material data sets (on page 361)).

6.8.1 Soil layers

Regarding soil data, material data sets can be assigned to individual soil layers in the boreholes. Therefore a borehole should be double-clicked to open the corresponding **Modify soil layers** window. In the **Modify soil layers** click the plus button next to the soil layer. The **Material dataset** window pops up, where the properties of the new material can be defined.

Materials button at the lower right hand side of the window should be clicked to open the material database. To assign a data set to a particular soil layer, select the desired data set from the material database tree view (click on the data set and hold the left hand mouse button down), drag it to the soil column in the borehole window (hold the mouse button down while moving) and drop it on the desired layer (release the mouse button). The layer should now show the corresponding material data set colour. The drag and drop procedure should be repeated until all layers have their appropriate data set. Note that material sets cannot be dragged directly from the global database tree view and must be copied to the project database first.

When multiple boreholes are used it should be noted that assigning a data set to a layer in one particular borehole will also influence the other boreholes, since all layers appear in all boreholes, except for layers with a zero thickness.

6.8.2 Structures

Regarding structures (fixed-end anchors, node-to-node anchors, embedded beam rows, plates, geogrids and interfaces), there are three different methods of assigning material data sets.

- The first method is based on an open **Material sets** window, showing the created material sets in the project database tree view. The desired material set can be dragged (select it and keep the left mouse button down)

to the drawing area and dropped on the desired component. It can be seen from the shape of the cursor whether or not it is valid to drop the material set.

- The second method is to select the desired structure in the drawing area or the *Model explorer* and select the desired material set from the *Set material* of the right hand mouse button menu.
- The third method is to select the desired structure in the drawing area or model explorer and use the *Material* drop-down menu in the explorer to define the material data set.

7 Meshing and calculation

When the geometry modelling process is complete, one can proceed with the calculations. This consists of generation of mesh and definition of the construction stages.

Generating a proper Finite Element mesh is an important intermediate step between the definitions of the geometry and the construction stages. In order to have a smooth and accurate calculation the finite element mesh has to fulfill several criteria. For the numerical stability of the calculation, the mesh should have a good quality, that is to say, the elements should be regular without being excessively long and thin. For the accuracy of the calculation, the elements should be small enough, especially in those areas where significant changes in stress or strain can be expected during the analysis. But that does not mean one should just generate an entire mesh of very small elements as this will lead to a very large calculation time. Therefore proper care should be taken to find the right balance between accuracy and calculation time while supervising on the mesh quality.

In the engineering practice, a project is divided into project phases. Similarly, a calculation process in PLAXIS 2D is also divided into calculation phases. Examples of calculation phases are the activation of a particular loading at a certain time, the simulation of a construction stage, the introduction of a consolidation period, the calculation of a safety factor, etc. Each calculation phase is generally divided into a number of calculation steps. This is necessary because the non-linear behaviour of the soil requires loadings to be applied in small proportions (called load steps). In most cases, however, it is sufficient to specify the situation that has to be reached at the end of a calculation phase. Robust and automatic procedures in PLAXIS 2D will take care of the sub-division into appropriate load steps.

The construction stages can be defined in the **Flow conditions** and **Staged construction** modes. The first calculation phase (Initial phase) is always a calculation of the initial stress field for the initial geometry configuration by means of *K0 procedure*, *Field stress* or *Gravity loading* . Alternatively, it may be indicated that the calculations only involve groundwater flow. After this initial phase, subsequent calculation phases may be defined by the user. In each phase, the type of calculation must be selected.

For deformation calculations distinction is made between *Plastic*, *Consolidation*, *Fully coupled flow-deformation*, *Dynamic*, *Dynamic with consolidation* or *Safety* calculation. The different types of calculations are explained in Types of analysis (on page 299).

7.1 Mesh generation - Mesh mode

When the geometry model is fully defined the geometry has to be divided into finite elements in order to perform finite element calculations. A composition of finite elements is called a mesh. The mesh is created in the **Mesh** mode.

The mesh should be sufficiently fine to obtain accurate numerical results. On the other hand, very fine meshes should be avoided since this will lead to excessive calculation times. The PLAXIS 2D program uses fully automatic generation of finite element meshes. The generation of the mesh is based on a robust triangulation

procedure. The mesh generation process takes into account the soil stratigraphy as well as all structural objects, loads and boundary conditions.

7.1.1 *Elements*

The basic soil elements to model soil layers and other volume clusters as described in Project properties (on page 21) are either *15-Node* or *6-Node* triangular elements (Figure 9 (on page 23)). The user chooses the type of element.

In addition to the soil elements, there are special elements for structural behaviour (plates, embedded beam rows, geogrids and anchors), as described in Structural elements (on page 97). The type of element for structures and interfaces is automatically taken to be compatible with the basic type of soil element.

To generate the mesh, click the *Generate mesh* button in the side toolbar of the **Mesh** mode or select the corresponding option in the *Mesh* menu.

The **Mesh options** window pops up where the general mesh properties can be defined (Figure 187 (on page 292)). The options are explained in Global settings (on page 292), Local refinement (on page 293) and Enhanced mesh refinements (EMR) (on page 294).

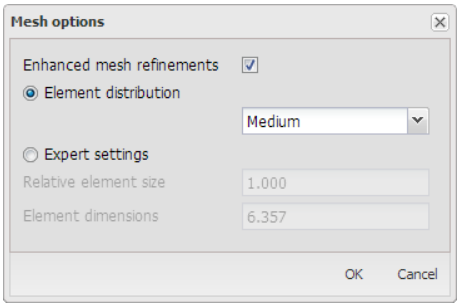


*Figure 187: **Mesh options** window*

The mesh is generated by clicking on the *OK* button in the **Mesh options** window. After the mesh was generated, the number of generated elements is displayed in the *Session* tab of the **Command line**.

To view the generated mesh click the *View mesh* button in the side toolbar.

Note:

Note that if meshing failed, the *Session* tab of the **Command line** might display the cause of the meshing failure, where in the geometry this occurs and what needs to be done to overcome this. Even when the meshing succeeds, the session history can still show some warnings on poor geometry and how this can be improved.

7.1.2 *Global settings*

The mesh generator requires a global meshing parameter l_e, which represents the target element dimension. In PLAXIS 2D this parameter is calculated from the outer geometry dimensions (x_{min}, x_{max}, y_{min}, y_{max}) and the

Element distribution selected in the **Mesh options** window. The target element dimension is calculated using the Eq. [72]Eq. [78].

$$l_e = r_e \times 0.06 \times \sqrt{(x_{\max} - x_{\min})^2 + (y_{\max} - y_{\min})^2} \qquad Eq.\ [72]$$

where the *Relative element size factor (r_e)* is derived from the *Element distribution*. There are five global levels. By default, the *Element distribution* is set to *Medium* but the user may select one of the other levels to make the mesh globally finer or coarser.

The predefined values of the parameter r_e (*Element distribution*) are:

Element distribution	r_e
Very coarse	2.00 (30 - 70 elements)
Coarse	1.33 (50 - 200 elements)
Medium	1.00 (90 - 350 elements)
Fine	0.67 (250 - 700 elements)
Very fine	0.50 (500 - 1250 elements)

Note:

Note that the *Relative element size* (r_e) does not correspond in value to the parameter value which is displayed in the command line.The value displayed is $r_e \times 0.06$ (Eq. [78]) .

The exact number of elements depends on the shape of the geometry and optional local refinement settings. The number of elements is not influenced by the *Type of elements* parameter, as set in the **Project properties** window. Note that a mesh composed of 15-node elements gives a much finer distribution of nodes and thus much more accurate results than a similar mesh composed of an equal number of 6-node elements. On the other hand, the use of 15-node elements is more time consuming than using 6-node elements.

As an alternative to the *Element distribution*, the mesh can be defined using *Expert settings*.

Relative element size	The relative element size factor (r_e) as defined above. The default value is *1.0* (medium).
Element dimension	The target element dimension (l_e) as defined above.
Max cores to use	The number of cores to be used in the meshing process.

The user specifies either the *Relative element size* or the *Element dimension* and the other one is then generated automatically as described above.

7.1.3 *Local refinement*

In areas where large stress concentrations or large deformation gradients are expected, it is desirable to have a more accurate (finer) finite element mesh, whereas other parts of the geometry might not require a fine mesh. Such a situation often occurs when the geometry model includes edges or corners or structural objects.

Local refinement is based on a local coarseness factor that can be specified for each geometry entity. This factor gives an indication of the relative element size with respect to the target element size as determined by the *Element distribution* parameter. By default, the *Coarseness factor* value is set to *1.0* for most geometry entities whereas this value is *0.25* for structural objects and loads. A *Coarseness factor* value of *0.5* reduces the element size to half the target element size. The coarseness factor can be changed by selecting the geometry entity and clicking on the *Coarseness factor* in the *Selection explorer*. Values in the range from *0.03125* to *8.0* are acceptable. Using a value larger than *1.0* coarsens the mesh locally.

To locally refine the mesh either click the *Refine mesh* button and select the geometry entity (line or point) where a finer mesh is required or right-click the geometry entity and select the *Finer mesh* option from the appearing menu.

To locally coarsen the mesh by a factor of $\sqrt{2}$ click the *Coarsen mesh* button and select the geometry entity (line or point) where a coarser mesh is required or right click the geometry entity and select the *Finer mesh* option from the appearing menu.

To reset the local refinement in a geometry entity right click it either in the drawing area or in *Selection explorer* and select the corresponding option from the appearing menu.

Note:

- In the **Mesh** mode, the colours indicating the material data sets are turned off and the whole model is shown in gray. However, refined and coarsened objects are displayed in green and yellow respectively.
- The more the object is refined, the lighter the shade of green in which it is displayed.
- The more the object is coarsened, the lighter the shade of yellow in which it is displayed.

7.1.4 *Enhanced mesh refinements (EMR)*

To supply a good quality mesh for every geometry which also takes into account the necessary mesh refinement around structural elements, loads and prescribed displacements, PLAXIS 2D will apply automatic mesh refinements.

These automatic refinements in terms of an implicit local element size multiplication factor will be applied in case of:

- the distance between points and/or lines is rather close, which requires a smaller element size to avoid large aspect ratios.
- an angle between geometry lines other than a multiple of *90°*, to allow for more accurate stresses around geometry discontinuities. The *Local element size* factor will be multiplied according to Figure 188 (on page 295).

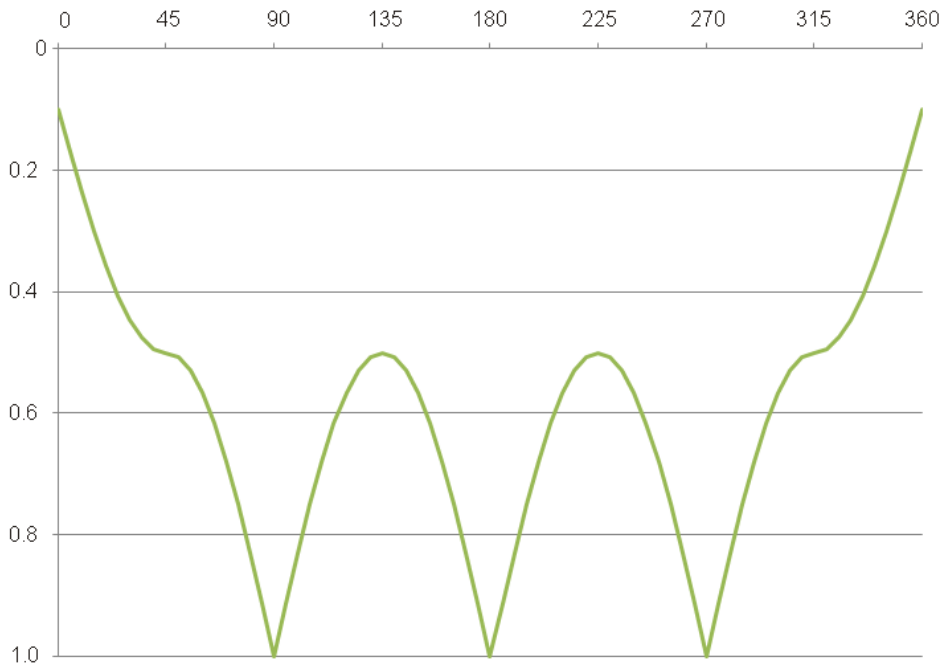


Figure 188: The implicit Coarseness factor as a function of the angle between two lines.

Note:

The automatic refinement considered can be turned off by unchecking the *Enhanced mesh refinements* option in the **Mesh options** window (Figure 187 (on page 292)).

7.2 Defining calculation phases

Finite element calculations can be divided into several sequential calculation phases. Each calculation phase corresponds to a particular loading or construction stage. The construction stages can be defined in the **Staged construction** mode. The calculation phases are listed in the *Phases explorer*.

7.2.1 *Phases explorer*

The phases defined in a project are displayed in the *Phases explorer*. The *Phases explorer* is accessible in the **Calculation** modes. However, it is not editable in the **Mesh** mode. The general layout is shown in Figure 189 (on page 296).

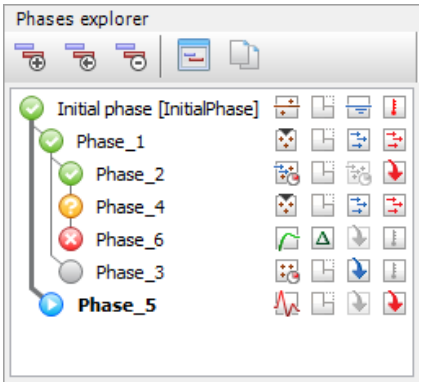


Figure 189: Phases explorer

Note:

Note that for a new project the initial phase is automatically added. It cannot be deleted. More information on the initial phase is given in Initial stress generation (on page 300).

Toolbar

The buttons in the toolbar enable introducing new phases, removing phases and accessing the **Phases** window where the settings of phases are defined.

To introduce a new calculation phase (a child phase), select a reference phase (parent phase) in the list and click the *Add phase* button. A new phase is added under the parent phase.

To insert a new calculation phase before a selected one, click the *Insert phase* button. The inserted phase will be the new parent phase for the phase selected before the insertion, whereas the original parent phase will become the parent phase of the inserted phase.

The user has to define the new settings for the inserted phase in a similar way as defining a new phase at the end of the phase list. The phases have the settings of the parent phase unless they are defined differently. In this case it is required that the next phase is fully redefined, since the start conditions have changed. This may also have consequences for the phases thereafter.

To delete a calculation phase, select it and click the *Delete phase* button. Before deleting a phase it should be checked which of the subsequent phases refer to the phase to be deleted. The parent of the deleted phase will be automatically the new parent phase. Nevertheless, it is required that the modified phases are redefined, since the start conditions have changed.

To define the phase settings either select the phase and click the *Edit phase* button or double click the phase. The **Phases** window pops up (Phases window (on page 298)).

The *Copy* button enables copying to clipboard general information about the phase.

Besides the buttons available in the *Phases explorer*, the right mouse button menu can be used to modify the phases in the model (Figure 190 (on page 297)).

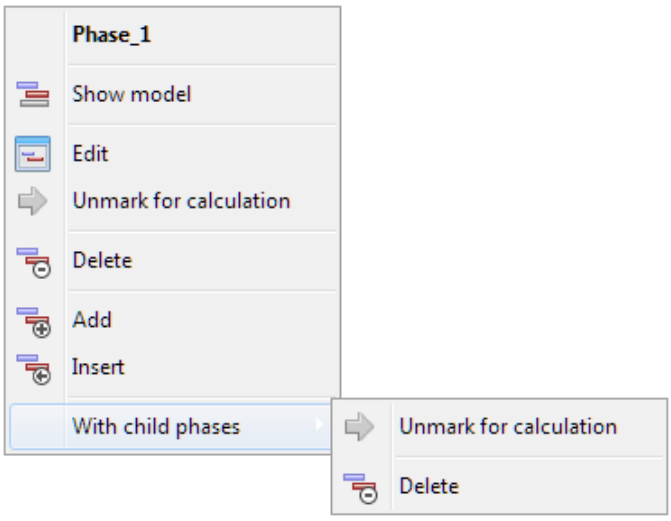


Figure 190: The expanded view of the right mouse menu of Phases explorer

Calculation state indication

The calculation state of a phase is indicated by the symbol at the beginning of each line in the *Phases explorer*.

- The phase is to be calculated.
- The phase is not to be calculated.
- The phase was calculated. No error occurred during calculation.
- The phase was calculated but an assumption is made by the program enabling the continuation of the calculation. Information is provided in the *Log info for last calculation step* box in the **Phases** window.
- The calculation failed. Information is provided in the *Log info for last calculation step* box in the **Phases** window.
- The calculation failed, but the calculation of the child phases is still possible. Information is provided in the *Log info for last calculation* box in the **Phases** window.

Phase identification

The ID of the phases is displayed in the *Phases explorer*. The *ID* of the phase consist of the caption and the name (part inside the square brackets). The name of the phases is determined consecutively by the program and it can not be modified by the user. The user may redefine the caption part of the ID of the phase in the **Phases** window (General phase parameters (on page 331)).

Calculation type indication

The *Calculation type*, the *Loading type* and the *Pore pressure calculation type* are indicated by the corresponding icons next to the phase ID in the *Phases explorer*. More information is given in Types of analysis (on page 299), Loading type (on page 316) and Water pressure calculation (on page 321) respectively.

7.2.2 *Order of calculation phases*

The order of calculation phases is defined either by selecting the reference phase (parent phase) first and then adding a phase, or selecting the reference phase in the *Start from phase* drop-down menu in the **Phases** window. By default, the previous phase is selected as parent phase. Note that a phase that appears later in the phase list cannot be selected.

Special cases

In some special cases, the order of calculation phases is not straightforward. Examples of such cases are:

- The *Initial phase* may be selected more than once as reference if different loadings or loading sequences are to be considered separately for the same project.
- For a certain situation, a load is increased until failure to determine the safety margin. When continuing the construction process, the next phase should start from the previous construction stage rather than from the failure situation.
- A third example where the phase ordering is not straightforward is in calculations where safety analysis for intermediate construction stages is considered. The calculation type in this case is *Safety*. In general, such a phase results in a state of failure. When continuing the construction process, the next stage should start from the previous phase rather than from the results of the safety analysis. Alternatively, safety analyses for the various construction stages can be performed at the end of the calculation process. In that case, the reference phase selected in the *Start from phase* drop-down menu should refer to the corresponding construction stage.

7.2.3 *Phases window*

Each calculation involves a number of parameters that control the calculation process. These parameters are listed and can be edited in the **Phases** window (Calculation control parameters (on page 331)). The **Phases** window can be opened by double clicking a phase in the *Phases explorer* or clicking the *Edit phase* button.

There are two possible views to display information in the **Phases** window. The view of the phases window can be selected by clicking the icon next to the *Copy* button in the **Phases** window. The icon will change according to the selected view.

The information in the **Phases** window is displayed in three separate panels (Figure 191 (on page 299)). In the left panel the *Phases explorer* is displayed giving an overview of all the phases in the model and the relation between them. The middle panel displays information regarding the selected phase in the *Model explorer*. The right panel displays log info for the last calculation of the selected phase and provides space to write comments.

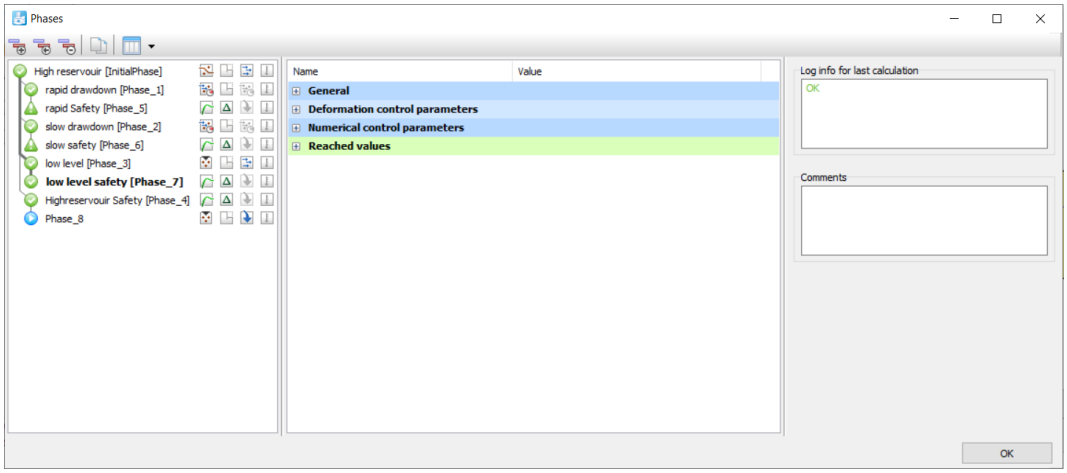


*Figure 191: Three panel view of the **Phases** window*

The *Spreadsheet* view displays all the calculation information in a spreadsheet form (Figure 192 (on page 299)) to facilitate a comparison of parameters for different phases. Parameters can be added or removed from the view by right-clicking on the bar with column header.

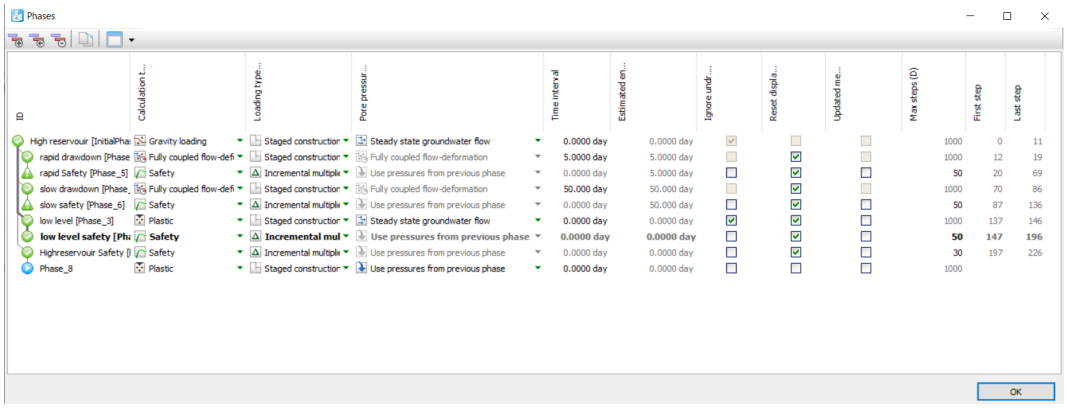


*Figure 192: Spreadsheet view of the **Phases** window*

In the **Phases** window, users need to select at least the *Calculation type* and the *Loading type* for each new phase. PLAXIS 2D provides convenient default values for most calculation control parameters, but the user can change these values. A description of the calculation types and control parameters is given in the next section.

7.3 Types of analysis

The first step in a PLAXIS 2D analysis is defining a calculation type of a phase in the *Calculation type* drop-down menu in the **Phases** window. The options available are *K0 procedure*, *Field stress* and *Gravity loading* for the initial phase to generate the initial stress state of soil. The *Flow only* option can be used if only groundwater flow analysis will be performed. For deformation analysis options such as *Plastic*, *Consolidation*, *Safety*, *Dynamic*, *Dynamic with consolidation* and *Fully coupled flow-deformation* are available.

7.3.1 *Initial stress generation*

Many analysis problems in geotechnical engineering require the specification of a set of initial stresses. The initial stresses in a soil body are influenced by the weight of the material and the history of its formation. This stress state is usually characterised by an initial vertical effective stress ($\sigma'_{v,0}$). The initial horizontal effective stress $\sigma'_{h,0}$ is related to the initial vertical effective stress by the coefficient of lateral earth pressure $K_0 (\sigma'_{h,0} = K_0 \cdot \sigma'_{v,0})$.

In PLAXIS 2D, initial stresses may be generated by using the *K0 procedure*, the *Gravity loading* or the *Field stress* option. Note that these options are available in the *Calculation type* drop-down menu only for the Initial phase. It is recommended to generate and inspect results from initial stresses first before defining and executing other calculation phases.

Note:

The *K0 procedure* differs from the *Gravity loading* as stress field equilibrium is not checked at the end of the initial phase. The *K0 procedure* is particularly suitable in cases with a horizontal surface and with all soil layers and phreatic levels parallel to the surface. In such a case, the equilibrium is systematically satisfied (vertical stresses = gravity weight, horizontal stresses = lateral reaction forces along the model boundaries). For all other cases, the use of the *K0 procedure* may lead to out-of-balance forces.

The initial stress state may involve pre-loading or overconsolidation. In particular, advanced soil models may take the effects of overconsolidation into account. This requires information about the overconsolidation ratio (OCR) or the pre-overburden pressure (POP). Such information may be provided in boreholes or in the material data set of the corresponding soil layer.

As part of the initial stress generation, the principal stress history parameter $\sigma'_{1,max}$ is initialised. The principal stress history ($\sigma'_{1,max}$, σ'_2, σ'_3) is used to calculate the equivalent preconsolidation pressure p_p^{eq}, which is used in advanced soil models to initialise a cap-type yield surface.

For more information about the calculation of $\sigma'_{1,max}$, see Initial tabsheet (on page 251).

K0 procedure

K0 procedure is a special calculation method available in PLAXIS 2D to define the initial stresses for the model, taking into account the loading history of the soil. The parameters required in the initial stresses development procedures are defined in the **Initial** tabsheet of material data sets for soil and interfaces (Initial tabsheet (on page 251)).

Only one K_0 value can be specified:

$$K_{0,x} = \sigma'_{xx} / \sigma'_{yy} \qquad K_{0,z} = \sigma'_{zz} / \sigma'_{yy} = K_{0,x} \qquad Eq.\ [73]$$

In practice, the value of K_0 for a normally consolidated soil is often assumed to be related to the friction angle by Jaky's empirical expression:

$$K_0 = 1 - \sin\varphi \qquad Eq.\ [74]$$

In an overconsolidated soil, K_0 would be expected to be larger than the value given by this expression.

For the Mohr-Coulomb model, the default value K_0-value is based on Jaky's formula. For the advanced models, (Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model, Sekiguchi-Ohta model), the default value is based on the K_0^{nc} parameter and is also influenced by the overconsolidation ratio (OCR) or the pre-overburden pressure (POP) (see Initial tabsheet (on page 251) and the Material Models Manual):

$$K_{0,x} = K_0^{nc}\text{OCR} - \frac{\nu_{ur}}{1-\nu_{ur}}(\text{OCR}-1) + \frac{K_0^{nc}\text{POP} - \frac{\nu_{ur}}{1-\nu_{ur}}\text{POP}}{\left|\sigma_{yy}^{0}\right|} \qquad \textit{Eq. [75]}$$

Using very low or very high K_0-values in the *K0 procedure* may lead to stresses that violate the Mohr-Coulomb failure condition. In this case PLAXIS 2D automatically reduces the lateral stresses such that the failure condition is obeyed. Hence, these stress points are in a plastic state and indicated as plastic points. Although the corrected stress state obeys the failure condition, it may result in a stress field which is not in equilibrium. It is generally preferable to generate an initial stress field that does not contain Mohr-Coulomb plastic points.

Note:

The plot of plastic points may be viewed after the presentation of the initial effective stresses in the Output program by selecting the *Plastic points* option from the *Stresses* menu (see Plastic points (on page 438)).

For a cohesionless material it can easily be shown that to avoid Mohr-Coulomb plasticity, the value of K_0 is bound by:

$$\frac{1-\sin\varphi}{1+\sin\varphi} < K_0 < \frac{1+\sin\varphi}{1-\sin\varphi} \qquad \textit{Eq. [76]}$$

When the *K0 procedure* is adopted, PLAXIS 2D will generate vertical stresses that are in equilibrium with the self-weight of the soil. Horizontal stresses, however, are calculated from the specified value of K_0. Even if the value of K_0 is chosen such that plasticity does not occur, the *K0 procedure* does not ensure that the complete stress field is in equilibrium, since it does not generate shear stresses. Full equilibrium is only obtained for a horizontal soil surface with any soil layers parallel to this surface and a horizontal phreatic level. Therefore, the *K0 procedure* is not recommended when dealing with non-horizontal surfaces, which require shear stresses to form an equilibrium stress field. Examples of non-horizontal surfaces, and non-horizontal weight stratifications are:

If the stress field requires only small equilibrium corrections, then these may be carried out using the *K0 procedure* followed by a *Plastic nil-phase* (Plastic nil-phase (on page 305)). If the stresses are substantially out of equilibrium, then the *K0 procedure* should be abandoned in favor of the *Gravity loading* procedure.

At the end of the *K0 procedure*, the full soil weight is activated (when *ΣMweight = 1.0*).

Field stress

In addition to *K0-procedure* and *Gravity loading* there is another method to introduce an initial stress field in the model, which is called *Field stress*. This option is available only for the *Initial phase* and can be selected as a *Calculation type* in the phases window.

Field stress allows for setting up a homogeneous initial stress state in the model, taking into account a rotation of principal stresses. This might be relevant for applications in deep soil or rock layers where the formation of these layers in the geological history has caused a rotation of principal stresses (shearing).

In addition to the selection of *Field stress* as *Calculation type* for the initial phase, the user needs to define the magnitude of the three principal stresses σ_1, σ_2, σ_3, as well as the orientation of the principal stress directions (*Axis1*), see Figure 193 (on page 302). σ_1 and σ_2 are the in-plane principal stresses and σ_3 is the out-of-plane stress.

Note: Compression is negative.

The principal stresses do not need to be ordered in the sense that σ_1 is the major principal stress and σ_3 is the minor principal stress. Moreover, the (x,y) coefficients of the orientation axis do not need to form a vector of unit length.

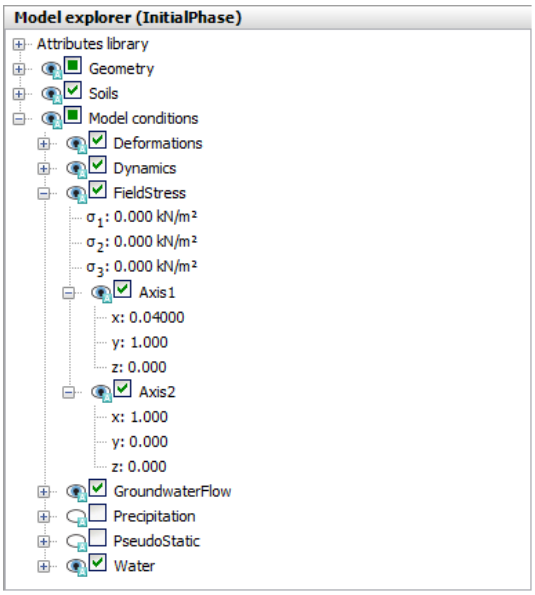


Figure 193: Field stress in Model explorer

Note:

Note that while visualising *Principal stresses* in Output, the values displayed for *Min* and *Max* indicate not only the visible in-plane stresses, but also the out-of-plane stress, which is not visible

As an alternative to the global definition of the field stress in the **Model conditions**, the user may define local field stress conditions in each soil polygon (cluster). To do so, in **Soil mode**, right click on the soil polygon. From the context menu choose *Soil_n > Create > Create field stress*. Selecting this option displays *ClusterFieldStress* in the selection explorer. Here the user needs to define the magnitude of the three principal stresses σ_1, σ_2, σ_3, as well as the orientation of the principal stress directions (*Axis1*) for the respective cluster. This option of defining different stress states to individual clusters might be relevant for applications in rock layers with disturbance zones where the weaker material may have a different initial stress state than the rock mass outside the disturbance zone.

Note that, after applying the *Field stress* initial calculation the user will still need to apply nil-phase in order to solve any unbalance in the model to get the correct initial stress state.

The calculation type *Field stress* does not consider the increase of stresses with depth due to gravity. Therefore, the *ΣMweight* multiplier is always set to zero in the case that the initial stress state is generated using the *Field stress* option. Still, nonzero unit weights may be defined in material data sets that are used in the soil/rock clusters. A proper unit weight may be required in order to generate the right mass and inertia in the case of dynamics calculations. Note that calculation phases that follow after a *Field stress* calculation should also have *ΣMweight = 0* in order to prevent a sudden unbalance from occurring.

Besides the input of field stresses, all model boundaries need to be fully fixed. Therefore, the user must manually set all global model boundaries (*BoundaryXMin*, *BoundaryXMax*, *BoundaryYMin*, *BoundaryYMax*,) to *Fully fixed* in the *Deformations* section of the *Model explorer*. Note that the *Field stress* option may generate shear stresses, which cannot be supported by the default boundary conditions; this is why the model boundaries must be fixed.

The result of a field stress calculation is a stress state according to the predefined principal stresses and the orientation of the first principal stress direction according to the *Axis1* parameter (Figure 194 (on page 303)). This stress state is not only generated in volume elements, but a corresponding stress state is also generated in interface elements, if applicable. Note that the *Field stress* option does not affect structural elements, and it will not take into account external loads and boundary conditions, just as the K0-procedure. Hence, structural elements and external loads should be inactive in the *Initial phase*.

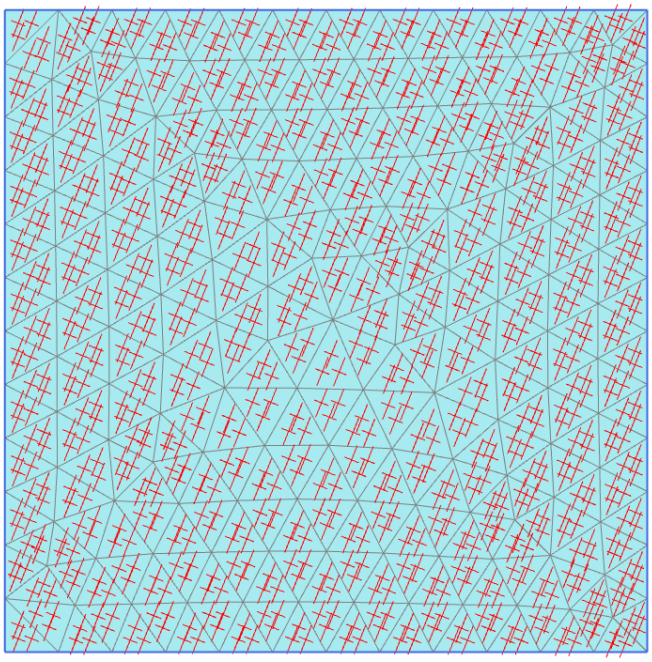


Figure 194: Initial stress generation using Field stress calculation

Gravity loading

Gravity loading is a type of *Plastic* calculation (Plastic calculation (on page 305)), in which initial stresses are generated based on the volumetric weight of the soil. If *Gravity loading* is adopted, then the initial stresses are set up by applying the soil self-weight in the first calculation phase. This is achieved by setting *ΣMweight = 1.0*. In this case, when using an elastic perfectly-plastic soil model such as the Mohr-Coulomb model, the ratio of horizontal effective stress over vertical effective stress, K_0, depends strongly on the assumed values of Poisson's ratio. It is important to choose values of Poisson's ratio that give realistic values of K_0. If necessary, separate material data sets may be used with Poisson's ratio adjusted to provide the proper K_0-value during gravity

loading. These sets may be changed by other material sets in subsequent calculations (Reassigning material data sets (on page 361)). For one-dimensional compression an elastic computation will give:

$$K_0 = \frac{\nu}{1 - \nu} \qquad Eq.\ [77]$$

If a value of K_0 of 0.5 is required, for example, then it is necessary to specify a value of Poisson's ratio of *0.333*. As Poisson's ratio must be lower than *0.5*, it is not straightforward to generate K_0 values larger than *1* using *Gravity loading*. If K_0 values larger than *1* are desired, it is necessary to simulate the loading history and use different Poisson's ratios for loading and unloading or use the *K0 procedure*.

When advanced soil models are used, the resulting K_0-value after gravity loading corresponds to the K_0^{nc} parameter in the material data set.

Note:

- To make sure that *Gravity loading* results in initial effective stresses in situations where undrained materials are used, the parameter *Ignore undrained behaviour* should be selected.
- Once the initial stresses have been set up using *Gravity loading*, the displacements should be reset to zero at the start of the next calculation phase. This removes the effect of the initial stress generation procedure on the displacements developed during subsequent calculations, whereas the stresses remain.
- The *OCR* and *POP* parameters are ignored in *Gravity loading*.

In some cases plastic points will be generated during the *Gravity loading* procedure. For cohesionless soils in one-dimensional compression, for example, plastic Mohr-Coulomb points will be generated unless the following inequality is satisfied:

$$\frac{1 - \sin\varphi}{1 + \sin\varphi} < \frac{\nu}{1 - \nu} < 1 \qquad Eq.\ [78]$$

Results of initial stress generation

After the generation of initial stresses the plot of the initial effective stresses can be inspected (Accessing the Output program (on page 395)). It is also useful to view the plot of plastic points.

Using K_0 values that differ substantially from unity may sometimes lead to an initial stress state that violates the Mohr-Coulomb criterion. If the plot of the plastic points shows many red plastic points (Mohr-Coulomb points), the value of K_0 should be chosen closer to *1.0*.

If there are a small number of plastic points, it is advisable to perform a plastic nil-phase. When using the Hardening Soil model and defining a normally consolidated initial stress state (*OCR* = 1.0 and *POP* = 0.0), the plot of plastic points shows many hardening points. Users need not be concerned about these plastic points as they just indicate a normally consolidated stress state.

Flow only [ULT]

This option allows pure groundwater flow calculations under saturated and unsaturated conditions. This option can only be selected for the *Inital phase*. Note that selection of this option effects the following phases as well. The calculation type for the following phases will be automatically set to *Flow only*. The *Calculation type* option is grayed out indicating that no change is possible. Note that it is not possible to define *Flow only* phases if the *Calculation type* of the initial phase is not specified as such.

When the *Flow only* option is selected for the initial phase only the *Steady state groundwater flow* option is available for *Pore pressure calculation type*.

Plastic nil-phase

If the *K0 procedure* generates an initial stress field that is not in equilibrium or where Mohr-Coulomb plastic points occur, then a plastic nil-phase should be adopted. A plastic nil-phase is a plastic calculation phase in which no additional load is applied (Plastic nil-phase (on page 314)). After this phase has been completed, the stress field will be in equilibrium and all stresses will obey the failure condition.

If the initial *K0 procedure* generates a stress field that is far from equilibrium, then the plastic nil-phase may fail to converge. This happens, for example, when the *K0 procedure* is applied to problems with very steep slopes. For these problems, the *Gravity loading* procedure should be adopted.

It is important to ensure that displacements calculated during a plastic nil-sphase (if it is applied immediately after generating the initial stresses) do not affect later calculations. This is achieved by selecting the *Reset displacements to zero* parameter in the subsequent calculation phase (Deformation control parameters (on page 334)).

7.3.2 *Plastic calculation*

A *Plastic* calculation is used to carry out an elastic-plastic deformation analysis in which it is not necessary to take the change of pore pressure with time into account. If the *Updated mesh* parameter has not been selected, the calculation is performed according to the small deformation theory. The stiffness matrix in a normal plastic calculation is based on the original undeformed geometry. This type of calculation is appropriate in most practical geotechnical applications.

Although a time interval can be specified, a plastic calculation does not take time effects into account, except when the Soft Soil Creep model is used (see Material Models Manual). Considering the quick loading of saturated clay-type soils, a *Plastic* calculation may be used for the limiting case of fully undrained behaviour using the *Undrained (A)*, *Undrained (B)* or *Undrained (C)* option in the material data sets. On the other hand, performing a fully drained analysis can assess the settlements on the long term. This will give a reasonably accurate prediction of the final situation, although the precise loading history is not followed and the process of consolidation is not dealt with explicitly.

An elastic-plastic deformation analysis where undrained behaviour (*Undrained (A)* or *Undrained (B)*) is temporarily ignored can be defined by checking the *Ignore undr. behaviour (A, B)* parameter. In this case the stiffness of water is not taken into account.

Note that *Ignore undrained behaviour* does not affect materials of which the drainage type is set to *Undrained (C)*.

When changing the geometry configuration (Geometry configuration - Staged Construction mode (on page 359)) it is also possible (for each calculation phase) to redefine the water boundary conditions and recalculate the pore pressures (Flow conditions (on page 350)). For more details on theoretical formulations of a plastic calculation reference should be made to the Scientific Manual.

In a *Plastic* calculation loading can be defined in the sense of changing the load combination, stress state, weight, strength or stiffness of elements, activated by changing the load and geometry configuration or pore pressure distribution by means of *Staged construction*. In this case, the total load level that is to be reached at the end of the calculation phase is defined by specifying a new geometry and load configuration, and/or pore pressure distribution, in the *Staged construction* mode (Geometry configuration - Staged Construction mode (on page 359)).

The options for *Pore pressure calculation type* for a *Plastic phase* are:

- *Phreatic*
- *Use pressures form previous phase*
- *Steady state groundwater flow*

More information on *Pore pressure calculation type* is given in Water pressure calculation (on page 321).

7.3.3 *Consolidation calculation* [ADV]

A *Consolidation* calculation is usually conducted when it is necessary to analyse the development and dissipation of excess pore pressures in a saturated clay-type soil as a function of time. PLAXIS 2D allows for true elastic-plastic consolidation analysis. In general, consolidation analysis without additional loading is performed after an undrained plastic calculation. It is also possible to apply loads during a consolidation analysis. However, take care when a failure situation is approached, since the iteration process may not converge in such a situation.

A consolidation analysis requires additional boundary conditions on excess pore pressures (Flow conditions (on page 350)).

Note:

- In PLAXIS 2D, pore pressures are divided into steady-state pore pressures and excess pore pressures. Steady state pore pressures are generated according to the water conditions assigned to the soil layers for each phase, whereas excess pore pressures are calculated as a result of undrained soil behaviour (*Undrained (A)* or *Undrained (B)*) or consolidation. A *Consolidation* calculation in PLAXIS 2D only affects the excess pore pressures.
- Rather than considering the drainage type settings Undrained (A) or Undrained (B), a Consolidation calculation considers the corresponding permeabilities as defined in the Groundwater tab of the material data set instead.
- A *Consolidation* calculation does not affect *Undrained (C)* materials since such materials do not allow (excess) pore pressures to be generated.

In a *Consolidation* analysis, the following options are available:

Consolidation and simultaneous loading in the sense of changing the load combination, stress state, weight, strength or stiffness of elements, activated by changing the load and geometry configuration by means of *Staged construction*. It is necessary to specify a value for the *Time interval* parameter, which has in this case the meaning of the total consolidation period applied in the current calculation phase. The load is linearly increased to the specified level within the time interval. The applied first time increment is based on the *First time step* parameter in the *Numerical control parameters* subtree. The *Staged construction* option should also be selected if it is desired to allow for a certain consolidation period without additional loading.

Consolidation without additional loading, until all excess pore pressures have decreased below a certain minimum value, specified by the *Minimum excess pore pressures* parameter. By default, the *Minimum excess pore pressures* is set to 1 stress unit, but this value may be changed by the user. Please note that the *Minimum excess pore pressures* parameter is an absolute value, which applies to pressure as well as tensile stress. The input of a *Time interval* is not applicable in this case, since it cannot be determined beforehand how much time is needed

to fulfill the minimum excess pore pressure requirement. The applied first time increment is based on the *First time step* parameter in the *Numerical control parameters* subtree.

Consolidation without additional loading, until a desired degree of consolidation, specified by the *Degree of consolidation* parameter, is reached. By default, *Degree of consolidation* parameter is set to 90.0 %, but this value may be changed by the user. The input of a *Time interval* is not applicable in this case, since it cannot be determined beforehand how much time is needed to fulfill the degree of consolidation requirement. The applied first time increment is based on the *First time step* parameter in the *Numerical control parameters* subtree.

Note:

Although the degree-of-consolidation is officially defined in terms of target settlement over final settlement, in PLAXIS 2D it is defined as the target minimum excess pore pressure over the maximum initial excess pore pressure $p_{max}/p_{max,initial}$.

7.3.4 *Fully coupled flow-deformation analysis* [ULT]

A *Fully coupled flow-deformation* analysis is conducted when it is necessary to analyse the simultaneous development of deformations and pore pressures in saturated and partially saturated soils as a result of time-dependent changes of the hydraulic boundary conditions. Examples where a fully coupled flow-deformation analysis may be required include draw down of the reservoir level behind a dam, embankment dams subjected to tidal waves and partially drained excavation and dewatering of a building site. In contrast to a consolidation analysis, which primarily affects the excess pore pressures, a fully coupled flow-deformation analysis directly operates on the total pore water pressures, i.e. the sum of steady-state and excess pore pressures. Still, in order to be consistent with other calculation types, steady-state pore pressures are calculated based on the hydraulic conditions at the end of the calculation phase, which enables the back-calculation of excess pore pressures from the total pore water pressures.

A fully coupled flow-deformation analysis takes into account unsaturated soil behaviour and suction in the unsaturated zone above the phreatic level. This is the most advanced and realistic type of analysis that considers a reduced permeability and degree of saturation in unsaturated zone. For this reason, the *Ignore suction* option is not available in *Fully coupled flow-deformation* analysis.

In principle, a fully coupled flow-deformation analysis takes into account unsaturated soil behaviour and suction in the unsaturated zone above the phreatic level. However, positive pore stresses in the unsaturated zone may be restricted by using the *Ignore suction* option.

When fully coupled flow - deformation analysis is combined with temperature calculation, a fully coupled Thermo - Hydro - Mechanical (THM) analysis is obtained. More detailed explanation, refer section Coupled analysis [ULT] (on page 325).

Note:

It is not possible to use *Updated mesh* in a fully coupled flow-deformation analysis.

7.3.5 *Safety calculation (phi/c reduction)*

The *Safety* calculation type is an option available in PLAXIS 2D to compute global safety factors. This option can be selected as a separate *Calculation type* in the **General** tabsheet.

In the *Safety* approach the shear strength parameters tan φ and c of the soil as well as the tensile strength are successively reduced until failure of the structure occurs. The dilatancy angle ψ is, in principle, not affected by the phi/c reduction procedure. However, the dilatancy angle can never be larger than the friction angle. When the friction angle φ has reduced so much that it becomes equal to the (given) dilatancy angle, any further reduction of the friction angle will lead to the same reduction of the dilatancy angle. The strength of interfaces, if used, is reduced in the same way. Optionally, the strength of structural objects like plates and anchors can also be reduced in a *Safety* calculation (Enhanced safety analysis (on page 310)).

The total multiplier *ΣMsf* is used to define the value of the soil strength parameters at a given stage in the analysis:

$$\Sigma Msf = \frac{\tan\varphi_{input}}{\tan\varphi_{reduced}} = \frac{c_{input}}{c_{reduced}} = \frac{s_{u,input}}{s_{u,reduced}} = \frac{\text{Tensile strength}_{input}}{\text{Tensile strength}_{reduced}} \qquad \textit{Eq. [79]}$$

where the strength parameters with the subscript 'input' refer to the properties entered in the material sets and parameters with the subscript 'reduced' refer to the reduced values used in the analysis. *Σ Msf* is set to *1.0* at the start of a calculation to set all material strengths to their input values.

A *Safety* calculation is performed using the *Load advancement number of steps* procedure (Load advancement - Number of steps (on page 330)). The incremental multiplier *Msf* is used to specify the increment of the strength reduction of the first calculation step. This increment is by default set to 0.1, which is generally found to be a good starting value. The strength parameters are successively reduced automatically until all *Additional steps* have been performed. By default, the number of steps is set to 100, but a larger value up to 10000 may be given here, if necessary. It must always be checked whether the final step has resulted in a fully developed failure mechanism. If that is the case, the factor of safety is given by:

$$SF = \frac{\text{available stregth}}{\text{stregth at failure}} = \text{value of } \Sigma\text{Msf at failure} \qquad \textit{Eq. [80]}$$

The principal results of a *Safety* calculation are the failure mechanism and the corresponding *ΣMsf*, which is the safety factor. The *ΣMsf*-value of a particular calculation step can be found in the **Calculation information** window displayed as the corresponding option is selected in the *Project* menu of the Output program. It is also recommended to view the development of *ΣMsf* for the whole calculation using the *Curves* option (Curves (on page 451)). In this way it can be checked whether a constant value is obtained while the deformation is continuing; in other words: whether a failure mechanism has fully developed. If a failure mechanism has not fully developed, then the calculation must be repeated with a larger number of steps.

Note:

The numerical parameters defined in the *Numerical control parameters* subtree are not supposed to have significant influence on the principal results of the *Safety* calculation. However, in *Safety* calculations involving safety factors close to 1, other outputs (e.g. internal forces of the structures) could be sensitive to the numerical parameters, especially to the *Desired min number of iterations* and the *Desired max number of iterations*.

To capture the failure of the structure accurately, the use of *Arc-length control* parameter is required. The use of a *Tolerated error* of no more than 1% is also required. Both requirements are complied with when using the default iteration parameters (Numerical control parameters (on page 337)).

Note:

- When performing *Safety* calculation without *Arc-length control*, the reduction factor *ΣMsf* cannot go down and an overestimation of safety factor can occur.
- To capture the failure of the structure accurately, a sufficiently fine mesh is required.

When using *Safety* calculation in combination with advanced soil models, these models will actually behave as a standard Mohr-Coulomb model, since stress-dependent stiffness behaviour and hardening effects are excluded from the analysis. In that case, the stiffness is calculated at the beginning of the calculation phase based on the starting stresses and kept constant until the calculation phase is complete.

Note:

- In case of the Jointed Rock model the strength on all the planes will be reduced by *ΣMsf*.
- In the case of the NGI-ADP model and UDCAM-S model, all undrained strength parameters are reduced by *ΣMsf*.
- Strength in the Modified Cam-Clay model and Sekiguchi-Ohta model is not reduced in *Safety analysis*.
- When using the Modified Cam-Clay modeland Sekiguchi-Ohta model, the strength is not reduced at all since these models do not have a cohesion or friction angle as model parameter. In other the reduction of the parameter values need to be explicitly defined in their code.

The strength reduction method, as adopted in a *Safety* calculation, gives similar safety factors as obtained from conventional stability analysis based on the Limit Equilibrium Method (LEM) slip-circle analysis. For a more detailed description of the strength reduction method (formerly called phi-c reduction) you are referred to Brinkgreve & Bakker (1991) (on page 530).

Note:

A more realistic value of the safety factor will be obtained for a *Fully coupled flow-deformation* analysis when suction is considered. This value is generally higher than a conventional safety factor ignoring suction. Therefore, be careful when interpreting this value.

Note:

In general, it is not recommended to perform a Safety analysis in combination with structural elements in which a residual strength is used. During a safety analysis in which structural elements are present in the model, there is a redistribution of the stresses and forces, generally leading to an increase in structural forces. When the strength of a structure is reached, and the structure has a residual strength lower than the normal strength, the structural force will reduce to the residual strength. This may lead to an unstable situation and, as a result, an unrealistic reduction of the factor of safety.

In a *Safety* calculation the following options are available:

- A *Safety* analysis is performed by reducing the strength parameters incrementally. The default increment of the strength reduction of the first calculation step, *Msf* is 0.1 but this value may be changed by the user.
- A *Safety* analysis is performed by reducing the strength parameters until a target value of the total multiplier *ΣMsf* is reached. The program will first try to find a safe value (above the target value) then it will return to the last step before passing the target and will do a last step to reach the target.

Enhanced safety analysis

For a *Safety* calculation, all soil clusters and interfaces are taken into account automatically. *Enhanced safety analysis* makes it possible to apply strength reduction to soil clusters as well as structures. Moreover, *Enhanced safety analysis* enables to exclude particular soil clusters or structures from the strength reduction procedure.

Safety calculations are controlled by the multiplier for strength reduction, *ΣMsf*. The purposes for the *Enhanced safety* can be the following:

- Surficial soil clusters along a slope can be excluded from the strength reduction procedure to avoid unrealistic shallow failure mechanisms.
- Along with the soil clusters, all structures can be selected for strength reduction, provided these structures behave as elastoplastic and have a limit strength.
- Enhanced safety analysis can be performed for a deliberate manual selection of certain soil clusters or structures for different phases.

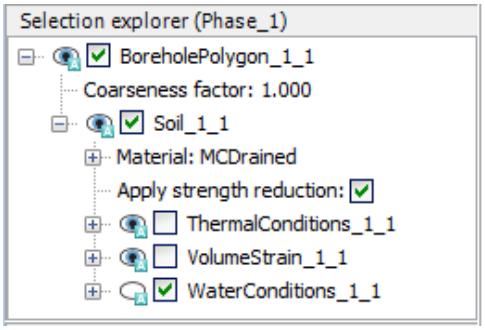


Figure 195: Enhanced safety for soil clusters

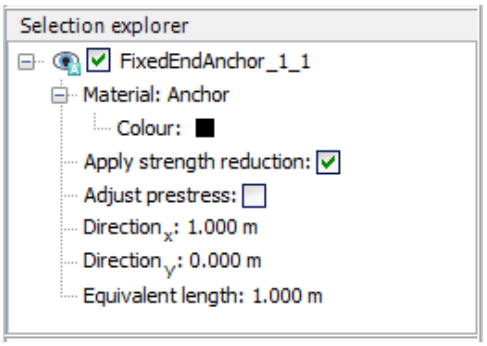


Figure 196: Enhanced safety for structures

Note:

Regarding structures, note that strength reduction can be applied only to *Elastoplastic* structures. Make sure that realistic mechanisms are not prevented in this way.

Soil clusters that have been excluded from the strength reduction procedure are indicated in grey in the *Connectivity plot*, provided that the *Hide items without strength reduction* in the *View* menu has been selected (Connectivity plot (on page 423)). Moreover, if this option has been selected, results are not shown in these clusters.

Note:

By performing a standard *Safety* calculation, unrealistic surficial mechanisms can be detected. An *Enhanced safety* calculation can be performed by de-selecting the surficial clusters.

Strength factorization in the Hoek-Brown model

When using the Hoek-Brown model to describe the behaviour of a rock section, the *Safety* calculation procedure is slightly modified, since the failure contour is not described by the Mohr-Coulomb criterion anymore. In order to have an equivalent definition of a safety factor as for the Mohr-Coulomb model, the Hoek-Brown yield function is reformulated to include the strength reduction factor *ΣMsf* for safety analyses:

$$f_{HB} = \sigma'_1 - \sigma'_3 + \overline{f}_{red}(\sigma'_3) \qquad Eq.\ [81]$$

with

$$\overline{f}_{red} = \frac{\overline{f}}{\eta} = \frac{\sigma_{ci}}{\eta}\left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^a \qquad Eq.\ [82]$$

and

$$\eta = \frac{1}{2}\left(\sum Msf\,(2 - \overline{f}\,')\sqrt{1 + \frac{\left(\frac{1}{\sum Msf^2} - 1\right)\overline{f}\,'^2}{(2 - \overline{f}\,')^2}} + \overline{f}\,'\right) \qquad Eq.\ [83]$$

where

$$\overline{f}\,' = \frac{\partial \overline{f}}{\partial \sigma'_3} = -am_b\left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^{a-1} \qquad Eq.\ [84]$$

More details and a derivation of the above equations can be found in Benz, Schwab, Vermeer & Kauther (2007) (on page 530).

Note:

Note that the safety factor obtained for Hoek-Brown material does not correspond to the safety factor obtained for Mohr-Coulomb material with equivalent strength properties.

Updated mesh

The geometry of the model considered in a *Safety* calculation depends on whether the *Updated mesh* option is selected or not. If *Updated mesh* is selected, the resulting safety factor will be from the deformed (updated) mesh.

Please note that during a safety calculation the mesh is not updated at the beginning of each load step even if the *Updated mesh* option is selected for the *Safety* phase. Hence, the *Updated mesh* mesh option in a *Safety* analysis only determines whether it starts from an updated mesh or not.

7.3.6 *Dynamics calculation* [ULT]

The *Dynamics* option should be selected when it is necessary to consider stress waves and vibrations in the soil. Dynamic loads can come in various types such as machine induced loads, impact loads, blast load, moving

vehicle load or as earthquake loads. With PLAXIS 2D it is possible to perform a dynamics analysis for these types of loads after a series of plastic calculations. It is possible to apply dynamic loads through displacement multipliers or load multipliers. They can be defined as harmonic or as input table. The applied dynamic load is the product of the input value of the defined dynamic load and the corresponding dynamic load multiplier. Besides the activation of the dynamic load or dynamic prescribed displacement, special absorbent boundary conditions can be defined for a Dynamics calculation. For a more detailed description of the boundary conditions, see Model conditions (on page 364). A Dynamics time interval can be defined to specify the calculation duration and an automatic time stepping scheme (see Automatic time stepping (Dynamics) [ULT] (on page 331)) takes care of the best combination of Max steps and Number of sub-steps, based on the estimated time steps.

Free vibration analysis

The possible vibrations of a system that occur when an existing static load is released can be analysed by performing a free vibration analysis. A free vibration analysis can be performed in a Dynamics calculation which can be used to determine the natural frequencies of a soil block or structure. To perform a free vibration analysis initially a plastic analysis is conducted with prescribed loads or displacements. This is followed by a dynamics phase in which the active static external load in the previous calculation phase is released (deactivated). Natural frequencies can be estimated by means of a power spectral density plot generated through output.

Earthquake Analysis

This section provides an overview of the Geotechnical earthquake modelling and analysis capabilities of PLAXIS. A variety of nonlinear native and UDSM models for earthquake analysis are offered in PLAXIS. In general, all native models offered in PLAXIS can be used in combination with Dynamics analysis. However, the user is expected to know the limitations of each of the models being used. Here are some models which are often used in combination with Seismic analysis. HSsmall and its UDSM version GHS can simulate strain dependency of stiffness and hysteretic damping. UBC3D-PLM (native) and PM4Sand (UDSM) are liquefaction models available in PLAXIS for which the parameters can be estimated from SPT or CPT data in the absence of lab tests. Processing and modification of input accelerograms are a key component of earthquake analysis. Options are available in the dynamics multiplier tab (Input accelerograms (on page 91)) to scale the signal to the required PGA. Furthermore, the input accelerograms can be easily transformed and visualized as Fourier amplitude spectrum, Power spectrum, PSA as well as Arias Intensity. A drift correction to correct the displacement drift due to instrument noise or background noise can also be applied automatically during a dynamics calculation. Special boundary conditions (Model conditions (on page 364)) are required for earthquake motion in order to minimize wave reflection at model boundaries. The compliant base boundary condition for the bottom boundary ensures that reflected waves from layers above are absorbed and allows direct application of an input (upward propagating) accelerogram. Free field boundary conditions for lateral boundaries impose free-field motion at the sides, additionally it absorbs the reflected secondary waves. Tied degrees of freedom connects the nodes on the same elevation at left and right boundaries, which can be used to simulate one-dimensional wave propagation.

Energy dissipation due to vibrational or cyclic loading can be defined through Damping. PLAXIS offers both hysteretic and viscous material damping, as well as numerical damping. Hysteretic damping is inherent to the HSsmall model, whereas, Rayleigh damping is a numerical feature to simulate viscous material damping, which can be applied in each individual material data set. Numerical damping can be imposed by changing the default Newmark alpha and beta parameters (Dynamics control parameters [ULT] (on page 348)). Dynamics calculations can be conducted as drained, undrained or as Dynamics with consolidation. Automatic time stepping (Automatic time stepping (Dynamics) [ULT] (on page 331)) scheme allows for a proper selection of the time step to accurately model wave propagation and reduce the numerical error due to integration of time history functions. The user can also control the number of steps using the semi-automatic or Manual option for accurate modelling.

Curves in Dynamics calculations [ULT] (on page 457) explains output curves generation options for dynamics analysis. It is possible to transform the generated curves from the time domain to the frequency domain automatically using Fast Fourier Transform (FFT). From FFT, Power Spectrum and Fourier Amplitude Spectrum(FAS) can be plotted for each acceleration component. From output curves it is possible to produce PSA spectrum to determine the predominant period as well as Relative displacement response spectrum, Amplification factor that displays the magnification of the response at a point with respect to the given excitation, and Arias intensity to determine the strength of a ground motion. Furthermore, plots can be generated for extreme accelerations, velocities and displacements for dynamics phases.

Note:

- Note that steady-state pore pressures in a *Dynamics* calculation are always taken from the steady-state pore pressures generated in the parent phase (see Calculation type (on page 322)).
- It is possible to calculate excess pore pressures in undrained soil layers in a dynamics analysis. However, the accuracy at which pore pressures are predicted depends on the capabilities of the soil models being used.
- A standard dynamics calculation may involve the generation of excess pore pressures, but not the dissipation of excess pore pressures. If the latter is required, a *Dynamics with consolidation* calculation should be performed.

Site response analysis

Wave propagation in the ground due to an earthquake can be analysed by means of a *site response analysis*. This can be performed in a *Dynamic* calculation on a narrow soil column using *tied degrees-of-freedom* and the earthquake signal applied at the base.

Updated mesh

During a dynamics calculation, large deformation effects and changes of the geometry can be taken into account by selecting the *Updated mesh* option. See Updated mesh analysis (on page 314) for more information about *Updated mesh* calculations.

In a *Dynamics* calculation loading can be defined in the sense of applying a predefined combination of external loads as dynamic forces using dynamics multipliers activated in the **Staged construction** mode. Dynamics calculations can also deal with moving loads.

7.3.7 *Dynamics with consolidation calculation* [ULT]

The *Dynamics with consolidation* option should be selected when the dynamic generation and subsequent dissipation of excess pore pressures in a short time domain is to be considered. This calculation type is based on the mixed *u-P* formulation (where *u* is displacement and *P* is excess pore pressure and can be used when partially drained soil conditions are present during a dynamics calculation. The applied dynamic load is the product of the input value of the defined dynamic load and the corresponding dynamic load multiplier.

Note:

A *Dynamics with consolidation* calculation requires the input of soil permeabilities and appropriate flow boundary conditions.

Besides the activation of the dynamic load or dynamic prescribed displacement, absorbent (viscous) boundary conditions can be defined for a *Dynamic with consolidation calculation*. For a more detailed description of the boundary conditions see Model conditions (on page 364).

In a *Dynamics with consolidation* calculation loading can be defined in the sense of applying a predefined combination of external loads as dynamic forces using dynamics multipliers activated in the **Staged construction** mode.

7.3.8 *Plastic nil-phase*

A plastic calculation may also be used to carry out a so-called plastic nil-phase. A plastic nil-phase is a plastic calculation phase in which no additional loading is applied. Each new phase introduced in the *Phases explorer* is initially a plastic nil-phase, until the calculation type, geometry or load configuration is changed. It may sometimes be required to solve large out-of-balance forces and to restore equilibrium. Such a situation can occur after a calculation phase in which large loadings were activated (for example gravity loading) or if the *K0 procedure* generates an initial stress field that is not in equilibrium or where plastic points occur. After this phase has been completed, the stress field will be in equilibrium and all stresses will obey the failure condition. In this case no changes should be made to the geometry configuration or to the water conditions. If necessary, such a calculation can be performed with a reduced *Tolerated error* to increase the accuracy of the equilibrium stress field.

If the original *K0 procedure* generates a stress field that is far from equilibrium, then the plastic nil-phase may fail to converge. This happens, for example, when the *K0 procedure* is applied to problems with very steep slopes. For these problems the *Gravity loading* procedure should be adopted instead.

It is important to ensure that displacements calculated during a plastic nil-phase (if it is applied immediately after generating the initial stresses) do not affect later calculations. This may be achieved by using the *Reset displacements to zero* option in the subsequent calculation phase (Deformation control parameters (on page 334)).

The *Staged construction* loading type is used to perform a plastic nil-phase to solve existing out-of-balance forces. No changes in the geometry, load level, load configuration and water pressure distribution should be made.

7.3.9 *Updated mesh analysis*

In conventional finite element analysis, the influence of the geometry change of the mesh on the equilibrium conditions is neglected. This is usually a good approximation when the deformations are relatively small as is the case for most engineering structures. However, there are circumstances under which it is necessary to take this influence into account. Typical applications where updated mesh analyses may be necessary include the analysis of reinforced soil structures, the analysis of large offshore footing collapse problems and the study of problems where soils are soft and large deformations occur.

Note:

By default the *Updated mesh* option in **Phases** window is unchecked.

When large deformation theory is included in a finite element program some special features need to be considered. Firstly it is necessary to include additional terms in the structure stiffness matrix to model the effects of large structural distortions on the finite element equations.

Secondly, it is necessary to include a procedure to model correctly the stress changes that occur when finite material rotations occur. This particular feature of large displacement theory is usually dealt with by adopting a definition of stress rate that includes rotation rate terms. Several stress rate definitions have been proposed by researchers working in this field although none of these are wholly satisfactory. In PLAXIS 2D the co-rotational rate of Kirchhoff stress (otherwise known as the Hill stress rate) is adopted. This stress rate would be expected to give accurate results provided that the shear strains do not become excessive.

Thirdly, it is necessary to update the finite element mesh as the calculation proceeds. This is done automatically within PLAXIS 2D when the *Updated mesh* option is selected.

It should be clear from the descriptions given above that the updated mesh procedures used in PLAXIS 2D involve considerably more than simply updating nodal coordinates as the calculation proceeds. These calculation procedures are in fact based on an approach known as an Updated Lagrangian formulation (Bathe, 1982 (on page 530)). Implementation of this formulation within PLAXIS 2D is based on the use of various advanced techniques that are beyond the scope of this manual. More details can be found in (van Langen, 1991 (on page 532)).

The three basic types of calculations (*Plastic*, *Consolidation* and *Safety*) can optionally be performed as an *Updated mesh* analysis, taking into account the effects of large deformations. Therefore, the *Updated mesh* parameter should be selected. It can also be selected whether water pressures should be continuously recalculated according to the updated position of the stress points. This option is termed *Updated water pressures* and is meant to take into account the effects of soil settling (partly) below a constant phreatic level. The *Updated mesh* option cannot be used in a fully coupled flow-deformation analysis.

Please note that an updated mesh calculation cannot be followed by a 'normal' calculation. Reversely, a normal calculation can be followed by an updated mesh calculation, provided that the option *Reset displacements to zero* is used (Deformation control parameters (on page 334)).

It should be noted that an updated mesh analysis takes more time and is less robust than a normal calculation. Hence, this option should only when really necessary.

Distributed loads

Distributed loads on deformed boundaries are taken into account as if those boundaries were not deformed. This is to ensure that the total force involved does not change when the boundary stretches or shrinks. This also applies to axisymmetric applications where the radius changes as a result of deformation.

Calculation procedures

In order to carry out an updated mesh analysis the *Updated mesh* checkbox in the *Deformation control parameters* subtree in the **Phases** window should be selected (Deformation control parameters (on page 334)).

Updated mesh calculations are carried out using iteration procedures similar to the conventional calculation options as described in preceding sections. Therefore an updated mesh analysis uses the same parameters. However, because of the large deformation effect, the stiffness matrix is always updated at the beginning of a load step. Due to this procedure and to the additional terms and more complex formulations, the iterative procedure in an updated mesh analysis is considerably slower than that for conventional calculations.

Safety calculations

The geometry of the model considered in a *Safety* calculation depends on whether the *Updated mesh* option is selected or not in the parent phase. If the mesh is updated, the resulting geometry at the end of the parent phase will be considered in the safety calculations.

During a safety calculation the mesh is not further updated at the beginning of each load step even if the *Updated mesh* option is selected for the *Safety* phase.

Practical considerations

Updated mesh analysis tends to require more computer time than an equivalent, conventional calculation. It is recommended, therefore, that when a new project is under study a conventional calculation is carried out before an updated mesh analysis is attempted.

It is not possible to give simple guidelines that may be used to indicate when an updated mesh analysis is necessary and where a conventional analysis is sufficient. One simple approach would be to inspect the deformed mesh at the end of a conventional calculation using the *Deformed mesh* option in the Output program. If the geometry changes are large (on a real scale) then significant importance of geometric effects might be suspected. In this case the calculation should be repeated using the updated mesh option. It cannot definitely be decided from the general magnitudes of the deformations obtained from a conventional plasticity calculation whether geometric effects are important or not. If the user is in any doubt about whether updated mesh analysis is necessary then the issue can only be resolved by carrying out the updated mesh analysis and comparing the results with the equivalent conventional analysis.

In general, it is not appropriate to use an updated mesh calculation for gravity loading to set up the initial stress field. Displacements resulting from gravity loading are physically meaningless and should therefore be reset to zero. Resetting displacements to zero is not possible after an updated mesh analysis. Hence, gravity loading should be applied in a normal plastic calculation.

Changing from a 'normal' plastic calculation or consolidation analysis to an updated mesh analysis is only valid when displacements are reset to zero, because a series of updated mesh analyses must start from an undeformed geometry. Changing from an updated mesh calculation to a 'normal' plastic calculation or consolidation analysis is not valid, because then all large deformation effects will be disregarded, resulting in a large force unbalance.

Updated water pressures

When the *Updated water pressures* option is selected in the *Deformation control parameters* subtree in the **Phases** window, pore pressures in stress points and external water pressures at model boundaries are updated during the calculation according to the deformed model boundaries and the displaced position of stress points. Basis for the update of water pressures is the general phreatic level and the cluster phreatic levels. In this way, the buoyancy effect of soil that is submerged below the phreatic level is taken into account.

Note that the pore pressures in clusters that have user-defined pore pressures are not updated. Also, pore pressures that are calculated from groundwater flow calculations are not updated.

7.4 Loading type

The loading type is specified in the *Loading input* drop-down menu in the **Phases** window. Only one of the described loading types can be activated in any single calculation phase. The available loading types depend on the selected type of calculation.

Note:

Note that the *Loading type* drop-down menu is NOT available for *Flow only* calculation type.

7.4.1 *Staged construction*

The *Staged construction* loading type enables the user to specify a new state that is to be reached at the end of the calculation phase. The water pressure distribution, the geometry, the input values of loads and the load configuration in the **Flow conditions** and **Staged construction** mode can be modified. The *Staged construction* option may also be used to perform plastic nil-phases to solve existing out-of-balance forces. In this case, no changes in the geometry, load level, load configuration and water pressure distribution should be made.

Before specifying the construction stage, the *Time interval* of the calculation phase should be considered. The *Time interval* is expressed in the unit of time. A non-zero value is only relevant in the case of a *Consolidation* analysis, a *Fully coupled flow-deformation* analysis or if a time-dependent soil model (such as *Soft Soil Creep model*) is used.

Since staged construction is performed using the *Load advancement ultimate level* procedure (Load advancement - Ultimate level (on page 328)), it is controlled by a total multiplier (*ΣMstage*).This multiplier starts at zero and is expected to reach the ultimate level of 1.0 at the end of the calculation phase. Note that the maximum number of additional steps as defined by the *Max steps* parameter will not be reached if *ΣMstage* criterion is met before.

In some special situations, however, it might be necessary to split the staged construction process into more than one calculation phase and to specify an intermediate value of *ΣMstage*. This can be done by assigning a value lower than 1.0 to the *ΣMstage* parameter in the *General* subtree in the **Phases** window which is only available for a *Plastic* calculation. However, care must be taken with an ultimate level smaller than 1.0, since this is associated with a resulting out-of-balance force. Such calculations must always be followed by another staged construction calculation.

Without specifying a value for *ΣMstage*, the program always assumes an ultimate level of *ΣMstage* = 1.0. Before starting any other type of calculation the *ΣMstage* parameter must first have reached the value 1.0. This can be verified in the *Reached values* subtree in the **Phases** window (Reached values (on page 349)).

Staged construction with *ΣMstage* < 1

In general, the total multiplier associated with the staged construction process, *ΣMstage*, goes from zero to unity in each calculation phase where staged construction has been selected as the loading input. In some very special situations it may be useful to perform only a part of a construction stage. This can be done by specifying a value lower than 1.0 tothe *ΣMstage*.

Unfinished staged construction calculation

At the start of a staged construction calculation, the multiplier that controls the staged construction process, that is to say *ΣMstage*, is zero and this multiplier is stepwise increased to the ultimate level (generally 1.0). When *ΣMstage* has reached the ultimate level, the current phase is finished. However, if a staged construction calculation has not properly finished, i.e. the multiplier *ΣMstage* is less than the desired ultimate level at the end of a staged construction analysis, then a warning appears in the **Log info** box. The reached value of the *ΣMstage* multiplier may be viewed in the *Reached values* subtree in the **Phases** window.

There are three possible reasons for an unfinished construction stage:

- An ultimate value of *ΣMstage* less than 1.0 is specified by the user. Note that the out-of-balance force is still partly unresolved. The remain out-of-balance forces must be solved in the next calculation phase.
- Failure of the soil body has occurred during the calculation. This means that it is not possible to finish the construction stage. Note that the out-of-balance force is still partly unsolved so that further calculations starting from the last calculation phase are meaningless.
- The maximum number of loading steps was insufficient. In this case the construction stage should be continued by performing another staged construction calculation that is directly started without changing the geometry configuration or water pressures. Alternatively, the phase may be recalculated using a larger number of *Max steps*. Note that it is advised against applying any other type of loading as long as the *ΣMstage* has not reached the value 1.0.

In the case of an unfinished staged construction calculation, the load that has actually been applied differs from the defined load configuration. The reached value of the *ΣMstage* multiplier may be used in the following way to estimate the load that has actually been applied:

$$f_{applied} = f_0 + \Sigma Mstage\left(f_{defined} - f_0\right) \qquad Eq.\ [85]$$

where $f_{applied}$ is the load that has actually been applied, f_0 is the load at the beginning of the calculation phase (i.e. the load that has been reached at the end of the previous calculation phase) and $f_{defined}$ is the defined load configuration.

A reduced ultimate level of *ΣMstage* may be reduced repetitively. In the case of multiple subsequent phases with *ΣMstage < 1*, it should be realised that *ΣMstage* starts at 0 in every phase. For example, if three phases are defined, where in phase 1 *ΣMstage = 0.5*; in phase 2 *ΣMstage = 0.5* and phase 3 *ΣMstage = 1.0* (without additional changes), it means that:

- At the end of phase 1: 50% of the unbalance is solved.
- At the end of phase 2: 50% of the remaining unbalance (= 75% of the initial unbalance) is solved.
- At the end of phase 3: 100% of the remaining unbalance (= 100%of the initial unbalance) is solved.

Deconfinement

In PLAXIS 2D it is possible to simulate the construction process of tunnels with a sprayed concrete lining (NATM). The major point in such an analysis is to account for the three-dimensional arching effect that occurs within the soil and the deformations that occur around the unsupported tunnel face. A method that takes these effects into account is described below.

There are various methods described in the literature for the analysis of tunnels constructed according to the New Austrian Tunnelling Method. One of these is the so-called Convergence confinement method or β-method (Schikora & Fink, 1982 (on page 532)), but others have presented similar methods under different names. The idea is that the initial stresses p_k acting around the location where the tunnel is to be constructed are divided into a part $(1 - \beta)\ p_k$ that is applied to the unsupported tunnel and a part $\beta\ p_k$ that is applied to the supported tunnel (Figure 198 (on page 319)). The β-value is an 'experience value', which, among other things, depends on

the ratio of the unsupported tunnel length and the equivalent tunnel diameter. Suggestions for this value can be found in literature (Schikora & Fink, 1982 (on page 532)).

In PLAXIS 2D it is possible to enter a *Deconfinement* value during staged construction as *1- β* in the model explorer (Figure 197 (on page 319)). This value is then taken into account during the calculation phases in *Staged construction*. The value of deconfinement is entered in percentage.

The way this works in PLAXIS 2D is that part of the stresses (*β*) in the soil cluster inside the tunnel are retained as a support pressure, while the cluster is de-activated. In subsequent phases, this support pressure can be further reduced (more deconfinement) until it is eventually zero (*β=0* or *100%* deconfinement).

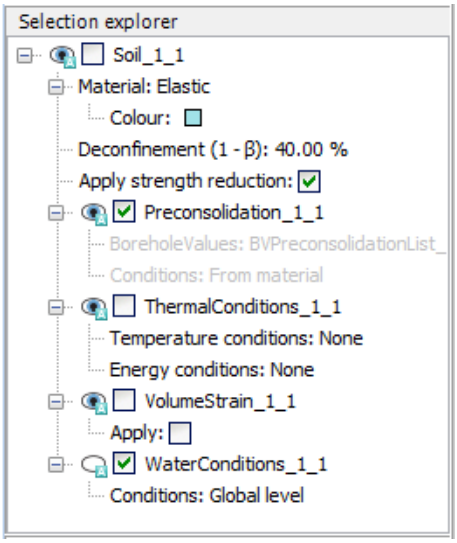


Figure 197: Deconfinement option in Selection explorer

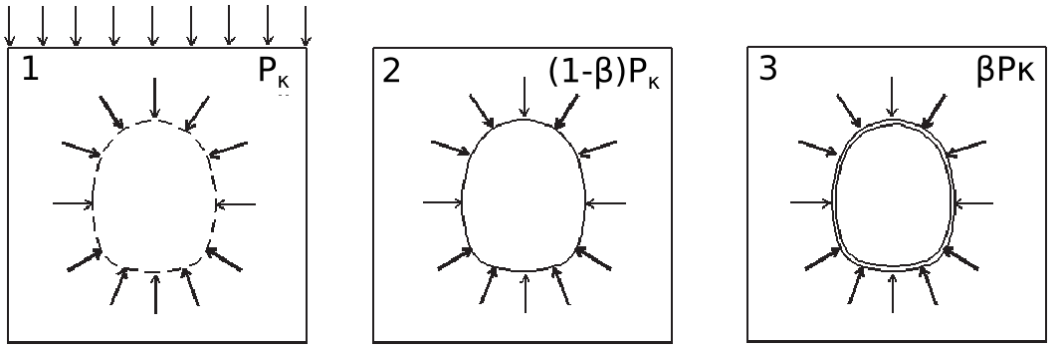


Figure 198: Schematic representation of the β-method for the analysis of NATM tunnels

The process is summarised below:

1. Generate the initial stress field and apply eventual external loads that are present before the tunnel is constructed.
2. De-activate the tunnel clusters without activation of the tunnel lining and apply a *Deconfinement* value of *1- β*. The value entered is in percentage.
3. Activate the tunnel lining to the soil clusters inside the tunnel and check that *Deconfinement* of the soil clusters inside the tunnel is set to 100%.

Deconfinement can be defined per soil cluster. That means that different clusters may have different levels of deconfinement. In the case of a single tunnel with different excavation sections, each excavation section may have a different level of deconfinement. In the case of multiple tunnels, each tunnel (or excavation section) may have a different level of deconfinement. Instead of only one intermediate level of deconfinement, the excavation process may involve different levels of deconfinement, divided over multiple calculation phases.

7.4.2 *Minimum excess pore pressure* [ADV]

The *Minimum excess pore pressure* option in the *Loading input* drop-down menu is a criterion for terminating a consolidation analysis. The calculation stops when the maximum absolute excess pore pressure is below the prescribed value of |*P-stop*|. Note that the maximum number of steps as defined by the *Max steps* parameter will not be reached if the |*P-stop*| criterion is met before. For example, when the maximum excess pore pressure has reached a certain value during the application of load, the user can make sure that the consolidation process is continued until all nodal values of excess pore pressure are less than |*P-stop*|, provided the value of *Max steps* is sufficient.

Note:

The *Minimum excess pore pressure* loading type is available only for *Consolidation* calculations.

7.4.3 *Degree of consolidation* [ADV]

The option *Degree of consolidation* is an alternative criterion for terminating a consolidation analysis. The calculation stops when the degree of consolidation, as defines herein, is below the value of *Degree of consolidation*. The degree of consolidation is an important indication of the consolidation state. Strictly, the degree of consolidation, *U*, is defined in terms of the proportion of the final settlement although the term is often used to describe the proportion of pore pressures that have dissipated to at least (100-*U*)% of their values immediately after loading. The *Degree of consolidation* option may be used to specify the final degree of consolidation in any analysis.

In this case the *Minimum excess pore pressure* parameter (see above) is set to a value as defined by the maximum excess pore pressure in the previous phase and the defined *Degree of consolidation (U)*:

$$\text{Minimim pore pressure} = (100 - U)\text{Pmax} \qquad \textit{Eq. [86]}$$

where *Pmax* is the maximum excess pore pressure reached in the previous phase which can be found in the *Reached values* subtree in the **Phases** window (Reached values (on page 349)). The calculation stops when the maximum absolute excess pore pressure is below this calculated value of *Minimum excess pore pressure*. Note that the maximum number of steps as defined by the parameter *Max steps* will not be reached if the *Minimum excess pore pressure* criterion is met before.

Note:

The *Degree of consolidation* loading type is available only for *Consolidation* calculations.

7.4.4 *Target ΣMsf*

Reduction of the soil and interface strength towards a target value of the total multiplier *ΣMsf*. The program first performs a full safety analysis until failure (Incremental multipliers (on page 321)) and then it recalculates the last step before the target value of *ΣMsf* in order to reach the target exactly.

Note:

The *Target ΣMsf* loading type is available only for *Safety* calculations.

7.4.5 *Incremental multipliers*

The *Incremental multipliers* loading type enables the user to perform a *Safety* analysis using the *Load advancement number of steps* procedure (Incremental multipliers (on page 321)). The incremental multiplier *Msf* is used to specify the increment of the strength reduction of the first calculation step. This increment is by default set to 0.1, which is generally found to be a good starting value. The strength parameters are successively reduced automatically until the value assigned to the *Max steps* parameter has been reached. By default, the maximum number of steps is set to 100, but a larger value up to 1000 may be given here, if necessary. It must always be checked whether the final step has resulted in a fully developed failure mechanism. If that is the case, the factor of safety is given by:

$$SF = \frac{\text{available strength}}{\text{strength at failure}} = \text{value of } \Sigma\text{Msf at failure} \qquad \textit{Eq. [87]}$$

The *ΣMsf*-value of a particular calculation step can be found in the **Calculation Information** window of the Output program. It is also recommended to view the development of *ΣMsf* for the whole calculation using the *Curves* option (Curves (on page 451)). In this way it can be checked whether a constant value is obtained while the deformation is continuing; in other words: whether a failure mechanism has fully developed. If a failure mechanism has not fully developed, then the calculation must be repeated with a larger number of steps.

Note:

The *Incremental multipliers* loading type is only available for *Safety* calculations.

7.5 Water pressure calculation

Water pressure can be 'external' water pressure (i.e. 'water load' on model boundaries) or 'internal' water pressure, which is known as pore water pressure. Pore water pressure is included in what is denoted in PLAXIS 2D as active pore pressure. Active pore pressure is composed of steady-state pore pressure and excess pore pressure, and may also include suction (positive pore water stress). In general, the steady-state component of water pressure (both the 'external' and 'internal' part) is considered to be input data and supposed to be known at the beginning of a deformation analysis, whereas excess pore pressure is the result of undrained loading or

consolidation. This section deals with the calculation of steady-state water pressure as input for a deformation analysis, which can be generated according to the options available in the **Phases** window.

7.5.1 *Calculation type*

Phreatic

The steady-state water pressure calculation by *Phreatic* is based on the input of a *Global water level* (Global water level (on page 323)) and the water conditions of the clusters (Defining water conditions (on page 53)). This generation is quick and straightforward. The following quantities are calculated as input for a deformation analysis:

- 'External' water pressures (i.e. 'water loads') on external model boundaries are calculated on the basis of the *Global water level*.
- Steady-state pore pressures in active clusters are calculated on the basis of the water conditions as defined for the corresponding clusters. If it is desired to exclude pore pressures from certain clusters, the *Dry* option should be used (Assigning water conditions to soil polygons (on page 353)) or the cluster should be *Non-porous* (General tabsheet (on page 162)).
- Steady-state pore pressures in inactive clusters are calculated on the basis of the water conditions as defined for the corresponding clusters, which forms the basis for the calculation of 'external' water pressures on boundaries between active and inactive clusters.

Steady-state pore pressure may include suction in the unsaturated zone above the phreatic surface. If it is desired to exclude suction from the steady-state pore pressure as input for a deformations analysis, the option *Ignore suction* may be used.

Groundwater flow (steady-state) [ADV]

The steady-state water pressure calculation by *Steady state groundwater flow* is based on the input of hydraulic boundary conditions (Hydraulic conditions [ADV] (on page 131) and Water pressure generation and previewing (on page 355)). It also requires the input of a non-zero permeability, in the material datasets for soil and interfaces. This generation is more time-consuming and the result is not always straightforward. The following quantities are calculated and are considered as input for a deformation analysis:

- The global water level. The 'internal' part of the global water level, is calculated as the level where the steady-state pore pressure is zero, whereas the 'external' part (if applicable) is calculated from the groundwater heads that are higher than the external model boundaries.
- 'External' water pressure (i.e. 'water load') on external model boundaries is calculated on the basis of the Global water level.
- Steady-state pore pressures in active clusters are calculated on the basis of the groundwater flow calculation. If it is desired to exclude pore pressures from certain clusters, the *Dry* option should be used (Assigning water conditions to soil polygons (on page 353)) or the cluster should be *Non-porous* (General tabsheet (on page 162)).
- Steady-state pore pressures in inactive clusters are interpolated (or extrapolated) on the basis of the water pressures at the boundaries between active and inactive clusters, as obtained from the groundwater flow calculation. These pressures also form the basis for the calculation of the 'external' water pressure on such boundaries.
- If the *Global water level* is used to define the hydraulic boundary conditions (groundwater head), the global water level will be replaced by the results of the groundwater flow calculation. In general, this will result in the same 'external' part of the global water level, but the 'internal (phreatic)' part will most likely change.

Note:

- If no groundwater flow boundary conditions are defined, the hydraulic boundary conditions for a steady-state groundwater flow calculation are taken from the *Global water level*, which means a prescribed hydraulic head below the water level and seepage above.
- The *Steady state groundwater flow* option is available for *K0 procedure*, *Gravity loading*, *Flow only*, *Plastic* and *Consolidation* calculation types.

Steady-state pore pressure may include suction in the unsaturated zone above the phreatic surface. If it is desired to exclude suction from the steady-state pore pressure, as input for a deformation analysis, the option *Ignore suction* may be used.

Groundwater flow (transient) [ULT]

This option is available if it is intended to perform ONLY groundwater flow calculations, to be indicated by selecting *Flow only* for the *Calculation type*. A transient groundwater flow calculation is based on the input of (time-dependent) hydraulic boundary conditions (Groundwater flow boundary conditions (on page 134)) and water levels (Water levels (on page 351)) A transient groundwater flow calculation requires a non-zero time interval. Moreover, it requires the input of a non-zero permeability in the material datasets for soils and interfaces and the selection of an appropriate flow model to describe the behaviour in the unsaturated zone (Groundwater tabsheet (on page 224)).

Note:

Note that the *Flow only* option is available only for the initial phase. When this option is selected for the initial phase, it will be automatically assigned for the following (child) phases as well and it will be grayed out, indicating that no change is possible.

Use pressures from previous phase

If the steady-state water pressure does not change compared to its parent phase, the option *Use pressures from previous phase* can be used to indicate that the calculation kernel should use the water pressures of the previous step (phase) as input for the current deformation analysis. This option should NOT be used if clusters change from active to inactive or vice versa. In that case the water pressures need to be generated using one of the other options.

7.5.2 Global water level

The *Global water level* can be used to generate a simple hydrostatic pore pressure distribution (*Phreatic* calculation type) for the full geometry. The global water level is by default assigned to all clusters in the geometry.

The *Global water level* can also be used to create boundary conditions for the groundwater head in the case that pore pressures are calculated on the basis of a *Groundwater flow* calculation or *Fully coupled flow-deformation analysis*.

The *Global water level* for a selected phase can be specified in the *Water* subtree under the *Model conditions* in the *Model explorer*. The drop-down menu displays the available water levels in the model (*Borehole water levels* or *User water levels*). More information on water levels is given in Water levels (on page 351).

7.6 Temperature calculation [ADV]

In case temperature (or actually a change in temperature) is of influence on the behaviour of soils or structures (for example thermal expansion or soil freezing), PLAXIS 2D offers the possibility to take thermal effects into account. The calculation of a temperature distribution in the ground is based on thermal calculations. Defining whether and how the temperature distribution should be calculated can be done by means of the *Thermal* calculation type field in the *General* section of the **Phases** window.

Note:

Note that the *Loading type* drop-down menu is NOT available for *Flow only* calculation type.

7.6.1 *Calculation type*

Ignore temperature

In the *Ignore temperature* option, no temperature distribution is calculated and the temperature effects are ignored.

Earth gradient

In this option, the (inital) temperature is calculated on the basis of the *ThermalFlow* parameters as defined in the *Model conditions* subtree of the *Model explorer* (Model explorer (on page 38)). This is NOT a finite element calculation.

Use temperatures from previous phase

In this option, the (initial) temperature distribution in the current phase is assumed to be equal to the temperature distribution (at the end) of the parent phase. This option can be used to indicate that the calculation kernel should use the temperature distribution of the previous step (phase) as input for the current deformation analysis. This option should NOT be used if clusters change from active to inactive or vice versa. In that case the temperature needs to be generated using one of the other options.

Note:

In the case the *Calculation type* is set to *Fully coupled flow-deformation* analysis, this option must be selected in order to take temperature effects into account. In this case, the temperature from the previous phase is only used to initialize the temperature at the beginning of the current phase, but it may still change during the phase as a result of a time-dependent change in thermal flow boundary conditions.

Steady state thermal flow

The temperature distribution is calculated by means of a steady-state thermal flow finite element calculation. A steady-state thermal flow calculation is based on the input of *Thermal flow boundary conditions* (Thermal

conditions [ADV] (on page 146)). It also requires appropriate thermal model parameters to be entered in the *Thermal* tabsheet of the **Materials** window (Thermal tabsheet [ADV] (on page 237)).

Transient thermal flow [ULT]

The temperature distribution is calculated by means of a transient thermal flow finite element calculation. A transient thermal flow calculation is based on the input of *Thermal flow boundary conditions* (Thermal conditions [ADV] (on page 146)). It also requires appropriate thermal model parameters to be entered in the *Thermal* tabsheet of the **Materials** window (Thermal tabsheet [ADV] (on page 237)) and a non zero time interval.

7.6.2 *Coupled analysis* [ULT]

Since groundwater flow plays an essential role in the transport of heat in the ground, thermal calculations can be coupled with groundwater flow calculations (thermo-hydraulic (TH) coupling). This requires appropriate values for the thermal constants as defined in the **Project properties** window. Moreover, results of coupled steady-state TH calculations may be used to analyse the effects of (changes of) temperature on stress and deformation (semi-coupled TH-M analysis). More generally, PLAXIS 2D allows for fully coupled transient thermo-hydro-mechanical (THM) calculations of problems in which the time-dependent effect of changes of temperature on stress, deformation and groundwater flow are to be taken into account simultaneously. Table 17 (on page 325)gives an overview of all possible combinations of thermal calculations and to what extend the three processes (T, H and M) are coupled.

Table 17: Overview of combinations for thermal calculations

Calculation type	Pore pressure calculation type	Thermal calculation type
Flow only	Phreatic	Earth gradient
		Steady-state thermal flow
		Transient thermal flow
Flow only	Steady-state groundwater flow	Earth gradient
		Steady-state thermal flow
Plastic	Phreatic	Earth gradient
		Use temperatures from previous phase
		Steady-state thermal flow
Plastic	Use pressures from previous phase	Earth gradient
		Use temperatures from previous phase
		Steady-state thermal flow

Calculation type	Pore pressure calculation type	Thermal calculation type
Plastic	Steady-state groundwater flow	Earth gradient
		Use temperatures from previous phase
		Steady-state thermal flow
Consolidation	Various	N/A
Fully coupled flow-deformation	Transient groundwater flow (automatic)	Use temperatures from previous phase
Safety	Use pressures from previous phase	Use temperatures from previous phase
Dynamic	Use pressures from previous phase	Use temperatures from previous phase

7.6.3 *Non-linearity during Phase-changes* [ADV]

The heat transfer behaviour becomes highly nonlinear when phase change (freezing/thawing) occurs. This is due to the latent heat effect, wherein a lot of energy must be added (or extracted) to change the phase of the material. This high non-linearity may create numerical instability, leading to divergence or very low convergence of the solution when using traditional iterative schemes such as Newton-Raphson or modified Newton-Raphson methods.

PLAXIS 2D addresses this non-linearity using several techniques. Firstly, a Trapezoidal Recursive Algorithm is used to compute the non-linear energy needed for phase change. Secondly, a Line Search algorithm is implemented to to detect the optimally corrected solution increment after each iteration automatically. This helps to limit the instability of the solution and thereby speed up convergence and avoid divergence due to the non-linearity.

However, for highly non-linear problems, the line search algorithm may not ensure fast convergence. To achieve convergence, modifications can be made to the numerical parameters defined in the *Numerical control parameters* subtree. The default values of *desired min number of iterations* and *desired max number of iterations* can be increased (for instance to 20 and 60 iterations respectively), and the *Max number of iterations* should be increased accordingly (to 120 for example).

7.7 Load stepping procedures

When soil plasticity is involved in a finite element calculation the equations become non-linear, which means that the problem needs to be solved in a series of calculation steps. An important part of the non-linear solution procedure is the choice of step size and the solution algorithm to be used.

During each calculation step, the equilibrium errors in the solution are successively reduced using a series of iterations. The iteration procedure is based on an accelerated initial stress method. If the calculation step is of a suitable size then the number of iterations required for equilibrium will be relatively small, usually around ten.

If the step size is too small, then many steps are required to reach the desired load level and computing time will be excessive. On the other hand, if the step size is too large then the number of iterations required for equilibrium may become excessive or the solution procedure may even diverge.

PLAXIS 2D has an automatic load stepping procedure for the solution of non-linear plasticity problems. The following procedures are available: *Load advancement ultimate level*, *Load advancement number of steps* and *Automatic time stepping*. Users do not need to worry about the proper selection of these procedures, since PLAXIS 2D will automatically use the most appropriate procedure by itself to guarantee optimum performance. The parameters for the load stepping procedures are available in the *Numerical control parameters* subtree in the **Phases** window (Figure 199 (on page 327)).

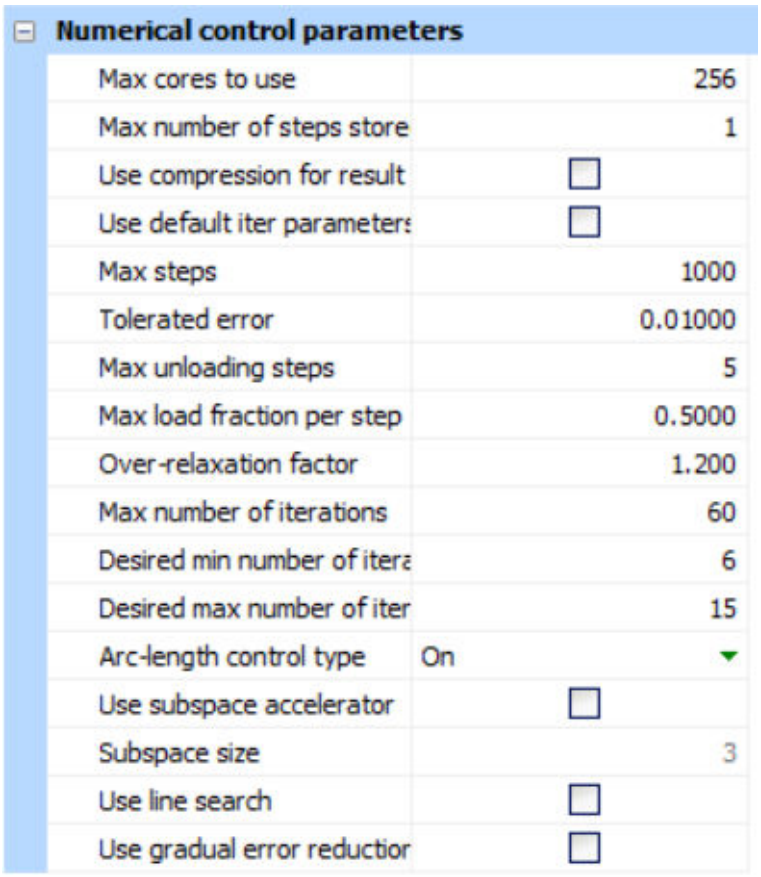


*Figure 199: Numerical control parameters subtree in the **Phases** window*

The automatic load stepping procedure is controlled by a number of calculation control parameters (Numerical control parameters (on page 337)). There is a convenient default setting for most control parameters, which strikes a balance between robustness, accuracy and efficiency. For each calculation phase, the user can influence the automatic solution procedures by manually adjusting the control parameters in the *Numerical control parameters* subtree in the **Phases** window.

In this way it is possible to have a stricter control over step sizes and accuracy. Before proceeding to the description of the calculation control parameters, a detailed description is given of the solution procedures themselves.

7.7.1 *Automatic step size procedure*

For each calculation phase the user specifies the new state or the total load that is to be applied at the end of this phase. The calculation program will compare the new situation (at the end of this phase) with the previous situation (at the end of the phase where it starts from) and will solve the difference during the current calculation phase by applying multiple load steps. In fact, the program will try to reach equilibrium for the new situation in the final load step of the current phase.

The size of the first load step in a calculation phase is automatically determined by performing trial calculations, taking into account the *Tolerated error* (Numerical control parameters (on page 337)). When a new load step is applied (first step or later steps), a series of iterations is carried out to reach equilibrium. There are three possible outcomes of this particular process. These outcomes are:

Case 1: The solution reaches equilibrium within a number of iterations that is less than the *Desired minimum* control parameter. By default, the *Desired minimum number of iterations* is 6, but this value may be changed in the *Numerical control parameters* subtree in the **Phases** window (Numerical control parameters (on page 337)). If fewer iterations than the desired minimum are required to reach the equilibrium state then the calculation step is assumed to be too small. In this case, the size of the load increment is multiplied by two and further iterations are applied to reach equilibrium.

Case 2: The solution fails to converge within a *Desired maximum number of iterations*. By default, the *Desired maximum number of iterations* is 15, but this value may be changed in the *Numerical control parameters* subtree in the **Phases** window (Numerical control parameters (on page 337)). If the solution fails to converge within the desired maximum number of iterations then the calculation step is assumed to be too large. In this case, the size of the load increment is reduced by a factor of two and the iteration procedure is continued.

Case 3: The number of required iterations lies between the *Desired minimum number of iterations* and the *Desired maximum number of iterations* in which case the size of the load increment is assumed to be satisfactory. After the iterations are complete, the next calculation step begins. The initial size of this calculation step is made equal to the size of the previous successful step.

If the outcome corresponds to either case 1 or case 2 then the process of increasing or reducing the step size continues until case 3 is achieved.

7.7.2 *Load advancement - Ultimate level*

This automatic step size procedure is used for calculation phases where a certain 'state' or load level (the 'ultimate state' or 'ultimate level') has to be reached, as in the case for a *Plastic* calculation where a construction stage is defined. The procedure terminates the calculation when the specified state or load level is reached or when soil failure is detected. By default, the *Max steps* parameter is set to 1000, but this parameter does not play an important role, since in most cases the calculation stops before the maximum number of steps is reached.

An important property of this calculation procedure is that the user specifies the state or the values of the total load that is to be applied. The size of the first load step is obtained automatically using one of the two following methods:

- PLAXIS 2D performs a trial calculation step and determines a suitable step size on the basis of this trial.
- PLAXIS 2D sets the initial load step size to be equal to the final load step size of any previous calculation.

The first method is generally adopted. The second method would only be used if the loading applied during the current load step is similar to that applied during the previous load step, for example if the number of load steps applied in the previous calculation proved to be insufficient.

In subsequent steps, the automatic load stepping procedures are adopted (Automatic step size procedure (on page 328)). If at the end of the calculation, the defined state or load level has been reached, the calculation is considered to be successful. A successful calculation is indicated by a check mark in a green circle in the *Phases explorer* and the **Phases** window.

If the defined state or load level has NOT been reached, the calculation is considered to have failed. A failed calculation is indicated by a cross mark in a red circle in the *Phases explorer* or the **Phases** window. A message describing the error is given in the **Log info for the last calculation** box in the **Phases** window:

Prescribed ultimate state not reached; Soil body collapses: A collapse load has been reached. In this case, the total specified load has not been applied. Collapse is assumed when the applied load reduces in magnitude in X successive calculation steps (where X is the maximum number of unloading steps; see Numerical control parameters (on page 337)) and the current stiffness parameter CSP is less than 0.015 (see Automatic error checks (on page 381) for the definition of CSP). It is also possible that the problem is failing but due to switched-off arc-length control, the program is not allowed to take negative step sizes. The user should check the output of the last step and judge whether the project is failing or not. In case of failure, recalculating the project with a higher *Max steps* parameter is useless.

Prescribed ultimate state not reached; load advancement procedure fails. Try manual control: The load advancement procedure is unable to further increase the applied load, but the current stiffness parameter CSP is larger than 0.015. In this case the total load specified has not been applied. The user can now attempt to rerun the calculation with slight changes to the iterative parameters in *Numerical control parameters* subtree in the **Phases** window, in particular turning off the *Arc-length control type* parameter.

Prescribed ultimate state not reached; Not enough load steps: The maximum specified number of load steps have been applied. In this case, it is likely that the calculation stops before the total specified load has been applied. It is advised to recalculate the phase with an increased value of *Max steps* or to add a similar subsequent calculation phase in which no changes in geometry configuration or water pressures are defined.

Cancelled by user: This occurs when the calculation process is terminated by clicking *Stop* in the **Active tasks** window.

Prescribed ultimate state not reached; Numerical error: A numerical error has occurred. In this case, the total specified load has not been applied. There may be different causes for a numerical error. Most likely, it is related to an input error. Careful inspection of the input data, the finite element mesh and the defined calculation phase is suggested.

Severe divergence: This is detected when the global error is increasing and has reached huge values. This error, for example, can be caused by very small time steps in a consolidation phase. The program scales down the step size when the tolerated error cannot be reached, resulting in small time steps. One of the reasons can be that a failure situation is reached. As for consolidation the arc-length procedure is not used, the program cannot really detect failure.

File xxxx not found: Such a message appears when a file that ought to exist does not exist.

Messages may indicate errors related to the iterative solution algorithm or the matrix condition. In the case of 'floating' elements (insufficient boundary conditions), one could get a message indicating that the matrix is nearly singular. Checking and improving the defined calculation phase usually solves the problem.

Note:

A list of warning and error messages appearing in PLAXIS 2D is available in Calculation warning and errors in PLAXIS (on page 602).

7.7.3 *Load advancement - Number of steps*

This automatic step size procedure always performs the number of steps specified in *Max steps* and is, in general, used for calculation phases where a complete failure mechanism should be developed during the analysis. This algorithm is therefore used in *Safety* analysis.

The size of the first step is determined by the incremental multiplier as defined for the particular calculation phase. For *Safety* calculations the *Loading type* parameter is *Incremental multipliers* and the default increment is $M_{sf} = 0.1$. This value may be changed in the *General* subtree of the **Phases** window. In subsequent steps, the automatic load stepping procedures are adopted (Automatic step size procedure (on page 328)).

If at the end of the calculation the value assigned to the *Max steps* parameter has been reached, the calculation is considered to be successful. A successful calculation is indicated by a tick mark in a green circle in the *Phases explorer* or the **Phases** window.

If the value assigned to the *Max steps* parameter has NOT been reached, the calculation is considered to have failed. A failed calculation is indicated by a cross mark in a red circle in the *Phases explorer* or the **Phases** window. A message describing the error is given in the **Log info for last calculation** box in the **Phases** window.

Cancelled by user: This occurs when the calculation process is terminated by clicking *Stop* in the **Active tasks** window.

Apart from cancellation by the user, a load advancement calculation will proceed until the number of steps defined in the *Max steps* parameter have been applied. In contrast to the *Ultimate level* procedure the calculation will not stop when failure is reached.

Note:

A list of warning and error messages appearing in PLAXIS 2D is available in Appendix H.

7.7.4 *Automatic time stepping (consolidation [ADV] and fully coupled analysis [ULT])*

Note: For analysis involving consolidation an [ADV] licence level is required. For fully coupled analysis an [ULT] licence is necessary.

When the *Calculation type* is set to *Consolidation*, the *Automatic time stepping* procedure is used. This procedure will automatically choose appropriate time steps for a consolidation analysis. When the calculation runs smoothly, resulting in very few iterations per step, then the program will choose a larger time step. When the calculation uses many iterations due to an increasing amount of plasticity, then the program will take smaller time steps.

The first time step in a consolidation analysis or a fully coupled analysis is generally based on the *First time step* parameter. This parameter is, by default, based on the advised minimum time step (overall critical time step) as described in Numerical control parameters (on page 337). The *First time step* parameter can be changed in the *Numerical control parameters* subtree in the **Phases** window when the *Use default iter parameters* checkbox is not selected. However, be careful with time steps that are smaller than the advised minimum time step.

During a *Consolidation* calculation or a *Fully coupled* analysis, arc-length control is always inactive.

7.7.5 *Automatic time stepping (Dynamics)* [ULT]

When the *Calculation type* is set to *Dynamics* or *Dynamics with consolidation*, the Newmark time integration scheme is used in which the time step is constant and equal to the critical time step during the whole analysis (Scientific Manual- *Bounds on the system response*). The maximum recommended time step for dynamics analyses is estimated in order to accurately model wave propagation and reduce error due to integration of time history functions. First, the material properties and the element size are taken into account to estimate the time step and then the time step is adjusted based on the time history functions used in the calculation. If the *Use default iteration parameters* is selected, the best combination of *Max steps* and *Number of sub-steps*, based on the estimated time step, will be calculated in the kernel. The user can always get and change these values by pressing the *Retrieve* button in the **Phases** window.

During a dynamics calculation, arc-length control is always inactive.

7.8 Calculation control parameters

The control parameters of a particular calculation phase and the corresponding solution procedure can be defined in the **Phases** window.

7.8.1 *General phase parameters*

The general properties of a phase can be defined in the *General* subtree in the **Phases** window (Figure 200 (on page 332)).

General	
ID	Phase_1
Start from phase	Initial phase
Calculation type	Plastic
Loading type	Staged construction
ΣM_{stage}	0.6000
ΣM_{weight}	1.000
Pore pressure calculation type	Phreatic
Thermal calculation type	Ignore temperature
Time interval	0.000 day
First step	7
Last step	8
Design approach	(None)
Special option	0

*Figure 200: General subtree in the **Phases** window*

Phase identification

The *ID* of the phase consists of the caption and the name (in square brackets). The name of the phases is determined consecutively by the program and it can not be modified by the user. The user may redefine the caption of the phase in the **Phases** window.

Calculation type

The calculation type of the selected phase can be defined in the corresponding drop-down menu in the **Phases** window. The options available are:

- *K0 procedure*
- *Field stress*
- *Gravity loading*
- *Flow only*
- *Plastic*
- *Consolidation*
- *Safety*
- *Dynamic*
- *Fully coupled flow-deformation*
- *Dynamic with consolidation*

A description of the different analysis types available in PLAXIS 2D is given in Types of analysis (on page 299).

Loading type

The options available for the *Loading type* parameter are:

- *Staged construction*
- *Incremental multipliers*
- *Target SumMsf*
- *Minimum excess pore pressure*
- *Degree of consolidation*

A description of the different loading types available in PLAXIS 2D is given in Loading type (on page 316).

ΣM_{stage}

ΣMstage is the total multiplier that controls the staged construction process. It is associated with the reduction of the out-of-balance forces that are to be solved in a staged construction calculation. By default, its target value is 1.0. The lowest allowed input value is 0.001. If *ΣMstage* is lower than this value, the load is considered to be negligible and no calculations take place. A value larger than 1.0 is not possible. By entering the default value of 1.0, the staged construction procedure is performed in the normal way.

In general, care must be taken with an ultimate level of *ΣMstage* smaller than 1.0, since this leads to a resulting out-of-balance force at the end of the calculation phase. Such a calculation phase must always be followed by another staged construction calculation. If *Σ Mstage* is not specified by the user, the default value of 1.0 is always adopted, even if a smaller value was entered in the previous calculation phase.

ΣM_{weight}

ΣM_{weight} is the total multiplier for the material weight. For $\Sigma M_{weight} = 1$, the material weight as specified by the unit weights in the material data sets is applied. In general, ΣM_{weight} remains at its default value of 1, but there are situations in which ΣM_{weight} may be changed:

- In the case of modelling simplified soil tests by means of finite element models, self weight of the material may be disregarded since the stresses are dominated by external loads rather than material weight. This can either be achieved by using zero unit weights in the material data set or by using $\Sigma M_{weight} = 0$ during the calculations. Note that in the former case ($\gamma = 0$) the material has no mass, which has consequences for a dynamics calculation.
- In the case of overconsolidated material in situations where the *K0 procedure* cannot be used (for example slopes), gravity loading can be used in the initial phase with an increased value of ΣM_{weight} equal to the overconsolidation ratio. In this way, the preconsolidation stress is properly initialised, at least when using advanced soil models. In the subsequent phase, ΣM_{weight} should be set back to 1.0 in order to have realistic initial soil stresses whilst the preconsolidation stress (in advanced models) will memorize the overconsolidated stress level.
- In the case of modelling a centrifuge test, ΣM_{weight} may be used to simulate the increased amount of gravity. Hence, in order to simulate a centrifuge test at 100 g, ΣM_{weight} should be set to 100.

Time interval

A non-zero value for the time-related parameters is only relevant when a transient groundwater flow calculation, a consolidation analysis, a fully coupled flow-deformation analysis or a dynamics analysis is performed or when using time-dependent material models (such as the Soft Soil Creep model).

Time interval	To define the total time period considered in the current calculation phase, expressed in the unit of time as defined in the **Project properties** window.
Dynamic time interval	To define the total time period considered in the current dynamics calculation phase, expressed in seconds [s].

A dynamics analysis uses a different time parameter than other types of calculations. The time parameter in a dynamics analysis is the *Dynamic time*, which is always expressed in seconds [s], regardless of the unit of time as specified in the **Project properties** window. In a series of calculation phases in which some of them are dynamic, the *Dynamic time* is only increased in the dynamic phases (even non-successive), while the *Dynamic time* is kept constant in other types of calculations (whether before, in-between or after the dynamic phases).

The *Dynamic time* is not affected by the regular time parameter. Reversely, the regular time parameter includes the *Dynamic time*.

7.8.2 *Deformation control parameters*

Ignore undrained behaviour

Ignore undrained behaviour excludes temporarily the effects of undrained behaviour in situations where undrained material data sets (*Undrained (A)* or *Undrained (B)*) are used. The selection of this option is associated with the selection of the *Plastic* calculation type. When the option is selected, the stiffness of water is not taken into account. As a result, all undrained material clusters (except for *Undrained (C)* materials) become temporarily drained. Existing excess pore pressures that were previously generated will remain, but no new excess pore pressures will be generated in that particular calculation phase.

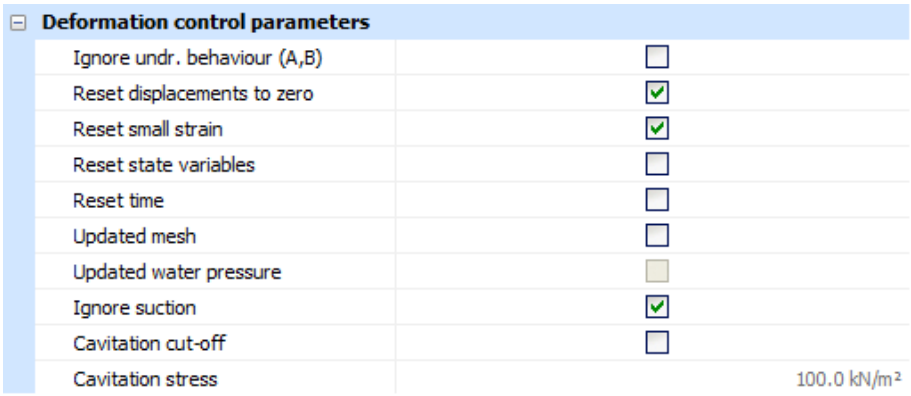


*Figure 201: Deformation control parameters subtree in the **Phases** window*

Gravity loading of undrained materials will result in unrealistic excess pore pressures. Stresses due to the self-weight of the soil, for example, are based on a long-term process in which the development of excess pore pressures is irrelevant. The *Ignore undrained behaviour* option enables the user to specify the material type from

the beginning as undrained for the main loading stages and to ignore the undrained behaviour during the *Gravity loading* stage, at least for data sets defined as *Undrained A* or *Undrained B*.

Note:

The *Ignore undrained behaviour* option is not available for a *Consolidation* analysis, *Fully coupled flow-deformation* analysis or a *Dynamics with consolidation analysis*, since these calculation types do not consider the *Drainage type* as specified in the material data sets, but use the material permeability instead.

Force fully drained behaviour of newly activated clusters

In order to use PLAXIS 2D properly for the application of consolidation of tailings and slurries in reservoirs, the user needs to be able to change the undrained/drained state of newly activated clusters (for fully coupled flow deformation or consolidation analysis).

By allowing clusters to act as undrained upon activation the user can model the gradual settlement of the tailing body correctly.

For the *Consolidation* and *Fully Coupled* calculation type, the parameter *Force fully drained behaviour newly activated clusters* is available to exclude temporarily the effects of undrained behaviour in situations where undrained material data sets (Undrained (A) or Undrained (B)) are used.

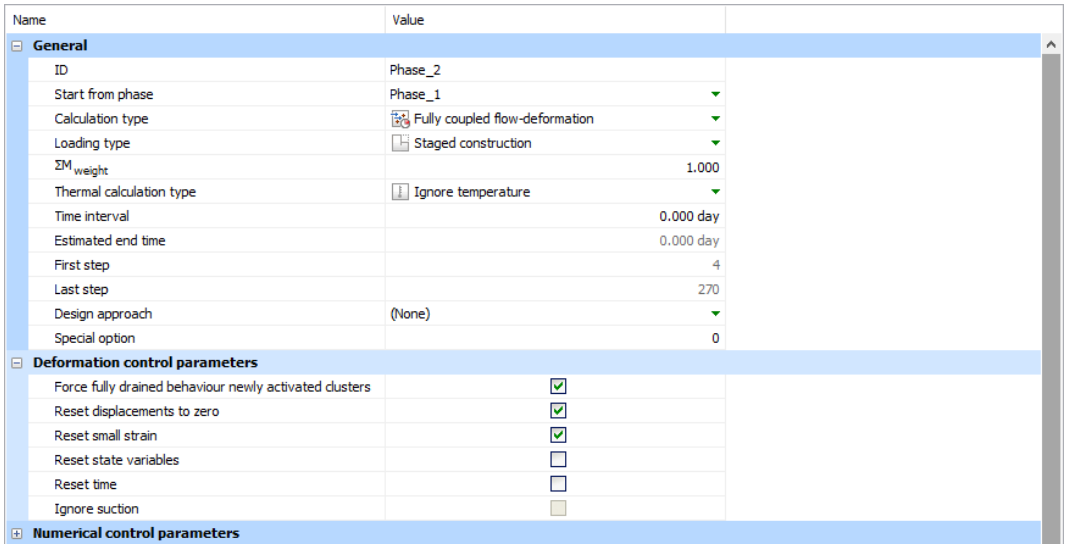


Figure 202: Deformation Control Parameters: Force Fully Drained Behaviour

By default, this parameter is switched on, which means that newly activated clusters will essentially behave as drained materials, irrespective of their drainage type setting.

This option does not affect already active clusters.

Reset displacements to zero

This option should be selected when irrelevant displacements of previous calculation steps are to be disregarded at the beginning of the current calculation phase, so that the new calculation starts from a zero displacement field. For example, deformations due to gravity loading are physically meaningless. Hence, this option may be chosen after *Gravity loading* to remove these displacements. If the option is not selected, the incremental displacements occurring in the current calculation phase will be added to those of the previous phase. The selection of the *Reset displacements to zero* option does not influence the stress field.

The use of the *Reset displacements to zero* option may not be used in a sequence of calculations where the *Updated mesh* option is used. However, if an *Updated mesh* analysis starts from a calculation where the *Updated mesh* option is not used, then the *Reset displacements to zero* option MUST be used in this *Updated mesh* analysis.

Note:

When the parameter *Reset displacements to zero* is selected, then automatically the check box for *Reset small strain* is selected. Hence if only the displacements is to be disregarded, then the box for *Reset small strain* should be unchecked.

Reset small strain

This option should be selected when an existing state of the small-strain tensor in the Hardening Soil model with small-strain stiffness, as reached at the end of the previous phase, is to be disregarded. As a result, the soil will behave as a 'virgin soil' without strain history. Other state parameters are not affected by this option. If it is the user's intention to reset all state parameters, then the option *Reset State Variables* should be used.

Reset time

The *Reset Time* option can be used in calculation phases to reset the *Time* parameter to zero. This applies to the regular *Time* as well as to *Dynamics Time*. As a result, the new phase will start from t = 0 [days]. Resetting the time does not have consequences for existing stresses, deformations or dynamic motions, and it does not influence time-dependent processes like consolidation or creep. Resetting time mainly affects history curves where *Time* or *Dynamics Time* is used on one of the axes.

As an example of where this option might be used is if the Soft Soil Creep model is used to set up the initial stress state using *Gravity Loading* (with a non-zero time interval). Similar as for the option *Reset displacements to zero*, it might be desired to reset time to zero and to start the new project at t = 0 [days]. This will give a clear history curve that starts from zero.

Reset state variables

This option should be selected when the values of the state parameters in advanced soil models, as reached at the end of the previous phase, are to be disregarded. As a result, the soil will behave as 'virgin' soil. The reset parameters are given in Table 18 (on page 336).

Table 18: The material models and the corresponding reset parameters

Material model	Parameter
Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model, Hardening Soil model, Hardening Soil model with small-strain stiffness	p_p reset to the OCR value from the **Initial** tab of the material set
Hardening Soil model with small-strain stiffness	Strain history tensor reset to zero
Hardening Soil model, Hardening Soil model with small-strain stiffness	Mobilised shear reset to current effective stress state

Note:

Note that resetting the OCR only adjust the preconsolidation pressure and not the current stresses. Therefore the user should be aware that, after resetting the OCR, the current stress state (particularly the ratio between horizontal and vertical stresses) may not be representative for the new OCR.

Updated mesh

This option must be selected when the calculation should be performed as a large deformation analysis according the *Updated Lagrange formulation*. For a more detailed information see Updated mesh analysis (on page 314).

Updated water pressure

This option can only be selected when the calculation is performed as a large deformation analysis according the *Updated Lagrange formulation*. It is used to recalculate the steady-state pore pressure at the updated position of the stress points. For more detailed information see Updated mesh analysis (on page 314).

Ignore suction

When steady-state pore pressures are generated by the phreatic level option or by a groundwater flow calculation, tensile pore water stresses (suction) will be generated above the phreatic level. Although suction is a realistic phenomenon, the use of suction in a deformation analysis may lead to an increased shear strength when effective strength parameters are used for the soil. In order to avoid this, suction can be cut off by selecting the *Ignore suction* option. Note that the *Ignore suction* option does not affect the excess pore pressures generated in undrained soil layers, except in the case of a fully coupled flow-deformation analysis. The default setting is such that *Ignore suction* is selected.

When *Ignore suction* is selected, soil below the phreatic level is considered to behave fully saturated, whereas soil above the phreatic level is considered to behave as dry soil, although excess pore pressures may still occur here. Positive values of steady-state pore stresses will be set to zero. However, any excess pore pressure above the phreatic level, both positive and negative, will be taken into account. This requires the effective degree of saturation, S_{eff}, to be set to 1. Hence, positive excess pore stresses will always be taken into account to their full extent until the cavitation cut-off is reached, except in the case of a consolidation analysis.

When *Ignore suction* is NOT selected, suction is allowed and included in the pore water stresses (both steady state and excess pore stresses). In this case the effective degree of saturation, S_{eff}, determines the proportion of the suction that is included in the active pore stresses. This depends on the soil-water retention curve as defined in the material data set of the soil layer.

Cavitation cut-off

In case of unloading of undrained materials (undrained A or B) tensile excess pore stresses may be generated. These excess pore stresses might give rise to tensile pore water stresses. In case the cavitation cut-off option is activated, excess pore pressures are limited so that the tensile pore water stress is never larger than the cavitation stress. By default, the cavitation cut-off option is not activated. If it is activated, the default cavitation stress is 100 kN/m^2. The *Cavitation cut-off* option is not available for a consolidation analysis or a fully coupled flow-deformation analysis.

7.8.3 *Numerical control parameters*

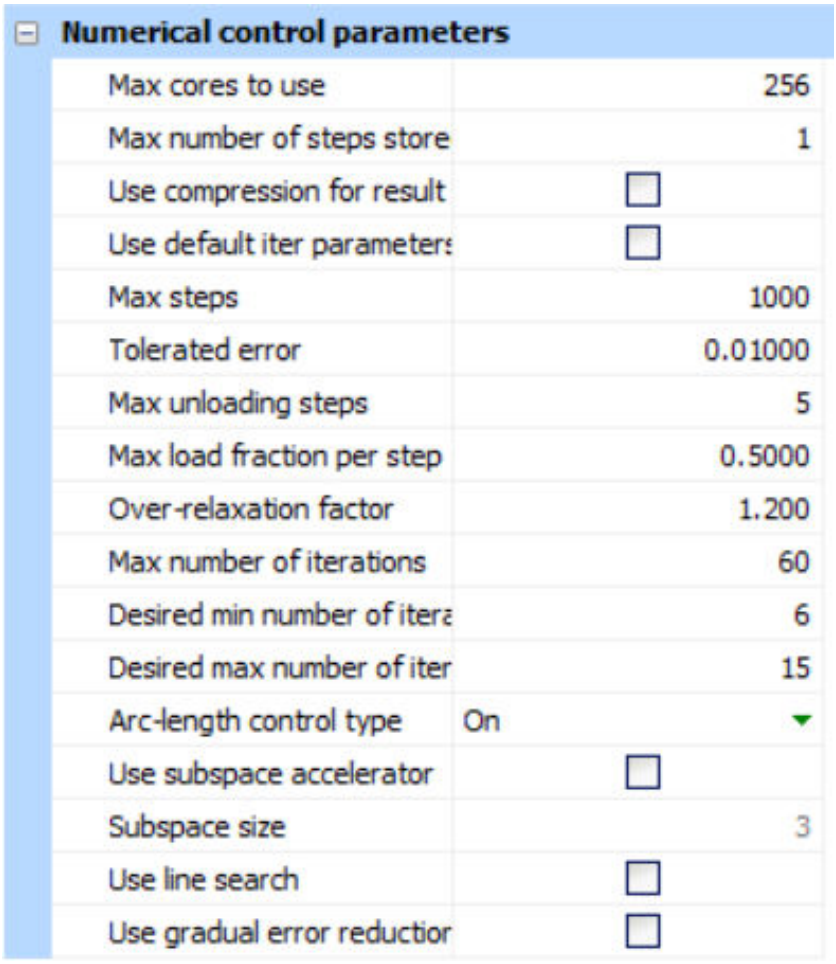


*Figure 203: Numerical control parameters subtree in the **Phases** window*

Max cores to use

The number of cores to be used by the solvers in the calculation process can be defined in the *Numerical control parameters* subtree of the **Phases** window.

Note:

- For PLAXIS 2D - licence the **max core** usage is limited to **1**.
- For PLAXIS 2D - [ADV] or [ULT] licence the **max number** of cores can be adjusted up to **256**.

Max number of steps stored

The *Max number of steps stored* parameter defines the number of steps of a calculation phase to be stored. In general, the final output step contains the most relevant result of the calculation phase, whereas intermediate steps are less important.

The final step of a calculation phase is always stored.

In case the *Max number of steps stored* is larger than 1, intermediate steps are stored when the corresponding calculation control parameter has passed another 'Interval' of the calculation phase. The 'Interval' is defined by the difference between the start value and the target value of the control parameter, divided by the number of steps stored.

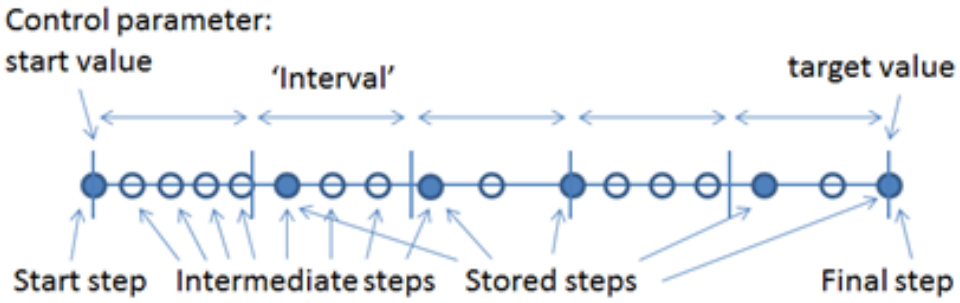


Figure 204: Max number of steps stored

The calculation control parameter and the 'Interval' depend on the type of calculation which is described in table below (Figure 204 (on page 339)).

Table 19: Calculation control parameters

Calculation type	Control parameter	'Interval'
Gravity loading	*ΣMweight*	target *ΣMweight* / *Max number of steps stored*
Plastic calculation	*ΣMstage*	target *ΣMstage* (usually 1) / *Max number of steps stored*
Consolidation (staged construction) and *Fully coupled analysis*	Time	Time interval / *Max number of steps stored*
Consolidation (min. pore pressure, degree of consolidation) and *Safety analysis*	Step number	Applied number of steps / *Max number of steps stored*
Safety analysis (target ΣMsf)	*ΣMsf*	(*ΣMsf* - 1) / *Max number of steps stored* (in this case the last step - 1 is also stored)
Dynamics analysis	Dynamics time	Dynamics time interval / *Max number of steps stored*

If a calculation phase does not finish successfully, as indicated by a red cross in front of the phase, then up to 50 steps of that phase will be stored. This enables a stepwise evaluation of the cause of the problem.

Note:

When it is intended to create animations of the results (e.g. in the case of transient calculations), the *Max number of steps stored* parameter should be set to a higher value.

Use compression for result files

Enabling the *Use compression for result files* option . Compression reduces file size, but causes a small performance overhead when opening the project in Output.

Iterative procedure control parameters

The iterative procedures, in particular the load advancement procedures, are influenced by some control parameters. These parameters can be set in the *Numerical control parameters* subtree in the **Phases** window. PLAXIS 2D has an option to adopt a default iterative procedure, which in most cases leads to a good performance. Users who are not familiar with the influence of the control parameters on the iterative procedures are advised to select the *Use default iter parameters*.

In some situations, however, it might be desired or even necessary to change the standard setting. In this case the user should deselect the *Use default iter parameters* and change the desired parameters. A description of the parameters is given below.

Max steps

This parameter specifies the maximum number of calculation steps (load steps) that are performed in a particular calculation phase.

If *Plastic*, *Consolidation* or *Fully coupled flow-deformation* is selected as the calculation type, then the *Max steps* parameter should be set to an integer number representing the upper bound of the required number of steps for this calculation phase. In this case the defined value is an upper bound to the actual number of steps that will be executed. In general, it is desired that such a calculation is completed within the defined number of steps and stops when either the prescribed ultimate state is reached or the soil body collapses. If such a calculation reaches the maximum number of steps, it usually means that the ultimate level has not been reached. By default, the *Max steps* parameter is set to 1000, but this number can be changed within the range 1 to 10000.

If *Safety* or *Dynamics* is selected as the calculation type, then the number of steps is always exactly executed. In general, it is desired that such a calculation is completed within the defined number of steps and stops when either the prescribed ultimate state is reached or the soil body collapses. If such a calculation reaches the maximum number of steps, it usually means that the ultimate level has not been reached. By default, the *Max steps* parameter is set to 100 for both *Safety* and *Dynamics* calculations which is generally sufficient to complete the calculation phase. However, this number may be changed within the range 1 to 10000.

Time step determination

This parameter determines how the time step parameter is calculated in a *Consolidation*, a *Fully coupled flow-deformation* or a *Dynamics* analysis. The user can select either the automatic procedure or specify the parameters manually.

First time step

The *First time step* is the increment of time used in the first step of a consolidation analysis (Automatic time stepping (consolidation [ADV] and fully coupled analysis [ULT]) (on page 330)). By default, the first time step is equal to the overall critical time step. The overall critical time step is calculated based on the material properties and the geometry of the model. For a more detailed description see the Scientific Manual - *Consolidation theory - Critical Time Step*. The *Min time step* and *Max time step* parameters are the minimum and maximum time step values.

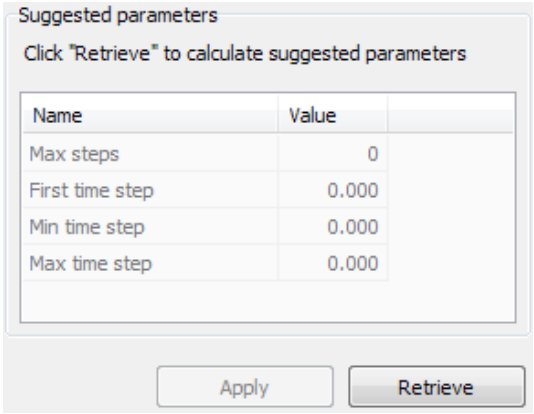


Figure 205: Dialog box displaying the retrieved value for the First time step parameter

To retrieve the value suggested by the program click the cell next to the *First time step* parameter. The value is displayed when the *Retrieve* button is clicked. The value is used when the *Apply* button is clicked and the *Time step determination*.

Sub steps in *Dynamics* calculation [ULT]

The time step used in a *Dynamics* calculation is constant and equal to $\delta t = \Delta t / (m \cdot n)$, where Δt is the duration of the dynamic loading (*Dynamics time interval*), m is the value of *Max steps* and n is the value of the *Number of sub steps* parameter. The result of the multiplication of the *Max steps* number (m) and the *Number of sub steps* number (n) gives the total number of steps to be used in the time discretisation. It is important to define a proper number of steps such that the dynamics signal used in dynamic loading is properly covered.

The *Max steps* parameter specifies the number of the steps that are stored which can be used in plots in the Output program. A higher value of the *Max steps* parameter provides more detailed plots and animations, however the processing time required by the Output program is increased as well.

In general, the total number of steps, which is the product of the values defined for the *Max steps* and *Number of sub steps* parameters, should be the same as the total number of data points used in the dynamics calculation. If the *Time step determination* is set to default option *Automatic*, PLAXIS 2D automatically calculates the proper number of steps and sub steps based on the material, mesh and number of data points in all active time history function (dynamics multipliers). In this case, the *Number of sub steps* parameter is automatically calculated in the kernel. When the option *Semi-automatic* is selected, the *Number of sub steps* will be automatically calculated inside the kernel, based on the *Max steps* as given by the user. When the *Manual* option is selected, both *Max steps* and *Number of sub steps* will not be automatically calculated inside the kernel but must be provided by the user.

To be able to see and change these two parameters, the user has the possibility to retrieve them by pressing the *Retrieve* button. By pressing the *Apply* button, these values will be applied and will be used by kernel without any change. It should be noted that PLAXIS 2D always tries to find the closest number of steps to the *Max steps*

that the user specifies (in case of using the default option, the number of steps will be the closest number to 100 as shown in Figure 206 (on page 342)).

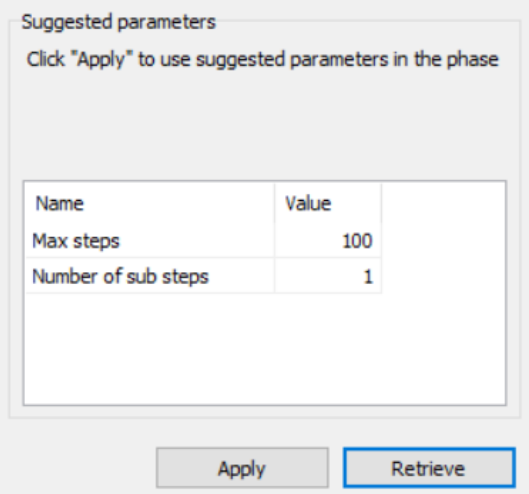


Figure 206: Dialog box displaying the retrieved value for the Number of sub steps parameter

Tolerated error

In any non-linear analysis where a finite number of calculation steps are used there will be some drift from the exact solution, as shown in Figure 207 (on page 343). The purpose of a solution algorithm is to ensure that the equilibrium errors, both locally and globally, remain within acceptable bounds (Automatic error checks (on page 381)). The error limits adopted in PLAXIS 2D are linked closely to the specified value of the *Tolerated error*.

Within each step, the calculation program continues to carry out iterations until the calculated errors are smaller than the specified value. If the tolerated error is set to a high value then the calculation will be relatively quick but may be inaccurate. If a low tolerated error is adopted then computer time may become excessive. In general, the standard setting of 0.01 is suitable for most calculations.

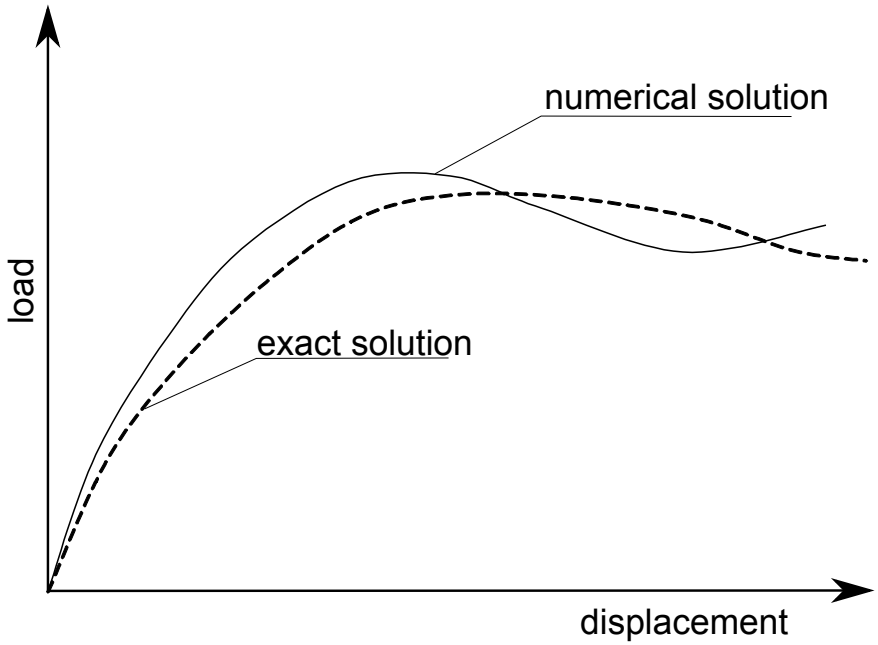


Figure 207: Computed solution versus exact solution

Note:

Be careful when using a tolerated error larger than the default value of 0.01, as this may give inaccurate results which are not in equilibrium.

If a calculation gives failure loads that tend to reduce unexpectedly with increasing displacement, then this is a possible indication of excessive drift of the finite element results from the exact solution. In these cases, the calculation should be repeated using a lower value of the tolerated error. For further details of the error checking procedures used in PLAXIS 2D see Automatic error checks (on page 381).

Most Validation examples have been performed with a tolerated error lower than the default settings.

Maximum unloading steps

This value gives the number of steps that PLAXIS 2D allows before declaring that the soil body has collapsed. The default value is 5. However, sometimes, in the case of a changing mechanism the load level may temporarily decrease in order to pick up another mechanism after which the load can still increase. For example in uplift problems there is a calculation stage when the top soil structure exhibits buckling behaviour (that lasts longer than 5 steps), which results in a small unloading. This small unloading is not a collapse of the whole model, as the final loading finally grows above the previous levels. Hence, the user might decide to set this value to a higher number and later judge on the presence of collapse based on the loading curve.

Max load fraction per step

This value controls the size of the load step in staged construction. Since this is a fraction, it determines what maximum part of the stage can be solved in one step. For instance, a value of 0.5 (default) means that the applied load or unbalance will be solved at least in 1/0.5 = 2 steps. More steps are possible if convergence is slow, but not less. The user might want to use small values (like 0.02 to force at least 50 steps) in order to observe the kinetics of the deformation process, prevent divergence in case of high nonlinearity or in combination with Gradual Error reduction.

Over-relaxation factor

To reduce the number of iterations needed for convergence, PLAXIS 2D makes use of an over-relaxation procedure as indicated in Figure 210 (on page 347). Over-relaxation is a deliberate over-estimation of the equilibrium error under the assumption that by just solving the exact unbalance, the next iteration is still far from equilibrium. The purpose of this is to reduce the number of the iterations in order to speed up the calculation. The parameter that controls the degree of over-relaxation is the over-relaxation factor. The theoretical upper bound value is 2.0, but this value should never be used. For low soil friction angles, for example $\varphi < 20°$, an over-relaxation factor of about 1.5 tends to optimise the iterative procedure. If the problem contains soil with higher friction angles, however, then a lower value may be required. The standard setting of 1.2 is acceptable in most calculations.

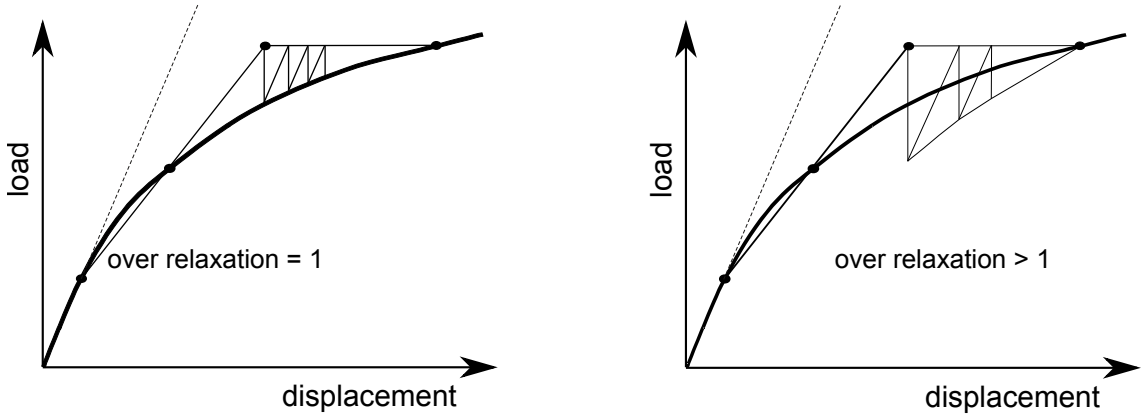


Figure 208: Influence of over-relaxation

Maximum number of iterations

This value represents the maximum allowable number of iterations within any individual calculation step. In general, the solution procedure will restrict the number of iterations that take place. This parameter is required only to ensure that computer time does not become excessive due to errors in the specification of the calculation. The standard value of *Maximum number of iterations* is 60, but this number may be changed within the range 1 to 100.

If the maximum allowable number of iterations is reached in the final step of a calculation phase, then the final result may be inaccurate. If this is the case then the message '*Maximum iterations reached in final step*' is displayed in the *Log info for last calculation* box of the **Phases** window. Such a situation occasionally occurs when the solution process does not converge. This may have various causes, but it mostly indicates an input error.

Desired min and desired max number of iterations

PLAXIS 2D makes use of an automatic step size algorithm (Automatic error checks (on page 381)). This procedure is controlled by the two parameters *Desired min number of iterations* and *Desired max number of iterations*, specifying the desired minimum and maximum number of iterations per step respectively. The default values of these parameters are 6 and 15 respectively, but these numbers may be changed within the range 1 to 100. For details on the automatic step size procedures see Automatic step size procedure (on page 328).

It is occasionally necessary for the user to adjust the values of the desired minimum and maximum from their standard values. It is sometimes the case, for example, that the automatic step size procedure generates steps

that are too large to give a smooth load-displacement curve. This is often the case where soils with very low friction angles are modelled. To generate a smoother load-displacement response in these cases, the calculations should be repeated with smaller values for these parameters, for example:

Desired minimum = 3 Desired maximum = 7

If the soil friction angles are relatively high, or if high-order soil models are used, then it may be appropriate to increase the desired minimum and maximum from their standard values to obtain a solution without the use of excessive computer time. In these cases the following values are suggested:

Desired minimum = 8 Desired maximum = 20

In this case it is recommended to increase the *Maximum iterations* to 80.

Arc-length control

The *Arc-length control* procedure is a method that is by default selected in PLAXIS 2D to obtain reliable collapse loads for load-controlled calculations. The iterative procedure adopted when arc-length control is not used is shown in Figure 209 (on page 345) for the case where a collapse load is being approached. In the case shown, the algorithm will not converge. If arc-length control is adopted, however, the program will automatically evaluate the portion of the external load that must be applied for collapse as shown below.

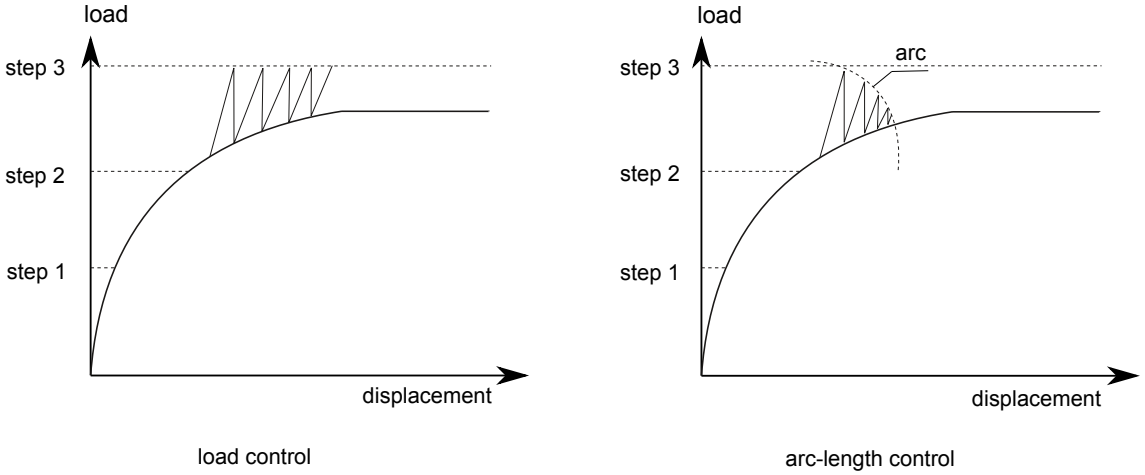


Figure 209: Influence of Arc-length control

Arc-length control is activated by selecting the corresponding check box in the *Numerical control parameters* subtree. The arc-length control procedure should be used for load-controlled calculations, but it may be deactivated, if desired, for displacement-controlled calculations.

Setting the *Arc-length control type* parameter to *Auto* enables the procedure only when the CSP (current stiffness parameter) is less than 0.5. In this way arc-length is only used when the material exhibits significant plastic behaviour.

Use subspace accelerator

A common way to solve the system of equations describing the relation between forces and displacement, $Ku = F$, is through a Newton-Raphson iteration that finds the solution of the equation $f(u) = F - Ku = 0$. In case the stiffness matrix K is constant and the material behaviour is elastic, only 1 iteration is necessary. But generally material behaviour is not elastic and K is for instance stress dependent, in which multiple iterations are required.

For iteration i in a certain calculation step, the displacements are found using the Newton-Raphson iteration:

$u_i = u_{i-1} - f(u_{i-1})/f'(u_{i-1})$

If the stiffness matrix K is stress dependent, it should be recalculated for every iteration in order to get the most accurate results and this is exactly what the traditional Newton-Raphson method does. However, making a new stiffness matrix is quite time consuming, which makes the traditional Newton-Raphson method quite slow.

The modified Newton-Raphson method on the other hand only determines a new stiffness matrix at the beginning of new calculation step without updating it per iteration. This considerably speeds up the calculation, but if during 1 calculation step the stress or strain state changes a lot due to strong non-linear behaviour, the solution reached in the iterations can deviated from the real solution which then will result in slower convergence and thus undoing the positive effect of the modified Newton-Raphson method.

Scott & Fenves (2010) describe a new accelerated Newton-Raphson algorithm that is based on the modified Newton-Raphson algorithm. In this accelerated algorithm, for every iteration the results are additionally improved by using the displacement increments from a limited number of previous iterations. The effect of the acceleration is the largest for those nodes that had the largest incremental displacements in the previous iterations, which are the nodes that had the highest non-linearity. Hence, the acceleration procedure has the most effect on those nodes that would have the largest deviation from the real solution in a modified Newton-Raphson iteration.

The calculation gathers the displacement increment vectors from previous iterations in a special matrix that is a so-called "Krylov subspace", hence the name *Subspace acceleration*.

The amount of previous iterations that is taken into account for the acceleration is the *Subspace size* parameter. Taking into account more results from previous iterations improves the accuracy of the results which then reduces the amount of iterations needed, but on the other hand the acceleration procedure itself also requires memory and calculation time and so there is a balance between how much the acceleration procedure can improve the convergence vs the calculation time for the acceleration procedure itself. In order to reduce the memory and calculation time needed, the *Subspace size* is usually chosen in the range of 3 - 6. Based on experience, PLAXIS uses a default value of 3, which means the displacements increments from the last 3 iterations are used to accelerate the current iteration.

In PLAXIS subspace acceleration can be successfully applied to solve nonlinear equations of geotechnical problems especially for computationally challenging situations when there are strong non-linearities involved such as drastic changes in the load-resisting characteristics of the geotechnical model components like development of local failure mechanisms, soil freezing/thawing, liquefaction and complex couplings among different physics.

Note: Subspace accelerator is not available for calculation types K_0 and *field stress*.

Use line search

This method aims to improve the convergence of non-linear problems in a relatively inexpensive way. It can also be seen as an improvement on the over-relaxation method that exists in PLAXIS 2D. In its essence line search tries to scale the correction of the solution increment that is calculated every iteration but the scaling parameter is not fixed as in the over-relaxation method but calculated in a special way to bring the whole system closer to equilibrium. In general, whereas *Subspace acceleration* is best used in situations with major non-linearity, the usage of *line search* is advised for problems without severe non-linearity, like serviceability calculations. Critical state calculations, like footing failure or embankment failure generally do not benefit from line search with respect to calculation time.

Use gradual error reduction

Gradual error reduction is a simple numerical trick that aims to speed up the calculation in case of very slow convergence, as it happens sometimes for non-associated plastic flow. It works as follows: initially the tolerated error (the one that controls the convergence of the nonlinear calculations) is increased 10-fold and then, by the end of the phase, it linearly reduces to the error as defined by the *Tolerated error* parameters. Hence, when using the default error of 0.01, the calculation will start with 0.1 and end with 0.01. The rationale behind is that in case of non-associative plastic flow one can obtain an infinite number of alternative (but still correct) solutions with the same tight tolerance, but with, for instance, different step size. Hence, is it not really necessary to enforce a tight tolerance during the whole calculation, as a feasible solution is located in a certain band, defined by all alternative paths. When the gradual error reduction procedure is enabled, the solution does not follow one of the paths, but, possibly, jumps from one to another. However, in the end of the calculation the error is forced to reduce to the levels, as defined for the calculation phase.

Nonetheless, application of this procedure should be done with caution. If the initial tolerated error is too high, hypothetically, the solution can switch to a path that is not even connected to the current initial conditions (hence infeasible). One way to prevent this is to use much smaller allowable *Max load fraction per step* than the default value used in PLAXIS 2D. The default maximum step size is 50% of the whole stage load. The user might consider reducing the step size to something like 2% or less, hence requiring at least 50 steps (100% / 2%) to obtain the solution of the calculation phase.

Extrapolation

It is a numerical procedure, which is automatically used in PLAXIS 2D if applicable, when a certain loading that was applied in the previous calculation step is continued in the next step. In this case, the displacement solution to the previous load increment can be used as a first estimate of the solution to the new load increment. Although this first estimate is generally not exact (because of the non-linear soil behaviour), the solution is usually better than the solution according to the initial stress method (based on the use of the elastic stiffness matrix) (See Figure below).

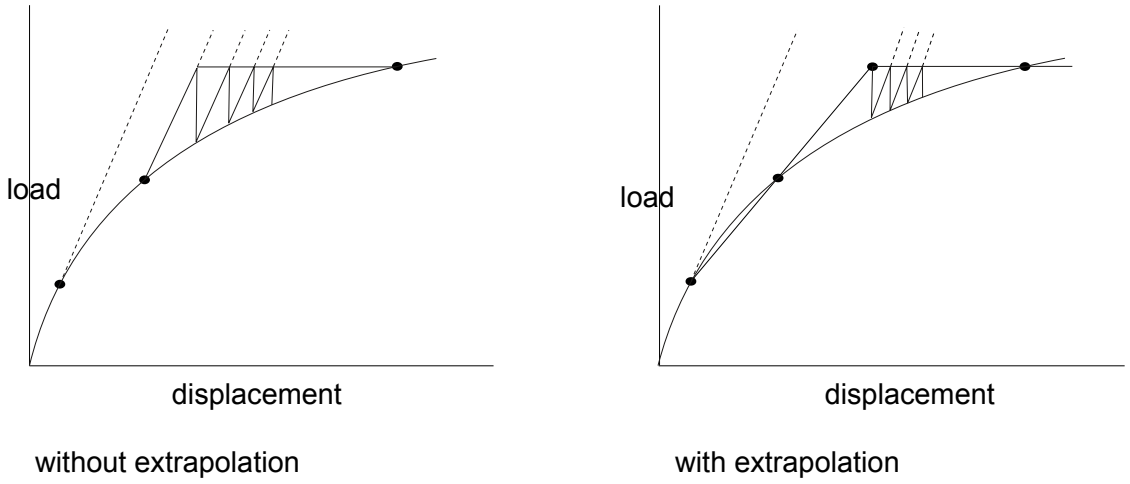


Figure 210: Difference between elastic prediction (a) and extrapolation from previous step (b)

After the first iteration, subsequent iterations are based on the elastic stiffness matrix, as in the initial stress method (Zienkiewicz, 1977 (on page 532)). Nevertheless, using *Extrapolation* the total number of iterations needed to reach equilibrium is less than without extrapolation. The extrapolation procedure is particularly useful when the soil is highly plastic.

7.8.4 *Flow control parameters* [ADV]

*Figure 211: Flow control parameters subtree in the **Phases** window*

Iterative procedure

Groundwater flow, optionally coupled with thermal flow, involves extra parameters to control the iterative procedure to solve the corresponding system of equations. For *Steady state flow* calculations two extra parameters are available, namely *Tolerated error* and *Over-relaxation factor*. For *Transient flow* a higher number of parameters are required to define the iterative procedure.

Max steps

This parameter specifies the maximum number of calculation steps that are performed in a phase in which the *Pore pressure* calculation type is set to *Steady state groundwater flow* or *Transient groundwater flow* or the *Thermal* calculation type is set to *Steady state thermal flow* or *Transient thermal flow*. By default, the *Max steps* parameter is set to 1000.

Tolerated error

Similar to the *Tolerated error* option in deformation analyses (Numerical control parameters (on page 337)), the *Tolerated error* option in flow calculations is used to check if the result is accurate enough. In the case of unsaturated groundwater flow, checking the local error (error which occurs in a few elements) is also needed. In this case, due to the fact that the permeability can change significantly, it is possible to have isolated water in some elements in unsaturated zones. The specified *Tolerated error* is used for checking both the global flow error and local errors. PLAXIS 2D used higher *Tolerated error* to check the local flow errors (usually 10 times larger). If both global and local errors are less than the *Tolerated error*, the groundwater flow calculation is terminated.

Over-relaxation factor

Since the unsaturated flow analyses may be highly non-linear, PLAXIS 2D uses an over-relaxation factor to speed up the steady state calculations and to decrease the number of steps needed. This factor is by default 1.5. For confined flow analyses which are linear a factor of 1 is enough. In case of highly non-linear analyses for which PLAXIS 2D needs more steps, higher values than 1.5 might result in faster convergence. The maximum possible value is 2.0.

7.8.5 *Dynamics control parameters* [ULT]

The parameters controlling a *Dynamics* and *Dynamics with consolidation* analysis can be defined in the *Dynamics control parameters* subtree in the **Phases** window.

Newmark alpha and beta

The Newmark alpha and beta parameters determine the numerical time-integration according to the implicit Newmark scheme. In order to obtain an unconditionally stable solution, these parameters must satisfy the following conditions:

$$\text{Newmark } \beta \geq 0.5 \text{ and Newmark } \alpha \geq 0.25\ (0.5 + \beta)^2$$

Dynamic control parameters	
Alpha - Newmark time integration	0.2500
Beta - Newmark time integration	0.5000
Mass matrix	0.000

*Figure 212: Dynamics control parameters subtree in the **Phases** window*

For an average acceleration scheme you can use the standard settings ($\alpha = 0.25$ and $\beta = 0.5$). Using a higher β-value and corresponding α-value results in a damped Newmark scheme (e.g. $\alpha = 0.3025$ and $\beta = 0.6$).

Mass matrix

In dynamics calculations, PLAXIS 2D uses, by default, a lumped mass matrix, i.e. a mass matrix that contains only diagonal terms in which the contributions of the off-diagonal terms are incorporated. This provides an efficient dynamics calculation process, but it may lead to slightly inaccurate results. For example, results of block waves in soil columns may show small oscillations as well as some dispersion. This can be improved by using a consistent mass matrix instead of a lumped mass matrix.

The *Mass matrix* parameter can be used to distinguish between a lumped mass matrix (*Mass matrix* parameter = *0*) and a consistent mass matrix (*Mass matrix* parameter = *1*). In fact, any intermediate value between a lumped mass matrix and a fully consistent mass matrix can be entered for dynamics calculations (*0≤ Mass matrix ≤ 1*), allowing for a mixed matrix to be used.

7.8.6 *Reached values*

*Figure 213: Reached values subtree in the **Phases** window*

Reached total time	It is the actual accumulated time at the end of a finished calculation phase.
CSP - Relative stiffness	It is the relative stiffness parameter calculated at the end of the phase, which is a measure for the amount of plasticity that occurs during the calculation (Automatic error checks (on page 381)). A value of 1.0 means that the full model is elastic; a value close to zero indicates that failure is approached.
Reached total force	They are the component of the resulting reactions on the nodes where non-zero prescribed displacement is applied.
Pmax - Reached max pp	It is the value of the maximum pore pressure that has been actually reached at the end of a calculation phase. This value corresponds to the maximum excess pore pressure for a *Consolidation* analysis, the steady state pore pressure for a *groundwater flow* analysis and the maximum active pore pressure for a *Fully coupled flow-deformation* analysis.
ΣM_{stage} - Reached phase proportion	It is an indication of the proportion of the unbalance solved in a phase where the *Staged construction* loading type is applied (Staged construction (on page 317)).
ΣM_{weight} - Reached material weight proportion	It is an indication of the total proportion of the material weight applied in a calculation (General phase parameters (on page 331)). A reached value of 1.00 means that the specified weight of soil and structural materials is fully applied.
ΣM_{sf} - Reached safety factor	It is the resulting value of the ΣM_{sf} parameter in a *Safety* phase (Safety calculation (phi/c reduction) (on page 307)).

7.9 Flow conditions

PLAXIS 2D is generally used for effective stress analysis in which total stresses are divided into effective stresses, σ', and active pore pressures, p_{active}.

$$\sigma = \sigma' + p_{active} \qquad Eq.\ [88]$$

Active pore pressure (p_{active}) is defined as the effective saturation, S_{eff}, times the pore water pressure, p_{water}.

$$p_{active} = S_{\text{eff}} \cdot p_{water} \qquad Eq.\ [89]$$

Pore water pressure differs from active pore pressure when the degree of saturation is less than unity. PLAXIS 2D can deal with saturated soil below the phreatic level, as well as with partially saturated soil above the phreatic level. More details about partially saturated soil behaviour are provided in Groundwater tabsheet (on page 224).

In the pore water pressure a further division is made between steady state pore pressure, p_{steady}, and excess pore pressure, p_{excess}.

$$p_{water} = p_{steady} + p_{excess} \qquad Eq.\ [90]$$

Excess pore pressures are pore pressures that occur as a result of stress changes in undrained materials. In this respect, changes in stress may be a result of loading, unloading, a change in hydraulic conditions or consolidation. Hence, excess pore pressures are a result of a deformation analysis. In a *Plastic* calculation, a *Safety analysis* or a *Dynamics analysis*, excess pore pressures can occur in clusters for which the *Drainage type* is set to *Undrained A* or *Undrained B*. In *Consolidation analysis*, *Dynamics with consolidation analysis* and *Fully coupled flow-deformation analysis*, excess pore pressures can occur in any material (except *Non-porous materials*), depending on the permeability as defined in the corresponding material data set. In the case of a *Fully coupled flow-deformation analysis*, excess pore pressure is calculated by subtracting the steady-state pore pressure, p_{steady}, from the pore water pressure, p_{water}.

Steady-state pore pressures are pore pressures that represent a stable state. Since steady-state pore pressures are not supposed to change during a deformation analysis, they are considered input data. There are various ways to define and generate steady-state pore pressures. The way to indicate how pore pressures are generated for a particular calculation phase is indicated by means of the *Pore pressure calculation type* parameter in the **Phases** window. In contrast to other deformation calculation types, a *Fully coupled flow-deformation* analysis does not allow for the selection of the *Pore pressure calculation type*, because in this case total pore water pressures, p_{water}, are calculated together with displacements. To enable a division in steady-state pore pressures and excess pore pressures, the steady-state pore pressures, p_{steady}, are automatically calculated on the basis of a preliminary steady-state groundwater flow calculation using the hydraulic boundary conditions at the end of the calculation phase. This enables the calculation and output of excess pore pressures, p_{excess}, in all steps:

$$p_{excess} = p_{water} - p_{steady} \qquad Eq.\ [91]$$

The definition of water conditions is needed to generate pore pressures in the soil and to generate external water pressures, if applicable. The definition of water conditions can be divided into the creation of water levels and assigning water conditions to polygon clusters. These features will be described in more detail below.

Note: Analysis involving consolidation analysis, steady-state groundwater flow or changes in hydraulic conditions the minimum required is an [ADV] licence. For dynamics analysis, transient flow or fully coupled calculations an [ULT] licence needs to be active.

7.9.1 *Water levels*

Water levels can either be generated according to the information specified in boreholes (Defining water conditions (on page 53)) or can be defined in the **Flow conditions** mode. A water level can be used to generate external water pressures (for water levels outside the model) and to generate pore pressures in soil layers. In the latter case the water level can act as a phreatic level in partially saturated soil layers as well as a pressure head level in aquifer layers. Water levels created in the model are grouped under the *Water levels* subtree in the

Attributes library in the *Model explorer*. Due to the global nature of the water levels, any change to an existing water level in the model will affect it in all the phases. If a water level in a time-dependent phase has the same geometry as previously defined but it varies with time, a flow function needs to be assigned to it. Moreover, it is required to create a copy of a water level with the same geometry. To create a copy of the water level, right-click the existing water level in the *Model explorer* and select the *Duplicate* option in the appearing menu (Figure 214 (on page 352)). A copy of the water level is created. Now a flow function can be created and assigned to this new water level. Note that copies of *Borehole water levels* are listed as *User water levels* rather than *Generated water levels*. No modifications, such as *Time dependency* are possible for *Borehole water levels* For water levels consisting of different sections, time-dependent conditions can only be applied to the horizontal parts of such water levels.

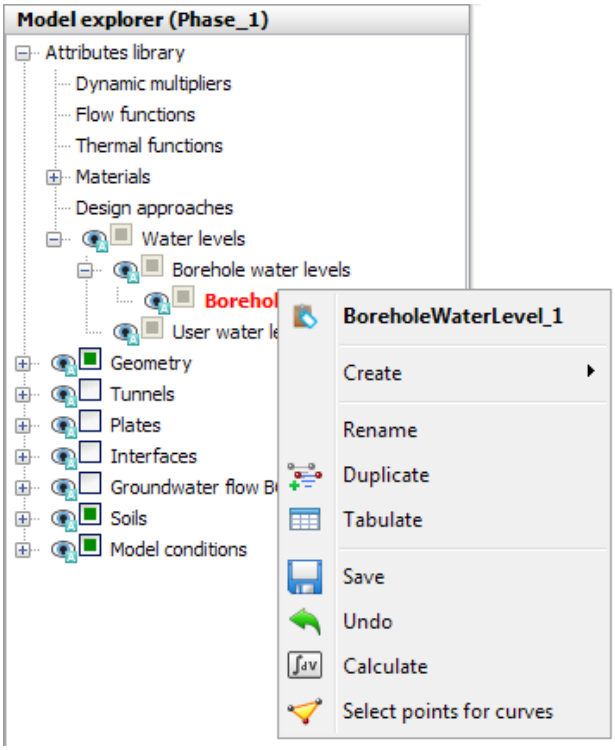


Figure 214: Copying water levels in Model explorer

The continuity of the flow functions assigned to water levels in consecutive phases can be maintained by using the options available in the pop up menu displayed when the water level is right-clicked. These options are:

Create the reached water level	When this option is selected a new water level is created at the position reached at the end of the previous phase. The location of the water level will be constant in the current phase.
Create the reached water level and continue	When this option is selected a new water level is created at the position reached at the end of the previous phase. The location of the water level in the current phase will change from that point on according to the flow function assigned to the original water level in the previous phase. PLAXIS 2D ensures the continuity of the flow function, whereby the accumulated time from previous phases is used as an offset to enter the flow function for the current phase.

Borehole water levels

The easiest way to define a water level is in the **Soil** mode by specifying a *Head* in the boreholes (Defining water conditions (on page 53)). A single borehole can be used to create a horizontal water surface that extends to the model boundaries. When multiple boreholes are used, a non-horizontal water surface can be created by combining the heads in the various boreholes. The water level defined in this way is termed a 'Borehole water level'. This water level is, by default, used as the *Global water level* (see further). In principle, a pore pressure distribution underneath this generated water level is hydrostatic. However, a non-hydrostatic distribution in the soil may be specified in the **Water** tabsheet of the **Modify soil layers** window, which may lead to additional generated water levels representing the (virtual) level where the extrapolated pore pressure in a particular soil layer is zero. For more details about this feature, see Defining water conditions (on page 53).

User water levels - Water levels mode

As an alternative to the *Generated water levels*, the user may define 'User water levels' using the *Create water level* option, which is available in the **Flow conditions** mode.

After selecting this option, a first water level point can be created by clicking in the model at a position where the water pressure is supposed to be zero. If no other point is specified,a horizontal water level is created through this point, extending to the model boundaries.

A non-horizontal water level can be created by defining more points in the geometry, while the *Create water level* option is still active. A new user water level may also be created by duplicating an existing water level and modifying it afterwards. To duplicate an existing water level, right-click it and select *Duplicate* from the appearing menu. User water levels can be deleted by right-clicking them and selecting the *Delete* option from the appearing menu.

Time dependency can be specified for horizontal sections of *User water levels*. The time dependent behaviour can be specified by selecting the corresponding flow function.

Note:

- The *Borehole water levels* cannot be deleted.
- A time-dependent flow function cannot be assigned to water levels that are non-horizontal.

7.9.2 *Assigning water conditions to soil polygons*

Pore pressures in soil polygons are generated on the basis of the water conditions assigned to these polygons. By default, the water conditions as defined in the **Modify soil layers** window are assigned to all soil polygons clusters. This could be the *Global water level*, but (in case of non-hydrostatic pressures defined in the boreholes) it could also be a different generated water level or one of the other water conditions as defined for soil layers in **Soil** mode.

To change the water conditions in the calculation phase considered, polygons objects have to be selected and the desired option should be selected from the *Water conditions feature* in the *Selection explorer*. Similar options can be selected in the *Model explorer* or from the right-hand mouse menu. A summary of all available options is given below:

Global level	Water conditions defined by the Global water level (default).The Global water level is defined in the **General** group in the **Phases** window.
Custom level	Another predefined water level is selected to set the water conditions.
Head	A new *Head* level is defined to set the water conditions.
User-defined	A reference level, reference pressure and pressure increment with depth are defined to set the water conditions.
Interpolate	Water pressures are linearly interpolated (in vertical direction) between the polygons cluster above and the polygons cluster below.
Dry	Set water pressures to zero.
Unsaturated	A saturation level is defined to set the water conditions to partially saturated. The resulting pore stress (suction) will be calculated according to the hydraulic model as defined in the material data set.

Global level

When the *Global level* option is selected, the pore pressure distribution in the soil layer is generated according to the global water level of the phase considered. The resulting pore pressure distribution in the soil layer is then hydrostatic. This does not mean that the pore pressure distribution in the soil as a whole is hydrostatic, since other soil layers may have other hydraulic conditions.

Custom level

When the *Custom level* option is selected in the *Water conditions* drop-down menu, a further selection should be made from the list of available water levels (Borehole water levels and User water levels). The resulting pore pressure distribution in the soil layer is then hydrostatic.

Head

The *Head* option is a quick alternative for the generation of hydrostatic pore pressures based on a horizontal water level. After selecting the *Head* option in the *Water conditions* drop-down menu, a reference level (y_{ref}) should be defined, indicating the level where the pore pressure will be zero.

Note:

- The multi-select option may be used to efficiently set water conditions for multiple soil layers at once.
- When a non-horizontal water level is used to generate a hydrostatic pore pressure distribution, this distribution may not be fully realistic, since in reality a non-horizontal water level would be the result of a groundwater flow process in which case the pore pressure distribution may be non-hydrostatic.

User-defined

If the pore pressure distribution is linear but not hydrostatic, the *User-defined* option should be selected in the *Water conditions* drop-down menu. Subsequently, values for the reference level (y_{ref}), the reference pressure (p_{ref}) and the pressure increment (p_{inc}) should be entered. Note that pressure is entered as a negative value.

Interpolate

It is possible to generate pore pressures in a soil polygons based on the pore pressures of the adjacent soil polygons (above and below) by selecting the *Interpolate* option in the *Water conditions* drop-down menu. This option is, for example, used if a relatively impermeable layer is located between two permeable layers with a different groundwater head. The pore pressure distribution in the relatively impermeable layer will not be hydrostatic, so it cannot be defined by means of a phreatic level.

On selecting the option *Interpolate* the pore pressure in that soil polygons is interpolated linearly in vertical direction, starting from the value at the bottom of the soil polygons above and ending at the value at the top of the soil polygon below. The *Interpolate* option can be used repetitively in two or more successive soil polygons (on top of each other). In the case that a starting value for the vertical interpolation of the pore pressure cannot be found, then the starting point will be based on the *Global water level*.

Dry

In addition to the global water pressure distribution it is possible to remove water pressures from individual groups of polygons in order to make them 'dry'. Deactivation of water can be done independent from the soil itself. Hence, if the soil is deactivated and the water level is above the excavation level, then there is still water in the excavated area. If it is the user's intention to simulate a dry excavation, then the water must be explicitly deactivated. Please note that the water pressure in adjacent soil polygons is not affected and may need to be changed manually.

Note:

The *Dry* clusters behave as non-porous materials. As a result, neither initial nor excess pore pressure is taken into account and flow is not possible through the cluster.

Unsaturated

It is possible to generate pore stresses (suction) based on an explicit value of initial saturation in a soil polygons, by selecting the *Unsaturated* option from the *Water conditions* drop-down menu. The *Saturation* value has to be in the range *0.1* and *0.99*. The saturation is uniform throughout the soil cluster. The resulting pore stess (suction), will be calculated according to the hydraulic model and parameters as defined in the material dats set. In order to be able to use this suction during calculations, the option 'ignore suction' must be de-selected.

7.9.3 *Water pressure generation and previewing*

Water pressures (i.e. pore pressures in finite element stress points and external water loads) are calculated on the basis of the water conditions as defined for a calculation phase as soon as the phase is previewed or when the calculation process is started.

It is recommended to preview the water pressures in order to check that the generated pressures are satisfactory. This can be done by clicking the *Preview phase* button in the side tool bar. After the water pressures have been generated and written to data files, the Output program is started, showing the active mesh for the phase considered. The various types of pore pressures can be previewed by selecting the desired option from the *Stress* menu. The phreatic level or external water level can be seen when the *Phreatic level* option is active in the *Geometry* menu. If the *Loads* option is also active and the water level is outside the active mesh, external water pressures can be seen.

Water pressures for phases that are selected for calculation are also generated at the start of the calculation process, just before the execution the finite element calculation itself. The results of the water pressure generation are also included in the output steps with computational results, and can be viewed after the calculation has finished.

7.9.4 *Flow boundary conditions* [ADV]

The flow boundary conditions of a phase (at least which boundaries are 'open' and which are 'closed') can be specified in the *GroundwaterFlow* subtree under *Model conditions* in the *Model explorer*. Such boundary conditions are required for a groundwater flow calculation, a consolidation analysis or a fully coupled flow-deformation analysis. By default the bottom boundary of the model is closed. The prescribed groundwater head on external geometry boundaries is, by default, derived from the position of the general phreatic level, at least when the general phreatic level is outside the active geometry. Also internal geometry lines that have become external boundaries due to a de-activation of soil clusters are considered to be external geometry boundaries and are therefore treated similarly.

In a consolidation analysis the flow boundary conditions define where excess pore pressures may dissipate through the model boundary, and hence, it affects the excess pore pressures only. In a groundwater flow or fully coupled analysis the flow boundary conditions define where pore water may flow into or out of the soil, and hence, it affects the total pore pressures in these cases. In the latter case, additional conditions may be required to define the hydraulic head at 'open' boundaries. By default, the hydraulic head is implicitly defined by the *Global water level*, but this can be overwritten by using the *Create groundwater flow boundary conditions* feature in the **Structures** mode (Groundwater flow boundary conditions (on page 134)). The parts of an 'open' boundary above the level defined by the hydraulic head are considered to be 'seepage' boundaries whereas the parts below are assigned the given hydraulic head as a boundary condition.

Boundary conditions - Phases

Besides the options described in Groundwater flow boundary conditions (on page 134) for groundwater flow boundary conditions, two other options are available in **Calculation modes**. These options are:

Constant value from previous phase	The water level reached at the end of the previous phase will be kept constant during the current phase.
Maintain function from previous phase	The water level in the current phase will continue to vary according to the flow function assigned to the boundary in the previous phase. The accumulated time from previous phases is used as an offset to enter the flow function for the current phase.

7.9.5 *Precipitation* [ADV]

The *Precipitation* option can be used to specify a general vertical recharge or infiltration (q) due to weather conditions. This condition is applied at all boundaries that represent the ground surface. This option can be selected in the *Model conditions* subtree in the *Model explorer* (Figure 215 (on page 357)).

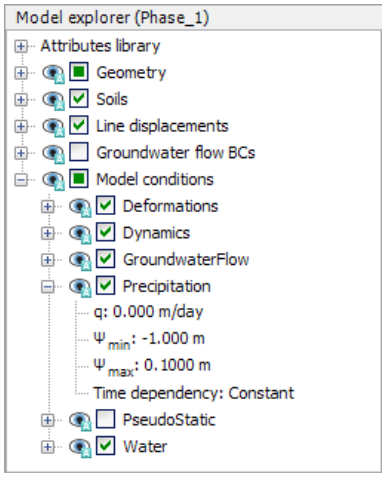


*Figure 215: The expanded **Precipitation** subtree in the Model explorer*

Note:

- Note that when both *Precipitation* and a groundwater flow boundary conditions is created at the top boundary of the model, the condition specified for the boundary condition is taken into account ignoring *Precipitation*.
- The *Precipitation* model condition is displayed on the Model explorer for PLAXIS 2D licence, however, this feature is only useful for [ADV] and [ULT] licences.

Note:

The parameters used to define precipitation are:

q	Recharge (infiltration), specified in the unit of length per unit of time. Negative values can be used to model evaporation.
ψ_{max}	Maximum pore pressure head, relative to the elevation of the boundary, specified in the unit of length (default 0.1 length units). When the groundwater head increases above this level, the infiltration discharge changes into the corresponding head to simulate run off.
ψ_{min}	Minimum pore pressure head, relative to the elevation of the boundary, specified in the unit of length (default -1.0 length units). When the groundwater head decreases below this level, the evaporation discharge changes into the corresponding head.

At horizontal ground surface boundaries, the full precipitation as specified by the value of q is applied as a recharge. At inclined ground surface boundaries (slopes) under an angle α with respect to the horizon, a recharge is applied perpendicular to the inclined boundary with a magnitude $q\ cos(\alpha)$.

If the resulting pore pressure head at a certain point of a boundary where a positive precipitation has been prescribed is increased such that it reaches the value $y + \psi_{max}$ (i.e. the water level comes above the ground surface at a depth of ψ_{max}) then the water is supposed to run-off. As a result, a constant head boundary condition equal to $y + \psi_{max}$ is applied instead.

If the resulting pore pressure head at a certain point of a boundary where a negative precipitation (evapotranspiration) has been prescribed is below a value $y + \psi_{min}$ (i.e. the upper part of the ground has become unsaturated), then the evapotranspiration is supposed to stop. As a result, a constant head boundary condition equal to $y + \psi_{min}$ is applied instead.

For transient groundwater flow calculations and fully coupled flow-deformation analysis, a variation of the precipitation in time can be specified resulting in time-dependent boundary conditions. This can be done by selecting the *Time dependent* option in the *Precipitation* subtree. The corresponding flow function describing the variation of the discharge with time can be selected from the drop-down menu. More information on *Flow functions* is available in Flow functions [ULT] (on page 141).

7.9.6 *Climate* [ADV]

The *Climate* option can be used for thermal calculations to specify a general convective condition due to weather conditions (air temperature). For details on *Convection* boundaries see Thermal flow boundary conditions [ADV] (on page 147). The *Climate* condition is applied at all boundaries that represent the ground surface. This option can be selected in the *Model conditions* subtree in the *Model explorer* (Figure 216 (on page 358)).

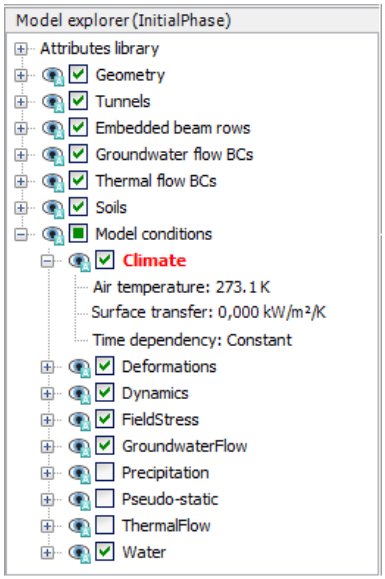


*Figure 216: The expanded **Climate** subtree in the Model explorer*

The parameters used to define climate conditions are:

Air temperature	Temperature of the air, entered in the unit of temperature.
Surface transfer	Heat transfer coefficient (the inverse of thermal resistance), entered in the unit of power per unit of length per unit of width in the out-of-plane direction per unit of temperature.
Time dependency [ULT]	Indicator whether the air temperature is *Constant* or *Time dependent*. In the latter case, a *Temperature* function needs to be assigned as defined in the *Thermal functions* of the *Attributes library* of the model explorer.

Note: The climate model condition is displayed on the Model explorer for PLAXIS 2D licence, however, this feature is only useful for [ADV] and [ULT] licences.

7.10 Geometry configuration - Staged Construction mode

In PLAXIS 2D, it is possible to change the geometry and load configuration by activating or deactivating loads, soil polygon clusters or structural objects created in the geometry input for each calculation phase. The material data sets can be reassigned and the water pressure distribution in the geometry can be changed as well.

Changes to the geometry configuration generally cause substantial out-of-balance forces. These out-of-balance forces are stepwise applied to the finite element mesh using the automatic load stepping procedures (Automatic step size procedure (on page 328)).

7.10.1 *Changing geometry configuration*

Loads, soil polygon clusters or structural objects may be activated or deactivated to simulate a process of construction or excavation in the **Staged construction** mode. Hence, it is possible, for example, to first make an excavation with soil retaining walls, then install a basement floor and subsequently constructing the building above. In this way, the effects around the excavation can be analysed realistically.

To activate/deactivate elements in a selected mode in *Phases explorer*, click the *Toggle activation* button and click the objects to be activated in the drawing area. All the elements assigned to the geometry selected after clicking *Toggle activation* will be activated/deactivate.

The right mouse button menu or the *Object explorers* can be used in cases where not all features should be deactivated. Note that in the right mouse button menu, besides the geometry entity, the features assigned to it are listed as well. To deactivate a feature, point that in the right mouse button menu and select the *Deactivate* option.

Entities and the features assigned to them can be activated or deactivated in *Object explorers* by clicking on the square in front of them (See Figure 217 (on page 360)).

- Not active object or group of objects.
- Active object or group of objects.
- Partly active group of objects.

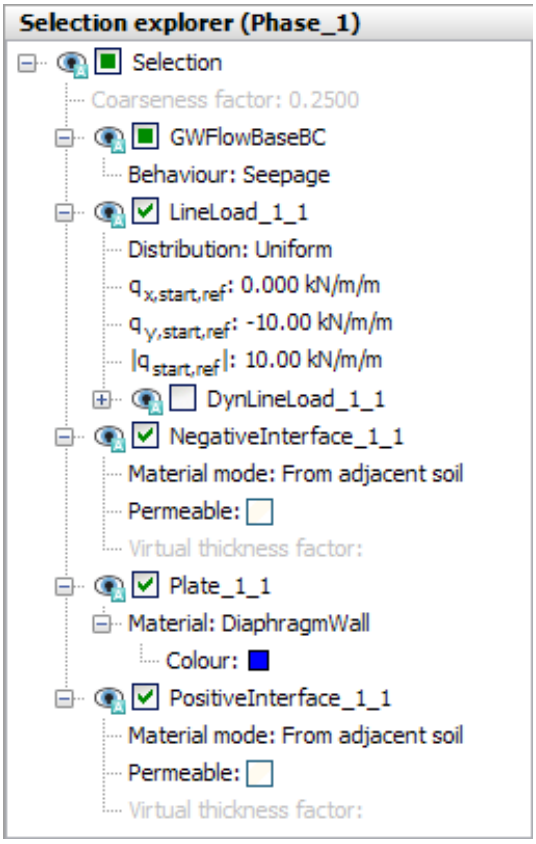


*Figure 217: Selection explorers in **Staged construction** mode*

The activation or deactivation of soil polygon clusters or structural objects or change of properties can introduce substantial out-of-balance forces. These out-of-balance forces are solved stepwise during the staged construction calculation process.

Note:

- The option to activate or deactivate objects should not be confused with the option to show or hide objects. A hidden object (invisible) may still be active.

7.10.2 *Staged construction procedure in calculations*

At the start of a calculation phase the information about active and inactive objects in the geometry model is transformed into information on an element level. Hence, deactivating a soil cluster results in 'switching off' the corresponding soil elements during calculation. The following rules apply to elements that have been switched off:

- Properties, such as weight, stiffness and strength, are not taken into account.
- All stresses are set to zero.
- All inactive nodes will have zero displacements.
- Boundaries that arise from the removal of elements are automatically taken to be free and permeable.

- As a soil cluster is deactivated, the *Water conditions* are maintained. The water conditions of the deactivated soil cluster can be changed to model a dry excavation.
- External loads that act on a part of the geometry that is inactive will not be taken into account.

For elements that have been inactive and that are (re)activated in a particular calculation, the following rules apply:

- Stiffness and strength will be fully taken into account from the beginning (i.e. the first step) of the calculation phase.
- Weight will, in principle, be fully taken into account from the beginning of the calculation phase. However, in general, a large out-of-balance force will occur at the beginning of a staged construction calculation since the initial stresses (if any) are not in equilibrium with the weight of the newly activated elements or loads. This out-of-balance force is stepwise solved in subsequent calculation steps.
- The stresses will develop from zero.
- When a node becomes active, an initial displacement is estimated by stressless pre-deforming the newly activated elements such that they fit within the deformed mesh as obtained from the previous step. Further increments of displacements are added to this initial value. As an example, one may consider the construction of a block in several layers, allowing only for vertical displacements (one-dimensional compression). Starting with a single layer and adding one layer on top of the first will give settlements of the top surface. If a third layer is subsequently added to the second layer, it will be given an initial deformation corresponding to the settlements of the surface.

The following rules apply for interfaces:

Deactivated interfaces	The interfaces can be deactivated when modelling soil-structure interaction is not desired. Nodes generated for interfaces in mesh generation process are still there. They have stiff elastic behaviour; fully permeable (*Consolidation* analysis).
Activated interfaces	Elasto-plastic behaviour; fully impermeable (*Consolidation* analysis).

7.10.3 *Changing loads*

During creation in the *Geometry* modes, a default value is given to a load which represents a unit load. These load values may be changed in each calculation phase to simulate changing loads in the various stages of construction. The change of loads can introduce substantial out-of-balance forces. These out-of-balance forces are solved during the staged construction calculation process.

7.10.4 *Connections in Staged construction*

Connections can be configured per phase. The active state of a connection does not affect the active state of the connecting plates. A warning message will be shown before calculation if a connection is active while one of its connected plates is inactive.

7.10.5 *Reassigning material data sets*

It is possible in a calculation phase to assign new material data sets to soil polygon or structural objects. This option may be used to simulate the change of material properties with time during the various stages of construction. The option may also be used to simulate soil improvement processes, e.g. removing poor quality soil and replacing it with soil of a better quality.

The change of certain properties, for example when replacing peat by dense sand, can introduce substantial out-of-balance forces due to a difference in unit weight. These out-of-balance forces are solved during the staged construction calculation process.

7.10.6 *Applying volumetric strains in clusters*

In addition to changing material properties for clusters, in PLAXIS 2D it is also possible to apply a *Volumetric strain* in individual soil clusters. This can be done in the *Selection explorer* by selecting the corresponding soil polygon and clicking the checkboxes corresponding to *VolumetricStrain* and to *Apply* (Figure 218 (on page 362)).

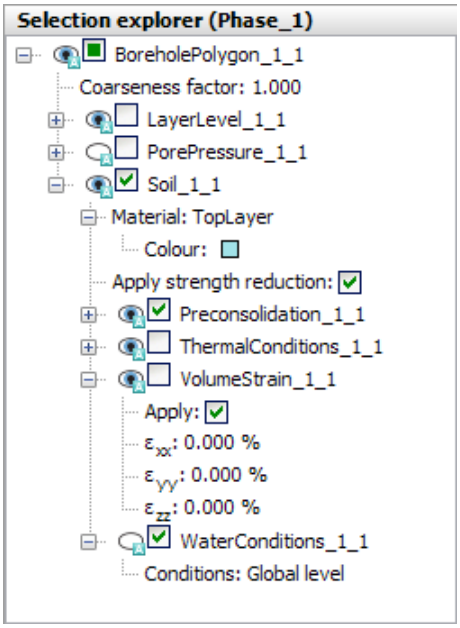


Figure 218: Volumetric strain in the Selection explorer

Values can be defined for the two strain components in *x*- and *y*-direction and for volumetric strain. A positive value of the strain component represents an expansion, whereas a negative value represents a shrinkage in that direction.

In contrast to other types of loading, volumetric strains are not activated with a separate multiplier. Note that the imposed volumetric strain is not always fully applied, depending on the stiffness of the surrounding clusters and objects.

7.10.7 *Prestressing of anchors*

Prestressing of anchors can be activated in the *Object explorer* for a selected phase. By default the *Adjust prestress* option is not selected. On selecting the *Adjust prestress force* check box it is possible to enter a value for the prestress force ($F_{prestress}$) in the corresponding edit box. A prestress force should be given in the unit of force. Note that tension is considered to be positive and compression is considered negative.

By default, in a calculation phase following a phase in which an anchor has been prestressed the prestress setting is NOT continued but the anchor force is maintained. In this way the existing anchor force is used as a start condition and will develop 'naturally' based on changes of stresses and forces in the model. If you intend to again apply a prestress force, this has to be done explicitly following the same procedure as described above.

If a previously entered prestress force needs to be 'reset' to retain a standard active anchor without prestressing but with existing anchor force, this should be done by de-selecting the *Adjust prestress* parameter in the *Object explorer*. It is generally NOT correct to set a prestress force to zero, since this will result in a zero anchor force.

7.10.8 *Advancing a tunnel* [GSE]

The construction of a tunnel can be regarded as a 'train' of construction steps ('tunnel train') that is advancing as a continuous process or in sections of a certain length. In the numerical modelling of tunnel construction, the advancement of the construction process always has to be discretised in sections. Therefore, the *Tunnel Designer* allows for a division of the full tunnel path (*Trajectory*) into sections, which are called 'slices'.

In order to simplify the definition of calculation phases in *Staged Construction* to account for the advancement of the 'tunnel train', the *Tunnel Designer* includes facilities to define the necessary steps (see Sequencing).

Advance to next tunnel step

In the *Model explorer* expand *Tunnels* and right-click on the corresponding *Tunnel*. The *Advance to next tunnel step* option can now be selected. The 'tunnel train' advances one step without regenerating the whole tunnel, maintaining the changes performed to the part of the model that is not affected by the advancement in the current phase.

In the case the construction and advancement of a tunnel consists of multiple phases, it might be useful to run the following commands in the commands runner (Click on the *Expert* menu and then *Run commands*, or *Ctrl +F9*):

```
phase Phases[-1]

setcurrentphase Phases[-1]

phase Phases[-1]advance Tunnel_1 Phases[-1]
```

These commands can be copied for the required number of phases. Alternatively, these commands can be copied to the clipboard. Commands on the clipboard can be executed directly by pressing *Ctrl+Shft+F9* without the need to have the *Run Commands* window open. As long as these commands are on the clipboard, pressing *Ctrl+Shft+F9* will automatically introduce another phase in which the tunnel is advanced over one slice.

Advancement step

In the *Model explorer* expand *Tunnels* and then expand the corresponding *Tunnel*. Scroll down the *Model explorer* until the option *Advancement step* is displayed. The number of advanced steps can be manually set. The 'tunnel train' advances to the chosen step regenerating the whole tunnel accordingly to the defined sequencing. All the changes previously performed to the part of the model that is not affected by the advancement are overwritten accordingly to the defined sequencing. The regeneration of the advancement regards only the current phase.

7.10.9 *Model conditions*

The global boundary conditions for a selected phase can be defined in the *Model conditions* subtree in the *Model explorer*. Note that user-defined conditions always prevail over the ones defined in the *Model conditions* subtree.

Deformation

PLAXIS 2D automatically imposes a set of standard fixities to the boundaries of the geometry model. These conditions are generated according to the following rules.

Soil polygon:

- Vertical model boundaries with their normal in *x*-direction , are fixed in *x*-direction ($u_x=0$) and free in *y*-direction.
- The model bottom boundary is fixed in all directions ($u_x = u_y = 0$).
- The 'ground surface' is free in all directions.

The options available for the model boundaries are (defaults in square brackets):

BoundaryXMin	*Free*, *[Normally fixed]*, *Horizontally fixed*, *Vertically fixed*, *Fully fixed*
BoundaryXMax	*Free*, *[Normally fixed]*, *Horizontally fixed*, *Vertically fixed*, *Fully fixed*
BoundaryYMin	*Free*, *Normally fixed*, *Horizontally fixed*, *Vertically fixed*, *[Fully fixed]*
BoundaryYMax	*[Free]*, *Normally fixed*, *Horizontally fixed*, *Vertically fixed*, *Fully fixed*

In the case that the model has no vertical model boundaries and/or no horizontal bottom boundary, prescribed displacements need to be defined in order to set the displacement boundary conditions.

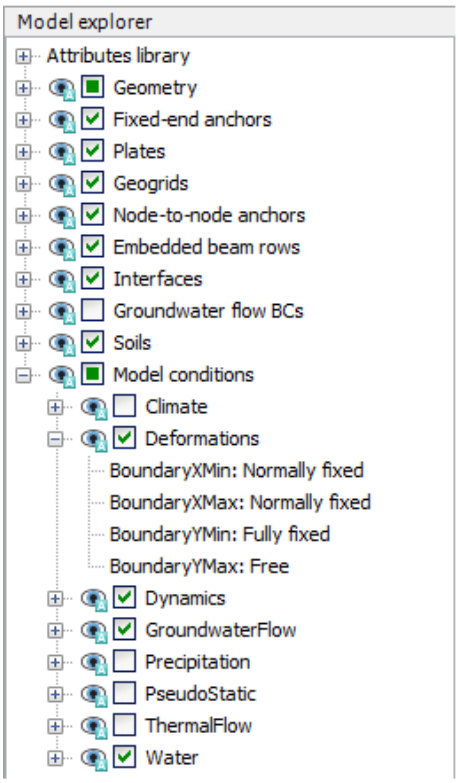


Figure 219: Boundary conditions

Structures: Structures that extend to the model boundary, where at least one displacement direction is fixed, obtain a fixed rotation in the points at the boundary.

- At vertical model edges and at the bottom boundary: $\varphi=0$.
- At the 'ground surface': φ= free.

Note:

- Note that the standard displacement fixities (Model conditions (on page 364)) apply to soil polygons and structures.
- User-defined prescribed displacements always prevail over the automatically defined fixities.

Dynamics boundary conditions [ULT]

For dynamics calculations, different boundary conditions than the standard fixities are required in order to represent the far-field behaviour of the medium. The reality is characterised by an infinite domain which has to be reduced to a finite domain when creating a geometry model. Appropriate boundary conditions can simulate the far-field behaviour by absorbing the increment of stresses caused by dynamic loading and avoiding spurious wave reflections inside the soil body. By default, PLAXIS 2D provides the option to use *Viscous* boundaries but other options are available as well.

When a *Dynamics* calculation is defined, the boundary conditions can be defined for the phase by selecting the proper options displayed when the *Dynamics* subtree is expanded under *Model conditions* in the *Model explorer* (Figure 220 (on page 366)).

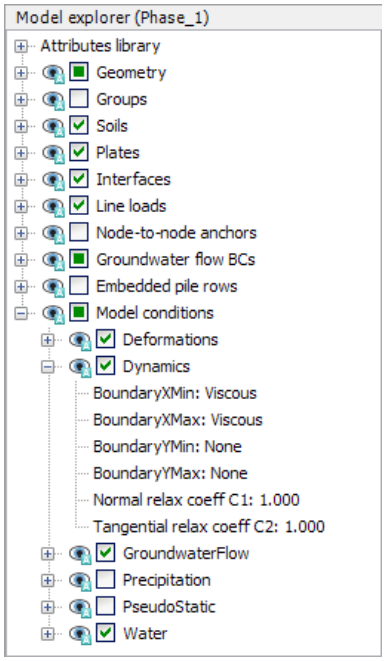


Figure 220: Boundary conditions for Dynamics calculations

The options available for the model boundaries are (defaults in square brackets):

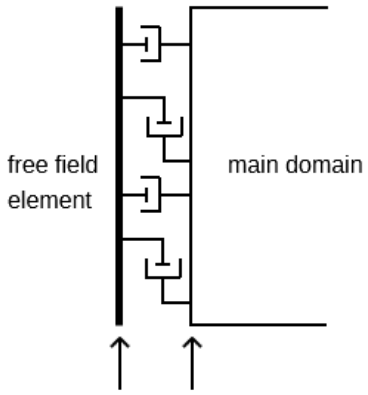
BoundaryXMin	*None*, *[Viscous]*, *Free-field* , *Tied degrees of freedom*
BoundaryXMax	*None*, *[Viscous]*, *Free-field* , *Tied degrees of freedom*
BoundaryYMin	*[None]*, *Viscous*, *Compliant base*
BoundaryYMax	*[None]*, *Viscous*
All nodes fixities	*[None]*, *Fixed in x-direction*, *Fixed in y-direction*

Note:

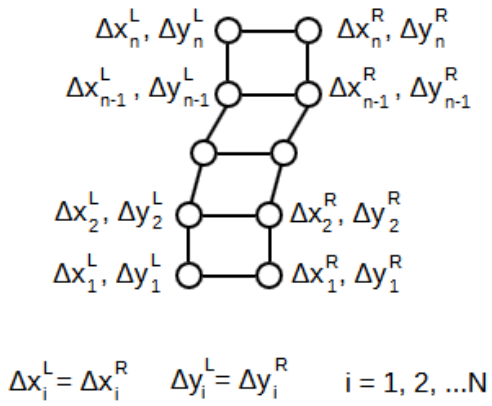
- Default *None* for XMin boundary in axi-symmetric models
- *Free-field* and *Tied degrees of freedom* conditions are ignored for axi-symmetric models

The meaning of these conditions is as follows:

None	Only the standard fixities are applied to this boundary. In earthquake analysis, this option can be used at the base of the geometry model in combination with a line prescribed displacement to simulate the boundary of two layers with a very high impedance contrast, for instance a soft soil layer over a rigid bedrock. This results in a full reflection of downward propagating waves.

Viscous	*Viscous* boundaries absorb the outgoing wave energy. This boundary condition corresponds to a situation in which viscous dampers are applied in the x and y direction along the boundary providing a resistant force in the normal and tangential direction at the boundary that is proportional to the velocity in the near-boundary material. The *Viscous* boundary condition has been introduced by Lysmer & Kuhlmeyer (1969) (on page 531). This option is suitable for problems where the dynamic load source is inside the mesh. If a *Viscous boundary* is selected at the base of the model for a seismic analysis, the required input has to be a load history. The *Relaxation coefficients* C_1 *and* C_2 need to be considered (see Scientific Manual - *Model Boundaries*).
Free-field	The *Free-field* boundary condition is only available for the lateral boundaries (i.e. x_{min} *and* x_{max}) and simulates the propagation of waves into the far field with minimum reflection at the boundary. Free field elements can be modelled at both sides of the main domain and are characterised by the same properties as the soil layers inside the mesh. The free field motion is transferred from the free field elements to the main domain by applying equivalent normal and shear forces. Two dashpots are added, in the normal and shear direction, at each node of the lateral boundary, to adsorb the waves reflected from the internal structures (Figure 221 (on page 367)). This option requires the creation of interface elements along the vertical model boundaries in the *Structures mode*. The interface elements must be added inside the model, else the *Free-field* boundary condition will be ignored. If this option is selected for one of the vertical model boundaries, it must also be selected for the opposite vertical model boundary. This option is preferred for earthquake analysis, where the dynamic load input is applied along the model bottom boundary. The input motion is then also applied at the base of the free-field elements.  *Figure 221: Free field boundary condition*

Compliant base	The *Compliant base* boundary condition is only available for the base (y_{min}) of the model and simulates the continuation of waves into the deep soil with minimum reflection at the bottom boundary. The compliant base (Joyner & Chen (1975) (on page 531)) is made up of a combination of a line prescribed displacement and a viscous boundary. The input signal (i.e. a displacement, velocity or acceleration time history) is transferred to the main domain by applying equivalent normal and shear forces. The viscous boundary consists of a series of dashpots, in the normal and shear direction, at each node of the base, to adsorb downward propagating compressive and shear waves. Note that only upward propagating waves should be considered in the signal applied at a compliant base boundary. In the case of a bedrock layer, this can be defined by taking only half of the input value of the corresponding prescribed displacement (e.g. 0.5 m instead of 1.0 m), together with the bedrock outcropping motion as dynamics multiplier. The combination of a load history and a viscous boundary allows for input of an earthquake motion while still absorbing outgoing waves. In earthquake analysis, this option can be used for site response analysis as well as for dynamic soil-structure interaction analysis where the model bottom boundary is within a homogeneous layer of soil or bedrock with a high shear wave velocity. In the case of a bedrock layer, it is recommended to include 1 m (3 ft) of bedrock at the base of the model below which the compliant based boundary is applied. Downward propagating waves from the main domain are free to propagate to the underlying layer which is not included in the finite element mesh. The equivalent forces are calculated taking into account the soil layer at the base. This option requires the presence of interface elements along the bottom boundary in the *Structures mode*. The interface elements must be added inside the model, else the *Compliant base* boundary condition will be ignored.

Tied degrees of freedom	The tied degree of freedom are only available for the lateral boundaries (i.e. x_{min} and x_{max}). This option, proposed by References (on page 530), connects the nodes on the same elevation at the left and right model boundaries, such that they are characterised by the same vertical and horizontal displacement (Figure 222 (on page 369)). This option may be chosen to model a one dimensional soil column to perform a site response analysis. If this option is selected for one of the vertical model boundaries, it must also be selected for the other vertical model boundary. The *Tied degrees of freedom* only work when the boundaries are free to move, i.e. no fixities should be applied on the vertical boundaries. In most cases, this will mean that the default fixities should be switched off during the dynamic analysis and that the desired boundary condition at the bottom should be applied. The *Tied degrees of freedom* can only be applied if the distribution of nodes along the vertical model boundaries is identical. In other word, the corresponding nodes left and right should have the same y-coordinate.  *Figure 222: Tied degrees of freedom*
Fixed in x-direction	All nodes at the boundary are fixed in x direction.
Fixed in y-direction	All nodes at the boundary are fixed in y direction.

Note:

- Viscous boundaries do not consider additional forces from structural elements. Hence when using viscous boundaries, do not use structural elements at these boundaries.
- In case the soil in the main domain is characterised by undrained behaviour (*Undrained A* or *Undrained B*), *Free-field* and *Compliant base* require a small layer or column of soil with drained soil behaviour near the model boundaries. The reason is that *Free-field* and *Compliant base* boundary conditions are only defined for drained soil behaviour.
- By default, *Free-field* or *Compliant base* elements are not displayed in the Output program. In order to be able to visualise the movement in the *Free-field* or *Compliant base*, a (inactive) geogrid or plate element can be added at these boundaries. Note that during calculations such a plate or geogrid must remain inactive.
- Tied degrees of freedom can only be applied if the distribution of nodes along the vertical model boundaries is identical. In other words, the corresponding nodes left and right should have the same y-coordinate.

The *Relaxation coefficients C_1 and C_2* are used to improve the wave absorption on the viscous boundaries. C_1 corrects the dissipation in the direction normal to the boundary and C_2 does this in the tangential direction. If the boundaries are subjected only to waves that come in perpendicular to the boundary, relaxation is not necessary ($C_1 = C_2 = 1$). When there are waves in arbitrary direction (which is normally the case), C_2 has to be adjusted to improve the absorption. The standard values are $C_1 = 1$ and $C_2 = 1$.

Note that the *Compliant base* and *Free field* conditions requires the manual creation of interface elements along the full model boundary, which enable the creation of a so called node pair. Between the two nodes of a node pair a viscous damper is created which allows to transfer the input motion and absorb the incoming waves. In the *Output* program only the parameters related to one side of the node pair can be visualised, i.e. the motion in these nodes is not equal to the input motion. To be able to see and to explicitly check the input motion, it is possible to create a so called 'dummy plate', i.e. a plate at the bottom line of the model, characterised by very low stiffness parameters so that it does not influence the results. A node has to be selected on the plate to be able to compare the motion in this node with the input signal. There is no specific need to activate the dummy plate during the calculation.

Field stress

The *Field stress* conditions option can be used when using the calculation type *Field stress* for initial phase calculation. This is used to specify rotated principal stresses as the initial stress conditions. This option can be selected in the *Model conditions* subtree in the *Model explorer* (See Figure 223 (on page 370)).

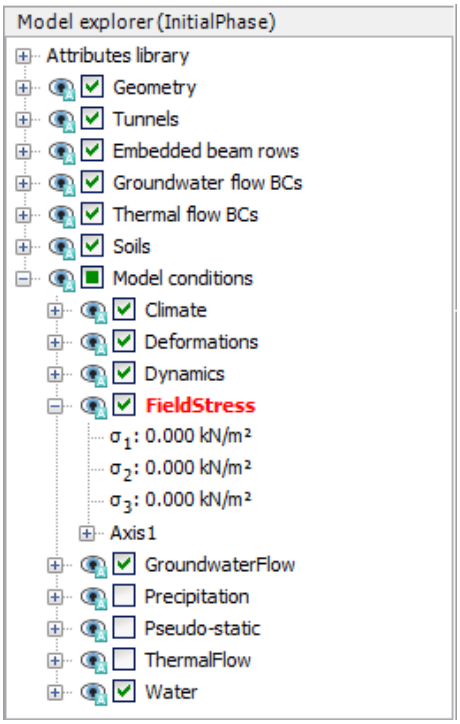


Figure 223: Field Stress conditions

The parameters used to define field stress are:

σ_1	The principal stress in first direction
σ_2	The principal stress in second direction

σ_3	The principal stress in third direction
Axis 1	Direction of the first principal stress
Axis 2	Direction of the second principal stress

Note:

- Note that the calculation is considered with ΣM_{weight} as zero
- Water weight is not considered when performing this type of calculation, but external loads may be applied if necessary.
- All model boundaries must be set to *Fully fixed* to ensure shear stress condition is simulated.

Refer to Initial stress generation (on page 300) for a more elaborate description of the Field stress option.

Groundwater flow boundary conditions [ADV]

The flow boundary conditions of a phase (at least which boundaries are 'open' and which are 'closed') can be specified in the *GroundwaterFlow* subtree under *Model conditions* in the *Model explorer* (See Figure 224 (on page 371)).

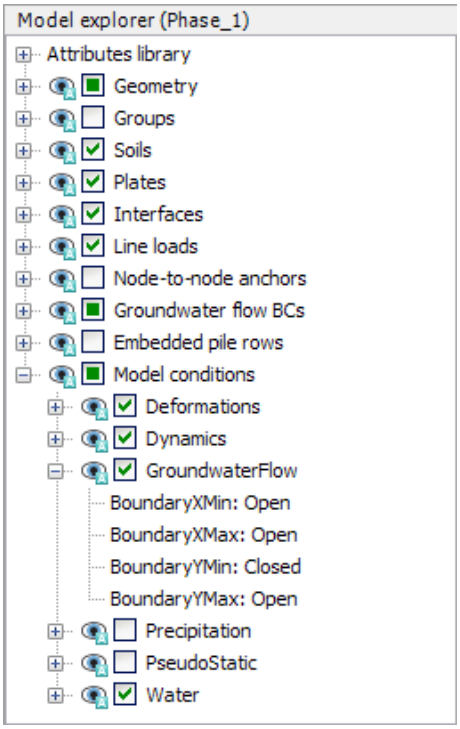


Figure 224: Boundary conditions for groundwater flow

Such boundary conditions are required for a groundwater flow calculation, a consolidation analysis or a fully coupled flow-deformation analysis. More information regarding the flow boundary conditions is available in Flow boundary conditions [ADV] (on page 356).

Precipitation [ADV]

The *Precipitation* subtree in the *Model explorer* (Figure 225 (on page 372)) can be used to specify a general vertical recharge or infiltration (q) due to weather conditions. This condition is applied at all boundaries that represent the ground surface. More information on *Precipitation* is available in Precipitation [ADV] (on page 356).

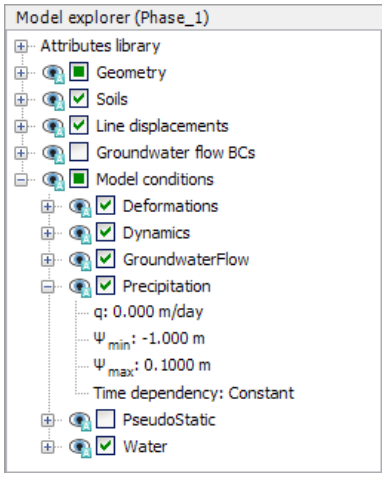


Figure 225: The Precipitation subtree in the Model explorer

Pseudo-Static [ADV]

Dynamic forces can be considered in a pseudo-static way by specifying a global acceleration (Figure 226 (on page 373)). A specified global acceleration is taken into consideration in *Gravity loading*, *Plastic*, *Consolidation* and *Safety* calculation type. Note that in a *Safety analysis* the settings of the parent phase are considered. It is possible to perform a safety analysis of a pseudo-static parent phase, however a change of global accelerations cannot be specified for the *Safety* phase itself.

When performing a groundwater flow calculation, the specified global accelerations are composed with the gravity affecting the unit weight of water used in the calculation of the pore water pressures.

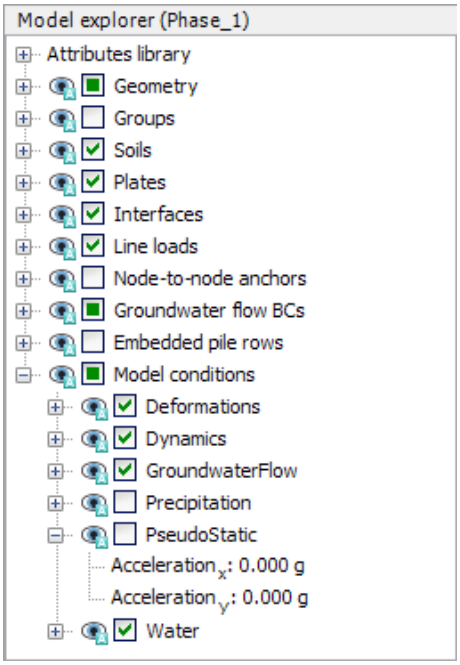


Figure 226: The PseudoStatic subtree in the Model explorer

Water

The global water level in a calculation phase can be defined in the *Water* subtree (Figure 227 (on page 373)). More information on *Global water level* is available in Assigning water conditions to soil polygons (on page 353).

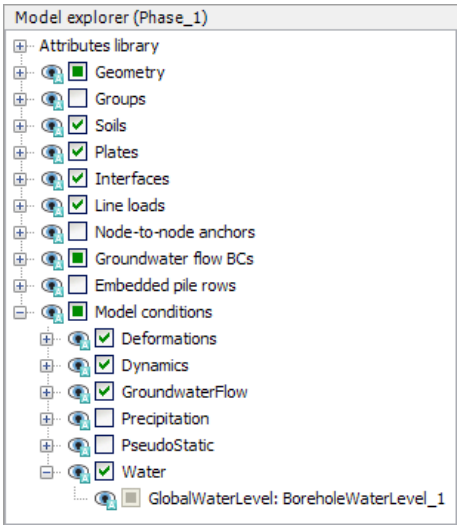


Figure 227: The Water subtree in the Model explorer

7.11 Special option for User Defined Soil Models [ADV]

The *General* section of the calculation parameters allows setting a Special option parameter. This parameter may be used in user-defined soil models to trigger a particular event or action at the beginning of the calculation phase based on the value of this parameter. The Special option parameter does not affect standard PLAXIS functionality.

7.12 Starting a calculation

When a calculation phase has been defined, its calculation can be started.

7.12.1 *Previewing a construction stage*

When a construction stage is fully defined, a view of the situation can be presented by pressing the *Preview phase* button in the side toolbar. This enables a direct visual check before the calculation is started. The active part of the model is presented in the Output program.

See Output program - General overview (on page 384) for more details on the inspection of models in the Output program. After the preview, press the *Close* button to return to the **Calculation** mode.

7.12.2 *Selecting points for curves*

After the calculation phases have been defined and before the calculation process is started, some points may be selected by the user for the generation of load-displacement curves or stress paths by clicking the *Select points for curves* button or by selecting this option in the *Tools* menu.

Nodes should be selected to plot displacements, whereas stress points should be selected to plot stresses and strains. Selection of points is described in detail in Selecting points for curves (on page 451).

7.12.3 *Execution of the calculation process*

When calculation phases have been defined and points for curves have been selected, then the calculation process can be executed. Before starting the process, however, it is useful to check the list of calculation phases. In principle, all calculation phases indicated with a blue arrow (→) will be executed in the calculation process. By default, when defining a calculation phase, it is automatically selected for execution. A previously executed calculation phase is indicated by a green tick mark (✓) if the calculation was successful, otherwise it is indicated by a red cross (×). To select or deselect a calculation phase for execution, the button in front of the phase should be clicked in the **Phases** window.

Alternatively, right click the corresponding line and select *Mark for calculation* or *Unmark for calculation* in the appearing menu.

· *Starting the calculation process*

The calculation process can be started by clicking the *Calculate* button in the tool bar. As a result, the program first performs a check on the ordering and consistency of the calculation phases. In addition, the first calculation phase to be executed is determined and all selected calculation phases in the list are subsequently executed, provided that failure does not occur.

7.12.4 *Aborting a calculation*

If, for some reason, the user decides to abort a calculation, this can be done by pressing the *Stop* button in the separate window that displays information about the iteration process of the current calculation phase.

If the *Stop* button is pressed, the total specified load will not be applied. In the **Phases explorer** the phase is preceded by a red cross and in the **General** subtree in the **Phases** window the following message is displayed in the *Log info* box: *Cancelled*.

In addition to aborting a calculation permanently, it is also possible to abort the calculation temporarily by clicking the *Pause* button. The calculation will be resumed after clicking the *Resume* button.

7.12.5 Output during calculations

During a 2D finite element deformation analysis, information about the calculation process is presented in a separate window (Figure 228 (on page 376)). The phase being calculated is indicated in the *Phase* tabs.

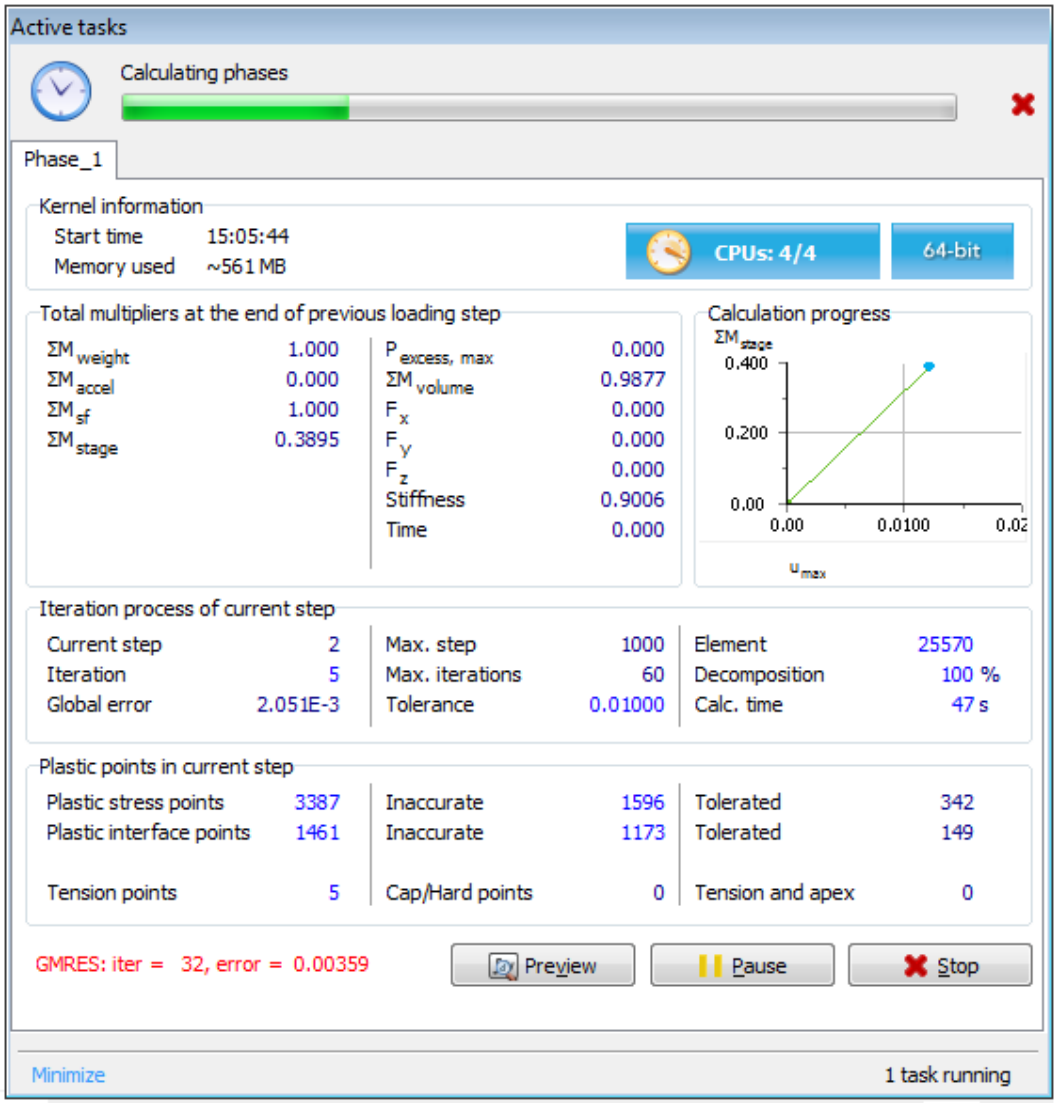


*Figure 228: The **Active tasks** window*

Note:

Parallel calculation is possible on multi-core computers when phases have the same parent phase.

Kernel information

Start time	The time indicating the start of the calculation is displayed.
Memory used	The memory occupied by the calculation process is displayed.

Total multipliers at the end of the previous loading step

ΣM_{dispX}, ΣM_{dispY}	Indicates the portion of the defined prescribed displacement applied in the current phase.
ΣM_{loadA}	Indicates the portion of the defined load applied in the current phase. In PLAXIS 2D this value is always 1 since load is applied as a staged construction process changing the input value of the load rather than the multiplier.
ΣM_{weight}	Indicates the total proportion of the material weights applied in a calculation. Its value is *0* at the beginning of the calculation and changes to *1.000*, indicating that all the materials weight is applied.
ΣM_{accel}	The value of this parameter is always *0*.
ΣM_{sf}	This parameter is related to the *Safety* analysis. It is defined as the ratio of the original strength parameters and the reduced strength parameters at a given stage of analysis. Its value is *1.000* at the beginning of an analysis. The increment of the strength reduction of the first calculation step is described in Safety calculation (phi/c reduction) (on page 307).
ΣM_{stage}	It gives the completed proportion of a plastic calculation. Its value is always *0* at the start of the calculation and it will be *1.000* at the end of a successful calculation. For other analysis types (*Consolidation* and *Safety*) it is always *0*.
$P_{excess,max}$	It represents the maximum excess pore pressure in the mesh, expressed in the units of stress. $P_{excess,max}$ is available in the deformation calculation tabsheet of the **Active task** window for the *Plastic*, *Consolidation*, *Safety* and *Dynamic* calculation types.
$P_{active,max}$	It represents the maximum active pore pressure in the mesh in *Fully coupled flow-deformation analysis*, expressed in the units of stress.
$P_{steady,max}$	It represents the maximum steady state pore pressure in the mesh, expressed in the units of stress. P_{steady} is only available for *Flow only* calculation types.
ΣM_{area}	It indicates the proportion of the total area of soil clusters in the geometry model that is currently active.
ΣF_x, ΣF_y	These parameters indicate the reaction forces corresponding to the non-zero prescribed displacements.
Stiffness	The *Stiffness* parameter gives an indication of the amount of plasticity that occurs in the calculation. The *Stiffness* is defined as: $$\text{Stiffness} = \int \frac{\Delta\varepsilon \cdot \Delta\sigma}{\Delta\varepsilon D^e \Delta\varepsilon}$$ When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the stiffness approaches zero. The *Stiffness* is used in determining the *Global error*. See Automatic error checks (on page 381) for more details.

Time	The current time within the specified time interval of the loading input for the calculated phase, defined in the **General** subtree in the **Phases** window.
Dyn. time	The current dynamic time within the specified time interval of the loading input for the calculated phase, defined in the **General** subtree of the **Phases** window.

Calculation progress

A small load-displacement curve for one of the pre-selected nodes for curves is shown in the *Calculation progress* group box. By default, the curve is shown for the first selected node. Curve for other pre-selected nodes is shown when the node is selected in the drop-down menu. The presented graph may be used to roughly evaluate the progress of the calculation.

Plastic analysis	For a plastic analysis the development of the ΣM_{stage} parameter is plotted against the displacement.
Consolidation analysis	In case of a *Consolidation* analysis, the maximum excess pore pressure, $P_{excess,max}$, in case of a *Consolidation* analysis based on excess pore pressure or the maximum active pore pressure, $P_{active,max}$, in case of a *Consolidation* analysis based on total pore pressure is plotted against the logarithm of time.
Fully coupled flow-deformation analysis	In case of a *Fully coupled flow-deformation* analysis, the maximum active pore pressure, $P_{active,max}$, is plotted against the logarithm of time.
Safety analysis	In case of *Safety* analysis, the development of ΣM_{sf} is plotted against the displacement.
Dynamic analysis	In case of *Dynamic* analysis, the displacement is plotted against the dynamic time.
Dynamic with consolidation analysis	In case of *Dynamic with consolidation* analysis, the displacement is plotted against the dynamic time.
Steady state groundwater flow	In case of *Steady state groundwater flow*, the maximum steady pore pressure, $P_{steady,max}$, at steady state is plotted.
Transient groundwater flow	In case of *Transient groundwater flow*, the maximum steady pore pressure, $P_{steady,max}$, is plotted against the logarithm of time.

Iteration process of current step

Current step	Indicates the number of the current calculation step.
Iteration	Indicates the number of the iterations in the current calculation step.
Global error	The value of this error is an indication of the global equilibrium error within the calculation step. As the number of iterations increases, its value tend to decrease. For further details on this parameter see Automatic error checks (on page 381).
Max. local error in flow	The value of this error is an indication of possible entrapment of water in saturated regions in the current calculation step. The tolerated value is 0.05.

Relative change in saturation	The value is an indication of variation of saturation degree in consecutive calculation steps. The tolerated value is 0.1. When the relative change in saturation is higher than the tolerated value, the time step is automatically decreased. When the relative change in saturation is lower than the tolerated value, the time step is automatically increased. Note that the values of the time step are always in the range defined by *Desired minimum* and *Desired maximum* parameters.
Relative change in relative permeability	The value is an indication of variation of relative permeability in consecutive calculation steps. The tolerated value is 0.1. When the relative change in relative permeability is higher than the tolerate value, the time step is automatically decreased. When the relative change in relative permeability is lower than the tolerated value, the time step is automatically increased. Note that the values of the time step are always in the range defined by *Desired minimum* and *Desired maximum* parameters.
Max. step	Indicates the last step number of the current calculation phase according to the *Max steps* parameter defined in the *Numerical control parameters* subtree in the **Phases** window.
Max. iterations	The value of *Maximum iteration steps* for the calculated phase, defined for the *Iterative procedure* in the **Numerical control parameters** subtree in the **Phases** window.
Tolerance	This value indicates the maximum global equilibrium error that is allowed. The value of the tolerance corresponds to the value of the *Tolerated error* in the settings for the iterative procedure. The iteration process will at least continue as long as the *Global error* is larger than the *Tolerance*. For details see Automatic error checks (on page 381).
Element	The number of soil elements in the calculated phase.
Decomposition	Progress of the decomposition of the phase being calculated.
Calc. time	Indicates the calculation time of the current calculation step.

Plastic points in current step

Plastic stress points	The total number of stress points in soil elements that are in plastic state.
Plastic interface point	The total number of stress points in interface elements that are in plastic state.
Inaccurate	This value indicates the number of plastic stress points in soil elements and interface elements respectively, for which the local error exceeds the tolerated error.
Tolerated	This value indicates the maximum number of inaccurate stress points in soil elements and interface elements respectively that are allowed. The iteration process will at least continue as long as the number of inaccurate points is larger than the tolerated number.
Tension points	A *Tension point* is a point that fails in tension. These points will develop when the *Tension cut-off* is used in any of the material sets in the model. This parameter indicates the total number of points that fail in tension.

Cap/Hardening points	A *Cap point* occurs if the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model or Soft Soil Creep model are used and the stress state in a point is equivalent to the preconsolidation stress, i.e. the maximum stress level that has previously been reached (*OCR≤1.0*). A *Hardening point* occurs if the Hardening Soil model or Hardening Soil model with small-strain stiffnessmall is used and the stress state in a point corresponds to the maximum mobilised friction angle that has previously been reached.
Apex points	These are special plastic points where the allowable shear stress is zero. The iterative procedure tends to become slow when the number of plastic apex points is large. Apex points can be avoided by selecting the *Tension cut-off* option in the material datasets for soil and interfaces.

Calculation status

The calculation status indicates what part of the calculation process is currently being executed. The following processes are indicated:

Decomposition...	Decomposing the global stiffness matrix.
Stresses...	Calculating the strain increments and constitutive stresses.
Write results...	Writing output data to disk.

Previewing intermediate results during calculation

The *Preview* button in the **Active tasks** window enables previewing the results of the intermediate calculation steps of the phase being calculated. The intermediate steps are listed in the drop-down menu (Selection of results (on page 401)) and the list is updated when the calculation of new intermediate steps is complete.

The results of the intermediate calculation steps can be used in curves as well. When a curve is created, the newly calculated steps can be included in the plot by using the *Regenerate* button available in the **Settings** window (Regeneration of curves (on page 475)).

Note that when the calculation of the phase is completed, a warning will appear indicating that the intermediate results are no longer available. A more detailed description on how to display the results of a calculated phase is given in Selecting calculation phases for output (on page 380).

7.12.6 *Selecting calculation phases for output*

After the calculation process has finished, the calculation list is updated. Calculation phases that have been successfully finished are indicated by a green tick mark (✓), whereas phases that did not finish successfully are indicated by a red cross (×). In addition, messages from the calculations are displayed in the *Log info* box of the **General** tabsheet.

When a calculation phase is selected that has been executed, the tool bar will show the *View results* button. Clicking this button will directly display the results of the selected phase in the Output program.

7.12.7 *Adjustment to input data in between calculations*

Be careful when changing the input data (in the Input program) in between calculation phases. In general, this should not be done since it causes the input to cease to be consistent with the calculation data. In most cases there are other ways to change data in between calculation phases instead of changing the input data itself.

Modification of geometry

When the geometry is slightly modified (small relocation of objects, slight modification of their geometry or deleting objects) in the Input program, the program will try to regenerate all data related to construction stages as soon as mesh regeneration is performed. However, in the Calculations program, the user has to check the construction stages carefully, since some settings may not have been generated properly. The calculation process must restart from the initial phase. If significant changes in the geometry are made then all settings need to be redefined, since PLAXIS 2D is not able to properly regenerate the settings automatically.

Modification of material parameters and feature properties

When changing material properties in existing data sets without changing the geometry, then all calculation information is retained as well. In this case, clusters refer to the same data sets, but the properties as defined in these data sets have changed. However, this procedure is not very useful, since PLAXIS 2D allows for a change of data sets within the *Staged construction* calculation option (Reassigning material data sets (on page 361)). Hence, it is better to create the data sets that will be used in later calculation phases beforehand and to use the *Staged construction* option to change data sets during calculations. The same applies to a change in water pressures and a change in input values of existing loads, since the latter is also possible using the *Staged construction* option (Geometry configuration - Staged Construction mode (on page 359) and Water pressure calculation (on page 321)).

7.12.8 *Automatic error checks*

During each calculation step, the PLAXIS 2D calculation kernel performs a series of iterations to reduce the out-of-balance errors in the solution. To terminate this iterative procedure when the errors are acceptable, it is necessary to establish the out-of-equilibrium errors at any stage during the iterative process automatically.

Two separate error indicators are used for this purpose, based on the measure of either the global equilibrium error or the local error. The values of both of these indicators must be below predetermined limits for the iterative procedure to terminate. These two error indicators and the associated error checking procedures are described below.

Global error check

The global error checking parameter used in the PLAXIS 2D calculation kernel is related to the sum of the magnitudes of the out-of-balance nodal forces. To obtain this parameter, the out-of-balance loads are non-dimensionalised as shown below:

$$\text{Global error} = \frac{\Sigma \| \text{ Out of balance nodal forces} \|}{\Sigma \| \text{ Active loads} \| \quad + CSP \cdot \| \text{ Inactive loads} \|} \qquad \textit{Eq. [92]}$$

where the *Out-of-balance nodal forces* are the difference between the external loads and the forces that are in equilibrium with the current stresses, the *Active loads* are the load difference between the actual calculation and the previous calculation phase (when $\Sigma M_{stage} = 0$ the *Active loads* are equal to the *Out-of-balance nodal forces*) and the *Inactive loads* are the active loads from the previous calculation phases.

In case of a flow calculation, the out-of-balance nodal flux will be used instead of the out-of-balance nodal forces.

CSP is the current value of the *Stiffness* parameter, defined as:

$$\text{Stiffness} = \int \frac{\Delta\varepsilon \cdot \Delta\sigma}{\Delta\varepsilon D^{e} \Delta\varepsilon} \qquad \textit{Eq. [93]}$$

which is a measure for the amount of plasticity that occurs during the calculation. See the Material Models Manual - *Preliminaries on material modelling - Elastic strains* for more information on the stiffness parameters. When the solution is fully elastic, the *Stiffness* is equal to unity, whereas at failure the *Stiffness* approaches zero. In the latter case the global error will be larger for the same out of balance force. Hence, it will take more iterations to fulfill the tolerance. This means that the solution becomes more accurate when more plasticity occurs.

Additional error check for structures

In case structures with rotational degree of freedom are included in the model, further checks are performed to ensure the robustness of the solution. The additional error check (AEC) for structures is defined as:

$$\text{AEC for structures} = \frac{\max \| \text{ Out of balance structure nodal moments} \|}{\max \| \text{ Structures nodal moments} \|} \qquad \textit{Eq. [94]}$$

where *max ||Out of balance structure nodal moments||* is the maximum out-of-balance moment of the structure nodes and *max ||Structure nodal moments||* is the maximum bending moment from the stress integration of all structures. In case of elastic structures, the *max ||Out of balance structure nodal moments||* is equal to 0.

Local error check

Local errors refer to the errors at each individual stress point. To understand the local error checking procedure used in PLAXIS 2D it is necessary to consider the stress changes that occur at a typical stress point during the iterative process. The variation of one of the stress components during the iteration procedure is shown in the figure below

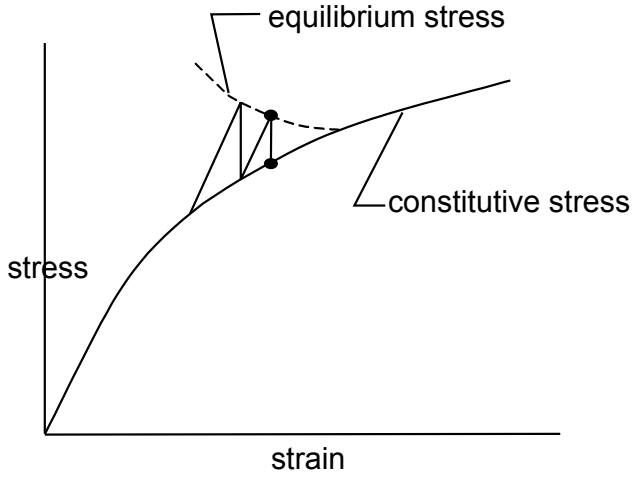


Figure 229: Equilibrium and constitutive stresses

At the end of each iteration, two important values of stress are calculated by PLAXIS 2D. The first of these, the 'equilibrium stress', is the stress calculated directly from the stiffness matrix (e.g. point **A** in Figure 229 (on page 382)). The second important stress, the 'constitutive stress', is the value of stress on the material stress-strain curve at the same strain as the equilibrium stress, i.e. point **B** in Figure 229 (on page 382).

The dashed line in Figure 229 (on page 382) indicates the path of the equilibrium stress. In general, this equilibrium stress path depends on the nature of the stress field and the applied loading. For the case of a soil element obeying the Mohr-Coulomb criterion, the local error for the particular stress point at the end of the iteration is defined:

$$\text{Local error} = \frac{\| \sigma^e - \sigma^c \|}{T_{\max}} \qquad \textit{Eq. [95]}$$

In this equation the numerator is a norm of the difference between the equilibrium stress tensor, σ^e, and the constitutive stress tensor, σ^c. This norm is defined by:

$$\| \sigma^e - \sigma^c \| = \frac{\sqrt{\left(\sigma^e_{xx} - \sigma^c_{xx}\right)^2 + \left(\sigma^e_{yy} - \sigma^c_{yy}\right)^2 + \left(\sigma^e_{zz} - \sigma^c_{zz}\right)^2 +}}{\left(\sigma^e_{xy} - \sigma^c_{xy}\right)^2 + \left(\sigma^e_{yz} - \sigma^c_{yz}\right)^2 + \left(\sigma^e_{zx} - \sigma^c_{zx}\right)^2} \qquad \textit{Eq. [96]}$$

The denominator of the equation for the local error is the maximum value of the shear stress as defined by the Coulomb failure criterion. In case of the Mohr-Coulomb model, T_{max} is defined as:

$$T_{\max} = \max\left(\frac{1}{2}\left(\sigma_3' - \sigma_1'\right),\ c \cdot \cos\varphi\right) \qquad \textit{Eq. [97]}$$

When the stress point is located in an interface element the following expression is used:

$$\text{Local error} = \frac{\sqrt{\left(\sigma^e_n - \sigma^c_n\right)^2 + \left(\tau^e - \tau^c\right)^2}}{c_i - \sigma^c_n \tan\varphi_i} \qquad \textit{Eq. [98]}$$

where σ_n and τ represent the normal and shear stresses respectively in the interface. To quantify the local accuracy, the concept of *inaccurate plastic points* is used. A plastic point is defined to be inaccurate if the local error exceeds the value of the user specified *tolerated error* (Numerical control parameters (on page 337)).

Termination of iterations

For PLAXIS 2D to terminate the iterations in the current load step, all of the following four error checks must be satisfied. For further details of these error-checking procedures, see Vermeer & van Langen (1989) (on page 532).

$$\begin{aligned} &\text{Maximum global error} \le \text{Tolerated error} \\ &\text{No. of inaccurate soil points} \le 3 + \frac{\text{No. of plastic soil points}}{10} \\ &\text{No. of inaccurate interface points} \le 3 + \frac{\text{No. of plastic interface points}}{10} \\ &\text{No. of inaccurate structure points} \le 3 + \frac{\text{No. of plastic structure points}}{10} \end{aligned} \qquad \textit{Eq. [99]}$$

where *Maximum global error* is the maximum between the *Global error* and the *Additional error check for structures*. The *Maximum global error* is the only one displayed during calculations.

8 Output program - General overview

This icon represents the Output program.

The main output quantities of a finite element calculation are the displacements and the stresses. In addition, when a finite element model involves structural elements, the structural forces in these elements are calculated. An extensive range of facilities exists within the PLAXIS 2D Output program to display the results of a finite element analysis. This chapter gives a description of the features available in the program.

If the Output program is activated by running its executable file or by clicking the *Output program* button in the Input program, the user has to select the model and the appropriate calculation phase or step number for which the results are to be viewed. More options on how to activate the Output are given in Accessing the Output program (on page 395).

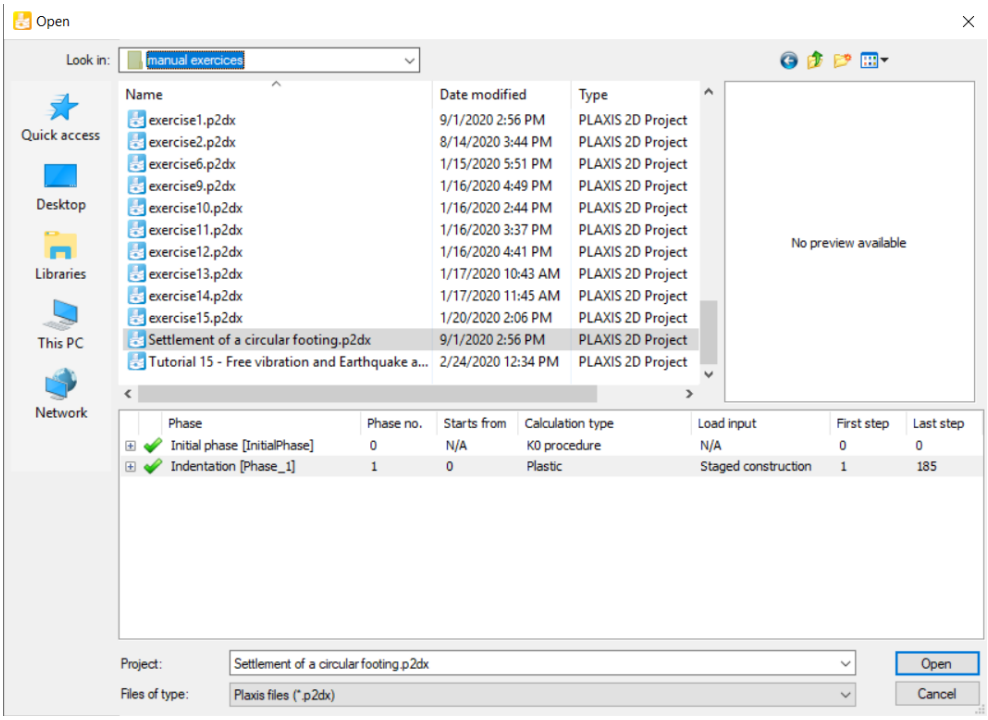


Figure 230: File requester of Output program

When a particular project is selected, the file requester displays the corresponding list of calculation phases from which a further selection should be made. If it is desired to select an intermediate calculation step, then a single mouse click should be given on the plus icon (+) at the left of the desired phase. As a result, the calculation list expands a list with all available step numbers for this phase, from which the desired step number can be selected.

Note:

Please note that the number of the individual steps available depends on the value assigned to *Max steps saved* in the **Parameters** tabsheet of the **Phases** window.

Once an output step of a particular project has been opened, the combo box in the toolbar will contain a list of available output steps, indicated by the step number and corresponding phase number.

8.1 Layout of the Output program

The layout of the Output program is shown below:

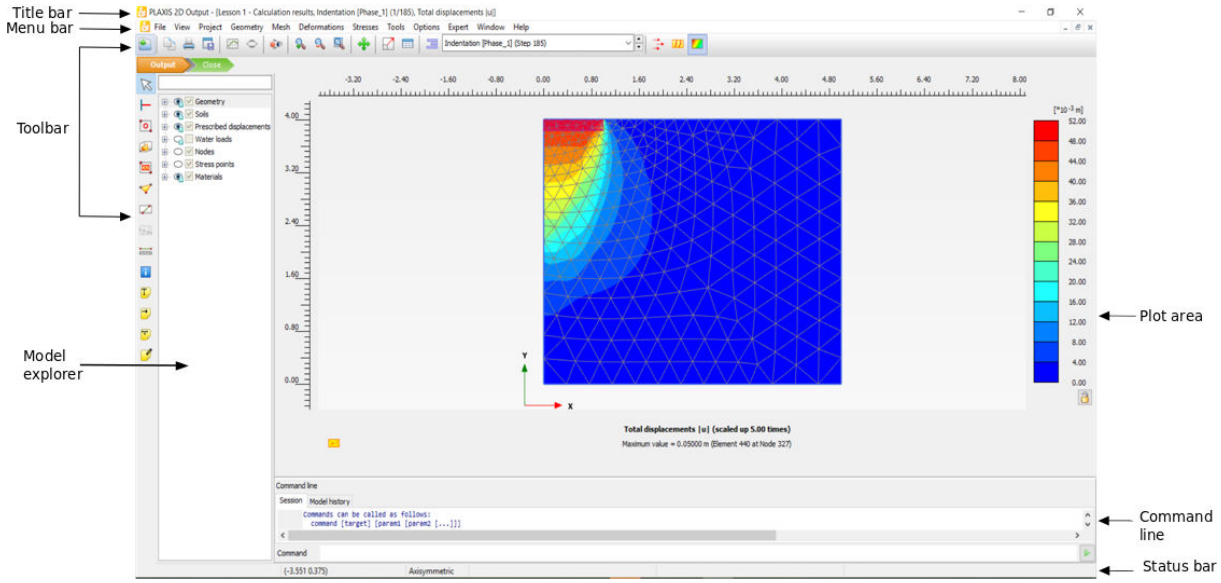


*Figure 231: **Main** window of the Output program*

8.1.1 *Title bar*

The title bar gives information about the project name, the step number and the type of information/results displayed.

8.1.2 *Menu bar*

The menu bar contains all output items and operations facilities of the Output program (Menus in the Menu bar (on page 386)).

8.1.3 *Toolbars*

Buttons for different features in the Output program are located above and at the left side of the plot area. A hint about the function of each tool is given as the cursor is located on it.

8.1.4 *Plot area*

The calculation results are displayed in the *Plot area*. The results can be displayed in graphical or tabular form. More information on how to handle the plot is given in Display area (on page 410).

8.1.5 *Status bar*

The status bar displays the locations of the cursor and the viewpoint and a hint about the object in the model and their element numbers.

8.1.6 *Command line*

PLAXIS 2D Output program enables to carry out certain actions using keyboard input by typing the corresponding commands in the command line.

More information about the available commands is provided when the *Command reference* option of the *Help* menu is selected.

8.1.7 *Explorers*

Information about the model and the project is given in the explorer.

Model explorer	It displays the properties of all the geometry objects and the features assigned to them in the model and gives the possibility to hide/show, activate/deactivate these objects, see the Model explorer (on page 38).

8.2 Menus in the Menu bar

The menu bar contains drop-down menus covering the options available in the Output program. The main results from a finite element calculation are deformations and stresses. Hence, these two aspects form the major part of the *Output* menu. When displaying a basic 2D geometry model, the menu consists of the *File*, *View*, *Project*, *Geometry*, *Mesh*, *Deformations*, *Stresses*, *Tools*, *Window* and *Help* menus. Note that the availability of the menus in the bar depends on the type of data that is presented on the output form.

8.2.1 *File menu*

Open project	To open the output of an existing project.
Close active project	To close all forms of the active project.
Close all projects	To close all forms of all opened projects.
Work directory	Set the default directory where PLAXIS 2D project files are stored.
Export to Paraview [GSE]	To export PLAXIS results to the visualization tool Paraview
Export to file	To export the information displayed, depending on the information type, to a text file (for results in tables) or image file (for plot).
Report generator	To generate a report of the project.
Create animation	Create an animation from selected output steps. The **Create animation** window is presented.
Print...	To print the active output on a selected printer.
(List of recent projects)	A list of the five most recent projects.
Exit	To leave the output program.

8.2.2 *View menu*

Zoom out	To restore to view before the most recent zoom action.
Reset view	To restore the original plot.
Save view	To save the current view (image or table). The saved views can be included in a report of the project.
Show saved views	To open or delete saved views.
Scale	To modify the scale factor of the presented quantity.
Legend settings...	To modify the range of values of the presented quantity in contour line plots and plots with shadings.

Scan line	To change the scan line for displaying contour line labels. After selection, the scan line must be drawn using the mouse. Press the left mouse button at one end of the line; hold the mouse button down and move the mouse to the other end. A contour line label will appear at every intersection of the scan line with a contour line.
Use result smoothing	To reduce the numerical noise resulting from the extrapolation of the results obtained in stress points (e.g. stress, force) to nodes. This option is available for plots and tables. Note that the option is by default selected in plot presentation of the results.
Rulers	To toggle the display of the rulers along the active plot.
Title	To toggle the display of the title of the active plot in the caption.
Model explorer	To toggle the model explorer panel.
Legend settings...	To modify the range of values of the presented quantity in contour line plots and plots with shadings.
Axes	To toggle the display of the global *x*- and *y*-axes in the active plot (displayed in the lower right corner).
Command line	To toggle the command line situated the bottom of the window.
Hide panels	To hide all the panels in the Output program.
Settings	To set various graphical attributes, such as object and background colours, symbol size, font size and diffuse shading.
Arrows*	To display displacement results as arrows.
Contour lines*	To display the results as contour lines.
Shadings*	To display the results as shadings.
Node labels*	To display the results at nodes.
Principal directions*	To display the principal stress directions in each stress point of the soil element.
Stress point labels*	To display the results at stress points.
Distribution*	To project the results perpendicularly to the structure, creating a distribution line for beams, embedded beams and anchors.
Wireframe distribution*	To project the results perpendicularly to the structure, creating a wireframe distribution line for beams, embedded beams and anchors.
Distribution envelope*	To project the envelope of all the phases results perpendicularly to the structure, creating a distribution line for beams, embedded beams and anchors.
Wireframe distribution envelope*	To project the envelope of all the phases results perpendicularly to the structure, creating a wireframe distribution line for beams, embedded beams and anchors.

Center principal directions*	To display the principal directions of stresses and strains at the center of each soil element.
Coloured principal directions*	To display the principal directions in each stress point of the soil element. Colours are used to distinguish the principal directions.
Coloured center principal directions*	To display the principal directions of stresses and strains at the center of each soil element. Colours are used to distinguish the principal directions.

Note:

- Menu options indicated with an asterisk (*) are only available for specific plot types.
- *Distribution envelope* and *Wireframe distribution envelope* show the whole history of the selected structures but the results (e.g. maximum and minimum value) correspond to the current phase and active part of the structure.

8.2.3 *Project menu*

Input nodes	View the table of the geometry input points.
Node fixities	View the table of the node fixities.
Water load information	View the table of the external water loads on the geometry boundaries in the current step.
Applied volume strain	View the table of the volume strain resulting at the end of the calculation phase.
Volume information	View the boundaries of the soil polygon, the total area of soil and the area of each cluster in the project.
Material information (all load cases)	View the material data of all load cases.
Material information (current load case)	View the material data of the current load case.
Deconfinement table	View the table with all applied deconfinements in the current step.
General information	View the general project information.
Calculation information	View the calculation information of the presented step.
Calculation info per phase	View the calculation information for each calculation phase.

Calculation info per step	View the calculation information for each calculation step.
Step info	View the step information of the presented step.
Structures per phase	View the active structures per calculation phase.

8.2.4 *Geometry menu*

Phreatic level	Toggle the display of the phreatic level in the model.
Frost line	Toggle the display of the frost line, i.e the boundary between frozen and unfrozen soil (T = 273.15 K = 0.0 °C = 32.0 °F)
Fixities	Toggle the display of the fixities in the model.
Connection axes	Toggle the display of the connection axes in the model.
Filter	Filter the nodes displayed in the model according to a defined criteria.

8.2.5 *Mesh menu*

Quality circles	The quality of the elements in the mesh is defined as inner circle divided by the outer circles of the soil or plate element where an equal sided triangleis normalised at 1.0.
Table of quality	View the table of the quality of the soil or plate elements according to different criteria.
Area	View the distribution of the area of the soil elements.
Table of element areas	View the table of the distribution of the areas of the soil elements.
Connectivity plot	View the connectivity plot (Connectivity plot (on page 423)).
Cluster borders	Toggle the display of the cluster borders in the model.
Element contours	Toggle the display of the element contours in the model.
Element deformation contours	Toggle the display of the deformed element contours in the model.
Materials	Toggle the display of the materials in the model.
Element numbers	Toggle the display of the soil element numbers.

Material set numbers	Toggle the display of the material set numbers in the soil elements.
Structure material set numbers	Toggle the display of the material set numbers of the structural elements.
Group numbering	Toggle the display of the group numbers. Groups are created according to the material setsand.
Cluster numbers	Toggle the display of the cluster numbers in the soil elements.
Node numbers	Toggle the display of the node numbers. Only possible when nodes are displayed.
Stress point numbers	Toggle the display of the stress point numbers. Only possible when stress points are displayed.
Selection labels	Toggle the display of the labels of the selected nodes or stress points.

8.2.6 *Deformations menu*

The *Deformations* menu contains various options to visualise the deformations (displacements, strains), the velocities and the accelerations (in the case of a dynamics analysis) in the finite element model (Deformations (on page 424)). These quantities can be viewed for the whole analysis (total values), for the last phase (phase values) or for the last calculation step (incremental values). In principle, displacements are contained in the nodes of the finite element mesh, so displacement related output is presented on the basis of the nodes, whereas strains are usually presented in integration points (stress points).

8.2.7 *Stresses menu*

The *Stresses* menu contains various options to visualise the stress state and other state parameters in the finite element model (Stresses (on page 428)). Stresses are contained in the integration points of the finite elements mesh, so stress related output is presented on the basis of the integration points (stress points).

8.2.8 *Forces menu*

The *Forces* menu contains various options to visualise the resulting forces in structural elements (Structures and interfaces (on page 441)).

8.2.9 *Interface stresses menu*

The *Interface stresses* menu contains various options to visualise the resulting stresses in interface elements (Structures and interfaces (on page 441)).

8.2.10 *Discontinuity stresses menu*

The *Discontinuity stresses* menu contains various options to visualise the resulting stresses in discontinuity elements (Structures and interfaces (on page 441)).

8.2.11 *Tools menu*

Copy	To copy the active output to the Windows clipboard.
Select points for curves	To enable selection of nodes and stress points to be considered in curves. All the nodes and stress points in the project are displayed enabling selection by clicking on them. The **Select points** window is activated, where the location of interest can be defined and the appropriate nodes or stress points can be selected form the list.
Mesh point selection	To activate the **Mesh selection** window. This option is active when the *Select points for curves* has been previously selected and the **Select points** window is closed.
Curves manager	To activate the **Curves manager** (Curves (on page 451)).
Table	To open a new form with a table of numerical values of the presented quantity.
Cross section	To select a user-defined cross section with a distribution of the presented quantity. The cross section must be selected by the mouse or by defining two points. Press the left mouse button at one end of the cross section; hold the mouse button down and move the mouse to the other end of the line. The cross section is presented on a new form.
Forces view	To open a new form with the possibility to visualise contact stresses and resulting forces on an arbitrary configuration of elements.
Structural forces in volume plates	To compute the forces in a structure which is modeled using soil with the material properties of the structure (e.g. concrete), after the calculation has already been finished. More information on the usage of this option is given in Structural forces in volume plates (on page 445).
Cross section curves	To display a plot of the results along the cross sections. The values in the x-axis in the plot are the distances of the points from the first point in the cross section.
Hint box	To display a hint box with information in individual nodes or stress points (if nodes or stress points are displayed).

Cross section points	To display the points defining the cross section. These points are displayed as greyed out in the **Cross section points** window. Their location can not be modified. This option is valid only when the *Cross section* view is active.
Distance measurement	To measure the distance between two nodes in the model both for the original mesh as well as the deformed mesh. This option is valid in the *Model* view only when nodes and/or stress points are displayed in the plot (Miscellaneous tools (on page 407)).
Remove annotations	To remove the annotations either partially (current view) or completely from the plots displayed in the output.

8.2.12 Options

Autocomplete [GSE]	To select the *Significant digits* for the project.
Licencing information	To switch on or off showing the licence information at startup.

8.2.13 Expert menu [GSE]

Run commands	To open the **Commands runner** window.
Macro library	To modify and run macros. Macros can be defined and indexed in the **Macro library** window, displayed as the corresponding option is selected in the sub-menu. To run a macro click the corresponding option in the sub-menu.
Python	To open and use **Interpreter**, **Editor**, **Command prompt** and to run python scripts (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Run Python script	To configure remote scripting server and open python script to run them (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Run Python tool	To configure remote scripting server and run python scripts (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).
Configure remote scripting server	To specify an available port and open it for connections by local or remote clients (Appendices Python HTTP REST API wrapper [GSE] (on page 558) and HTTP REST API [GSE] (on page 584)).

8.2.14 *Window menu*

Project manager	To view the projects and forms currently displayed in Output.
Duplicate model view	To duplicate the active view.
Close window	To close the active output form.
Cascade	To cascade the displayed output forms.
Tile horizontally	To tile horizontally the displayed output forms.
Tile vertically	To tile vertically the displayed output forms.
(List of recent views)	A list of the output forms.

8.2.15 *Help menu*

Manuals	To display the manuals.
Command reference	To display information about commands in the program.
Instruction movies	To reach the Bentley communities website where instruction movies are displayed.
Request support	To send a request for support.
Licence configuration	To view and configure licences.
Visit website	To reach the PLAXIS 2D product page.
News feed	Latest news and events of PLAXIS products.
Disclaimer	The complete disclaimer text is displayed.
About	Information about the program version and licence are displayed.
Graphics card information	Gathers your computer's graphics card information to be sent to our support department, if requested.

8.3 General information on command line

At the bottom of the Output application, a command line panel is available. The command line provides an alternative way of executing certain operations, by entering commands as text.

A feedback panel is situated above the command line panel. Here, the executed commands are shown, together with the feedback of the command. Successful commands return green coloured feedback, consisting of details on newly generated geometry or elements. Erroneous commands return red coloured feedback, containing an error report.

In the PLAXIS 2D Output program, a global command is included that shows information on the commands that can be executed: `cms`. This command lists the signatures of all commands available in the PLAXIS 2D Output program. An extended overview and description of all available commands is accessible via the *Help* menu in the user interface of the PLAXIS 2D Output program.

More information is available in General information on command line (on page 44).

8.3.1 *Indexing in commands*

Array indexing syntax can be used in the command line. This consists of square brackets behind an object name, followed by an integer and a matching closing bracket. Integer indexing is zero-based and works on any listable object, i.e. any object on which you can fire a filter or tabulate command. Both positive and negative numbering is used in indexing. The positive numbering begins at the start of the list, in which 0 is the first item from the table, while negative numbering starts from the end and goes backward through the list (see Indexing in commands (on page 46)).

8.4 Tools in the Output program

Besides displaying the calculation results, the Output program provides tools to handle the view and enable a better examination of the results. The buttons are grouped in the toolbar below the menu bar and in the side tool bar. The tools and their functionality is described in the following sections.

8.4.1 *Accessing the Output program*

All the results are displayed in the Output program. There are different ways to access the Output program. Besides the option of activating the program as described at the beginning of this chapter, the results can be displayed before or after the calculation of the phases is completed.

The results that can be displayed before calculating the phases are:

The generated mesh	The generated mesh is automatically displayed in the Output program as it is generated in the Input program.

Pore pressures	The generated pore pressures can be displayed when they are generated according to the phreatic level.
Connectivity plot	The *Connectivity plot* displays the distribution of the finite elements in the mesh and the nodes and stress points available. The *Connectivity plot* is displayed when the *Select point for curves* button is selected in the Calculations program. A more detailed description is given in Connectivity plot (on page 423).

Note:

Note that the groundwater calculations are performed when the phase is calculated. As a result, the pore pressure distribution is available only after the phase is calculated.

The calculation results of a project are displayed in the Output program by selecting a calculated phase in the *Phases explorer* and clicking the *View calculation results* button in the Calculation program.

While the Output program is already active, the results of other projects can be accessed either by clicking the *Open project* button or by selecting the corresponding option in the *File* menu (File menu (on page 386)).

8.4.2 *Exporting output data*

The PLAXIS 2D Output program enables exporting the displayed results such as plots or values. This is possible by clicking the corresponding button in the toolbar.

Copy to clipboard

Data as displayed in output forms may be exported to other programs using the Windows clipboard function. When clicking on the *Copy to clipboard* button, the **Copy** window appears in which selections can be made of the various plot components that are to be included in the copy (Figure 232 (on page 396)).

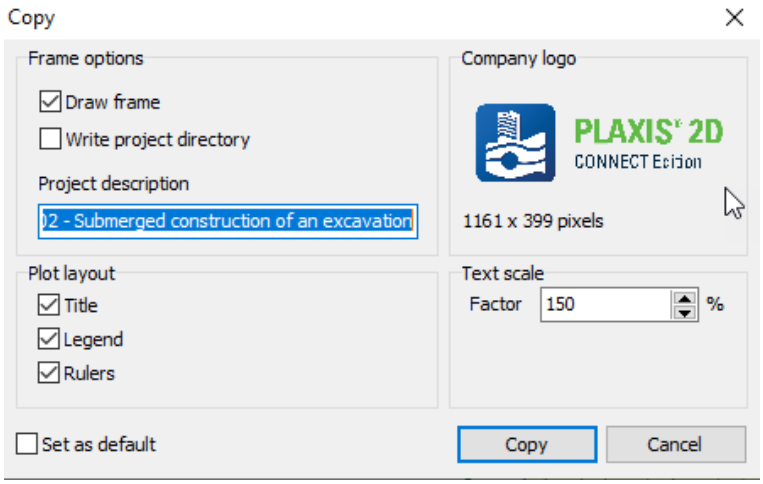


*Figure 232: **Copy** window*

Print

Hardcopies of graphs and tables can be produced by sending the output to an external printer. When the *Print* button is clicked or the corresponding option is selected in the *File* menu, the **Print** window appears, in which various plot components that are to be included in the hardcopy can be selected (Figure 233 (on page 397)).

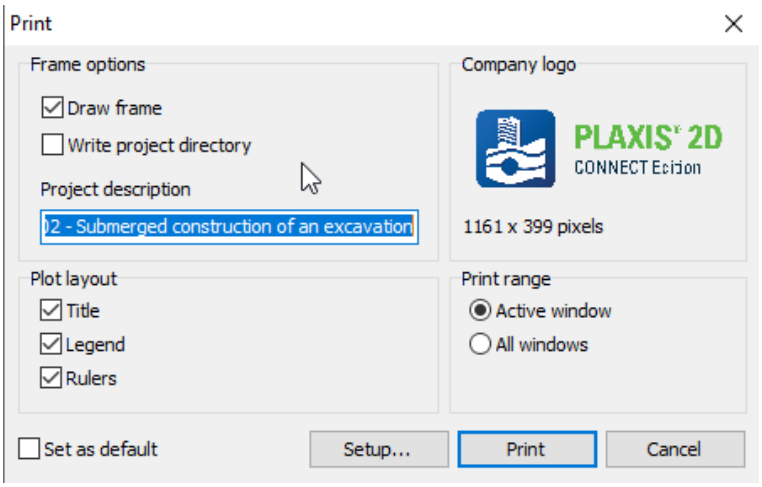


*Figure 233: **Print** window*

When pressing the *Setup* button, the standard printer setup window is presented in which specific printer settings can be changed. When the *Print* button is clicked, the plot is send to the printer. This process is fully carried out by the Windows® operating system.

Note:

When the *Copy to clipboard* option or the *Print* option is used on a plot that shows a zoomed part of the model, only the part that is currently visible will be exported to the clipboard or the printer.

Export

Data in output forms may be exported to files. When the *Export to file* button is clicked, the **Export** window appears. Note that a text scaling factor can be defined.

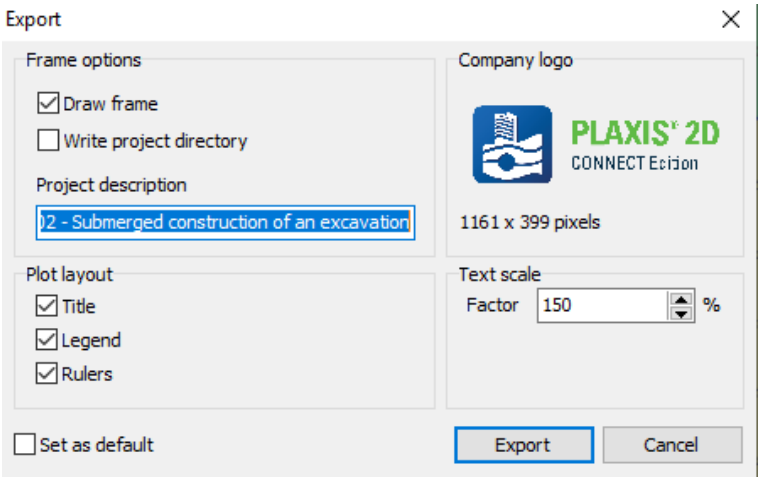


*Figure 234: **Export** window*

Instead of the PLAXIS 2D logo in the frame, it is also possible to insert a company logo. This logo has to be provided as a bitmap and can be selected in the **Print** window after clicking on the logo.

8.4.3 *Curves manager*

Clicking the *Curves manager* button activates the **Curves manager** window where curves can be generated to evaluate the results at specified locations in the model. Selection of points of interest and the generation of curves is described in detail in Curves (on page 451).

8.4.4 *Store the view for reports*

The views in the Output program can be saved to be used when reports are generated by clicking on the *Store the view for reports* button. The **Save view** window pops up as the button is clicked. Description can be given to the view in the Figure 235 (on page 398) window which can be beneficial when the report is generated. Report generation is described in detail in Report generation (on page 415).

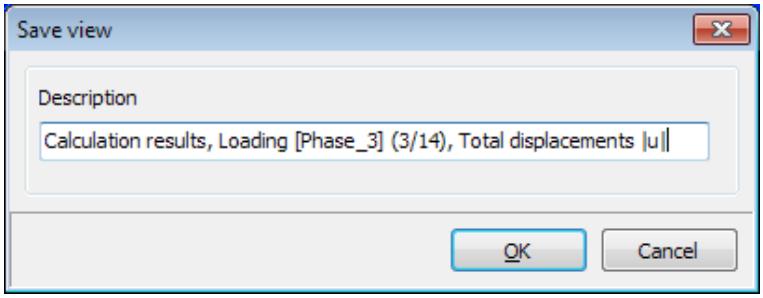


*Figure 235: **Save view** window*

8.4.5 *Zooming the plot*

It is possible to zoom in and out in the view of the plots by scrolling the mouse wheel. Other options for zooming are available by clicking the corresponding buttons in the toolbar.

- As this feature is selected, the mouse can be dragged on the model to define a local zooming rectangle. In the window, only the results in the defined rectangle will be displayed.
- Clicking the *Zoom out* button or selecting the corresponding option in the View menu (on page 387) restores the view of before the most recent zoom action.
- Clicking the *Reset view* or selecting the corresponding option in the View menu (on page 387) enables restoring the original plot.

8.4.6 Relocation of the plot

The plot can be moved using the mouse.

When this button is clicked, the plot will be relocated (moved) by clicking on the plot area and dragging it while keeping the left mouse button pressed.

8.4.7 *Scaling the displayed results*

Whenever the results are indicated by length entities such as *Arrows*, *Distribution*, *Axis*, etc. (Display type of results (on page 401)), the *Scale factor* button can be used to receive a better overview. When the button is clicked or the corresponding option in the *View* menu is selected, a window pops up where the factor can be defined. Note that this option is also available in the right mouse click pop-up menu.

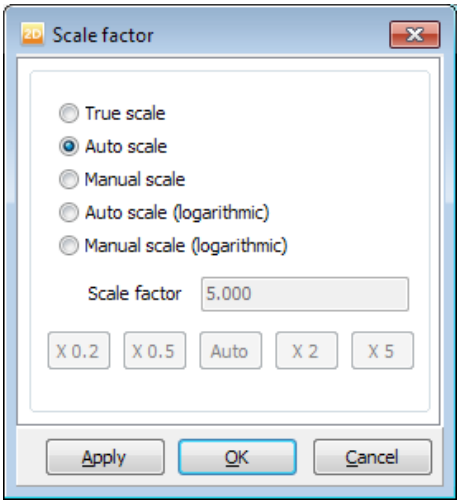


Figure 236: ***Scale factor*** *window*

Note:

- The default value of the *Scale factor* depends on the size of the model.
- The *Scale factor* may be used to increase or reduce the displayed (virtual) thickness of interfaces in the **Connectivity plot**.

8.4.8 *Tables*

The tabular form of the results given in the plot can be obtained by clicking on the *Table* button or by selecting the corresponding option in the menu. Note that this option is also available in the right mouse click pop-up menu.

Note:

The table of displacements may be used to view the global node numbers and corresponding coordinates of individual elements.

Displaying of tables

By default, a table is presented in ascending order according to the global element number and local node or stress point. However, a different ordering may be obtained by clicking on the small triangle in the column header of the desired quantity on which the ordering should be based. Another click on the same column header changes the ordering from ascending to descending.

The options available in the right click pop-up menu are:

Select for curves	To select the right clicked point in the table to be considered in curves.
Create chart	To create a chart from the values presented in the table.

Align	To align the text in the selected part of the table.
Decimal	To display data in decimal representation.
Scientific	To display data in scientific representation.
Decimal digits	To define the number of decimal digits displayed.
View factor	To define a factor to the values in the table.
Copy	To copy the selected values in the table.
Find value	To find a value in the table.
Find soil element	To find a soil element with a specified ID in the table when the results are displayed for soil elements.
Find structural element	To find a structural element with a specified ID in the table when the results are displayed for structures.
Filter	To filter the results in the table.

Note:

The values in the tables contain the most accurate information, whereas information in plots can be influenced or be less accurate due to smoothing or extrapolation of information from stress points to nodes.

8.4.9 *Selection of results*

As the type of result is selected from the *Deformations*, *Stresses* or *Structures* menu, the results are displayed either in plots or tables according to the selection made.

While the Output program is running, other steps of the project can be selected from the drop-down menu in the toolbar. The button in front of the drop-down menu can be used to toggle between the end results of phases, or individual output steps:

- A list of the calculation phases and their final calculation steps is given. The results at the end of the final calculation steps can be shown for each phase.
- A list of the saved calculation steps and the calculation phase they belong to is given. The results of each calculation step can be shown.

In addition to the drop-down menu, the spinner at the right of the drop-down menu or using the *Ctrl-Up* and *Ctrl-Down* keys will select the end results of the previous or next calculation step or calculation phase.

8.4.10 *Display type of results*

The plot type options are located at the right of the drop-down menu:

- The results are displayed as contours.
- The results are displayed as shadings.
- The results for *Displacements* can be displayed as arrows. Scaling of the results is possible.
- The results are displayed at each stress point of the soil elements. The length of each line represents the magnitude of the principal quantity (stress or strain) and the direction indicates the principal direction. Positive direction is indicated by arrows. Scaling of the results is possible.
- The average results are displayed at the center of each soil element. The length of each line represents the magnitude of the principal quantity (stress or strain) and the direction indicates the principal direction. Positive direction is indicated by arrows. Scaling of the results is possible.
- The results are displayed in different colours at each stress point of the soil elements. The length of each line represents the magnitude of the principal quantity (stress or strain) and the direction indicates the principal direction. Positive direction is indicated by arrows. Scaling of the results is possible.
- The average results are displayed in different colours at the center of each soil element. The length of each line represents the magnitude of the principal quantity (stress or strain) and the direction indicates the principal direction. Positive direction is indicated by arrows. Scaling of the results is possible.
- The deformed shape of cross sections, plates, geogrids or interfaces is displayed. The relative deformation is indicated by arrows. Scaling of the results is possible.
- The distribution of the results in cross sections, plates, geogrids or interfaces is displayed. Scaling of the results is possible.
- The wireframe distribution of the results in cross sections, plates, geogrids or interfaces is displayed. Scaling of the results is possible.
- The distribution of the maximum and the minimum values of the resulting forces in plates, geogrids and node-to-node anchors up to the current calculation step is displayed. Scaling of the results is possible.
- The wireframe distribution of the maximum and the minimum values of the resulting forces in plates, geogrids and node-to-node anchors up to the current calculation step is displayed. Scaling of the results is possible.
- The *Plastic points* option shows the stress points that are in a plastic state, displayed in a plot of the undeformed geometry (Plastic points (on page 438)). Scaling of the results is possible. When scaling is used, it is possible to pull the interfaces out of the plates, however the stress points will remain at their physical locations.

Note:

For (enhanced) safety calculations, results are NOT shown in soil clusters that are excluded from the strength reduction procedure as long as the *Hide items without strength reduction* option in the *View* menu has been selected.

The availability of the display type buttons in the toolbar can be toggled on/off by selecting the corresponding options in the *View* menu.

8.4.11 *Select structures*

By default, all the active structures and interfaces in the selected phase are displayed in the plot. The disabled structures can be displayed by selecting the corresponding option in the *Geometry* menu.

Output for structures and interfaces can be obtained by clicking the *Select structures* button and then double clicking the desired object in the 2D model. As a result, a new form is opened on which the selected object appears. At the same time the menu changes to provide the particular type of output for the selected object.

Another option of selecting structural elements in the output is by clicking on the *Select structures in a rectangle* and drawing a rectangle in the model. As a result, the structures in the rectangle will be selected.

To clear the selection, press *Esc*. Only structural elements of the same type can be selected at the same time. For example, if a geogrid is selected, only other geogridscan be added to the selection and no embedded beam rows or plates.

8.4.12 *Partial geometry*

To enable the inspection of certain internal parts of the geometry (for example individual layers or volume clusters) it is possible to make other parts of the geometry invisible in the *Model explorer* by clicking the button in front of them as shown in Figure 237 (on page 403).

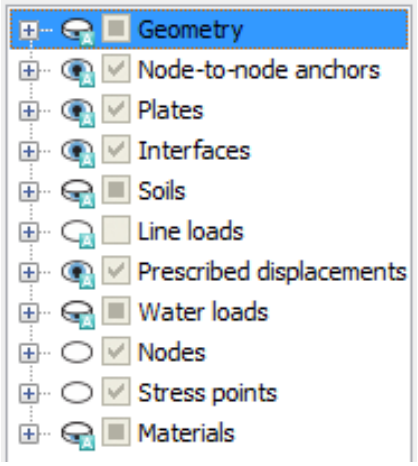


Figure 237: Model explorer in Output

Visible model components are indicated by an open eye, whereas invisible ones are indicated by a closed eye. By clicking on the button, the view of the components (individual and/or groups) can be toggled from being visible to being invisible and vice versa. A group is expanded by clicking on the + sign in front of the group. Clusters that have been set inactive in the framework of staged construction are always invisible and cannot be made visible.

The user will be able to show/hide clusters in the *Model explorer* under the option of *Soils* based on the materials assigned to them.

Note:

The cluster numbers are activated by selecting the *Cluster numbers* option in the *Mesh* menu.

The information in the *Model explorer* can be narrowed according to the filtering criteria specified at the corresponding cell.

The *Show all* option will make all the object active in the selected phase visible. The *Hide all* will revert the action. The *Invert selection* option will toggle all visible elements invisible and all invisible elements visible. The *Deselect all* button will set all elements to invisible. On pressing the *Close* button the **Partial geometry** window is closed without further changes.

Apart from the *Model explorer*, individual volume elements or entire clusters of volume elements can be made invisible by holding down the *Ctrl* key, the *Shift* key or both keys at the same time, respectively, while clicking on an element in the 2D model. These elements can be visible again by clicking the corresponding check boxes in the *Model explorer*.

Clicking the *Hide soil* button in the side bar menu enables hiding parts of the soil. To hide soil elements, click the *Hide soil* button first and hold the *Ctrl* key pressed while clicking on the soil elements. To hide soil clusters, click the *Hide soil* button first and hold the *Shift* key pressed while clicking on the soil clusters.

Clicking on the *Hide soil in the rectangle* button enables hiding the soil in the rectangle drawn in the model. The drawing order of the rectangle effects the resulting hidden soil elements.

To hide only the soil elements that fall completely in the defined rectangle, first click the *Hide soil in the rectangle* button. In the model, click at the point defining the upper left corner of the rectangle, drag the mouse to the point defining the lower right corner of the rectangle and click again.

To hide all the soil elements that are intersected by the defined rectangle, first click the *Hide soil in the rectangle* button. In the model, click at the point defining the lower right corner of the rectangle, drag the mouse to the point defining the upper left corner of the rectangle and click again.

8.4.13 *Viewing results in cross sections*

To gain insight in the distribution of a certain quantity in the soil it is often useful to view the distribution of that quantity in a particular cross section of the model. This option is available in PLAXIS 2D for all types of stresses and displacements in soil elements .

A cross section can be defined by clicking the *Cross section* button in the side button bar or by selecting the *Cross section* option in the *View* menu. Note that this option is also available in the right mouse click pop-up menu. Upon selection of this option, the **Cross section points** window pops up in which the two cross section coordinates can be defined.

After the cross section has been selected, a new form is opened in which the distribution of a quantity is presented on the indicated cross section. At the same time, the menu changes to allow for the selection of all other quantities that may be viewed on the indicated cross section.

Note:

The distribution of quantities in cross sections is obtained from interpolation of nodal data and may be less accurate than data presented in the 2D model.

Multiple cross sections may be drawn in the same geometry. Each cross section will appear in a different output form. To identify different cross sections, the end points of a cross section are indicated with characters in alphabetical order. The points defining the cross section can be viewed by selecting the *Cross section points* option in the *Tools* menu.

In addition to the output quantities that are available for the 2D model, a cross section allows for the display of cross section stresses, i.e. effective normal stresses σ'_N, total normal stresses σ_N, vertical shear stresses τ_s and horizontal shear stresses τ_t.

Note:

It is possible to move a cross section in the direction of its normal while the presentation of results is updated for the new location of the cross section.

- Using the *Ctrl+-* and *Ctrl++* keys will move the cross section 1/100 times the diagonal of the geometry model.
- Using the *Ctrl+Shift+-* and *Ctrl+Shift++* will move the cross section 1/1000 times the diagonal of the geometry model.

8.4.14 *Plot annotations*

PLAXIS 2D enables addition of user-defined information to the output plots. The buttons in the side toolbar provide different annotation options.

Label annotations in plots

PLAXIS 2D enables adding labels as annotations in plots. To add a label annotation to a plot:

- Click the *Add label annotation* button in the side toolbar.
- Double click the location on the plot where the label is to be located. The **Annotation window** for the label pops up (Figure 238 (on page 406)).
- Define the type by selecting the corresponding option in the **Caption** drop down menu. The options available for the *Caption type* are:

user-defined	To enter a label defined by the user.
Node ID	To display the ID of the double clicked node.
Result value	To display the result (i.e. the displacement, groundwater head or other result, depending on the selected Output plot) at the double clicked node.

Note:

The information available for the annotation depends on whether a node is double clicked or not. When a node is double clicked, besides *User-defined* text, information such as the node ID, result value at the node, the type of

element to which the node corresponds and the number of that element is provided. If a random location is double-clicked in the plot the only option available for the *Caption* is *User-defined*.

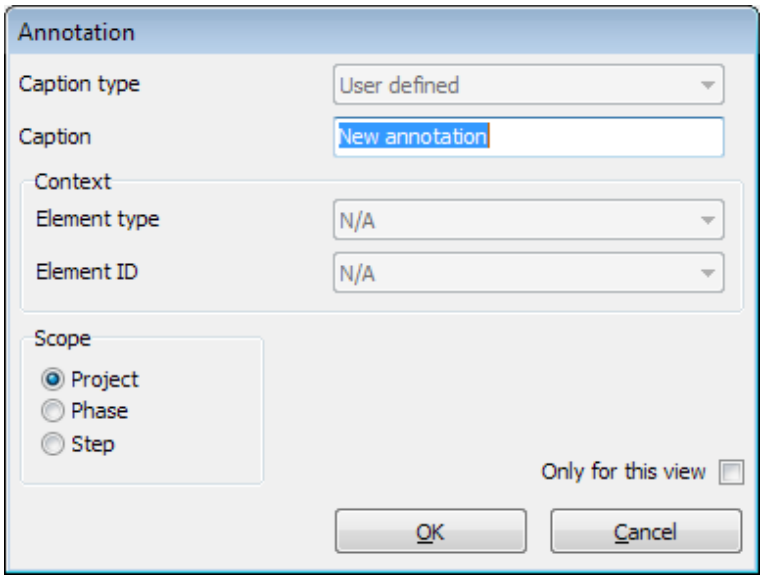


Figure 238: Annotation window for labels

- If the *User-defined* option is selected, specify the label in the *Caption* cell.
- If either the *Node ID* or the *Result value* is selected from the *Caption* drop-down menu, in the *Context* box, select the element type to which the node belongs in the *Element type* drop down menu. Depending on the model, the options might be *Soil-element* or *Structural-element*.
- A node can be shared by multiple elements. To specify the element of interest select the corresponding option from the *Element ID* drop down menu in the *Context* box. Note that the *Context* box is only available if an annotation is assigned to a node (a node is double-clicked).
- Select one of the options available for the *Scope* box to prevent undesired display of the annotation in the plot. The defined annotations can be relevant for the whole project (*Project* option), only the current phase (*Phase* option) or only the current calculation step (*Step*).
- To limit the display of the annotation to the current view select the corresponding checkbox in the window. Note that if this option is selected, the current view should be saved to preserve the defined annotation.

Line annotations in plots

PLAXIS 2D enables adding lines or arrows as annotations in plots. To add a line annotation to a plot:

- Click the *Add line annotation* button in the side toolbar.
- Define the start and the end points of the line by clicking on the plot. When the end point is defined the **Annotation** window for lines pops up (Figure 238 (on page 406)).
- Use the *Style* options available in the drop down menus for the start and end point of the line and for the line itself to modify the style of the line.
- Specify the thickness of the line and the arrows in the corresponding cell.
- Select one of the options available for the *Scope* to prevent undesired display of the annotation in the plot. The defined annotations can be relevant for the whole project (*Project* option), only the current phase (*Phase* option) or only the current calculation step (*Step*).

- To limit the display of the annotation to the current view select the corresponding checkbox in the window. Note that if this option is selected, the current view should be saved to preserve the defined annotation.
- The *Delete* button is only available when editing the annotations (see below).

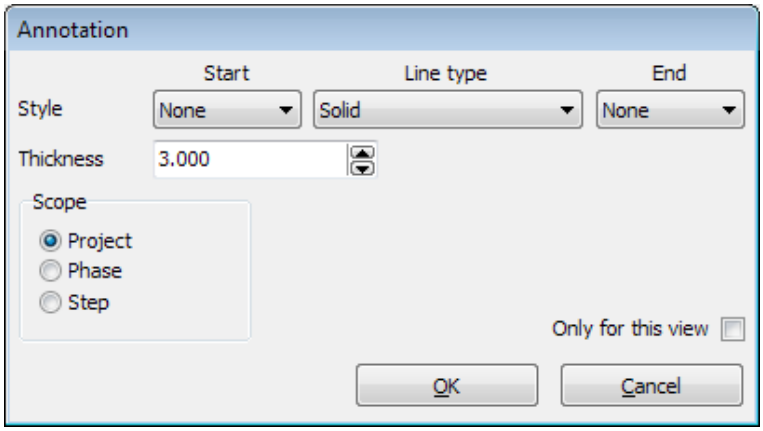


Figure 239: Annotation window for editing

Measurement annotations in plots

PLAXIS 2D enables adding measurement annotations in plots displaying the distance between two locations in the model. To add a measurement annotation to a plot:

- Click the *Add measurement annotation* button in the side toolbar.
- Define the start and the end points of the line by clicking on the plot. The distance between the specified locations is displayed in the model.

Editing annotations

To modify or remove an annotation from the plot:

- Click the *Edit annotation* button in the side toolbar.
- Click the annotation to be modified. The *Annotation* window will appear displaying the modification options depending on the clicked annotation. Note that a new button (*Delete*) is available in the window, enabling the removal of the annotation (Figure 239 (on page 407)).

8.4.15 *Miscellaneous tools*

Distance measurement

The distance between two nodes in the model can be measured by either clicking on the *Distance measurement* button or by selecting the *Distance measurement* option in the *Tools* menu and by subsequently selecting the nodes in the model. The **Distance information** window pops up displaying the information about the distance (Figure 240 (on page 408)).

The distance can be given according to the original node position or in deformed shape (i.e. using shifted node positions according to their displacements).

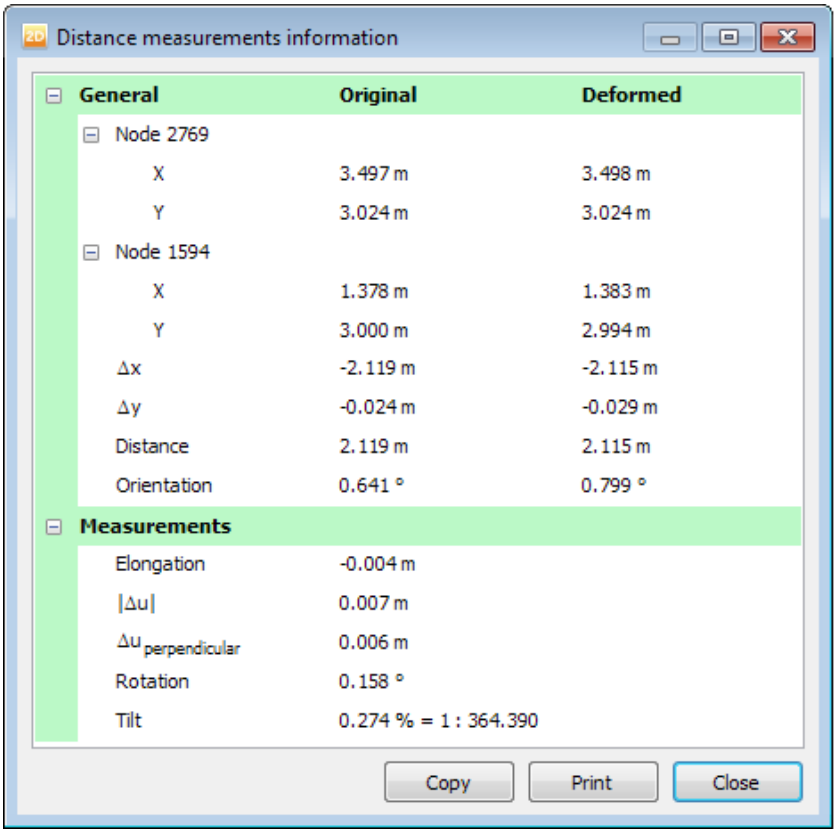


Figure 240: Distance information

A description of the information available in the table is given as follows:

Coordinates	The *Original* and *Deformed* coordinates for the first node/stress point and the second node/stress point.
Δ x	The *Original* and *Deformed* x-component of the distance between the nodes.
Δ y	The *Original* and *Deformed* y-component of the distance between the nodes.
Elongation	The difference between the original vector and the projection of the deformed vector onto the original vector (Figure 241 (on page 409)). Hence, *Elongation* does not consider the rotation of the line between the two points. If the *Updated mesh* analysis is performed, *Elongation* is the difference in distance between the original and deformed vectors.
Distance	The *Original* (v) and *Deformed* (v') distance between the points.
Orientation	The angle between the measurement line and the x-axis. Orientation is positive if the vector describing the line points upwards (*Y>0*) and negative if it points downwards (*Y<0*). The *Orientation* is calculated for the undeformed and deformed measurement line.
|Δ u|	The change in the distance between the selected points before and after deformation.

$\Delta u_{perpendicular}$	The deformation in the direction perpendicular to the original line between the selected points.
Rotation	The difference between the deformed orientation and the original orientation (Figure 241 (on page 409)). The sign of the rotation is determined using the right-hand rule (clockwise rotation is negative, counter-clockwise is positive).
Tilt	The ratio of the deformation in the direction perpendicular to the line between the selected points, $\Delta u_{perpendicular}$, to the original distance between the selected points, v, () given both as ratio and percentage: $$\text{Tilt} = \tan(\alpha_{\text{Tilt}}) = \frac{\Delta u_{\text{perpendicular}}}{v}$$

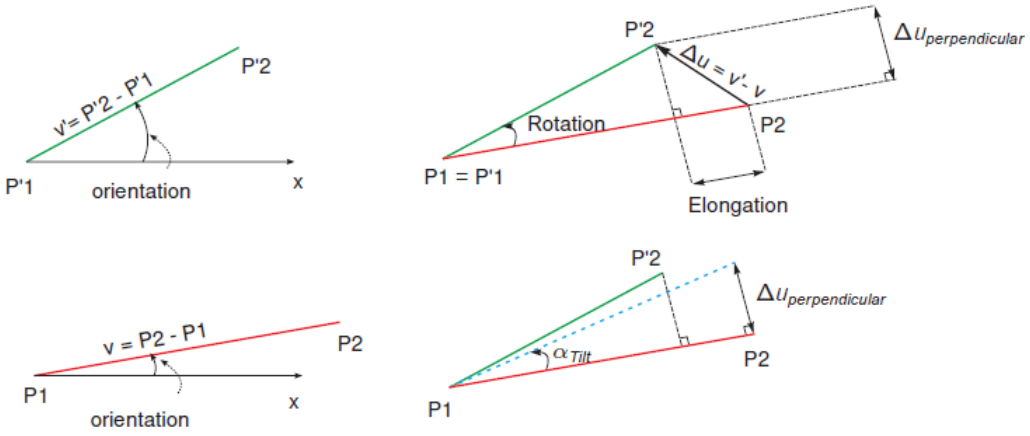


Figure 241: Deformation measurement

Draw scanline

When the *Contour lines* option is selected , a distribution of the values can be displayed by clicking on the *Draw scanline* button in the side toolbar and drawing a line on the regions of interest. Note that this option is also available in the right mouse click pop-up menu.

Hint box with node or stress point data

When nodes or stress points are displayed in the model using the corresponding option in the *Tools* menu, it is possible to view data of these points in a hint box. This can be done by clicking on the corresponding button in the side bar menu. If this option is active and the mouse is moved over a node, the hint box shows the global node number, the node coordinates and the current displacement components.

If the *Hint box* option is active and the mouse is moved over a stress point, the hint box shows the global stress point number, the current Young's modulus E, the current cohesion c, the current overconsolidation ratio *OCR*, the current principal stresses and a sketch of Coulomb's envelope and Mohr's circles for that stress point.

Selecting nodes or stress points for curves

Nodes and stress points can be selected in the Output program by clicking the *Select nodes and stress points* button in the side toolbar. Make sure the *Nodes* and/or *Stress points* option has been selected in the *Mesh* menu. Nodes are generally used to draw displacements whereas stress points are generally used to draw stresses or strains.

Note that for the nodes and stress points selected after the calculation process information is only valid for the saved calculation steps. For a more detailed description see Selecting points for curves (on page 451).

Interactive ruler

The result value at a specific location in a structure or cross section can be displayed by clicking the *Interactive ruler* button in the side toolbar and by moving the cursor to the point of interest. The current value (corresponding to the point on the cross section line), a minimum value (based on the minimum value in the distribution), and a maximum value (based on the maximum value in the distribution) are shown along the ruler. The *Interactive ruler* is available in the **Structure** and **Cross section** views.

8.5 Display area

The distribution of the results in the model is shown in the display area.

The presence of the legend, title bar, and axes in the drawing area is arranged using the options in the *View* menu (View menu (on page 387)).

Note:

The icon in the title bar indicates the view in which the results are displayed. A more detailed description on *Views* is given in View menu (on page 387).

8.5.1 *Legend*

The *Legend* is available for the display options where a variation in colour describes the variation in the displayed result values. It is activated by selecting the corresponding option in the *View* menu.

When the *Legend* is double clicked, the *Legend settings* window pops up, where the scaling and the colouring can be defined (Figure 242 (on page 411)). Note that this option is also available in the right mouse click pop-up menu. *Automatic*, *Manual* or *Logarithmic* options are possible for scaling of results.

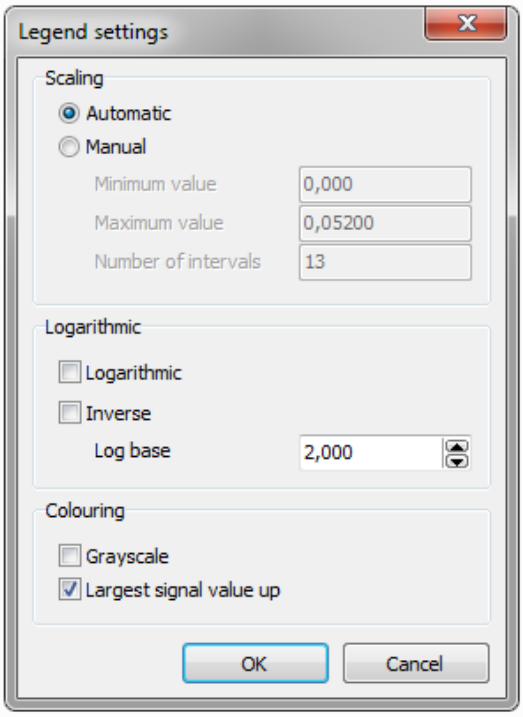


Figure 242: ***Legend settings*** *window*

The distribution of values in the legend can be locked by clicking the *Lock the legend* button. When the legend is locked, the value distribution will not change as the *Ctrl*+*+* or *Ctrl*+- keys are used to move the cross section through the model or when similar result of other calculation phases or projects are displayed.

8.5.2 *Modifying the display settings*

The view settings can be defined in the **Settings** window, that is activated when the corresponding option in the *View* menu is selected. Note that this option is also available in the right mouse click pop-up menu.

The visualization settings can be defined in the **Visualization** tabsheet of the **Settings** window (Figure 243 (on page 412)).

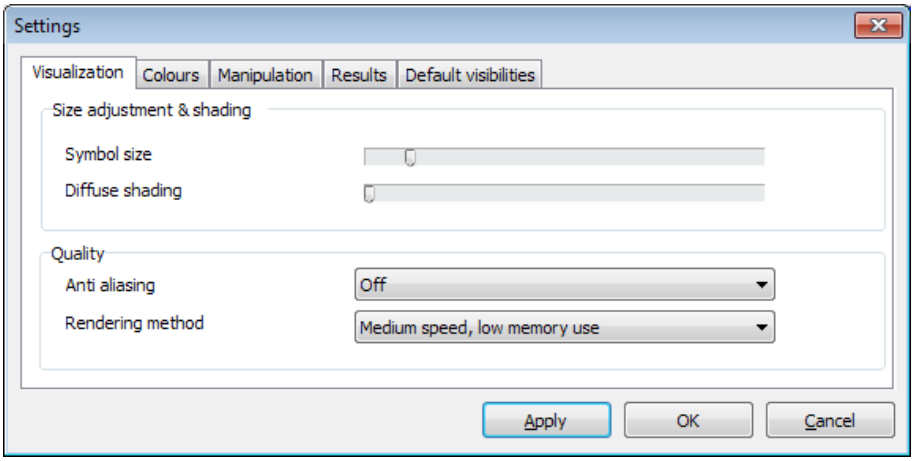


*Figure 243: **Visualization** tabsheet of the **Settings** window*

Symbol size	To modify the size of the symbols in the display for nodes, forces, etc.
Diffuse shading	To make the appearance of the model even more realistic, the *Diffuse shading* option may be used. Using this option, object surfaces that have the same colour by definition (such as soil elements with the same material data set) appear 'brighter' or 'darker', depending on their orientation with respect to the viewer. Object surfaces appear the brightest when the normal to the surface points in the direction of the viewer. The surfaces become darker the more the normal deviates from this direction. The contrast can be set to the desired magnitude using the slide bar.
Anti aliasing	To select a convenient anti aliasing method from the options available in the drop-down menu.
Rendering method	To select a convenient rendering method from the options available in the drop-down menu.
Display	Toggle the display of the *Cluster borders*.

The displaying colours can be arranged in the **Colours** tabsheet of the **Settings** window (Figure 244 (on page 413)).

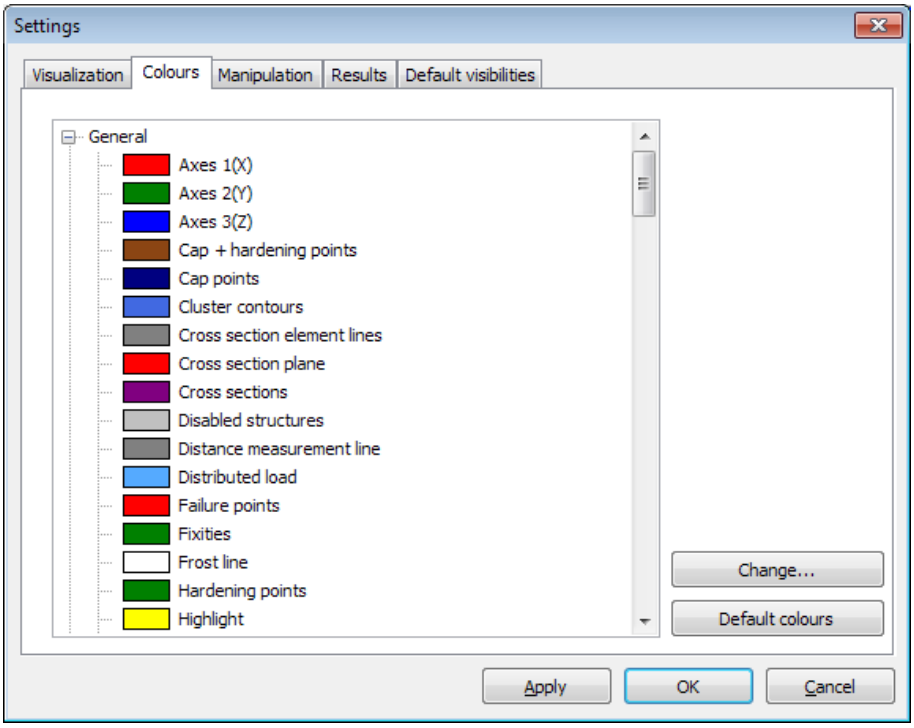


*Figure 244: **Colours** tabsheet of the **Settings** window*

The function of the left and the middle mouse buttons can be defined in the **Manipulation** tabsheet of the **Settings** window (Figure 245 (on page 413)).

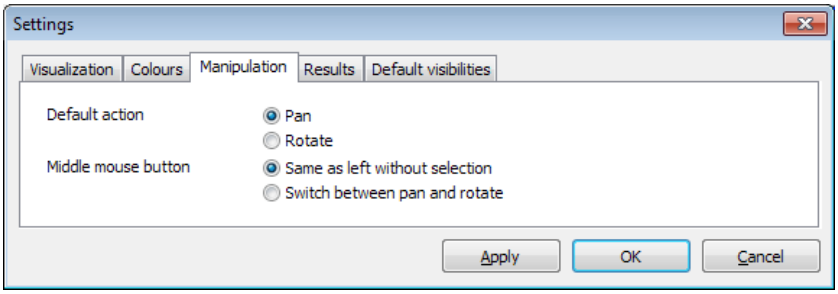


*Figure 245: **Manipulation** tabsheet of the **Settings** window*

The display of particular results can be toggled on/off in the **Results** tabsheet of the **Settings** window (Figure 246 (on page 413)).

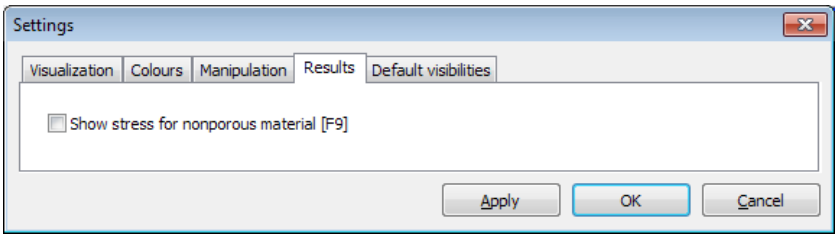


*Figure 246: **Results** tabsheet of the **Settings** window*

The default visibility of *Nodes*, *Stress points* and *Disabled structures* can be toggled on/off in the **Default visibilities** tabsheet of the **Settings** window (Figure 247 (on page 414)).

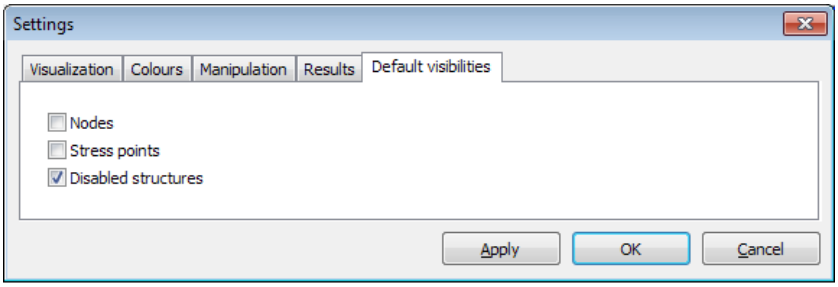


*Figure 247: **Default visibilities** tabsheet of the **Settings** window*

8.6 Views in Output

In the Output program the results are displayed in different views. The view type is indicated by the corresponding icon in the title bar. The available views are:

8.6.1 *Model* view

In the *Model* view the results are displayed in the whole model. This is the default display of results.

8.6.2 *Structures* view

When a structure (or multiple structures) is selected and double clicked, the variation of the result is displayed in the *Structures* view.

8.6.3 *Cross section* view

In the *Cross section* the results in the defined cross section are displayed. A cross section can be moved perpendicular to the cross section using the *Ctrl-[* and *Ctrl-]* keys. Simultaneously pressing the *Shift* key moves in small steps.

8.6.4 *Forces* view

The *Forces* view enables a view of the mesh with contact stresses and (resulting) forces on the boundaries of the visible active parts of the mesh. This option can be selected from the *Tools* menu.

For stresses:

Water load	Only external water pressures and pore pressures are shown

Normal stress	Only effective normal stresses are shown
Shear stress	Only shear stresses are shown
Total stress	Effective normal stresses (red) as well as external water pressures and pore pressures (blue) are shown

It can be selected whether external loads and resulting forces from prescribed displacements are taken into account. It can be selected whether resulting forces from water loads, effective soil stresses, forces from structures, gravity forces, external loads and forces from prescribed displacements are taken into account.

The *Partial geometry* option can be used to make parts of the mesh invisible, if necessary. In this way, all stresses and forces on sub-structures can be visualised.

The *Table* option may be used to view the actual values of stresses and forces. The table of forces also shows the resulting force below the table, both as an actual value and as a percentage of the total applied forces. The latter can be used to evaluate if there is a significant unbalance of the (sub-)structure. If necessary, the calculation may be repeated using a smaller tolerated error or a finer mesh.

8.7 Report generation

To document project input data and computational results, a *Report Generator* facility is available in the PLAXIS 2D Output program. The *Report generator* option can be selected from the *File* menu. The data files for the report are generated in the following eight steps.

Step 1: The report can be generated in a group of files or all the information can be combined in a single file (a RTF, PDF or HTML document). The directory where the report is stored should be defined (Figure 248 (on page 415)).

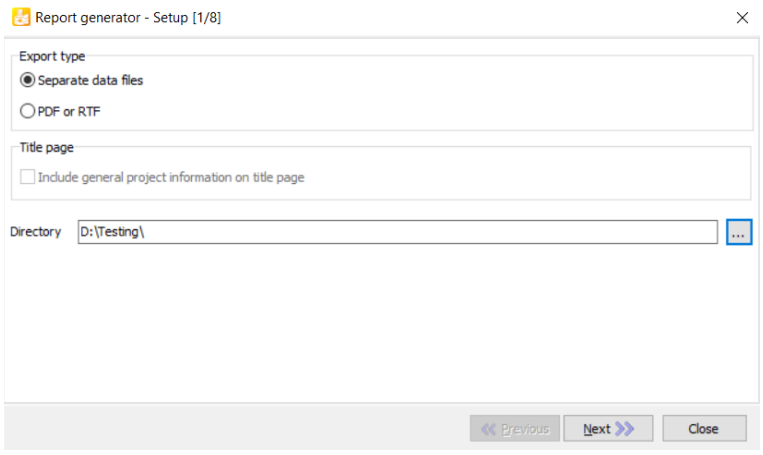


Figure 248: Report generator - Setup

Step 2: Select the phases for which results will be included in the report (Figure 249 (on page 416)).

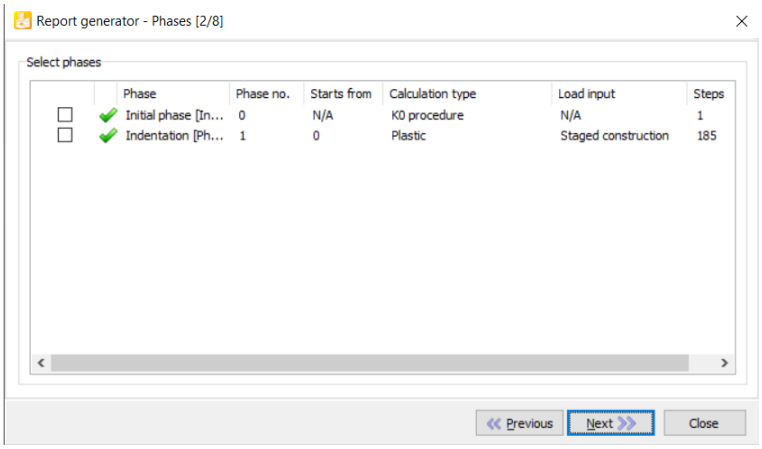


Figure 249: Report generator - Phases

Step 3: Select general information sets to be included in the report. Note that the selection can be saved as a new set besides *All* and *None* sets (Figure 250 (on page 416)).

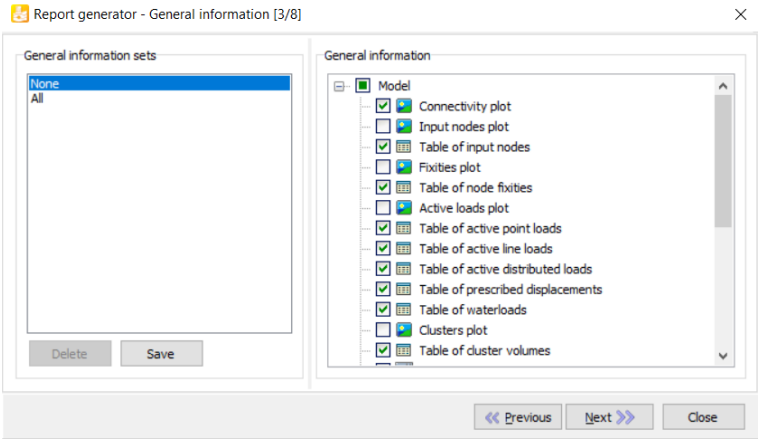


Figure 250: Report generator - General information

Step 4: Select model view sets to be displayed in the report. Note that the selection can be saved as a new set besides *All* and *None* sets (Figure 251 (on page 417)).

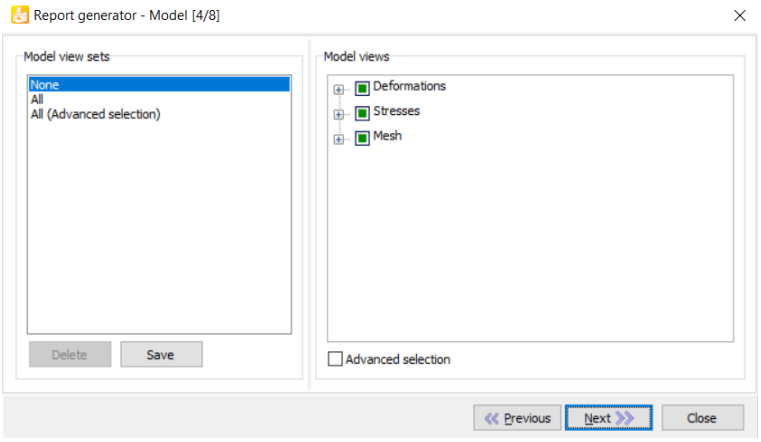


Figure 251: Report generator - Model

Step 5: Select structure view sets to be displayed in the report are selected. Note that the selection can be saved as a new set besides *All* and *None* sets (Figure 252 (on page 417)).

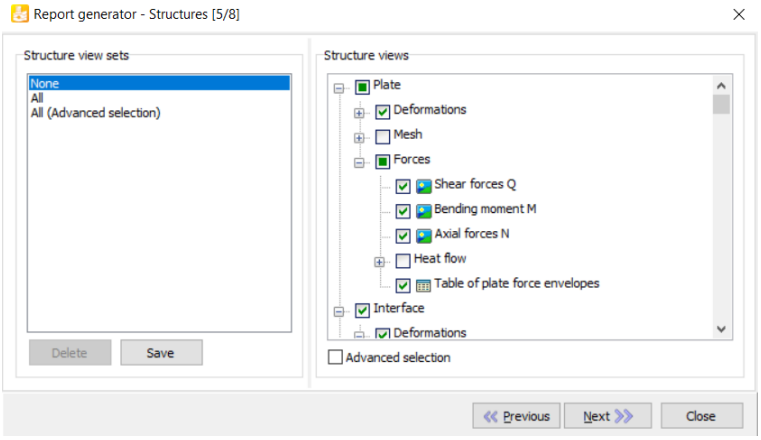


Figure 252: Report generator - Structures

Step 6: Select saved views to be included in the report (Figure 253 (on page 418)). For more details on saved views, see Store the view for reports (on page 398).

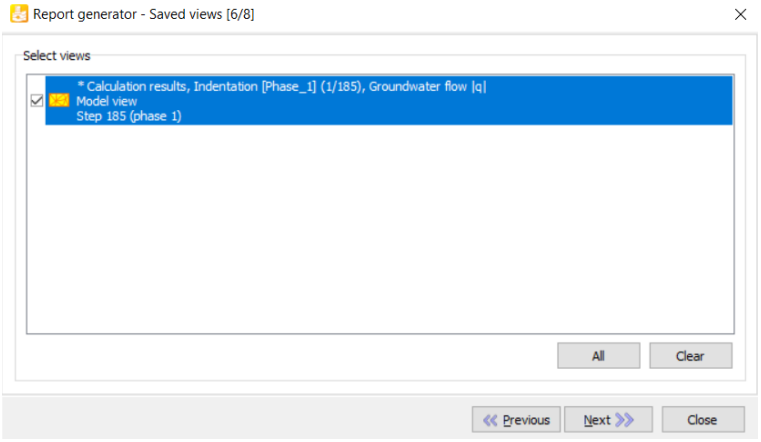


Figure 253: Report generator - Saved views

Step 7: Select generated charts to be included in the report (Figure 254 (on page 418)).

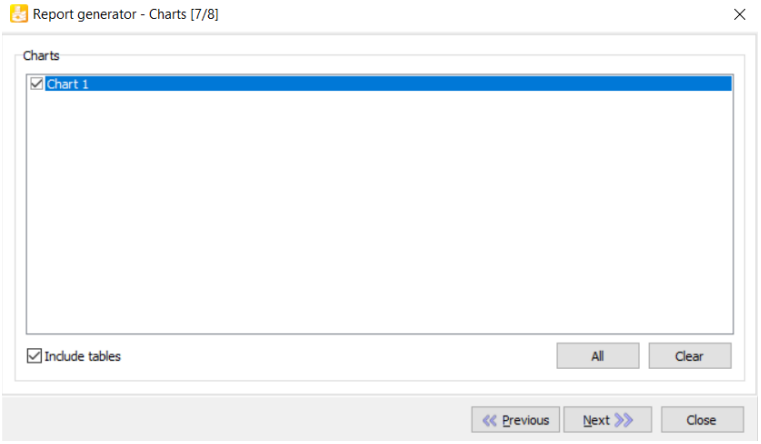


Figure 254: Report generator - Charts

Step 8: A summary of the number of rows and the number of figures in the report is given (Figure 255 (on page 419)). The report is created as the *Export* button is clicked. A progress bar appears displaying the number of the remaining rows and images.

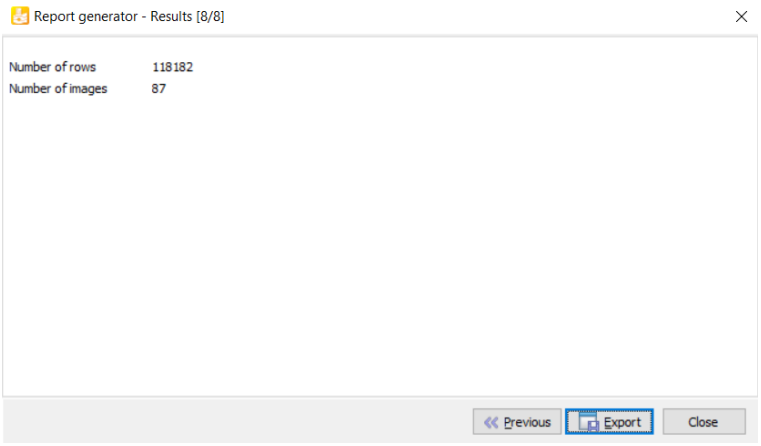


Figure 255: Report generator - Results

8.7.1 Configuration of the document

When the *RTF, PDF or HTML document* option is selected in the first step, after completing the steps required to generate the report another window pops up (Figure 256 (on page 420)) where the document type, name, the storage location and the display properties such as page setup (for RTF and PDF documents), the table configuration and the type and size of the used font can be defined.

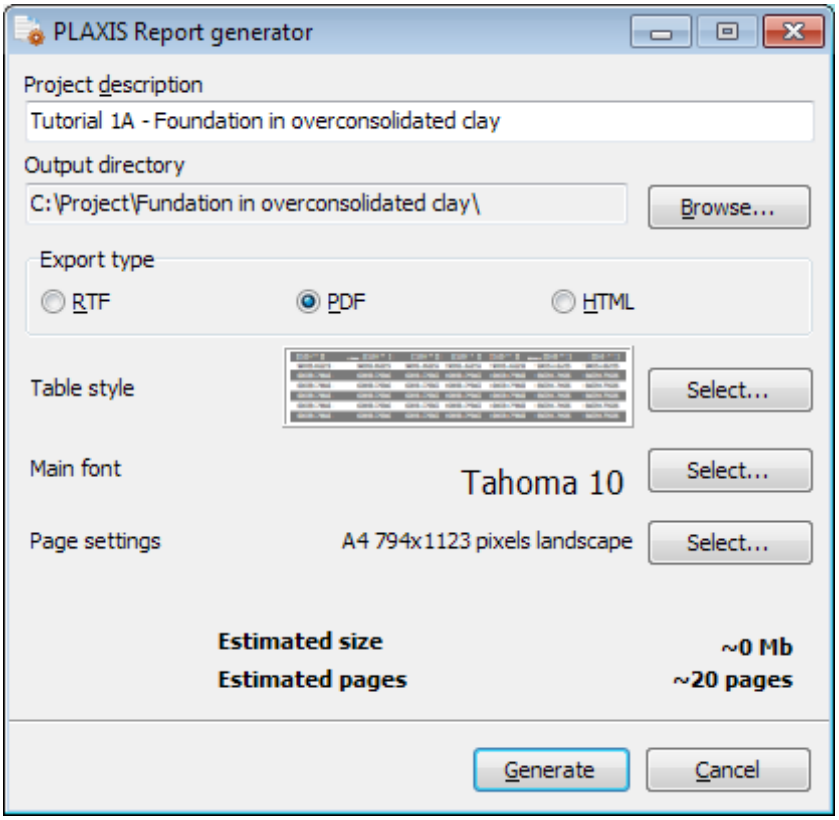


Figure 256: Document properties

8.8 Creating animations

The *Create animation* option is available in the *File* menu. If the option is selected, the **Create animation** window appears (see Figure 257 (on page 421)). The phases and calculation steps to be included in the animations can be selected. Note that the animation will include only available output steps. This depends on the *Max number of steps stored* parameter as defined for each phase in the **Phases** window. After selecting the phase(s), click *OK* to start the process. The progress of this process is indicated in a separate window.

If a large number of steps is to be included in the animation, the process may take some minutes after which the animation is presented. The result is stored in an animation file (*.AVI) in the project data directory.

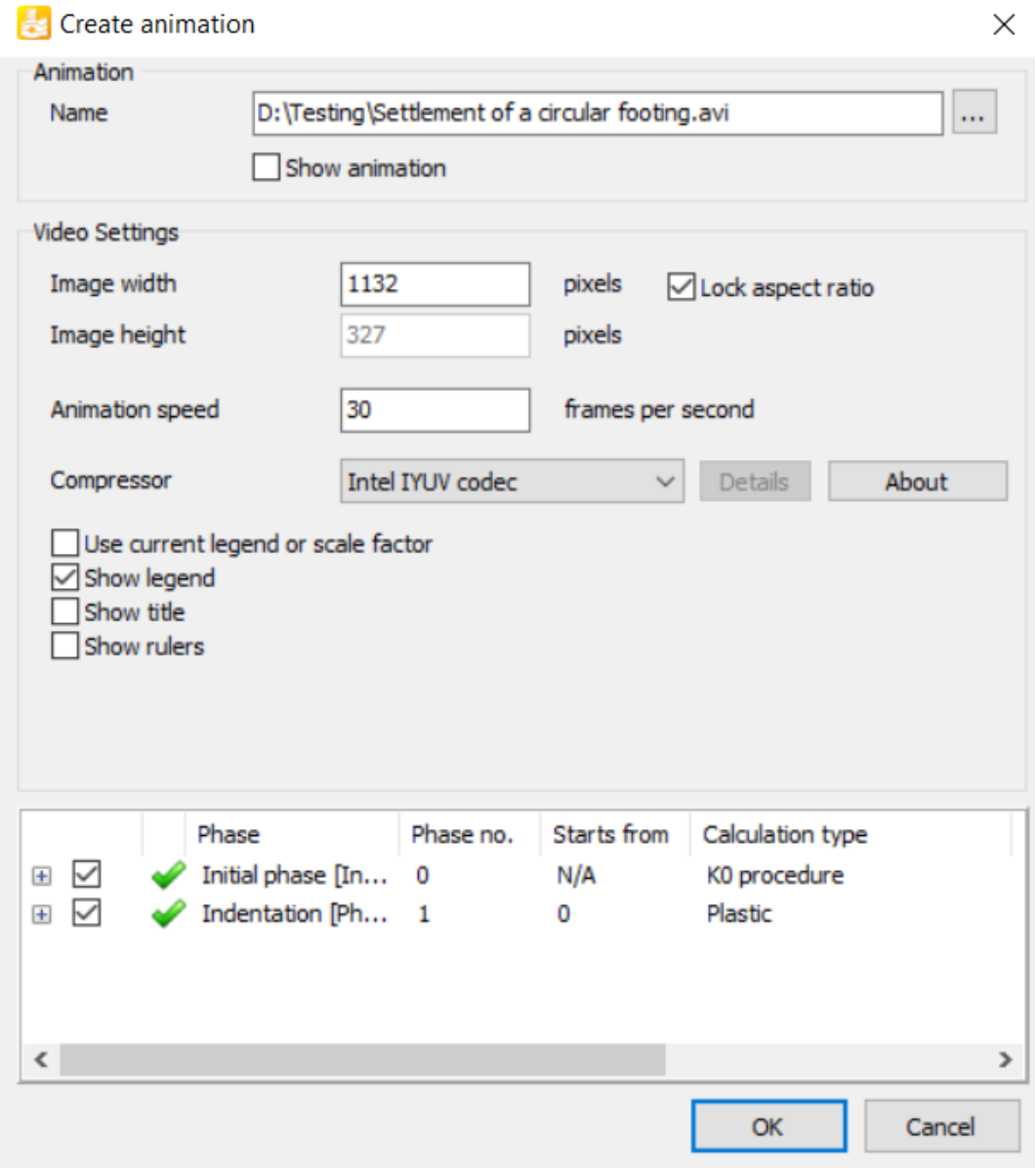


*Figure 257: Selection of phases from **Create animation** window*

8.9 Paraview Export [GSE]

The open source tool ParaView has been enabled to export the FEM model and the calculation results from PLAXIS. This tool allows to visualize and post-process large data from the calculations in PLAXIS 2D. The tool can be accessed in the **Output Program** and clicking the **File** menu (See Figure 258 (on page 422)).

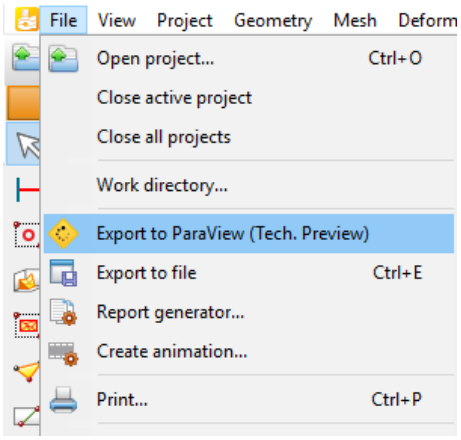


Figure 258: Export to ParaView

After selecting the ***Export ParaView*** option a new window is displayed with the results that can be exported at level of nodes and stress points (See Figure 259 (on page 422)). Finalize the process by selecting the necessary results and click in ***Export***.

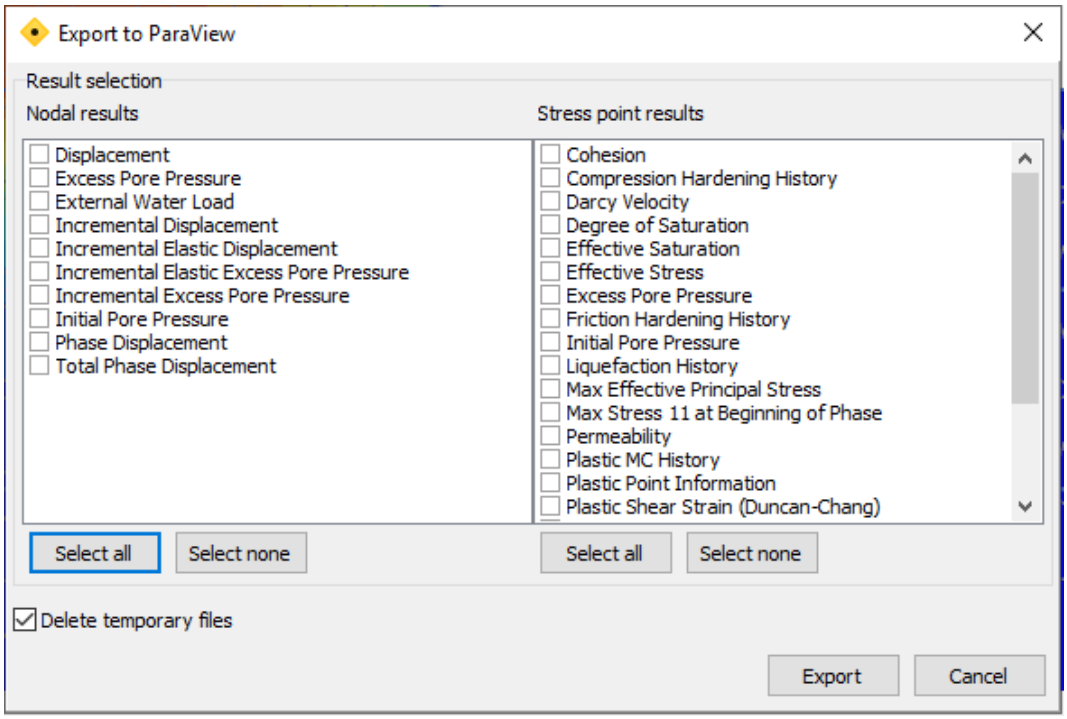


Figure 259: Export to ParaView -Result selection

Note: For more information visit Bentley Communities/Paraview export..

9 Results available in Output program

9.1 Connectivity plot

A *Connectivity plot* is a plot of the mesh in which the element connections are clearly visualised. It is the result of the meshing process. It is available only in the representation of spatial variation of the results. This plot is particularly of interest when interface elements are included in the mesh. Interface elements are composed of pairs of nodes in which the nodes in a pair have the same coordinates. In the *Connectivity plot* however, the nodes in a pair are drawn with a certain distance in between so that it is made clear how nodes are connected to adjacent elements. This option is available from the *Mesh* menu.

In the *Connectivity plot* it can, for example, be seen that when an interface is present between two soil elements, that the soil elements do not have common nodes and that the connection is formed by the interface. In a situation where interfaces are placed along both sides of a plate (*Positive interface* and *Negative interface*), the plate and the adjacent soil elements do not have nodes in common. The connection between the plate and the soil is formed by the interface. An example of *Connectivity plot* is given in Figure 260 (on page 423).

In the *Connectivity plot* it is possible to view soil clusters and structural chains that are excluded from the strength reduction procedure in a (enhanced) safety calculation (select **View > Hide items without strength reduction**). These clusters and structures are indicated by a custom colour; the default colour is grey, but this can be changed in the settings of Modifying the display settings (on page 411).

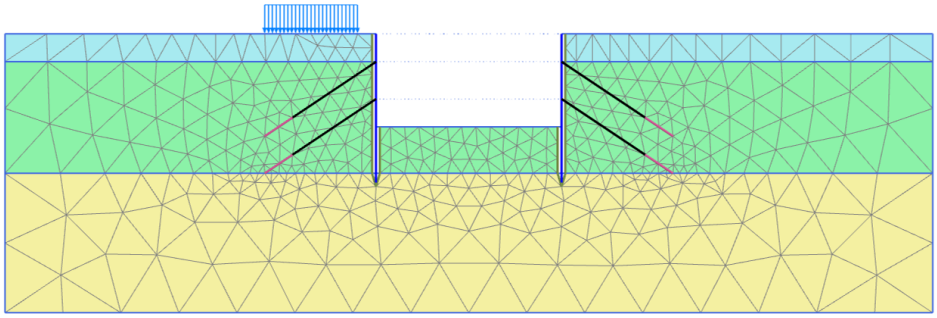


*Figure 260: Example of the **Connectivity plot***

Note:

For (enhanced) safety calculations, results are NOT shown in soil clusters that are excluded from the strength reduction procedure as long as the *Hide items without strength reduction* option in the *View* menu has been selected.

9.2 Deformations

The *Deformations* menu contains various options to visualise the displacements and strains in the finite element model. By default, the displayed quantities are scaled automatically by a factor (1, 2 or 5) $\cdot 10^n$ to give a diagram that may be read conveniently.

The scale factor may be changed by clicking the *Scale factor* button in the toolbar or by selecting the *Scale* option from the *View* menu. The scale factor for strains refers to a reference value of strain that is drawn as a certain percentage of the geometry dimensions. To be able to compare plots of different calculation phases or different projects, the scale factors in the different plots must be made equal.

9.2.1 *Deformed mesh*

The *Deformed mesh* is a plot of the finite element model in the deformed shape. By default, the deformations are scaled up to give a plot that may be read conveniently. If it is desired to view the deformations on the true scale (i.e. the geometry scale), then the *Scale* option (Scaling the displayed results (on page 399)) may be used. The deformed mesh plot may be selected from the *Deformations* menu.

9.2.2 *Total displacements*

The *Total displacements* option contains the different components of the accumulated displacements at the end of the current calculation step, displayed on a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the total displacement vectors, $|u|$, and the individual total displacement components, u_x and u_y. The total displacements may be presented as *Arrows*, *Contour Lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.3 *Phase displacements*

The *Phase Displacements* option contains the different components of the accumulated displacement increments in the whole calculation phase as calculated at the end of the current calculation step, displayed on a plot of the geometry. In other words, the phase displacements are the differential displacements between the end of the current calculation phase and the end of the previous calculation phase. This option may be selected from the *Deformations* menu.

A further selection can be made among the phase displacement vectors, $|Pu|$, and the individual phase displacement components, Pu_x and Pu_y. The phase displacements may be presented as *Arrows*, *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.4 *Sum phase displacements*

In **Staged construction** calculations elements that are switched from inactive to active are, by default, pre-deformed such that the displacement field across the boundary between the new elements and the existing elements is continuous. However, in some applications, such as the staged construction of dams and embankments, this will lead to the undesired situation that the top of the embankment shows the largest settlements (Figure 261 (on page 425)(a)) and is lower than what has been designed, because of the accumulated settlements of the different construction layers. When the *Sum phase displacements* option is selected, the pre-deformation of newly activated elements is ignored. In this way, the settlement of the last construction layer will be limited and the largest settlement will most likely occur in the middle of the embankment, as expected. When plotting the settlements in a vertical cross section through the embankment, the results are somewhat discontinuous, but the overall settlement profile is more realistic than without choosing this option (Figure 261 (on page 425)(b)). The more construction layers are used, the smoother the settlement profile is (Figure 261 (on page 425)(b)).

Figure 261: Settlement profile of an embankment on a stiff foundation layer: a. Phase displacement results; b. Sum phase displacements results (5 construction layers); c. Sum phase displacements results (10 construction layers)

Note that, if displacements are set to zero in intermediate phases, the *Sum phase displacements* results may (at some point) be larger than the total displacements.

Considering structural elements, the *Sum phase displacements* actually applies to the displacements of the corresponding *nodes* as long as they are active. This means that when a structural element (like a plate) is activated in *Phase (i)* whereas the corresponding nodes are already active from the initial phase, the *Sum phase displacements* in *Phase (i+j)* includes displacements of the corresponding nodes before the structure itself was activated.

9.2.5 *Incremental displacements*

The *Incremental displacements* option contains the different components of the displacement increments as calculated for the current calculation step, displayed on a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the displacement increment vectors, $|\Delta u|$, and the individual incremental displacement components, Δu_x and Δu_y. The displacement increments may be presented as *Arrows*, *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of

results (on page 401)). The contours of displacement increment are particularly useful for the observation of localisation of deformations within the soil when failure occurs.

9.2.6 *Extreme total displacements*

The *Extreme total displacements* option contains the different components of the extreme values of the total displacements in the model, displayed on a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the maximum and the minimum of the total displacement components ($u_{x,min}$, $u_{x,max}$, $u_{y,min}$, $u_{y,max}$) and the maximum overall value ($|u|_{max}$). The extreme total displacements may be presented as *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.7 *Velocities* [ULT]

The option *Velocities* contains the different components of the velocities at the end of the current calculation step, displayed on a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the velocity vectors, $|v|$, the individual velocity components, v_x and v_y, as well as the extreme values of the total velocities in the calculation phase. The velocities may be presented as *Arrows*, *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.8 *Accelerations* [ULT]

The option *Accelerations* contains the different components of the accelerations at the end of the current calculation step, displayed on a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the acceleration vectors, $|a|$, the individual acceleration components, a_x and a_y, as well as the extreme values of the total accelerations in the calculation phase. The accelerations may be presented as *Arrows*, *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.9 *Accelerations in 'g'* [ULT]

The option *Accelerations in 'g'* contains the different components of the accelerations at the end of the current calculation step, displayed on a plot of the geometry as multiples of the gravity acceleration. This option may be selected from the *Deformations* menu. A further selection can be made among the acceleration vectors, $|a('g')|$, the individual acceleration components, $a_x('g')$ and $a_y('g')$, as well as the extreme values of the total accelerations in the calculation phase. The accelerations may be presented as *Arrows*, *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.10 *Total cartesian strains*

The *Total cartesian strains* option contains the different components of the accumulated strains at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the three or four individual Cartesian strain components ε_{xx}, ε_{yy}, ε_{zz} (for axisymmetric models only) and γ_{xy}. In case of a plane strain model, ε_{zz} will be zero. In case of an axisymmetric model, the value of the strain in this direction can be calculated as $\varepsilon_{zz} = \partial u_z / \partial z = u_x / R = u_x / x$. The individual strain components may be presented as *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.11 *Phase cartesian strains*

The *Phase cartesian strains* option contains the different components of the accumulated strain increments in the whole calculation phase as calculated at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the three or four individual Cartesian strain components $P\varepsilon_{xx}$, $P\varepsilon_{yy}$, $P\varepsilon_{zz}$ (for axisymmetric models only) and $P\gamma_{xy}$. In case of a plane strain model, $P\varepsilon_{zz}$ will be zero. In case of an axisymmetric model, the value of the strain in this direction can be calculated as $P\varepsilon_{zz} = \partial Pu_z / \partial z = Pu_x / R = Pu_x / x$.

The individual strain components may be presented as *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar Display type of results (on page 401)).

9.2.12 *Incremental cartesian strains*

The *Incremental cartesian strains* option contains the different components of the strain increments as calculated for the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the three or four individual Cartesian strain components $\Delta\varepsilon_{xx}$, $\Delta\varepsilon_{yy}$, $\Delta\varepsilon_{zz}$ (for axisymmetric models only) and $\Delta\gamma_{xy}$. In case of a plane strain model, $\Delta\varepsilon_{zz}$ will be zero. In case of an axisymmetric model, the value of the strain in this direction can be calculated as $\Delta\varepsilon_{zz} = \partial \Delta u_z / \partial z = \Delta u_x / R = \partial \Delta u_x / x$.

The individual strain components may be presented as *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 401)).

9.2.13 *Total strains*

The *Total strains* option contains various strain measures based on the accumulated strains in the geometry at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the principal strain directions, the individual principal strain components ε_1, ε_2, ε_3 (for axisymmetric models only), $(\varepsilon_1 + \varepsilon_3)/2$, $(\varepsilon_1 - \varepsilon_3)/2$, the angle, the volumetric strain ε_v, the deviatoric strain ε_q and the void ratio e.

- Note that the principal strain components are arranged in algebraic order:

$$\varepsilon_1 > \varepsilon_2 > \varepsilon_3$$

Hence, ε_1 is the largest compressive principal strain and ε_3 is the smallest compressive principal strain.

- The volumetric strain is calculated as:

  In normal calculations:

  $$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} \qquad \textit{Eq. [100]}$$

  In *Updated mesh* calculations:

  $$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} + \varepsilon_{xx}\varepsilon_{yy} + \varepsilon_{xx}\varepsilon_{zz} + \varepsilon_{yy}\varepsilon_{zz} + \varepsilon_{xx}\varepsilon_{yy}\varepsilon_{zz} \qquad \textit{Eq. [101]}$$

- The deviatoric strain is calculated as:

  $$\varepsilon_q = \sqrt{\frac{2}{3}\left[\left(\varepsilon_{xx} - \frac{\varepsilon_v}{3}\right)^2 + \left(\varepsilon_{yy} - \frac{\varepsilon_v}{3}\right)^2 + \left(\varepsilon_{zz} - \frac{\varepsilon_v}{3}\right)^2 + \frac{1}{2}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right)\right]} \qquad \textit{Eq. [102]}$$

- The void ratio is calculated as:

  $$e = e_0 + (1 + e_0)\varepsilon_v \qquad \textit{Eq. [103]}$$

9.2.14 *Phase strains*

The *Phase strains* option contains various strain measures based on the accumulated strain increments in the whole calculation phase as calculated at the end of the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the volumetric strain ($P\varepsilon_v$) and the deviatoric strain ($P\varepsilon_q$).

9.2.15 *Incremental strains*

The *Incremental strains* option contains various strain measures based on the strain increments as calculated for the current calculation step, displayed in a plot of the geometry. This option may be selected from the *Deformations* menu. A further selection can be made among the volumetric strain ($\Delta\varepsilon_v$) and the deviatoric strain ($\Delta\varepsilon_q$).

9.3 Stresses

Various options are available to visualise the stress state in the finite element model. The *Stresses* menu also contains options to display the results of groundwater flow and thermal flow calculations.

Note:

By default, the stresses developed in non-porous materials are not displayed in the plot. To display them select the *Show stress for nonporous material* option in the **Results** tabsheet of the **Settings** window (Modifying the display settings (on page 411)).

9.3.1 *Cartesian effective stresses*

The *Cartesian effective stresses* are different components of the effective stress tensor (i.e. the stresses in the soil skeleton). A further selection can be made among the three individual Cartesian stress components σ'_{xx}, σ'_{yy}, σ'_{zz} (for axisymmetric models only), and σ_{xy}.

Figure 262 (on page 429) shows the sign convention adopted for Cartesian stresses. Note that pressure is considered to be negative.

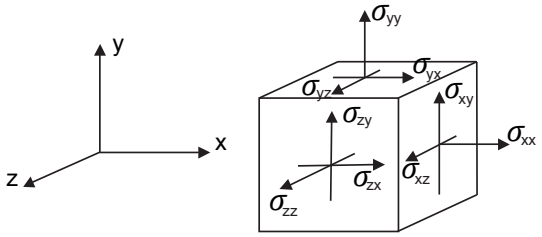


Figure 262: Sign convention for stresses

9.3.2 *Cartesian total stresses*

The *Cartesian total stresses* are different components of the total stress tensor (i.e. effective normal stresses + active pore pressures). A further selection can be made among the three individual Cartesian stress components σ_{xx}, σ_{yy}, σ_{zz} (for axisymmetric models only), and σ_{xy}. The latter quantity is equal to the corresponding one in the *Cartesian effective stress* option, but are repeated here for convenience (Cartesian effective stresses (on page 429)). The individual stress components may be presented as *Contour lines* or *Shadings* by clicking the appropriate button in the toolbar.

9.3.3 *Principal effective stresses*

The *Principal effective stresses* are various stress measures based on the effective stresses σ' (i.e. the stresses in the soil skeleton). A further selection can be made among the effective principal stresses, the individual principal effective stress components σ'_1, σ'_2, σ'_3, $(\sigma'_1+\sigma'_3)/2$, the principal stress directions, the mean effective stress p', the deviatoric stress q, the relative shear stress τ_{rel}, the maximum shear stress τ_{max} and the mobilised shear strength τ_{mob}.

Note that the effective stress components are arranged in algebraic order:

$$\sigma_1' \leq \sigma_2' \leq \sigma_3' \qquad \textit{Eq. [104]}$$

Hence, σ'_1 is the largest compressive (or smallest tension) principal stress and σ'_3 is the smallest compressive (or largest tension) principal stress.

The *Mobilised shear strength* τ_{mob} is the maximum value of shear stress (i.e. the radius of the Mohr stress circle or half the maximum principal stress difference).

The *Relative shear stress* τ_{rel} gives an indication of the proximity of the stress point to the failure envelope, and is defined as:

$$\tau_{\mathrm{rel}} = \frac{\tau_{\mathrm{mob}}}{\tau_{\mathrm{max}}} \qquad \textit{Eq. [105]}$$

where τ_{max} is the maximum value of shear stress for the case where the Mohr's circle is expanded to touch the Coulomb failure envelope while keeping the center of Mohr's circle constant.

$$\tau_{\mathrm{max}} = -\frac{\sigma'_1 + \sigma'_3}{2}\sin\varphi + c\cos\varphi \qquad \textit{Eq. [106]}$$

Note:

Particularly when the soil strength has been defined by means of effective strength parameters- *Undrained (A)*- it is useful to plot the mobilised shear strength τ_{mob} in a vertical cross section and to check this against a known shear strength profile.

When using the Hoek-Brown model to describe the behaviour of a rock section, the definition of the maximum shear stress τ_{max} is slightly modified. Starting from the Hoek-Brown failure criterion:

$$f_{HB} = \sigma'_1 - \sigma'_3 + \overline{f}(\sigma'_3) = 0 \qquad \textit{Eq. [107]}$$

the maximum shear stress is defined by :

$$\tau_{\mathrm{max}} = \frac{1}{2}\overline{f}(\sigma'_3) \quad \text{where} \quad \overline{f}(\sigma'_3) = \sigma_{ci}\left(m_b\frac{-\sigma'_3}{\sigma_{ci}} + s\right)^a \qquad \textit{Eq. [108]}$$

The relative shear stress is correspondingly defined by:

$$\tau_{rel} = \frac{\tau_{mob}}{\tau_{\mathrm{max}}} = \frac{|\sigma'_1 - \sigma'_3|}{\overline{f}(\sigma'_3)} \qquad \textit{Eq. [109]}$$

The principal stress directions are defined by:

$$\alpha = \frac{1}{2}\arctan\left(\frac{2\sigma_{zy}}{\sigma_{yy} - \sigma_{xx}}\right) \quad (-90° \le \alpha \le 90°) \qquad \textit{Eq. [110]}$$

For α = 0, the major principal stress is vertical and the minor principal stress is horizontal. In this case, the cartesian shear stress is zero (for example initial stress generated by the *K0 procedure*). This situation corresponds to an active stress state.

A passive stress state is equivalent to α= *+90°* or α= *-90°*. Zones of positive stress may show a jump from α= *+90°* to α= *-90°* and as a result discontinuous colour shadings are displayed.

A positive value of cartesian shear stress will lead to a clockwise rotation of the principal stress direction (α*>0*), whereas a negative cartesian shear stress will rotate the principal stress counter-clockwise (α*<0*). The plot of principal stress directions is only available in PLAXIS 2D. A graphical description of the principal stress directions is shown in Figure 263 (on page 431).

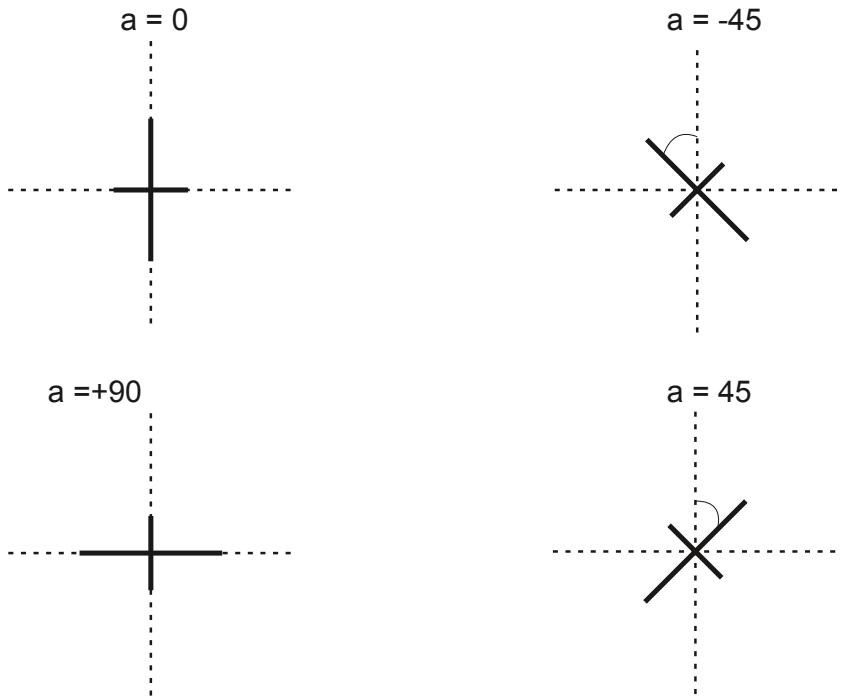


Figure 263: Example of principal stress directions

9.3.4 *Principal total stresses*

The *Principal total stresses* are various stress measures based on the total stresses σ (i.e. effective stresses + active pore pressures). A further selection can be made among the principal total stress directions, the individual principal total stress components σ_1, σ_2, σ_3, $(\sigma_1+\sigma_3)/2$, $(\sigma_1-\sigma_3)/2$, the principal stress directions, the mean total stress p, the deviatoric stress q, the relative shear stress τ_{rel} and the mobilised shear strength τ_{mob}. The latter three quantities are equal to the corresponding ones in the *Principal effective stress* option, but are repeated here for convenience (Principal effective stresses (on page 429)).

Note that the total stress components are arranged in algebraic order:

$$\sigma_1 \leq \sigma_2 \leq \sigma_3$$

Hence, σ_1 is the largest compressive (or the smallest tension) principal stress and σ_3 is the smallest compressive (or the largest tension) principal stress.

9.3.5 *Initial conditions*

The *Initial conditions* sub-menu (Figure 264 (on page 432)) is only available in the initial phase. The spatial distribution of the Pre-Overburden Pressure (POP), as defined in the **Preconsolidation** tab in the **Modify soil layers** window, can be displayed and checked (Figure 265 (on page 432)).

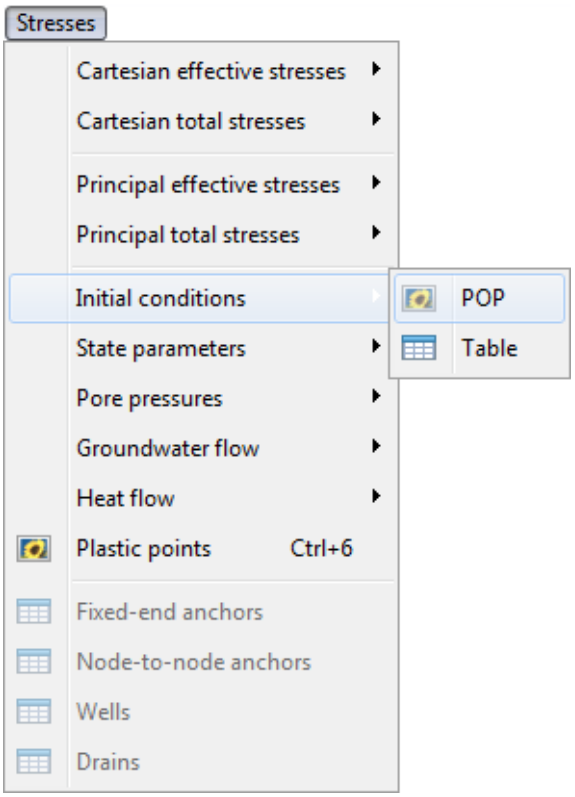


Figure 264: Initial conditions in the Stresses menu (Initial phase)

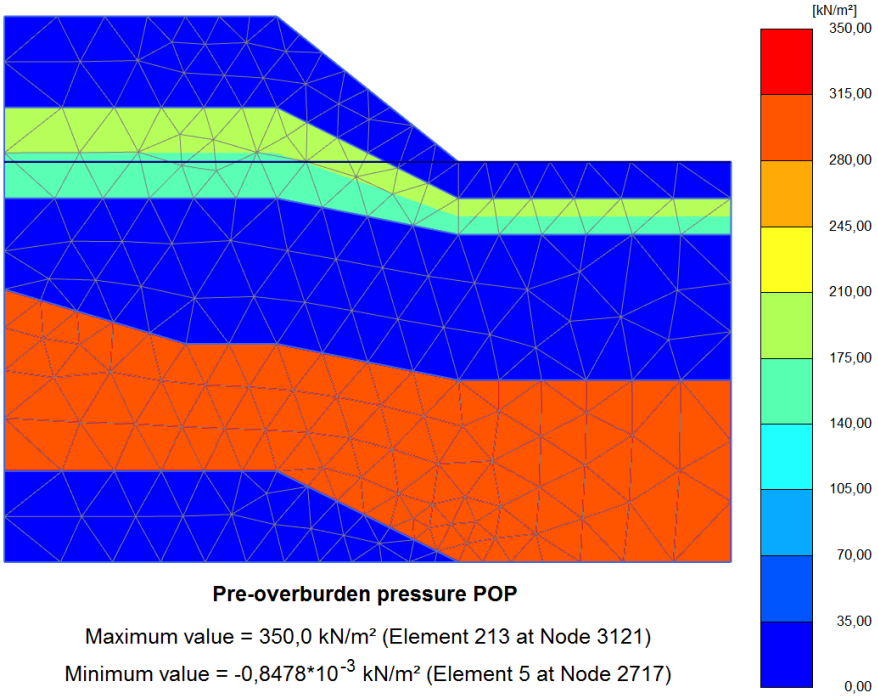


Figure 265: Example of POP distribution

9.3.6 *State parameters*

The *State parameters* are various additional quantities that relate to the state of the material in the current calculation step, taking into account the stress history. For more information regarding state parameters of each material model, see Material Models Manual.

The actual permeability

The actual permeability ($Permeability_{actual,x}$, $Permeability_{actual,y}$) is the relative permeability times the saturated permeability as defined in the flow parameters of the material set.

The strain history

The strain histories ε_{xx} - ε_v, ε_{yy} - ε_v, ε_{zz} - ε_v (for axisymmetric models only), and ε_{xy} are only available in the Hardening Soil model with small-strain stiffness.

The secant shear modulus

G_s: The secant shear modulus G_s is only available in the Hardening Soil model with small-strain stiffness. This option may be used to check the actual secant shear modulus at the reference level, used in the current calculation step. More information about this parameter can be found in the Material Models Manual - The Hardening Soil model with small-strain stiffness - Describing small-strain stiffness with a simple hyperbolic law.

The ratio between the actual shear modulus and the unloading reloading stiffness

G/G_{ur} : The ratio between the actual shear modulus G and the unloading reloading stiffness G_{ur} is only available in the Hardening Soil model with small-strain stiffness.

The equivalent isotropic stress

p_{eq}: The equivalent isotropic stress p_{eq} is only available in the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and Sekiguchi-Ohta model.The equivalent isotropic stress is defined as the intersection point between the stress contour (with similar shape as the yield contour) through the current stress point and the isotropic stress axis. Depending on the type of model being used it is defined as:

$p_{eq} = \sqrt{p^2 + \frac{q^2}{\alpha^2}}$	for the Hardening Soil model and Hardening Soil model with small-strain stiffness.
$p_{eq} = p' - \frac{q^2}{M^2(p' - c\cot\varphi)}$	for the Soft Soil model, Soft Soil Creep model and Modified Cam-Clay model. For the Modified Cam-Clay model, the cohesion c is defined as 0 kN/m^2.
$p_{eq} = \frac{p'}{exp\left(-\frac{q}{Mp'}\right)}$	for the Sekiguchi-Ohta model.

More information about this parameter for each model can be found in the Material Models Manual

The isotropic preconsolidation stress p_p

The isotropic preconsolidation stress p_p is only available in the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and Sekiguchi-Ohta model. The isotropic preconsolidation stress represents the maximum equivalent isotropic stress level that a stress point has experienced up to the current load step.

The isotropic overconsolidation ratio *OCR*

The isotropic overconsolidation ratio *OCR* is only available in the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and Sekiguchi-Ohta model. The isotropic overconsolidation ratio is the ratio between the isotropic preconsolidation stress p_p and the equivalent isotropic stress p_{eq}.

The hardening parameter γ^p

The hardening parameter γ^p is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness. This option may be used to check the actual hardening during the current calculation step.

The maximum major effective principal stress $\sigma'_{1,max}$

The maximum major effective principal stress $\sigma'_{1,max}$ is the maximum value of the major effective principal stress in the stress history up to the current results.

The overconsolidation ratio on major principal stress OCR_1

The overconsolidation ratio on major principal stress $OCR_1=\sigma'_{1,max}/\sigma'_1$, hence the maximum major effective principal stress in the stress history divided by the current major effective principal stress. Note that for the advanced models that include separate primary loading and unloading/reloading behaviour, the maximum major effective principal stress is also the principal preconsolidation pressure and so OCR_1 is a true overconsolidation ratio. However, for elastic and elastic-perfectly plastic models, such as the Linear Elastic, Mohr-Coulomb and Hoek-Brown models, there is no memory of preconsolidation and so the maximum major effective principal stress is just a maximum value that occurred and not a preconsolidation pressure. Hence, for these models OCR_1 is not a real overconsolidation ratio.

The overconsolidation ratio on major principal stress OCR_1 is mostly used for the SHANSEP models.

The actual stiffness E_{ur} for unloading and reloading

The actual Young's modulus E_{ur} for unloading and reloading is the unconstrained elastic stiffness modulus as used during the current calculation step. This option is only available in the Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay model and Sekiguchi-Ohta model.

The stiffness E_{ur} depends on the stress level. In models with stress-dependency of stress, the actual stiffness E_{ur} is calculated on the basis of the stresses at the beginning of the current step. The option may be used to check the actual stress-dependent stiffness used in the current calculation step.

The actual cohesion *c*

The actual cohesion *c* is the cohesive strength as used during the current calculation step. This option is only available in the Mohr-Coulomb model, Hardening Soil model, Hardening Soil model with small-strain stiffness, Soft Soil model and Soft Soil Creep model.

When the Mohr-Coulomb model, Hardening Soil model or the Hardening Soil model with small-strain stiffness is utilised with an increasing cohesive strength with depth ($c_{increment} > 0$), this option may be used to check the actual cohesive strength profile used in the calculation.

9.3.7 *Pore pressures*

Pore pressures are the quantities that relate to the stress in the pores of the material. The pores in soil are usually filled with a mixture of water and air. In most cases, stresses and pore pressures in the soil are negative (pressure). However, due to capillary action or undrained unloading, pore pressures may become positive (tension), which is denoted as suction.

PLAXIS 2D distinguishes between different types of pore pressures and related quantities. It is important to understand the differences between these quantities in order to properly interpret the results from a PLAXIS 2D calculation.

Total stresses are divided into effective stresses, σ', and active pore pressures, p_{active}.

$$\sigma = \sigma' + p_{active} \qquad Eq.\ [111]$$

Note that all the effective and total stresses are tensorial quantities. Then pore pressure should be also understood as a diagonal tensor (pore pressure only applies to normal stress components).

Active pore pressures are defined as the effective saturation, S_{eff}, times the pore water pressures, p_{water}.

$$p_{active} = S_{\text{eff}} \cdot p_{water} \qquad Eq.\ [112]$$

Pore water pressure differs from active pore pressure when the degree of saturation is less than unity, which is usually the case above the phreatic level. Below the phreatic level p_{active} and p_{water} are generally equal. For more details, refer Appendix D.

As an alternative to the pore water pressure, p_{water}, the groundwater head, h, can be viewed:

$$h = z - \frac{p_{water}}{\gamma_w} \qquad Eq.\ [113]$$

where z is the vertical coordinate and γ_w is the unit weight of water. In the pore water pressure a further distinction is made between steady state pore pressure, p_{steady}, and excess pore pressure, p_{excess}.

$$p_{water} = p_{steady} + p_{excess} \qquad Eq.\ [114]$$

where steady-state pore pressure is the steady-state or long term part of pore pressure, which is supposed to be input data for a deformation analysis.

Excess pore pressure is the result from undrained behaviour (*Undrained (A)* or *(B)*, or low-permeable materials), and is affected by stress changes due to loading or unloading, a (sudden) change in hydraulic conditions and consolidation.

Below an overview is given of the various pore pressure related quantities that are available in the *Stresses* menu of the PLAXIS 2D Output program. Most quantities can be presented as *Contour lines* or *Shadings*. Although pore pressures do not have principal directions, the *Principal stresses* presentation can be useful to view pore pressures inside the model. In that case the colour of the lines represents the magnitude of the pore pressure and the directions coincide with the x-, y- and z-axis.

Groundwater head

The groundwater head is an alternative quantity for the pore water pressure, equivalent to the top level of a free water column.

Active pore pressure

The part of total stress in the pores of the soil. In saturated soil, active pore pressure is equal to the pore water pressure. In unsaturated soil, active pore pressure is the effective saturation times the pore water pressure.

Pore water pressure

The water pressure in the pores of the soil consisting of steady-state pore pressure and excess pore pressure. In most cases, pore water pressure is negative (pressure). However, due to capillary action or undrained unloading, pore water pressure may become positive (suction).

Steady-state pore pressures

Long-term part of pore water pressure which is supposed to be input data for a deformation analysis. Steady-state pore pressure is the result of a direct pore pressure generation based on phreatic levels and cluster-related pore pressure definition, or the result of a steady-state groundwater flow calculation. In the case of a fully coupled flow-deformation analysis, steady-state pore pressure is calculated from a steady-state groundwater flow calculation using the hydraulic boundary conditions at the end of the calculation phase. When the option *Ignore suction* has been used, the steady-state pore pressure only includes negative values.

Excess pore pressure

Instationary part of pore water pressure that is a result of undrained behaviour (*Undrained (A)* or *(B)*, or low-permeable materials), affected by stress changes due to loading or unloading, a (sudden) change in hydraulic conditions and consolidation. In the case of a fully coupled flow-deformation analysis, excess pore pressure is the difference between the calculated pore water pressure and the steady-state pore pressure. The latter is calculated from a steady-state groundwater flow calculation using the hydraulic boundary conditions at the end of the calculation phase. Excess pore pressure may show positive values (suction) as a result of undrained unloading, even when the option Ignore suction has been selected.

Note: For Steady-state groundwater flow analysis a [ADV] licence is required (as minimum) . For fully coupled calculations an [ULT] licence needs to be active.

Extreme excess pore pressure

Maximum and minimum values of excess pore pressure over the whole calculation phase up to the selected calculation step.

Change in pore pressure per phase

The change in active pore pressure from the beginning of a phase up to the selected calculation step.

Suction

Any positive value (tension) in the pore water pressure. Suction can be a result of capillary action (included in the steady-state pore pressure if the option *Ignore suction* has NOT been used) or undrained unloading (included in the excess pore pressure).

Effective suction

The net effect of suction on the soil stresses taking into account the degree of saturation. Hence, the effective suction is the suction times the effective degree of saturation. The occurrence of effective suction results into an increase of effective stresses, which may lead to an increased strength of the soil.

Pore pressure ratio

The ratio (R_u) of the excess pore pressure (P_{excess}) to the major principal effective stress (σ'_1) at step 0 of the phase (initial stress).

9.3.8 *Groundwater flow* [ADV]

When a groundwater flow calculation has been performed to generate the pore pressure distribution, then the specific discharges at the element stress points are available in the Output program in addition to the pore pressure distribution. The specific discharges can be viewed by selecting the *Groundwater flow* option from the *Stresses* menu. A further selection can be made among the groundwater flow components, ($|q|$, q_x, q_y).

The flow field may be viewed as *Arrows*, *Contour lines* or *Shadings* by selecting the appropriate option from the presentation box in the tool bar. When the specific discharges are presented as arrows, then the length of the arrow indicates the magnitude of the specific discharge whereas the arrow direction indicates the flow direction.

Discharge ($|q|$, q_x, q_y)

The total discharge through a cross section can be obtained by drawing a cross section through the interface element and selecting *Stresses* → *Groundwater flow*→$|q|/q_x/q_y$. The value of total discharge is displayed below the plot in the unit of volume per unit of time , per unit of width in the out-of-plane direction.

Saturation

The PlaxFlow module within PLAXIS 2D may be used to calculate a pore pressure distribution for confined as well as for unconfined flow problems. The determination of the position of the free phreatic surface and the associated length of the seepage surface is one of the main objectives of an unconfined groundwater flow calculation. In this case a relationship is used between the pore pressure and the degree of saturation. Both quantities are calculated in a groundwater flow calculation and are made available in the Output program.

The degree of saturation is generally 100% below the phreatic level and it reduces to the residual saturation within a finite zone above the phreatic level. Note that the residual saturation value is equal to zero when suction is ignored. The saturation can only be presented as *Contour lines* or *Shadings*.

Hydraulic gradient (i)

The hydraulic gradient (also called groundwater head gradient or Darcy's slope) between any two points is the rate of spatial variation of the hydraulic head measurements over the length of the flow path that links them. It is more generally expressed as the spatial derivative of the hydraulic head field, $i = \nabla h$. The hydraulic gradient together with the permeability tensor determines the flux of groundwater in the space. In isotropic media, the magnitude and direction of this parameter are proportional and opposite to the flow vector (q), as water flows from higher to lower hydraulic head locations. For the general evaluation of seepage problems in anisotropic media, it is crucial to consider both vectors independently.

The hydraulic gradient can be visualised by selecting the *|i|* option or one of the available components i_x and i_y. These can be presented as *Contour lines*, *Shadings*, and *Arrows*.

Effective saturation (*Saturation*$_{eff}$)

The effective saturation is used as the Bishop coefficient in the definition of Bishop stress and also to calculate the weight of soil. The effective saturation can only be presented as *Contour lines* or *Shadings*.

Relative permeability (*Permeability*$_{rel}$)

The relative permeability can be visualised by selecting the *Permeability*$_{rel}$ option. The relative permeability can only be presented as *Contour lines* or *Shadings*.

9.3.9 *Heat flow* [ADV]

When a *Thermal flow* calculation has been performed to generate the temperature distribution, then the heat fluxes at the element stress points are available in the Output program, in addition to the temperature distribution. The heat fluxes can be viewed by selecting the *Heat flow* option from the *Stresses* menu.

A further selection can be made among the heat flux components ($|qt|$, qt_x, qt_y). Moreover, in case of partially saturated soils where part of the pore water is in the gas phase (vapour), the vapour flux can be selected. A further selection can be made among the vapour flux components ($|qv|$, qv_x, qv_y).

The heat flux and vapour flux may be viewed as *Arrows*, *Contour lines* or *Shadings*, by selecting the appropriate option from the presentation box in the tool bar. When the fluxes are presented as arrows, then the length of the arrow indicates the magnitude of the flux whereas the arrow direction indicates the flux direction.

9.3.10 *Plastic points*

The *Plastic points* option shows the stress points that are in a plastic state, displayed in a plot of the undeformed geometry. Plastic points can be shown in the 2D mesh or in the elements around a cross section.

Note:

The *Plastic point history* option in the *Stresses* menu enables displaying in the model all the points that have been plastic in any step in the calculation history (depending on the specified criteria, *Failure*, *Tension cut-off*, etc.) generated up to the current calculation phase.

The plastic stress points are indicated by small symbols that have different shapes and colours, depending on the type of plasticity that has occurred:

- A red cube (Failure point) indicates that the stresses lie on the surface of the failure envelope.
- A white cube (Tension cut-off point) indicates that the tension cut-off criterion was applied.
- A blue upside-down pyramid (Cap point) represents a state of normal consolidation (primary compression) where the preconsolidation stress is equivalent to the actual stress state. The latter type of plastic points only

occurs if the Hardening Soil model, the Hardening Soil model with small-strain stiffness, the Soft Soil model, the Soft Soil Creep model or the Modified Cam-Clay model is used.

- A brown diamond (Cap+Hardening point) represents points that are on the shear hardening and cap hardening envelope. Such plastic points can only occur in the Hardening Soil model or the Hardening Soil model with small-strain stiffness.
- A green pyramid (Hardening point) represents points on the shear hardening envelope. Such plastic points can only occur in the Hardening Soil model, the Hardening Soil model with small-strain stiffness and the UBC3D-PLM model.
- A purple inverted triangle (Liquefaction point) represents a stress point which is liquefied, characterised by the pore pressure ratio $R_u > 0.95$. Such plastic points can occur only in the UBC3D-PLM model, the PM4Sand and User-defined Soil Model. When viewed in the *Table* view (Table of Cartesian effective stresses), the *Status* shows as 'Liquefied'.

In case the Concrete model is adopted, the symbols have different meanings:

- A red cube (Failure point) indicates that the stresses lie on the Mohr-Coulomb peak surface or the Mohr-Coulomb softening surface.
- A white cube (Tension cut-off point) indicates that the stresses lie on the failure envelope in tension, including the intersection between the Rankine surface and the Mohr-Coulomb surface.
- A green pyramid (Hardening point) represents points on the Mohr-Coulomb shear hardening envelope.

The failure points are particularly useful to check whether the size of the mesh is sufficient. If the zone of plasticity reaches a mesh boundary (excluding the centre-line in a symmetric model) then this suggests that the size of the mesh may be too small. In this case the calculation should be repeated with a larger model.

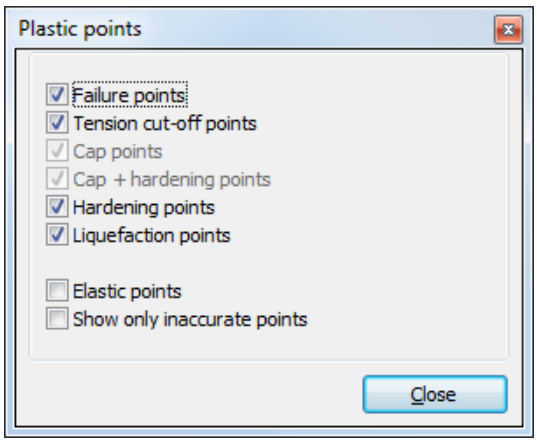


Figure 266: Plastic points window

When *Plastic points* is selected in the *Stresses* menu the *Plastic points* dialog is shown (Figure 266 (on page 439)). Here the user can select which types of plastic points are displayed. When the *Stress points* option is selected, all other stress points are indicated by a purple diamond shape . For details of the use of advanced soil models, the user is referred to the Material Models Manual.

By default both accurate and inaccurate plastic points are displayed in the model. Only the inaccurate plastic points are displayed as the corresponding check box is selected in the **Plastic points** window. Inaccurate plastic points are points where the local error is larger then the tolerated error (Automatic error checks (on page 381)). When the results are displayed with the *Table* option, an asterisk appears on the status column to identify the inaccurate plastic points.

Conventions used for Plastic point information with Paraview Export [GSE]

If *plastic point information* is exported from PLAXIS 2D to ParaView (see Paraview Export [GSE] (on page 421)), the point information will have associated different status described in Conventions used for Plastic point information with Paraview Export [GSE] (on page 440).

Table 20: Conventions for plastic point results on Paraview

Value	Plastic point status
1 or -1	MC accurate
2	Tension cut-off accurate
3	Cap Hardening accurate
4	Cap + Shear hardening accurate
5	Shear hardening accurate
6	Liquefaction accurate
11 or -11	MC inaccurate
12	Tension cut-off inaccurate
13	Cap Hardening inaccurate
14	Cap + Shear hardening inaccurate
15	Shear hardening inaccurate
16	Liquefaction inaccurate

9.3.11 *Fixed end anchors*

When *Fixed end anchors* is selected in the *Stresses* menu a table appears displaying the fixed end anchors available in the model, their location, the resulting axial force, the rotation angle and the equivalent length.

9.3.12 *Node to node anchors*

When *Node to node anchors* is selected in the *Stresses* menu a table appears displaying the node to node anchors available in the model, the location of the nodes and the resulting axial forces.

9.3.13 *Wells* [ADV]

When *Wells* is selected in the *Stresses* menu a table appears displaying the wells available in the model, the nodes representing the well and their location, the discharge of the entire well and the defined minimum groundwater head.

9.3.14 *Drains* [ADV]

When *Drains* is selected in the *Stresses* menu a table appears displaying the drains available in the model, the nodes representing the drain and their location, the total discharge and the defined groundwater head of the drain.

9.4 Structures and interfaces

By default, the structures active in the current phase are displayed in the model. Otherwise, these objects may be displayed in by switching the view on in the explorer.

Output for structures and interfaces can be obtained by clicking the *Select structures* button and then double-clicking the desired object in the 2D model. As a result, a new form is opened on which the selected object appears. At the same time the menu changes to provide the particular type of output for the selected object.

All objects of the same type with the same local coordinate system are automatically selected. When multiple objects or multiple groups of objects of the same type need to be selected, the *Shift* key should be used while selecting the objects. The last object to be included in the plot should then be double clicked. When all objects of the same type are to be selected, select one of the objects while pressing *Ctrl-A* simultaneously. If it is desired to select one or more individual elements from a group, the *Ctrl* key should be used while selecting the desired element.

Another option of selecting structural elements in the output is by clicking the *Drag a window to select structures* button and drawing a rectangle in the model. As a results, the structures in the rectangle will be selected.

9.4.1 *Deformation in structures and interfaces*

The deformation options for the structural elements are given in the *Deformations* menu. The user may select the *Total displacements*, the *Phase displacements* or the *Incremental displacements* (Deformations (on page 424)). For each item a further selection can be made among the displacement vectors $|u|$, and the individual total displacement components, u_x and u_y.

The deformation options in the direction of local axis of the structures are available as well. The user may select the *Total local displacements*, the *Phase local displacements* or the *Incremental local displacements*. For each item a further selection can be made among the individual displacement components u_1 and u_2.

The *Rotation* option is available for *Plates* displaying the total rotation (*Rotation*) and the incremental rotation (*ΔRotation*) of the selected plates with respect to the global axes.

Note:

The *Total Rotation* and *Incremental rotation* for plates is positive in counter clockwise direction with respect to the global axis system.

Options such as *Relative total displacements*, *Relative phase displacements* and *Relative increments* are available for interfaces. Relative displacements are differential displacements between node pairs. These options may be used to view if plastic shearing has occurred in the interface.

Only in the case of semi-permeable interfaces, the output program shows the resulting specific discharge through the interface elements. This output is not available in the case of fully impermeable or fully permeable interface elements.

In the case of fully permeable interfaces, the resulting discharge through the interfaces can be obtained by drawing a cross section just outside the wall or tunnel lining, and plotting the flow through this cross section. The title of the cross section plot will indicate the total discharge through the cross section.

9.4.2 *Resulting contraction in plates and geogrids*

When a *Line contraction* is used in the calculation and the correspondent plate or geogrid is displayed, the realised contractions are available from the *Deformations* menu. The contraction can be positive (e.g. concrete shrinkage) or negative (e.g. thermal expansion of geogrids).

The realised contraction outputs are C_{tunnel} and C_{planar}. The realised tunnel contraction, C_{tunnel}, returns the value as used in the calculation. The realised planar contraction, C_{planar}, is equal to $C_{tunnel}/2$. Assuming a case of line (circumference) contraction (C_{ref}^{line}) from an initial length L_0 to a final length L against circular surface contraction (C_{ref}^{surf}) from an initial area A_0 (with radius $R_0=L_0/(2\pi)$) to a final area A (with radius $R=L/(2\pi)$), the factor 2 is derived as such:

$$C_{ref}^{line} = \frac{\Delta L}{L_0} = \frac{L - L_0}{L_0} \qquad Eq.\ [115]$$

$$C_{ref}^{surf} = \frac{\Delta A}{A_0} = \frac{A - A_0}{A_0} = \frac{(R_0 + \Delta R)^2 - R_0^2}{R_0^2} \cong 2\frac{R - R_0}{R_0} = 2\frac{L - L_0}{L_0} \qquad Eq.\ [116]$$

The formulation above does not only apply to a circular surface but to all closed surfaces.

9.4.3 *Resulting stresses in interfaces*

Interface elements are formed by node pairs, i.e. two nodes at each node position: one at the 'soil' side and one at the 'structure' side or the other 'soil' side. Interfaces can be visualised by activating the corresponding option in the explorer at the left of the output window. Output for interfaces can be obtained by double clicking on the

interface elements in the two-dimensional model. The output for interfaces comprises deformations (Deformations (on page 424)) and stresses.

When an interface is displayed, the options *Effective* σ'_N, *Total* σ_N, *Shear* τ, *Relative Shear* τ_{rel}, *Groundwater flow*, steady-state pore pressure p_{steady}, excess pore pressure p_{excess}, active pore pressure p_{active}, pore water pressures p_{water}, effective degree of saturation S_{eff}, suction, effective suction and *Groundwater head* are available from the *Interface stresses* menu. The effective normal stress is the effective stress perpendicular to the interface. Note that pressure is considered to be negative. The relative shear stress τ_{rel} gives an indication of the proximity of the stress point to the failure envelope, and is defined as:

$$\tau_{rel} = \frac{\tau}{\tau_{\max}} \qquad \textit{Eq. [117]}$$

where τ_{max} is the maximum value of shear stress according to the Coulomb failure envelope for the current value of the effective normal stress.

Total discharge through an interface [ADV]

In the case of semi-permeable interfaces, the total discharge through an interface can be obtained by selecting the *Interface stresses* → *Groundwater flow* → q_n/Q_n. Beneath the interface plot, the value of total discharge is displayed in the unit of volume per unit of time per unit of length in the out of plane direction. This option is useful in scenarios where it is required to know the groundwater discharge through a tunnel lining or a retaining wall.

Note:

- It is strongly recommended to select only one interface, to view discharge results. If double interfaces (for example, in the case of interfaces on both sides of a retaining wall) are selected, the results for discharge through these interfaces can be misleading.
- This option is not viable in case of fully permeable interface elements as Darcy's law cannot compute discharge for these scenarios. A workaround in such a case can be to use a semi-permeable interface and assign a very low hydraulic resistance *(d/k)* in case of fully permeable interfaces.

9.4.4 *Resulting stresses in discontinuities*

Discontinuity elements are, like Interface elements, formed by node pairs, i.e. two nodes at each node position separating the rock mass on either side of the discontinuity. Discontinuities can be visualised by activating the corresponding option in explorer on the left side of the Output window. Output for discontinuities can be obtained by double clicking on the discontinuity elements in the two-dimensional model. The output for discontinuities comprises deformations (Deformations (on page 424)) and stresses.

When an discontinuity is displayed, the options *Effective* σ'_N, *Total* σ_N, *Shear* τ, *Relative Shear* τ_{rel}, *Groundwater flow*, pore water pressures p_{water}, steady-state pore pressure p_{steady}, excess pore pressure p_{excess} and *Plastic points* are available from the *Discontinuity stresses* menu. The effective normal stress is the effective stress perpendicular to the discontinuity. Note that pressure is considered to be negative. The relative shear stress τ_{rel} gives an indication of the proximity of the stress point to the failure envelope, and is defined as:

$$\tau_{rel} = \frac{\tau}{\tau_{\max}} \qquad \textit{Eq. [118]}$$

where τ_{max} is the maximum value of shear stress according to the Coulomb failure envelope for the current value of the effective normal stress.

9.4.5 *Resulting forces in plates*

When a plate is displayed, the options *Axial Forces N*, *Shear Forces Q* and *Bending Moment M* are available from the *Forces* menu. For axisymmetric models the *Forces* menu also includes the forces in the out-of-plane direction (*Hoop Forces* N_z). Hoop forces are expressed in unit of force per unit of length. The values are constant over the circumference. Integration of the hoop forces over the in-plane length of the plate will give the total hoop force. All of these forces represent the actual forces at the end of the calculation step.

In addition to the actual forces, PLAXIS 2D keeps track of the historical maximum and minimum forces in all calculation phases. These maximum and minimum values up to the current calculation step may be viewed after clicking the *Distribution envelope* button in the top toolbar.}

Note that axial forces or hoop forces are positive when they generate tensile stresses, as indicated in Figure 267 (on page 444) .

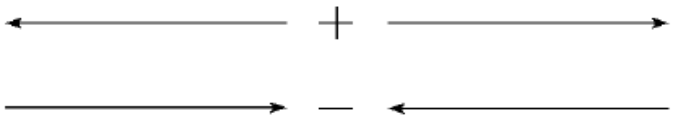


Figure 267: Sign convention for axial forces and hoop forces in plates

The *Axial force N* is the axial force along the element axis. The *Shear force Q* is the in-plane (*XY*) shear force. The *Bending moment M* is the bending moment due to bending over the out-of-plane axis (*Z*).

9.4.6 *Resulting forces in geogrids*

When a geogrid is displayed, the option *Axial force* is available. Forces in geogrids are always positive (tension). Compressive forces are not allowed in these elements.

In addition to the actual forces, PLAXIS 2D keeps track of the historical maximum and minimum forces in all calculation phases. These maximum and minimum values up to the current calculation step may be viewed after clicking the *Distribution envelope* button in the top toolbar.}

9.4.7 *Resulting forces in embedded beam rows*

When an embedded beam row is displayed, the options *Axial force N*, *Shear force Q* , *Bending moment M*, *Skin force* T_{skin} (in axial pile direction) the lateral force T_{lat}, the maximum shear stress $T_{skin,max}$ and the relative shear stress $T_{skin,rel}$ are available from the *Forces* menu. The latter four options relate to the pile-soil interaction (see below).

Note:

The *Axes* option from the *View* menu may be used to display the pile's local system of axes.

The pile-soil interaction forces are obtained from the special interface that is automatically applied between the embedded beam elements and the surrounding soil volume elements. The *Skin force* T_{skin}, expressed in the unit of force per unit of pile length per unit of width in the out-of-plane direction, is the force related to the relative displacement in the pile's first direction (axial direction). This force is limited by the skin resistance as defined in the embedded beam row material data set (General properties (on page 275)).

The interaction force T_{lat} relates to the relative displacement perpendicular to the pile in the pile's second direction. These quantities are expressed in the unit of force per unit of pile length per unit of width in the out-of-plane direction.

The maximum shear stress $T_{skin,max}$ is the limit defined for the material dataset. The relative shear stress $T_{skin,rel}$ gives an indication of the proximity of the stress point to the failure envelope.

The pile foot force F_{foot}, expressed in the unit of force per unit of width in the out-of-plane direction, is obtained from the relative displacement in the axial pile direction between the foot or tip of the pile and the surrounding soil. The foot force is shown in the plot of the *Axial force N*. The foot force is limited by the base resistance as defined in the embedded beam row material data set (General properties (on page 275)).

In addition to the actual forces, PLAXIS 2D keeps track of the historical maximum and minimum forces in all calculation phases. These maximum and minimum values up to the current calculation step may be viewed after clicking the *Distribution envelope* button in the top toolbar.

9.4.8 *Resulting force in anchors*

Output for anchors (fixed-end anchors as well as node-to-node anchors) involves only the anchor force expressed in the unit of force on the anchor (on the nodes in node-to-node anchor). The anchor force appears in a table after double clicking the anchor in the model. The program displays the values of the historical maximum and minimum forces in all calculation phases in node-to-node anchors.

9.4.9 *Results in connections*

The results in the connections available in the model are displayed when the *Connections information* option is selected in the *Project* menu. An example of the appearing information window is given in Figure 268 (on page 445).

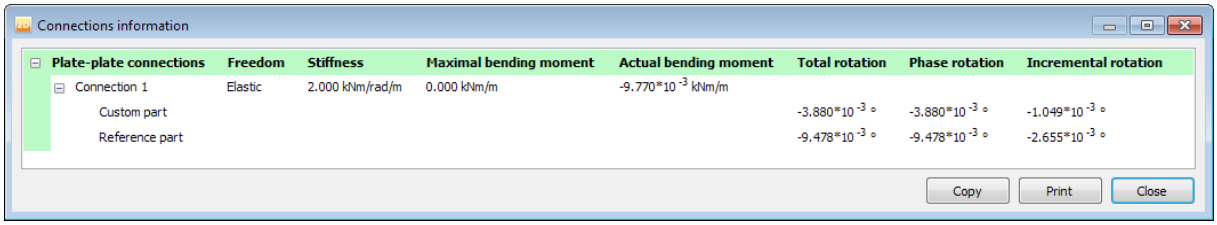


Figure 268: Resulting bending moment in a rotation spring

9.4.10 *Structural forces in volume plates*

The *Structural forces in volume plates* feature is available in the toolbar or as an option in the *Tools* menu in the Output program (Tools menu (on page 392)). Using this feature, it is possible to visualise structural forces (bending moments M, shear forces Q and axial forces N) in a regular structure (rectangular or tapered) that is composed of volume elements in which only stresses have been calculated. In this way it is possible, for example, to display the structural forces in a wall structure that is composed of volume elements with an assigned data set with concrete properties.

Note:

- Note that the structural forces are calculated by integrating the results in the stress points along the region perpendicular to the cross section line. Special care is required when the structural forces in the region of connection of the subparts are evaluated.

Creating a cross section line

When the feature is selected the *Draw a centerline* button in the side toolbar is automatically selected and the **Center line points** window pops up (Figure 269 (on page 446)).

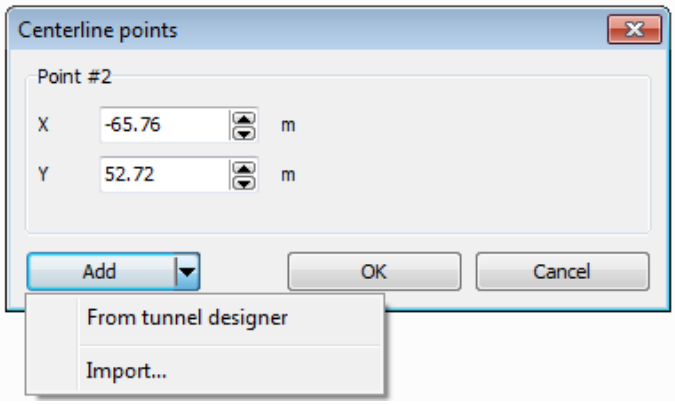


*Figure 269: **Center line points** window*

The cross section line may consist of several line sections. The points defining the segments composing the centerline can be defined either by directly clicking on the model or by defining the coordinates of the location in the **Center line points** window. The process of center line definition can be quit by either clicking the *OK* button in the **Centerline points** window or by right clicking and selecting the *Finish* option in the appearing menu. The structural forces are then computed on-the-fly and visualised along the created line.

From tunnel designer: It is also possible to define the *Center line points* using the option *From tunnel designer*, by clicking on the dropdown menu using *Add*. To be able to use this option, a thick lining must be present in the tunnel. This can be done in Input using the *Tunnel designer* (for more detailed information, refer Create polycurve). When the thick lining is added to a tunnel, PLAXIS 2D automatically generates a *data.otnl*, which is stored in the folder of the current project. This file contains all the information about the center line.

Click *OK* to view the *Forces*.

Note:

Note that when more than one thick lining is defined, the *data.otnl* is overwritten in the project folder. To save both files, the user must manually rename the exsiting *data.otnl* before saving the new thick lining.

Import: *Center line points* can also be imported using the *Import* option, by clicking on the dropdown menu using *Add*. The **.otnl*, **.tnl* or **.json* files are located in the project folders.

Click *OK* to view the *Forces*.

At the drawn cross section line, the selected structural force is calculated on the basis of the integral of the stresses perpendicular to the cross section line. The extent of the area that is used to integrate the stresses is limited by a radius. The default radius, for each point of the cross section, is defined by the elements that contain the same material data set as the element in which the cross section point is drawn. However, this radius may be redefined by the user.

Automatic center lines wizard [GSE]

As an alternative to the manual creation of a centerline, the centerline can be automatically generated using the *Generate centerlines* wizard.

The automatic center lines wizard can be used in two situations as described below.

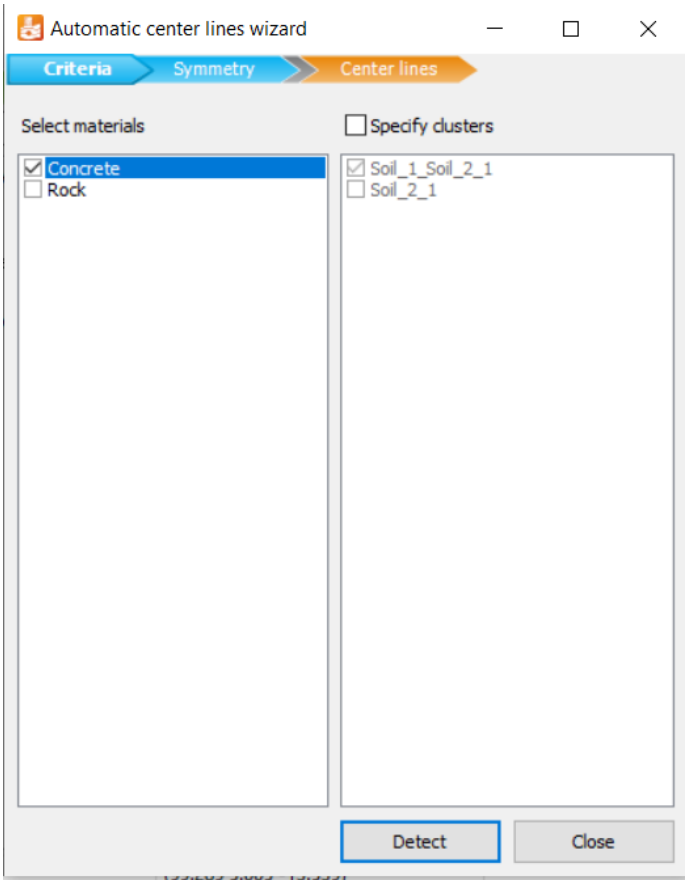


Figure 270: Automatic center lines wizard Criteria tabsheet

Criteria	In this blue tabsheet, it is possible to select the materials whose centerline is required for calculation of structural forces. In addition, individual volume clusters can be specified in the righthand panel. Click *Detect* button to generate the center lines.
Center lines	In this orange tabsheet, the results for the centerlines are shown. Here it is possible to select the specific centerline of interest. Each centerline can easily be toggled to show or hide it.

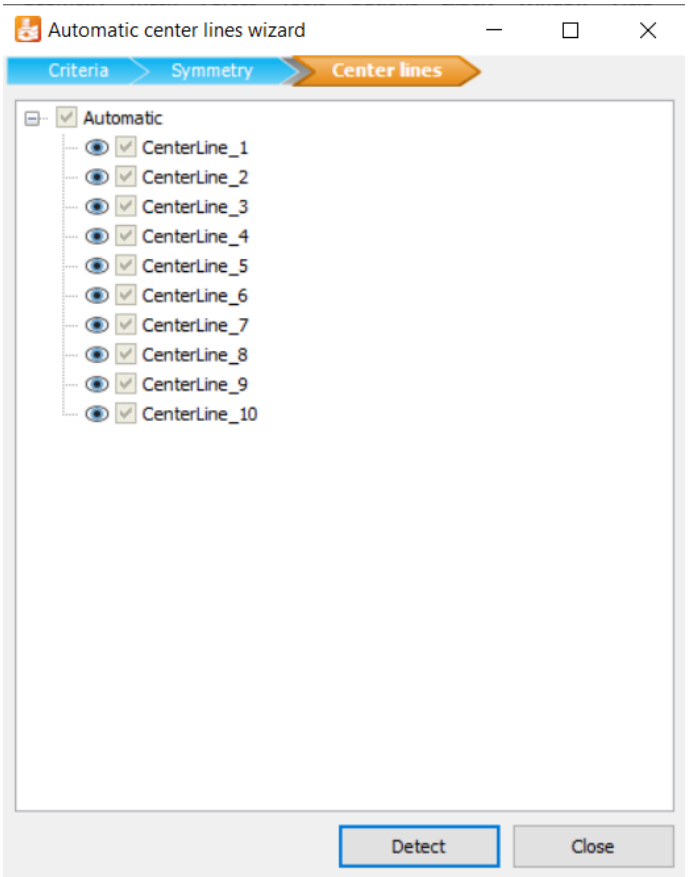


Figure 271: Automatic center lines wizard Center lines tabsheet

The structural forces are computed by integrating the stresses along the cross-sections which are perpendicular to the centerlines which have been generated in the previous step.

Changing the extent of the stress range

The range of the stress to be taken into account can be modified by specifying the radius of the extension from the centerline. The initial radius is determined by the distance, across the cross section line (centerline), to the nearest cluster which has a different material assigned.

When the *Edit radii* button is clicked in the side toolbar, the extent of the stress range is indicated in the plot by a transparent green colour (Figure 273 (on page 449)).

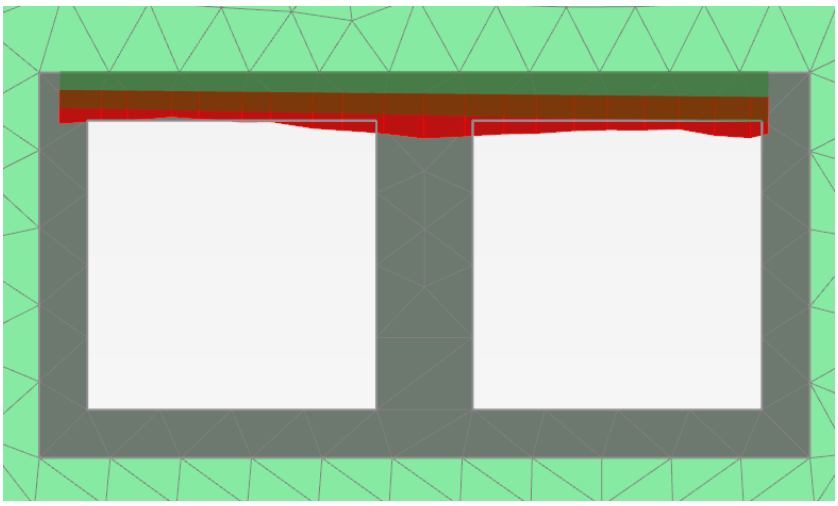


Figure 272: Figure displaying the structural force in a volume and the default radii

It is possible to modify the stress range by clicking on the highlighted area and dragging this area by the mouse (Figure 273 (on page 449)).

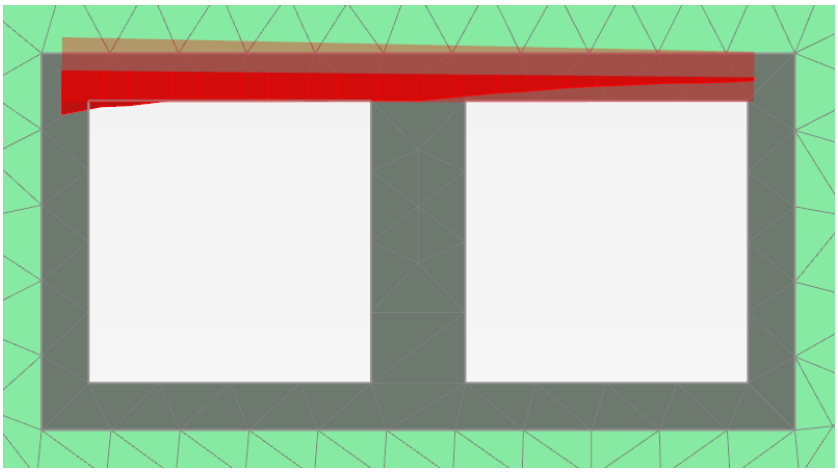


Figure 273: Figure displaying the structural force in a volume and the modified radii

Note that the dragging location affects the way how the radii are modified. If the area is dragged near the start or near the end of the centerline, only the radius for that location (start radius or end radius) will be modified. This option may be used for tapered structures. However if the dragging location is approximately in the middle of both ends of the centerline, both radii will be modified.

Alternatively, if the segment is double-clicked, a dialog will be opened in which the radii can be specified precisely (Figure 274 (on page 450)). The defined cross section lines (center lines) can be removed from the plot by right clicking on the plot and by selecting the *Clear all* option appearing when the *Lines* option is pointed in the appearing menu.

Note:

The *Create animation* feature can be used to view the evolution of the structural forces in volumes in calculation phases.

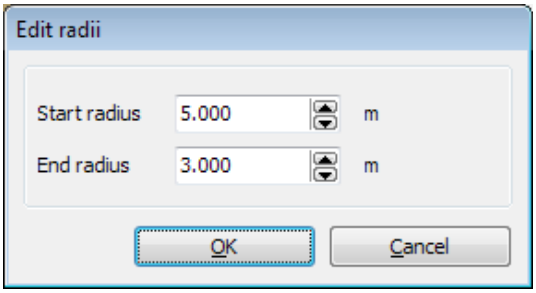


Figure 274: ***Edit radii*** *window*

10 Curves

The development of quantities over multiple calculation steps at a specified location in the model can be viewed using the *Curve manager* facility. This facility allows for the generation of load-displacement curves, force-displacement curves, stress-paths, strain paths, stress-strain curves and time-related curves.

10.1 Selecting points for curves

The location in the model where the variation of results through calculation steps is to be analysed is specified by selecting nodes or stress points in the model. The selection of points should be done preferably before but may also be done after calculating the project.

In order to specify points to be considered in curves, the *Select points for curves* option should be selected. This option is available as a button in the toolbar of the **Staged Construction mode** in PLAXIS input and as an option in the *Tools* menu of the **Staged Construction mode**. Selecting this option will open the Output program displaying the *Connectivity plot* and the **Select points** window.

Nodes and stress points can be selected in the Output program either by clicking the *Select points for curves* button in the side toolbar or by selecting the corresponding option in the *Tools* menu. More information on selecting procedure is given in Mesh point selection (on page 451).

It is important to consider the differences in selecting the points before or after starting the calculation process. A more detailed description is given in Pre-calculation points (on page 452) and Post-calculation points (on page 453).

10.1.1 *Mesh point selection*

Nodes and stress points can directly be selected by clicking them in the 2D model. Make sure that the *Nodes numbers* and/or *Stress point numbers* option has been selected in the *Mesh* menu.

The amount of visible nodes and stress points can be decreased using the *Partial geometry* option in the *Geometry* menu or by clicking the *Hide soil* button in the side toolbar.

In the **Select points** window (Figure 275 (on page 452)), the coordinates of the location of interest can be specified. The program lists the number of the nearest node and stress points at the lower part of the window when the *Search closest* button is clicked. The nodes and stress points can be selected by defining their ID as well. The displayed nodes or stress points are selected as the corresponding button at the right of the cell is checked. The selections are listed in the upper part of the window.

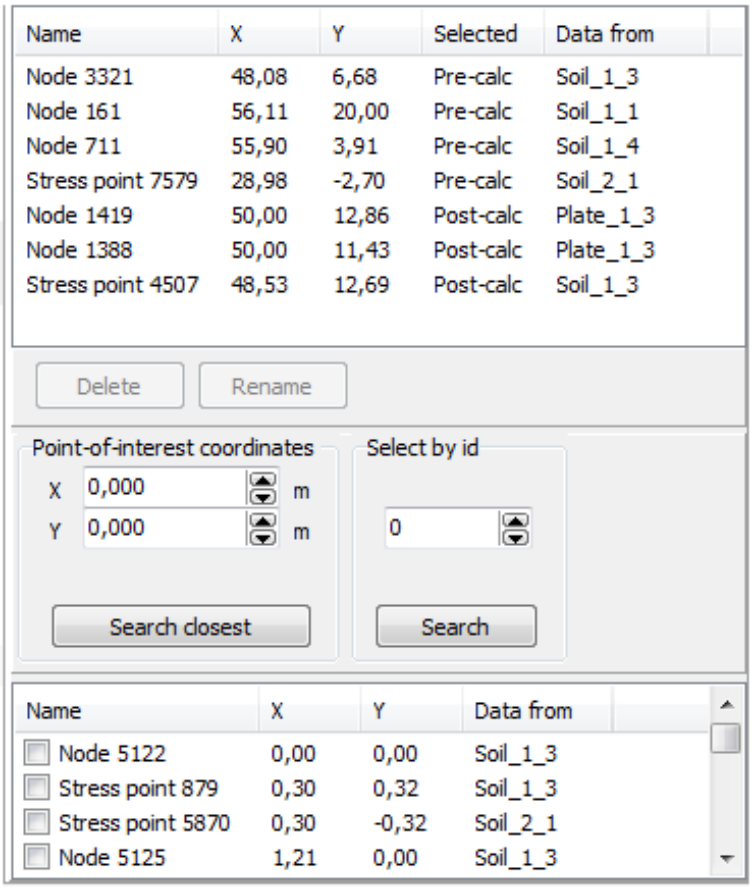


Figure 275: ***Select points*** *window*

Selected nodes can be deselected by selecting the point in the list and pressing *Delete* or by clicking the point in the model.

Note:

When the *Select points for curves* option is selected but the **Select points** window is closed, it can be displayed by selecting the *Mesh point selection* option in the *Tools* menu.

If the finite element mesh is regenerated (after being refined or modified), the position of nodes and stress points will change. As a result, previously selected nodes and stress points may appear in completely different positions. Therefore nodes and stress points should be reselected after regeneration of the mesh.

10.1.2 *Pre-calculation points*

After the calculation phases have been defined and before the calculation process is started, some points may be selected by the user for the generation of load-displacement curves or stress paths. During the calculation, information for these selected points for all the calculation steps is stored in a separate file. The precalculation points provide more detailed curves.

Note:

Pre-calculation points provide detailed information related to stress and strain at those points. However, information about structural forces and state parameters is not provided.

10.1.3 *Post-calculation points*

When the calculations are started without the selection of nodes and stress points for curves, the user will be prompted to select such points. The user can then decide to select points or, alternatively, to start the calculations without pre-selected points. In this case, it is still possible to generate load-displacement curves or stress-strain curves after the calculation, but such curves may be less detailed.

When a node or stress point is selected after calculating the project, only the information for the saved calculation steps is available. For more detailed curves the value of the *Max number of steps stored* should be increased.

Note:

In generating curves, the user can use pre-calculation selected points and post-calculation selected points. However, with structural forces, this can be done only using post-calculation selected structural nodes.

The information available for selected points (nodes or stress point) depends on the view in which they have been selected in the Output program.

The points selected in the *Model* view, can be used to generate curves related to displacements, stresses, strains and state parameters in soil elements. The *Model* view is the default view in the Output program.

 The points selected in the *Structure* view, can be used to generate curves related to resulting structural forces. The points should be selected after selecting the structure first (Select structures (on page 403)). The *Structure* view is displayed when structures are selected and double clicked.

Note:

The type of the active view is indicated by the corresponding icon under the plot.

10.2 Generating curves

To generate curves, the *Curves manager* option should be selected from the *Tools* menu or the corresponding button in the toolbar should be clicked. As a result, the **Curves manager** window appears with three tabsheets named **Charts**, **Curve points** and **Select points**.

The **Charts** tabsheet contains the saved charts that were previously generated for the current project. The **Curve points** tabsheet gives an overview of the nodes and stress points that were selected for the generation of curves, with an indication of their coordinates. The list includes the points selected before the calculation (*pre-calc*) as well as the points selected after the calculation (*post-calc*) (Figure 276 (on page 454)). For points that are part of a structure further information is given in the list about the type of structure and the corresponding structure element number. The **Select points** window is described in Mesh point selection (on page 451).

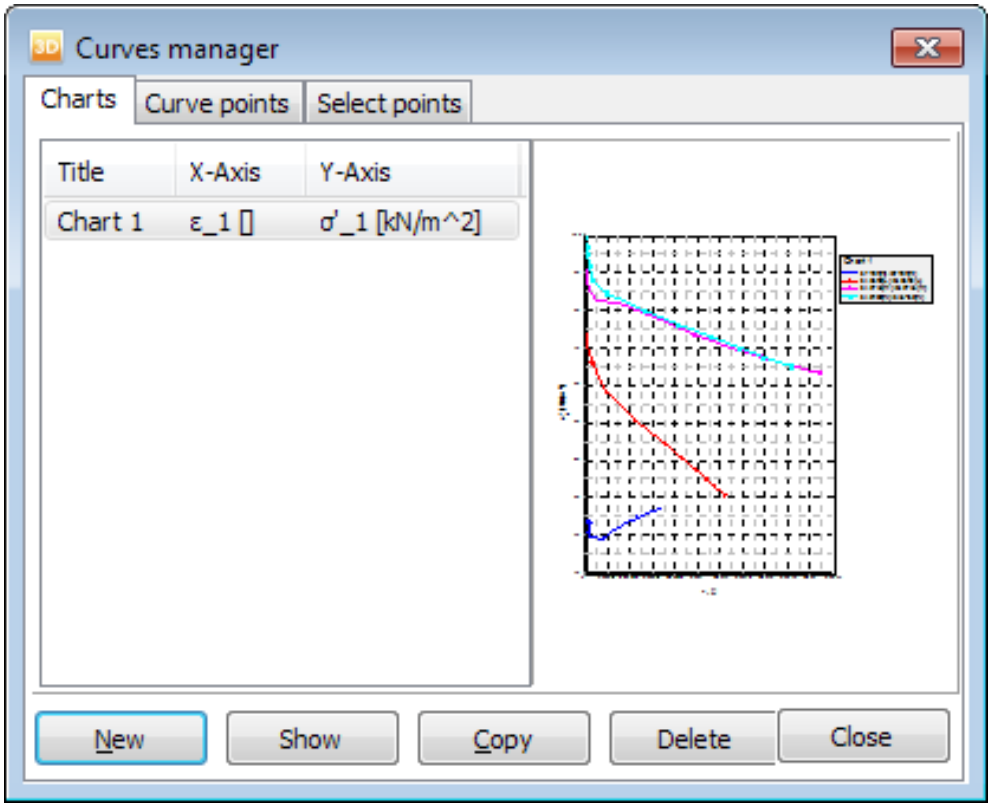


Figure 276: ***Curve manager*** *window*

As a next step to generate curves, the *New* button should be pressed while the **Charts** tabsheet is active. As a result, the **Curve generation** window appears, as presented in Figure 277 (on page 455).

Two similar groups with various items are shown, one for the *x*-axis and one for the *y*-axis of the curve. The *x*-axis corresponds to the horizontal axis and the *y*-axis corresponds to the vertical axis. For each axis, a combination of selections should be made to define which quantity is plotted on that axis. First, for each axis a selection should be made whether the data to be shown is related to the general project (*Project*) or a particular selected node or stress point. The tree in the **Curve generation** window will then show all quantities which are available depending for this type of data. The tree can be expanded by clicking the + sign in front of a group. The *Invert sign* option may be selected to multiply all values of the *x*-quantity or the *y*-quantity by -1. When both quantities have been defined and the *OK* button is pressed, the curve is generated and presented in a chart window.

The combination of the step-dependent values of the *x*-quantity and the *y*-quantity forms the points of the curve to be plotted. The number of curve points corresponds to the available calculation step numbers plus one. The first curve point (corresponding to step 0) is numbered as 1.

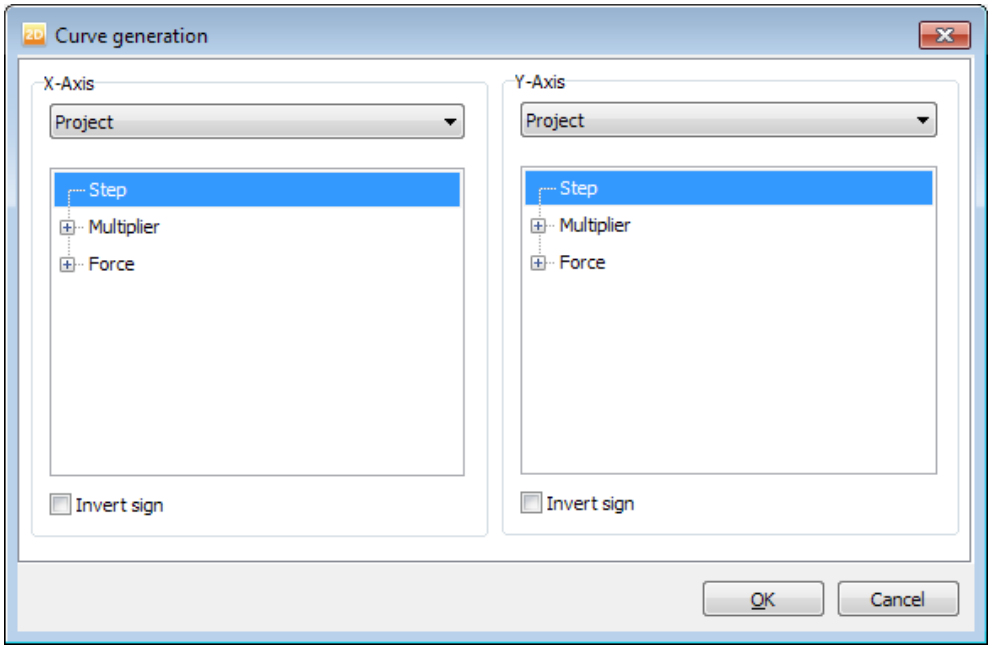


Figure 277: ***Curve generation*** *window*

Note:

- When curves are generated from points selected after the calculation, only information of saved steps can be considered. The number of the saved steps for each calculation phase is defined by the *Maximum number of steps stored* option in the *Numerical control parameters* subtree in the **Phases** window (Numerical control parameters (on page 337)).
- All the calculation results are available for the pre-selected points.

10.2.1 *Load-displacement curves*

Load-displacement curves can be used to visualise the relationship between the applied loading and the resulting displacement of a certain point in the geometry. In general, the *x*-axis relates to the displacement of a particular node (*Deformations*), and the *y*-axis contains data relating to load level. The latter is related with the value of *ΣMstage* in the following way: Applied load = Total load applied in previous phase + *ΣMstage* · (Total load applied in current phase - Total load applied in previous phase). Also other types of curves can be generated.

The selection of *Displacement* must be completed with the selection of a node in the drop-down menu and the selection of a displacement component in the *Deformations* subtree. The type of displacement can be either the length of the displacement vector ($|u|$) or one of the individual displacement components (u_x, u_y). The displacements are expressed in the unit of length, as specified in the **Project properties** window of the Input program.

To define a multiplier on the *y*-axis, first the *Project* option should be selected as the activation of a load system is not related to a particular point in the geometry. The selection must be completed with the selection of the desired load system, represented by the corresponding multiplier in the *Multiplier* subtree. Note that the 'load' is

not expressed in units of stress or force but in a multiplier value without unit. To obtain the actual load, the presented value should be multiplied by the input load as specified by means of staged construction.

Another quantity that can be presented in a curve is the *Pore pressure*. This quantity is available for selected nodes as well as stress points. In the *Pore pressures* subtree of the *Stresses* tree p_{active}, p_{steady} or p_{excess} can be selected. Pore pressures are expressed in the unit of stress.

When non-zero prescribed displacements are activated in a calculation, the reaction forces against the prescribed displacements in the *x*- and *y*-direction are calculated and stored as output parameters. These force components can also be used in the load-displacement curves by selecting the option *Project* and then selecting one of the forces in the *Forces* subtree.In plane strain models the *Force* is expressed in the units of width in the out-of-plane direction. In axisymmetric models the *Force* is expressed in the unit of force per radian. Hence, to calculate the total reaction force under a circular footing that is simulated by means of prescribed displacements, the F_y value should be multiplied by 2π.

10.2.2 *Force-displacement curves*

Force-displacement curves can be used to visualise the relationship between the development of a structural force quantity and a displacement component of a certain point in the geometry. A structural force quantity can only be selected for nodes being selected after the calculation. In general, the *x*-axis relates to the displacement of a particular node (*Displacement*), and the *y*-axis relates to the corresponding structural force of a node of a structural element.

To define a displacement on the *x*-axis, first the desired node should be selected. The selection must be completed with the selection of the type of displacement. This displacement can be either the length of the displacement vector ($|u|$) or one of the individual displacement components (u_x, u_y). The displacements are expressed in the unit of length, as specified in the **General settings** window of the Input program.

To define a structural force on the *y*-axis, first the desired node of the structural element should be selected. The selection of *Structural force* must be completed with the selection of type of force. Depending on the type of structural element, a selection can be made among axial forces *N*, shear forces *Q* or bending moments *M*. In case of interfaces, a selection can be made among the interface stresses (Structures and interfaces (on page 441)).

10.2.3 *Displacement-time or force-time curves*

Time-dependent curves can be useful to interpret the results of calculations in which the time-dependent behaviour of the soil plays an important role (e.g. consolidation and creep). In this case, the *Time* option is generally selected for the *x*-axis, and the *y*-axis contains data for a displacement component, a stress component, a structural force quantity, or the temperature of a particular node or stress point. The selection of *Time* requires the *Project* option to be selected. *Time* is expressed in the unit of time, as specified in the **Project properties** window of the Input program.

Instead of selecting time for the horizontal axis, it is also possible to select the calculation step number (*Step*). This may also give useful curves for time independent calculations. When interpreting such a curve it should be noted that during the calculation the step size might change as a result of the automatic load stepping procedures.

10.2.4 *Stress and strain diagrams*

Stress and strain diagrams can be used to visualise the development of stresses (stress paths) or strains (strain paths) or the stress-strain behaviour of the soil in a particular stress point. These types of curves are useful to analyse the local behaviour of the soil. Stress-strain diagrams represent the idealised behaviour of the soil according to the selected soil model. Since soil behaviour is stress-dependent and soil models do not take all aspects of stress-dependency into account, stress paths are useful to validate previously selected model parameters.

First a stress point should be selected before the desired quantity can be selected in the *Stress* or *Strain* tree. The selection must be completed with the selection of the type of stress or strain. As a stress quantity all scalar quantities available in the *Stresses* menu can be selected (Stresses (on page 428)). However, the *State parameters* option is only available for stress points selected after the calculation Post-calculation points (on page 453)). As a strain quantity all scalar strain quantities available in the *Deformations* menu can be selected (Deformations (on page 424)).

See the Scientific Manual for a definition of the stress and strain components. The phrase 'in absolute sense' in the description of the principal components is added because, in general, the normal stress and strain components are negative (compression is negative). Stress components are expressed in the units of stress; strains are dimensionless. A definition of the stress and strain components is given in the Material Models Manual.

10.2.5 *Curves in Dynamics calculations* [ULT]

The **Curve generation** window differs when dynamics calculations are executed in the project. The normal tabsheet is similar to the tabsheet when no dynamics calculations are performed. However, the *Dynamics time* option is available in the tree when *Project* is selected in the axis parameter drop down menu. When a node is selected, the *Velocities*, *Accelerations* and *Acceleration ('g')* options are available under *Deformation* (Figure 278 (on page 458)).

Note:

In generating curves, the user can use pre-calculation selected points and post-calculation selected points. However, with structural forces, this can be done only using post-calculation selected structural nodes. For more information, see Post-calculation points (on page 453).

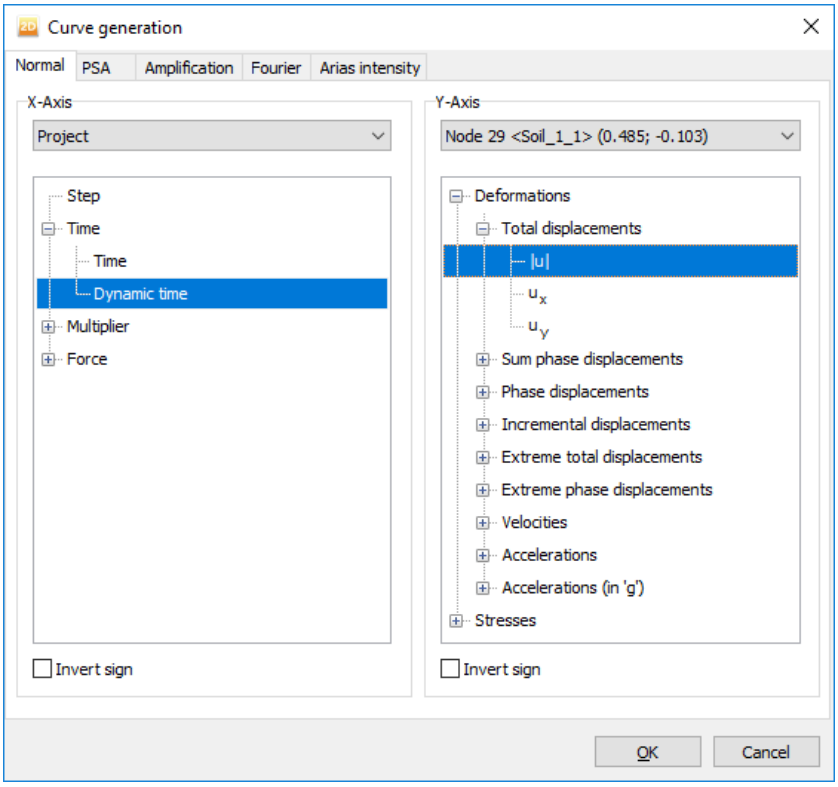


*Figure 278: Options available in **Normal** tabsheet for dynamics calculations*

PSA spectrum

During an earthquake, the soil vibration causes forced vibrations on the overlying structure. The structure can be idealised as a single degree of freedom (SDOF) system, i.e. a simple model characterized by a mass *m*, concentrated in a point, a stiffness *k* and a damping ratio ξ. The dissipation, due to structural elements, is simulated through viscous actions depending on the velocity (See Figure below).

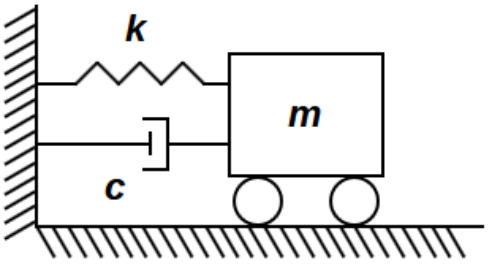


Figure 279: Single Degree Of Freedom System

The dynamics equation of motion is given by:

$$m\ddot{u} + c\dot{u} + ku = -m\ddot{u}_g \qquad Eq.\ [119]$$

where u, $\dot{u}$ and $\ddot{u}$ represent the displacement, velocity and acceleration of the mass with respect to the base, respectively, c is a damping coefficient and $\ddot{u}_g$ is the input motion. The relationship between c and ξ is expressed by the following equation:

$$\xi = \frac{c}{2\sqrt{km}} \qquad \textit{Eq. [120]}$$

The solution *u(t)* of a SDOF system subjected to an earthquake is given by Duhamel's integral and its maximum value is used to plot the relative displacement response spectrum. In general, the response spectrum is the locus of the maximum response of SDOF systems characterised by different *k* and subjected to the same earthquake, for a given damping ratio. The maximum response can be calculated in terms of displacements, velocities or accelerations. The relative displacement response spectrum S_D is given by:

$$S_D = \left| u \right|_{\max} \qquad \textit{Eq. [121]}$$

The relative pseudospectral velocity PSV and the absolute pseudospectral acceleration PSA are related to the relative displacement response spectrum according to the following equations:

$$PSV = \omega_0 \cdot S_D \qquad \textit{Eq. [122]}$$

$$PSA = \omega_0^2 \cdot S_D \qquad \textit{Eq. [123]}$$

where ω_0 represents the natural angular frequency of the structure, $\omega_0 = \sqrt{k / m}$. PSV and PSA are indicated as pseudo quantities because, in a damped system, they do not coincide exactly with the maximum relative velocity ($\left| \dot{u} \right|_{max}$) and the maximum absolute acceleration ($\left| \ddot{u} + \ddot{u}_g \right|_{max}$).

The maximum response is plotted on the y-axis of the response spectrum, while the natural period *T* is represented on the x-axis, which contains the information about the structure stiffness *k* and mass *m*, as shown in the following equation:

$$T = \frac{2\pi}{\omega_0} = 2\pi\sqrt{\frac{m}{k}} \qquad \textit{Eq. [124]}$$

By varying *k*, it is possible to obtain the response of all the possible SDOF systems for that specific seismic signal for a given damping ratio ξ. For typical structures in reinforced concrete where the damping is mainly due to non structural elements such as partition walls, ξ is generally set equal to 5%. Higher values may be used in other specific cases, for instance when the structure is isolated at the base with dampers, which results in lower y-values in the response spectrum.

Considering the PSA spectrum, the value that can be read on the Y-axis corresponding to the natural period of vibration of the overlying structure allows to calculate the maximum shear stress at the base of the structure, equal to the maximum inertial force of a structure with mass *m* subjected to an acceleration given by the selected *PSA* value (Figure 280 (on page 460)):

$$F_{\max} = m \cdot PSA = \frac{PSA}{g} w \qquad \textit{Eq. [125]}$$

where *g* is the gravity acceleration and *w* is the weight of the structure.

To generate the response spectrum, select the PSA tab from the Curve generation window when selecting *New* in the *Curve manager* (Figure 280 (on page 460)). In the *PSA* tab, a selection has to be made among horizontal acceleration (a_x), vertical acceleration (a_y) or absolute value of the acceleration (vectorial length). Subsequently, the structural damping ratio, ξ, has to be specified, together with the maximum natural period of the structure, T.

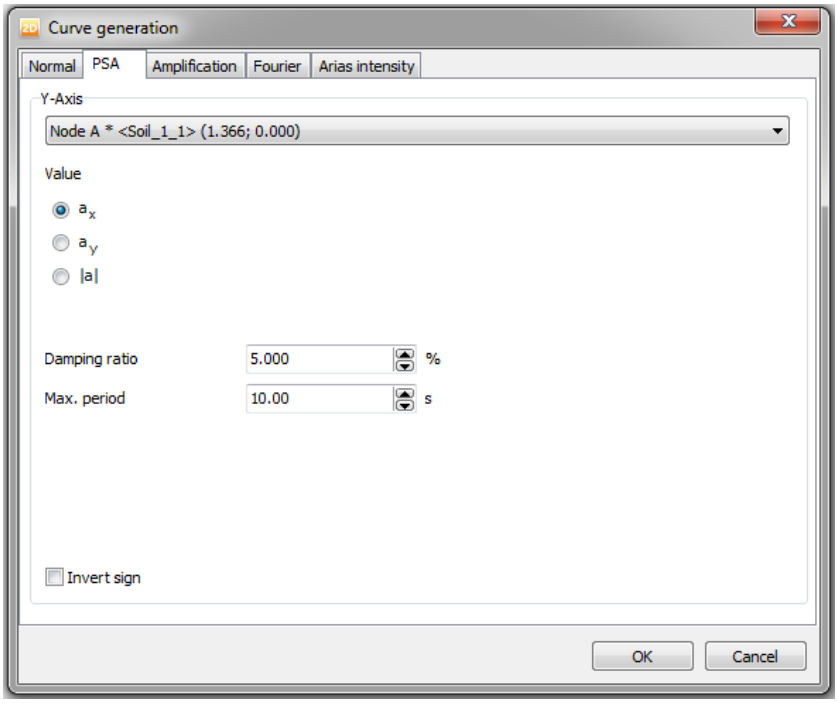


Figure 280: Generate PSA

An example of a pseudo-acceleration response spectrum determined at the foundation level at the end of a site response analysis is shown in the figure below.

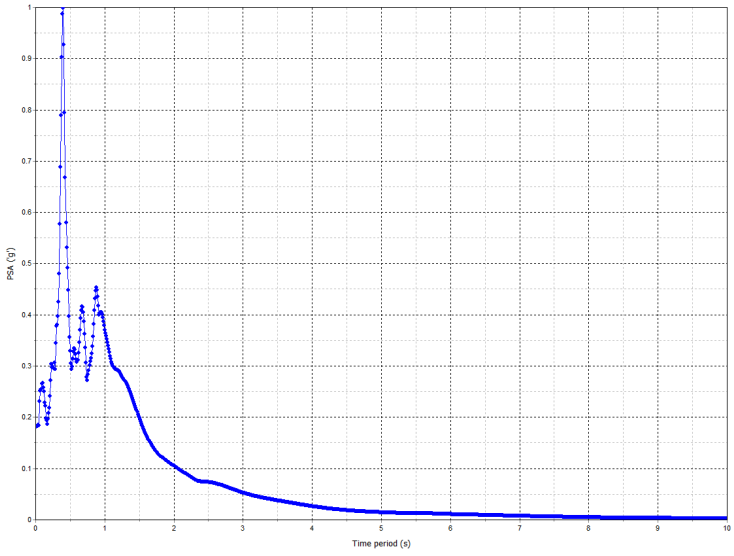


Figure 281: PSA

Based on the PSA diagram, it is possible to determine the predominant period, i.e. the period related to the peak PSA value. In the PSA diagram in Figure 281 (on page 460), this is equal to 0.4 s. If the natural period of the structure corresponds to the predominant period, the structure, subjected to the selected earthquake and in that

specific soil condition, is in resonance, i.e. the system oscillates with very high amplitudes which can cause great damages in the building.

Once the PSA spectrum has been generated, it is possible to display the relative displacement response spectrum by selecting *Settings* under the *Format* menu. The *Settings* window opens and it is possible to select the *Chart* tabsheet and the *Relative displacement* response spectrum option (Figure 282 (on page 461)).

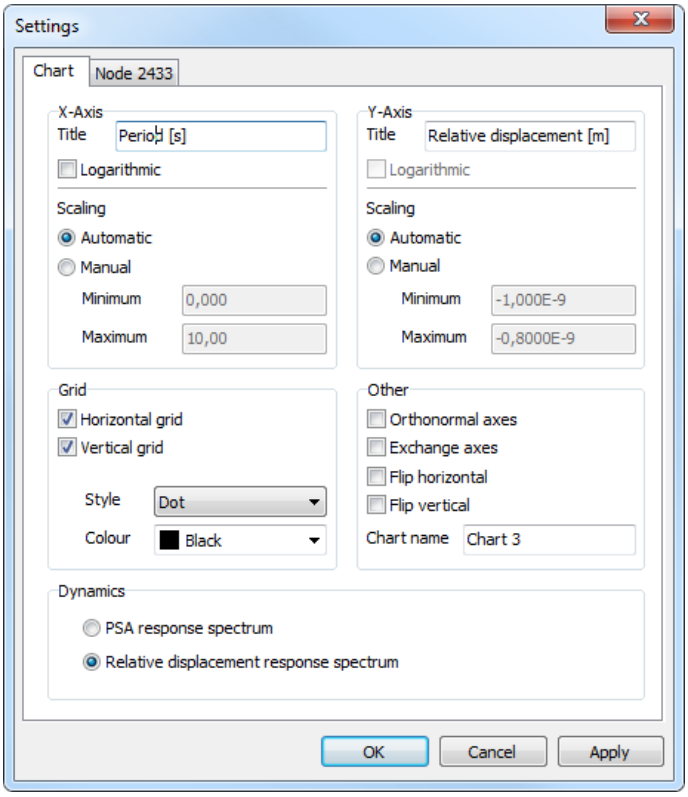


Figure 282: Generate Relative displacement response spectrum

The corresponding chart (Figure 281 (on page 460)) shows the maximum relative displacement that can occur in the structure, characterised by a natural period of vibration T and subjected to an earthquake, for a given damping ratio ξ.

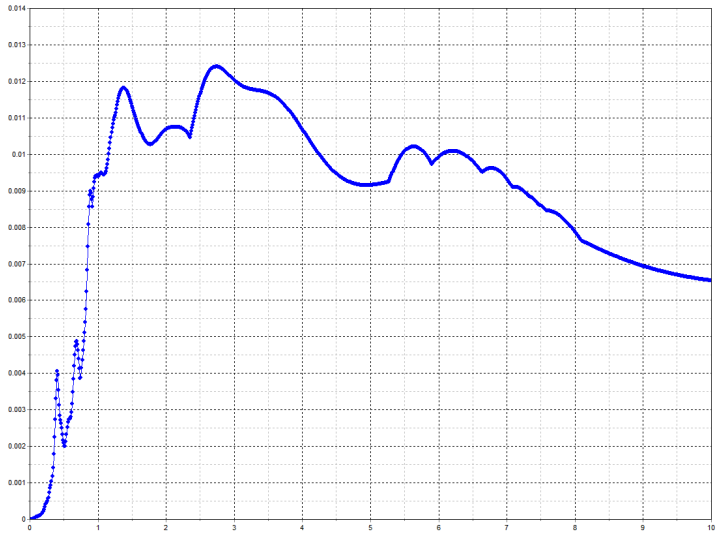


Figure 283: Relative displacement response spectrum

The product of the maximum relative displacement and the stiffness *k* represents the maximum equivalent static force, i.e. the static force that causes the same maximum deformation induced by the earthquake:

$$F_{\max} = k \cdot S_D \qquad Eq.\ [126]$$

By comparing the PSA spectrum with the *Relative displacement* response spectrum, it can be seen that at low periods, that characterize rigid structures, the displacement is close to zero. Ideally, in an undamped perfectly rigid structure (T = 0) subjected at the base to an acceleration $\ddot{u}_g(t)$, the mass moves together with the soil, i.e. there is no relative displacement of the mass ($| u |_{max} = 0$) and the pseudo spectral acceleration coincides with the maximum absolute acceleration at that level and so, with the maximum acceleration of the soil ($| \ddot{u} + \ddot{u}_g |_{max} = | \ddot{u}_g |_{max}$).

By increasing the period, both the PSA and the relative displacement increase. For very high periods, the maximum response in terms of acceleration tends to decrease and reaches values even lower than the maximum acceleration at the selected level. The relative displacement tends to the absolute displacement of the soil. Ideally, in an undamped highly flexible structure ($T = \infty$) subjected at the base to an earthquake, the relative displacement of the mass coincides with the absolute displacement of the soil.

Reinforced concrete structures are generally characterised by periods of vibration corresponding to an amplification of the acceleration of the soil.

Amplification

The *Amplification* tabsheet enables obtaining the plot which shows the ratio of the acceleration response of any point (*Top*) to the acceleration response of another point (*Bottom*) which is preferably the point where input load is applied (“Figure 284 (on page 463)”). This gives the magnification of the response at one point with respect to given excitation.

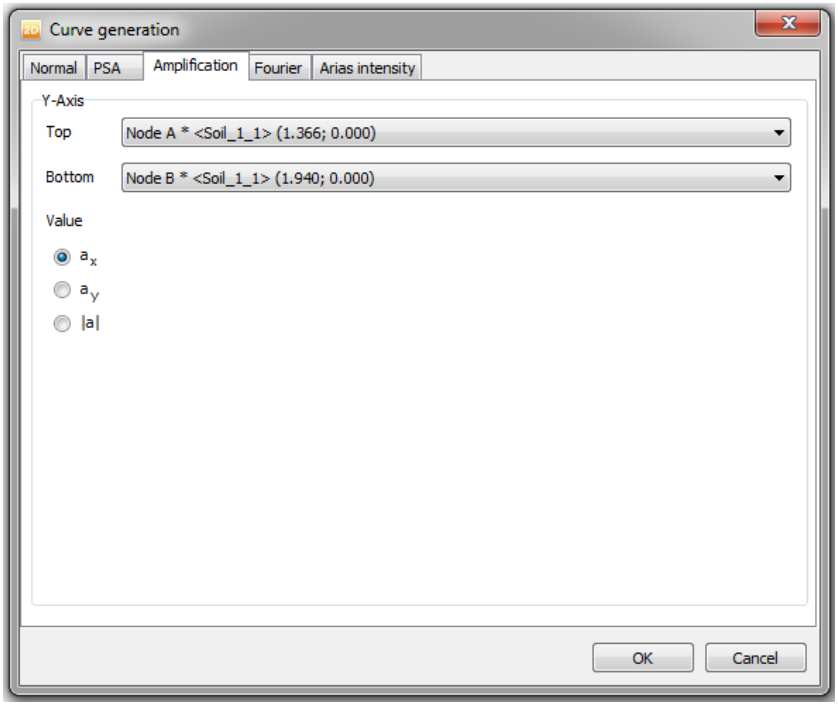


Figure 284: Amplification spectrum generation

Fourier

A quantity plotted in the time domain can be transformed into the frequency domain through the Fast-Fourier Transform (FFT), which contains both the amplitude and the phase information of the signal. From the computed FFT, the user can plot the *Power Spectrum* and the *Fourier Amplitude Spectrum(FAS)* for each acceleration component.

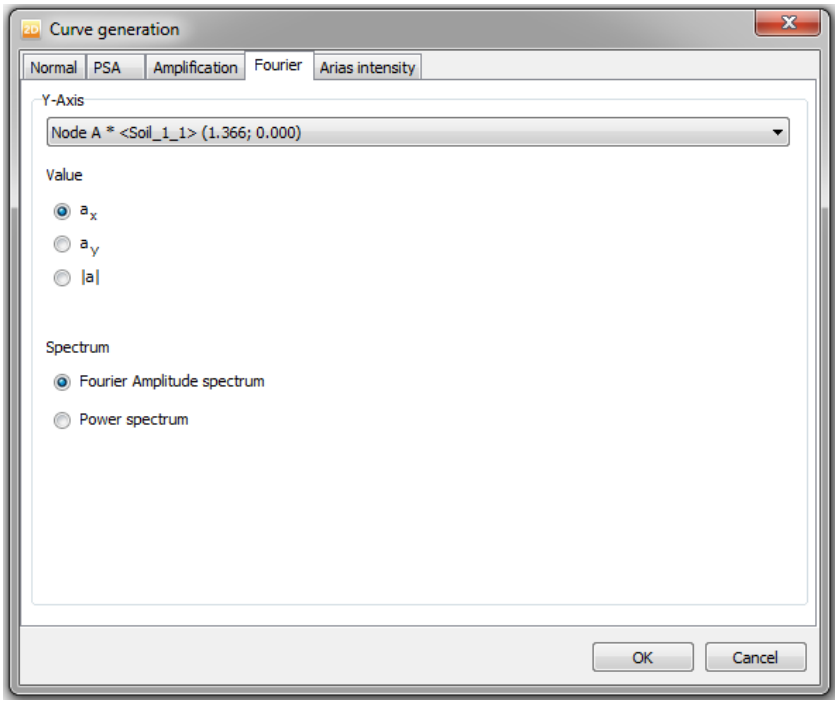


Figure 285: Fourier curve generation

Arias intensity

The *Arias intensity*, I_a (length/dyn. time) is used to determine the strength of a ground motion (intensity of shaking). It is calculated based on the time history acceleration, *a(t)*, in a node:

$$I_a = \frac{\pi}{2g}\int_0^T d[a(t)]^2 \, d\, t \qquad Eq.\ [127]$$

where g is the gravity acceleration and T_d is the duration of the motion (i.e. maximum dynamics time in the time history acceleration).

The *Arias Intensity* has the dimension of velocity (Figure 286 (on page 465)). In the **Chart** tabsheet, the user can choose to represent I_a in percentage of the maximum *Arias intensity*.

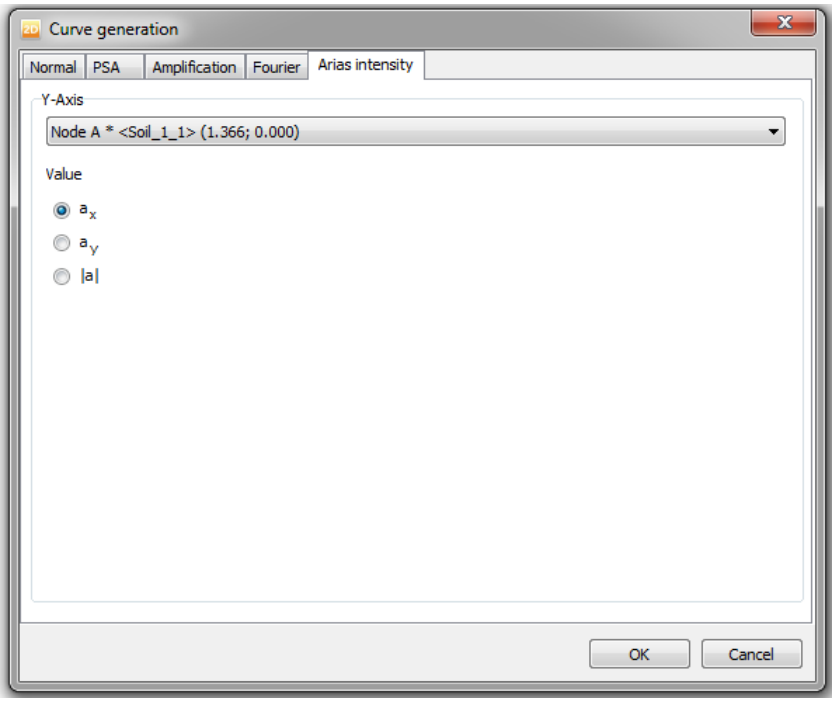


Figure 286: Arias curve generation

In the top toolbar, two options are available for specifying conditions for the *Arias Intensity* curves. Clicking the *Arias duration* button, the user can specify the minimum and maximum value of I_a in order to perform the calculation of the effective duration (based on I_a in length/dyn. time) or the significant duration (based on I_a as a percent) (Figure 288 (on page 466)). Clicking the *Extreme values* button, the user can view the *Extreme values* for the curve (Figure 287 (on page 465)).

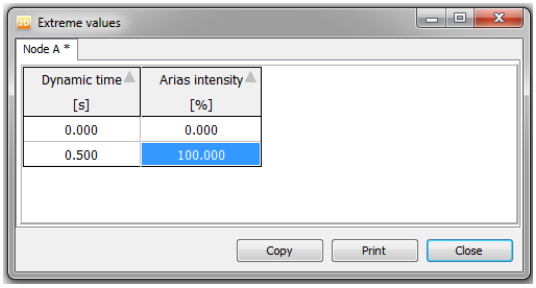


Figure 287: Extreme values for Arias intensity curves

Figure 288: Significant duration (left) and Effective duration for Arias intensity curves (right)

Transformation of curves from time to frequency domain

Once a time curve has been generated, it is possible to transform this curve into a frequency spectrum using the Fast Fourier Transform (FFT). This can be done in the *Chart* tabsheet of the **Settings** window (Figure 289 (on page 467)).

For curves created in the **Normal** and **Amplifications** tabsheets of the **Curve generation** window, you can select the option *Use frequency representation (spectrum)* and one of the three types of spectrum (*Standard frequency (Hz)*, *Angular frequency ({rad/s}*) or *Wave period*(s)).

The three optional quantities on the horizontal axis are related as follows:

$$\omega = 2\pi f = 2\frac{\pi}{T} \qquad Eq.\ [128]$$

where ω is the Angular Frequency, f is the Standard Frequency, T is the Wave Period.

Upon clicking on the *OK* button the existing time curve will be transformed into a spectrum. The original curve can be reconstructed by de-selecting the frequency representation in the **Chart** tabsheet.

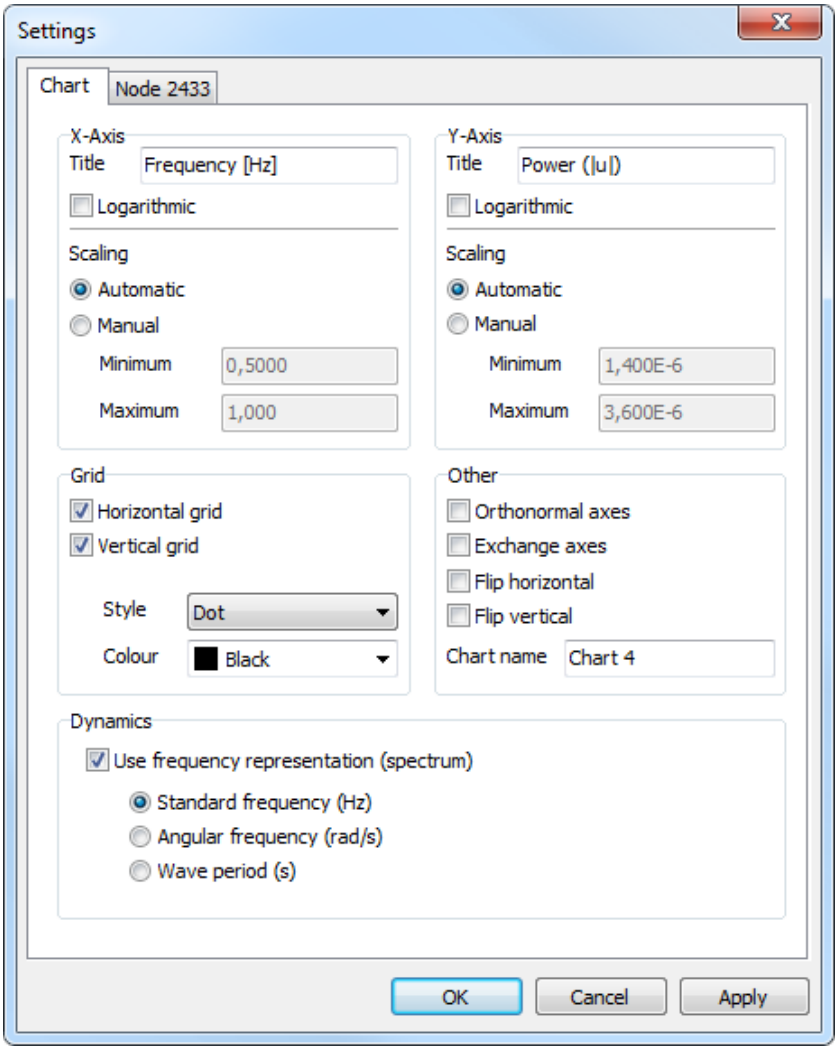


Figure 289: Fast Fourier Transform

Note:

The **Settings** window is displayed by right clicking the chart and selecting the corresponding option in the appearing menu or by selecting the option in the *Format* menu.

10.3 Formatting curves

Once a curve has been generated, a new chart window is opened in which the generated curve is presented. The quantities used to generate the curve are plotted along the *x*- and *y*-axis. By default, a legend is presented at the right hand side of the chart. For all curves in a chart, the legend contains the *Curve title*, which is automatically generated with the curve. An example of curves in Output program is given Figure 290 (on page 468).

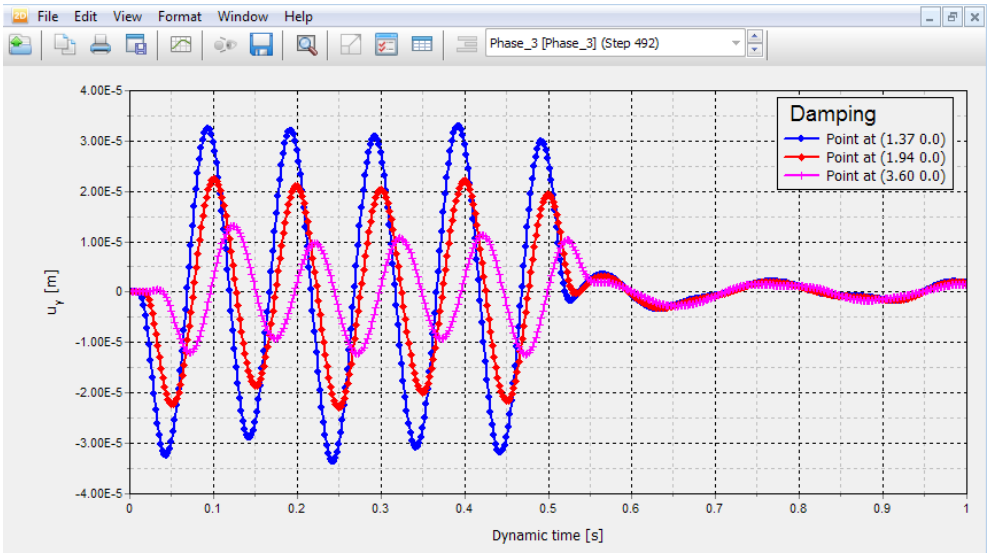


Figure 290: Curves in Output program

10.3.1 *Menus for curves*

The menus in menu bar when curves are displayed vary slightly from the ones in the Output program. A description of the menus and the options available in them is given as follows.

File menu

The *File* menu is basically the same with the one available in the Output program. For a more detailed description see File menu (on page 386).

Edit menu

Note that *Edit* menu is only available when the curves are displayed. The options available can be used to include curves in the current chart. These options are:

Copy	To export the chart to other programs using the Windows clipboard function. This feature is described in detail in Exporting output data (on page 396).
Add curve from current project	To add a new curve to the active chart from the current project.
Add curve from another project	To add a new curve to the active chart from another project.
Add curve from clipboard	To add a new curve to the active chart from clipboard.

Note:

- The added curves are redefined using the data from either the current project, another project or clipboard. It is not possible to mount a generated curve to the current chart.
- It is possible to add a curve to the active chart using the *Add curve* option in the corresponding option in the right mouse click pop-up menu.

View menu

The display of the results in the window is arranged using the options available in the *View* menu. These options are:

Reset view	To reset a zoomed view.
Table	To display the data series in a table. More information on tables is given in the Editing curve data in table (on page 469).
Legend	To toggle the display of the legend in the chart.
Legend in chart	To locate the legend in the chart.
Value indication	To toggle the display of information about the points in the curves when the mouse pointer is located on them.

Note:

- For a more detailed view of particular regions in curves, press the left mouse button at a corner of the zoom area; hold the mouse button down and move the mouse to the opposite corner of the zoom area; then release the button. The program will zoom into the selected area. The zoom option may be used repetitively.
- The zoomed view can be reset by clicking the corresponding button in the toolbar as well.

Format menu

The *Format* menu contains the *Settings* option, selecting which displays the corresponding window where the layout of the chart and curves can be modified.

Window and Help menus

These menus contain the same options as defined in Window menu (on page 394) and Help menu (on page 394).

10.3.2 *Editing curve data in table*

In contrast to the general Output program, the Curves part allows for editing of the table by the user using the options in the menu appearing as the table is right clicked.

Delete rows	To delete selected rows in the table.

Update chart	To update chart according to the modifications made in the table.
Align	To align the text in the selected part of the table.
Decimal	To display data in decimal representation.
Scientific	To display data in scientific representation.
Decimal digits	To define the number of decimal digits displayed.
View factor	To define a factor to the values in the table.
Copy	To copy the selected values in the table.
Find value	To find a value in the table.
Filter	To filter the results in the table.

Editing load-displacement curves is often needed when gravity loading is used to generate the initial stresses for a project. As an example of the procedures involved, consider the embankment project indicated in Figure 291 (on page 470).

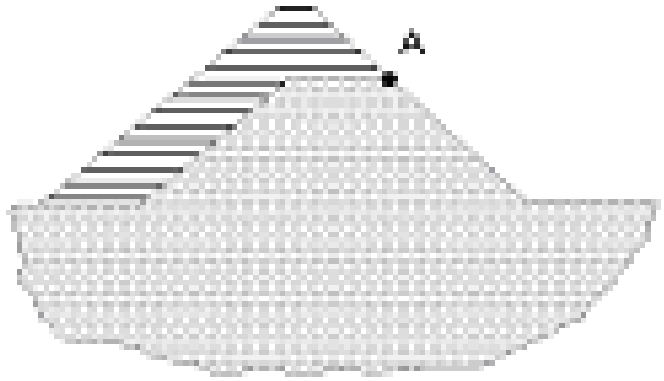


Figure 291: Raising an embankment

In this example project soil is to be added to an existing embankment to increase its height. The purpose of this example analysis is to calculate the displacement of point A as the embankment is raised. One approach to this problem is to generate a mesh for the final embankment and then deactivate the clusters corresponding to the additional soil layer by using the *Initial geometry configuration* item of the Input program.

An alternative procedure would be to generate the initial stresses for the project, i.e. the stresses for the case where the original embankment has been constructed but the new material has not yet been placed. This should be done using the gravity loading procedure. In this procedure the soil self-weight is applied by increasing *ΣMweight* from zero to 1.0 in a *Plastic calculation* using *Total multipliers* as *Loading input*.

The settlement behaviour of point A when gravity loading is applied is shown by the initial horizontal line in Figure 292 (on page 471) a). This line will, in general, consist of several plastic calculation steps, all with the same value of *ΣMarea*.

To model the behaviour of the soil structure as a whole as the additional material is placed, then the cluster of the additional material should be activated using a staged construction calculation. At the start of this staged construction calculation, all displacements should be reset to zero by the user. This removes the effect of the physically meaningless displacements that occur during gravity loading.

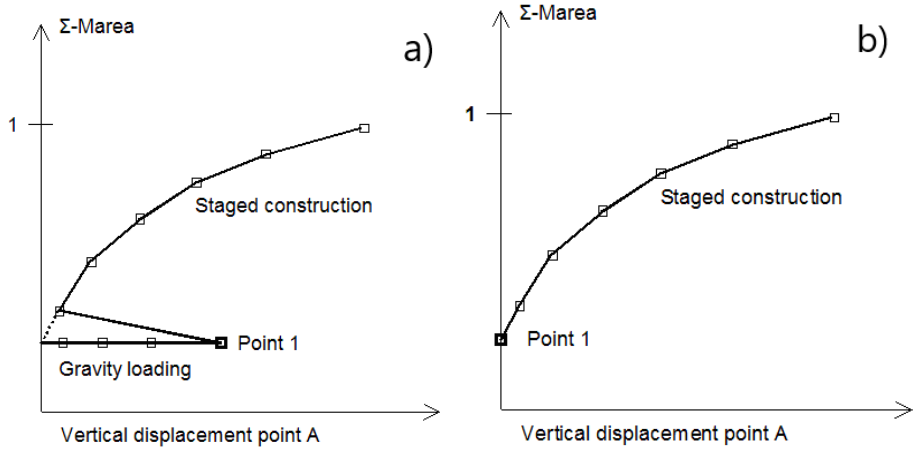


Figure 292: Load-displacement curves of the embankment project. a) Before editing; b) After editing

The load-displacement curve obtained at the end of the complete calculation for point A is shown in Figure 292 (on page 471) a). To display the settlement behaviour without the initial gravity loading response it is necessary to edit the corresponding load-displacement data. The unwanted initial portion, with the exception of point 1, should be deleted. The displacement value for point 1 should then be set to zero. The resulting curve is shown in Figure 292 (on page 471) b).

As an alternative to the above editing procedure, the gravity loading phase can be excluded from the list of calculation phases that are included in the curve (Formatting curves (on page 467)).

10.3.3 *Value indication*

If the *Value indication* option in the *View* menu is active and the mouse is moved over a data point in a curve, the hint box shows the precise value of the *x*- and *y*-quantities at that point. In addition, it shows the curve point number and the step and phase numbers corresponding with that curve point.

10.4 Formatting options

The layout and presentation of charts can be modified by clicking the *Settings* button available in the toolbar or by selecting the corresponding option in the *Format* menu. Alternatively, the *Settings* option can be selected from the *Format* menu of the right mouse button menu. As a result, the **Settings** window will appear. Distinction is made between the chart settings displayed on the first tabsheet and the curve settings displayed on a separate tabsheet for each curve. The options available in the **Chart** tabsheet can be used to customize the frame and axes of the chart (Chart settings (on page 472)). The options available in the tabsheets of the curves can be used to customize the plot (Curve settings (on page 473)).

If the correct settings are defined, the *OK* button may be pressed to activate the settings and to close the window. Alternatively, the *Apply* button may be pressed to activate the settings, keeping the **Settings** window active. The changes to the settings can be ignored by pressing the *Cancel* button.

10.4.1 *Chart settings*

The **Settings** window contains a tabsheet with options to customise the layout and presentations of the chart (see Figure 293 (on page 472)).

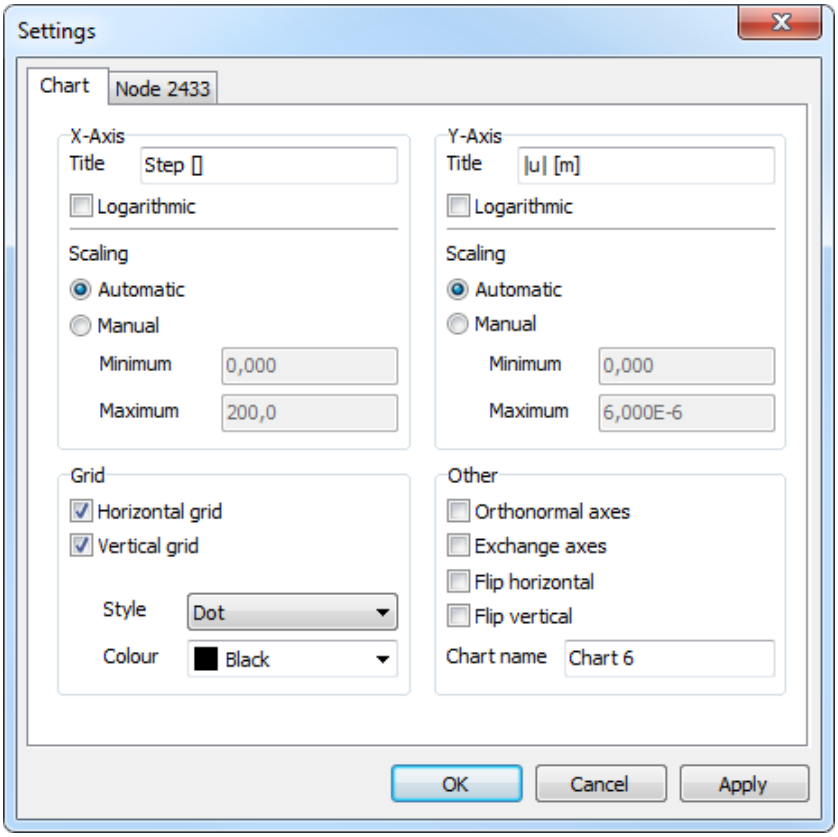


Figure 293: ***Chart settings*** *tabsheet*

Titles	By default, a title is given to the *x*-axis and the *y*-axis, based on the quantity that is selected for the curve generation. However, this title may be changed in the *Title* edit boxes of the corresponding axis group. In addition, a title may be given to the full chart, which can be entered in the *Chart name* edit box. This title should not be confused with the *Curve title* as described in above.

Scaling of x- and y-axis	By default, the range of values indicated on the *x*- and *y*-axis is scaled automatically, but the user can select the *Manual* option and enter the desired range in the *Minimum* and *Maximum* edit boxes. As a result, data outside this range will not appear in the plot. In addition, it is possible to plot the *x*- and/or *y*-axis on a logarithmic scale using the *Logarithmic* check box. The use of a logarithmic scale is only valid if the full range of values along an axis is strictly positive.
Grid	Grid lines can be added to the plot by selecting items *Horizontal grid* or *Vertical grid*. The grid lines may be customised by means of the *Style* and *Colour* options.
Orthonormal axes	The option *Orthonormal axes* can be used to ensure that the scale used for the *x*-axis and the *y*-axis is the same. This option is particularly useful when values of similar quantities are plotted on the *x*-axis and *y*-axis, for example when making diagrams of different displacement components.
Exchange axes	The option *Exchange axes* can be used to interchange the *x*-axis and the *y*-axis and their corresponding quantities. As a result of this setting, the *x*-axis will become the vertical axis and the *y*-axis will become the horizontal axis.
Flip horizontal or vertical	Selecting the option *Flip horizontal* or *Flip vertical* will respectively reverse the horizontal or the vertical axis.

10.4.2 *Curve settings*

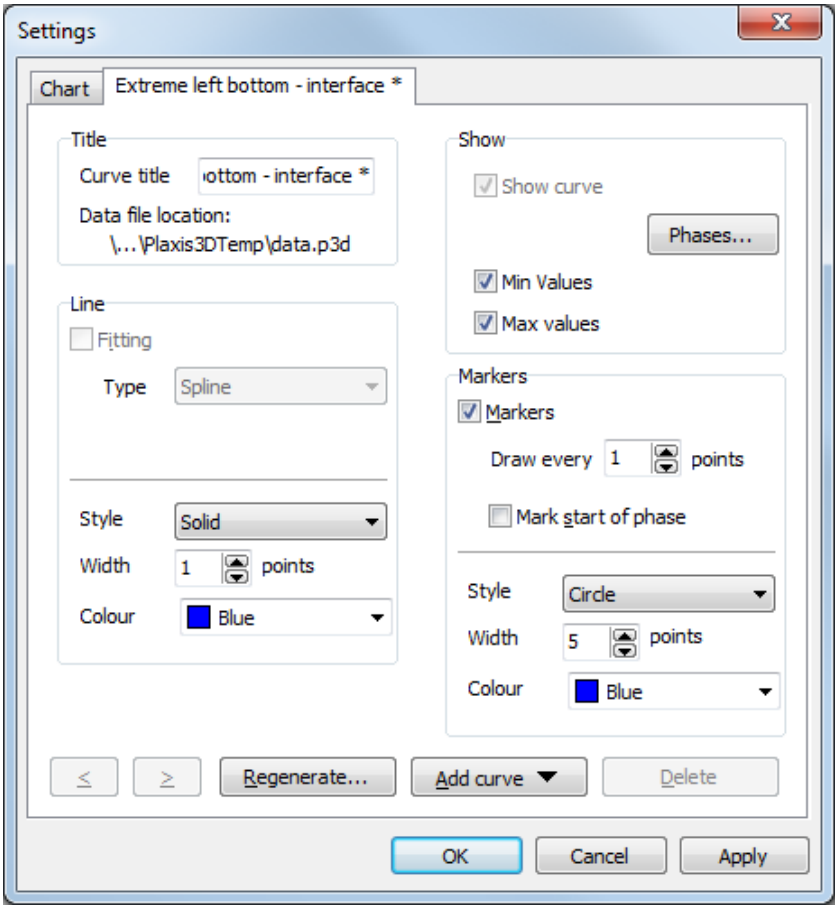


Figure 294: ***Curve settings*** *tabsheet*

The **Settings** window contains for each of the curves in the current chart a tabsheet with the same options (Figure 294 (on page 474)).

Title	A default title is given to any curve during its generation. This title may be changed in the *Curve title* edit box. When a legend is presented for the active chart in the main window, the *Curve title* appears in the legend.
Show	When multiple curves are present within one chart, it may be useful to hide temporarily one or more curves to focus attention on the others. The *Show curve* option may be deselected for this purpose. It is possible to select the check boxes for *Min values* and *Max values* to display the minimum and maximum values in the chart with an orange circle.
Phases	The *Phases* button may be used to select for which calculation phases the curve has to be generated. This option is useful when not all calculation phases should be included in the curve.

Fitting	To draw a smooth curve, the user can select the *Fitting* item. When doing so, the type of fitting can be selected from the *Type* combo box. The *Spline* fitting generally gives the most satisfactory results, but, as an alternative, a curve can be fitted to a polynomial using the least squares method.
Line and marker presentation	Various options are available to customise the appearance of the curve lines and markers. It is possible to *Mark start of phase* by clicking on the checkbox, which shows a distinction for the start of a new phase in the chart.
Arrow buttons	The arrow buttons can be used to change the order of the curves in the legend.
Regenerate	The *Regenerate* button may be used to regenerate a previously generated curve to comply with new data (Regeneration of curves (on page 475)).
Add curve	The *Add curve* button may be used to add new curves to the current chart (Multiple curves in one chart (on page 475)).
Delete	When multiple curves are present within one chart, the *Delete* button may be used to erase a curve.

10.5 Regeneration of curves

If, for any reason, a calculation process is repeated or extended with new calculation phases, it is generally desirable to update existing curves to comply with the new data. This can be done by means of the *Regenerate* facility. This facility is available in the **Settings** tabsheet (Figure 293 (on page 472)), which can be opened by selecting the *Settings* option from the *Format* menu. When clicking on the *Regenerate* button, the **Curve generation** window appears, showing the existing setting for *x*- and *y*-axis. Pressing the *OK* button is sufficient to regenerate the curve to include the new data. Another *OK* closes the *Settings* window and displays the newly generated curve.

When multiple curves are used in one chart, the *Regenerate* facility should be used for each curve individually. The *Regenerate* facility may also be used to change the quantity that is plotted on the *x*- or *y*-axis.

10.6 Multiple curves in one chart

It is often useful to compare similar curves for different points in a geometry, or even in different geometries or projects. Therefore PLAXIS 2D allows for the generation of more than one curve in the same chart. Once a single curve has been generated, the *Add curve* options in the *Edit* menu can be used to generate a new curve in the current chart. As an alternative, the *Add curve* option from the **Settings** window or from the right mouse button menu can be used. Distinction is made between a new curve from the current project, a new curve from another project or curves available on the clipboard.

The *Add curve* procedure is similar to the generation of a new curve (Generating curves (on page 453)). However, when it comes to the actual generation of the curve, the program imposes some restrictions on the selection of data to be presented on the *x*- and the *y*-axis. This is to ensure that the new data are consistent with the data of the existing curve.

When the *Add curve* option is used, the current chart is modified. In order to preserve the current chart, a copy of it can be created by selecting it first in the list and then by clicking the *Copy* button in the **Curves manager** window.

11 PLAXIS Advanced Tools

11.1 Sensitivity Analysis and Parameter variation [GSE]

The *Sensitivity Analysis and Parameter Variation* facility can be used to evaluate the influence of model parameters on calculation results. This might be useful in the case that parameters cannot be determined accurately. In such a case, a sensitivity analysis can be performed in which parameters are varied individually in order to evaluate the so-called sensitivity score against certain criteria. The sensitivity score can be used to evaluate which of the parameters have a major influence and which have a minor influence. Next, a parameter variation analysis can be performed with the parameters having a major influence in order to evaluate the range of results that can be expected when these parameters are varied all together within their specified range.

Before starting the *Sensitivity Analysis and Parameter Variation* facility, it is necessary to completely create a PLAXIS 2D model, define calculate phases, select points for curves and perform the calculations with the values as defined in the material data sets.

To start the process it is required to first configure the remote server. This can be done in both *Input* and *Output* by clicking on *Configure remote scripting server* from the drop-down menu in *Expert* menu. Next, click the *Start server* to establish a remote scripting server connection. This window can now be closed. To end the connection the button *Stop server* can be clicked.

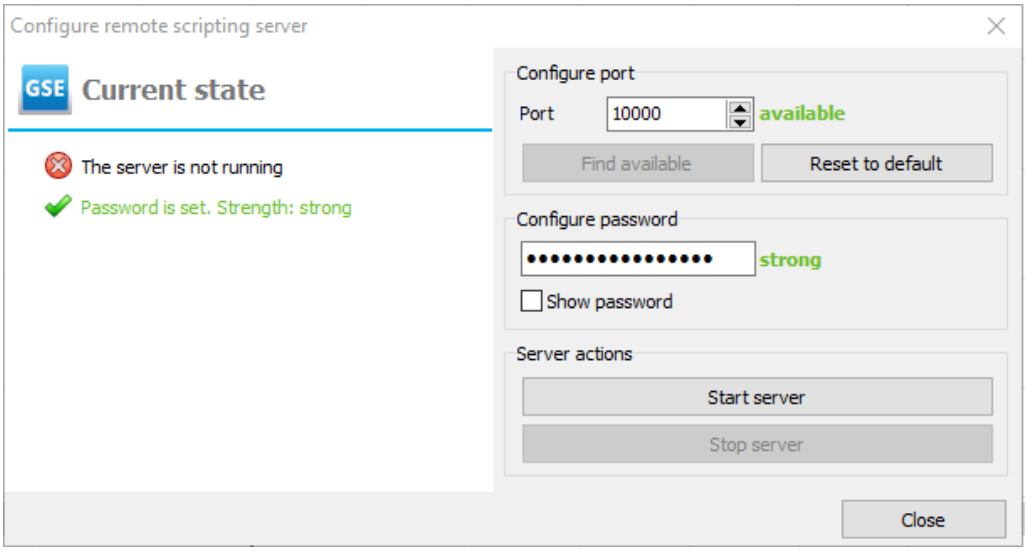


Figure 295: ***Configure remote scripting server*** *window*

To start the tool, the *Sensitivity analysis and parameter variation* is clicked from the *Expert* drop-down menu. This now automatically starts both *Output* and *Sensitivity and parameter variation* windows. This window has two blue tabs and two orange tabs.

The first blue *Settings* tabsheet, gives information regarding the input and output connection and also the option to reconnect. It is recommended to test the *Input* and *Output* connections.

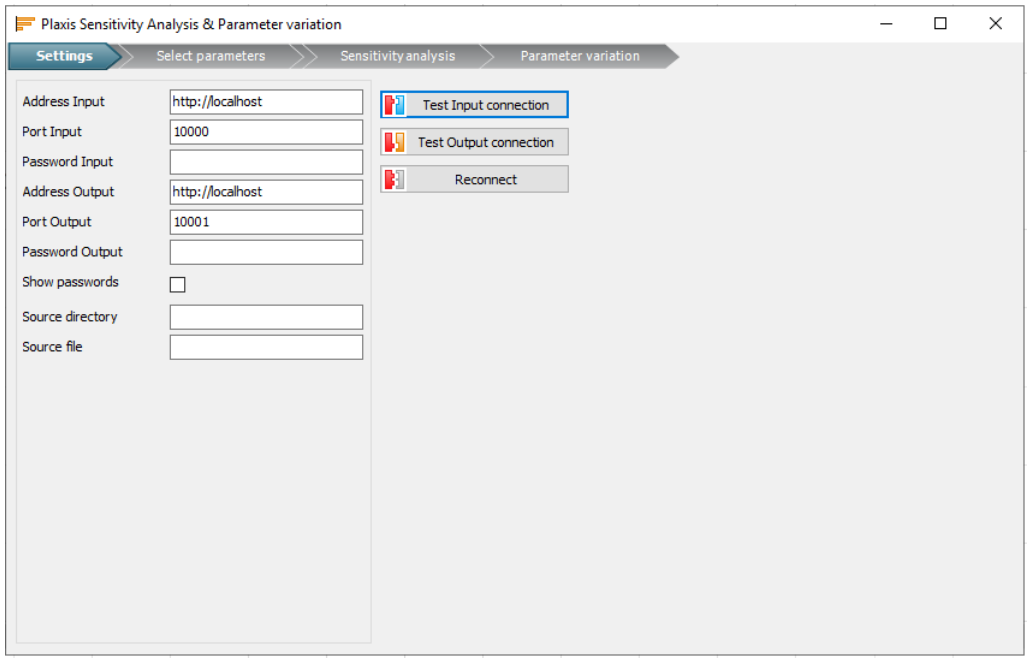


Figure 296: Settings tabsheet

11.1.1 *Select parameters*

The **Select parameters** tabsheet, gives information about all the parameters that can be changed in order to perform the sensitivity analysis. Available parameters are most model parameters of the data sets for soil and interfaces as well as parameters of the data sets of structural elements if applicable.

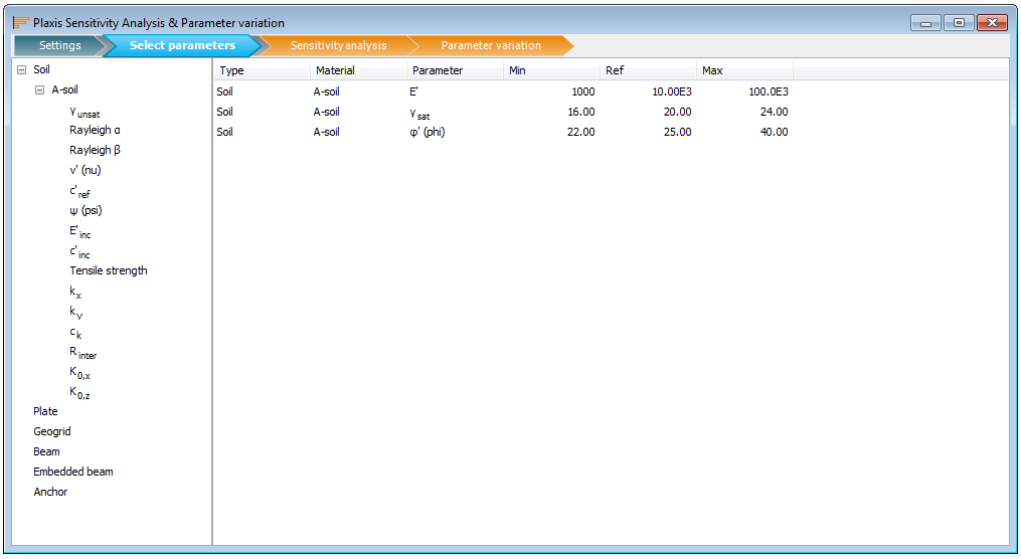


Figure 297: Select parameters tabsheet

Data sets maybe expanded by clicking on the + box in front of the data set. It is possible to add parameters to the analysis by right-clicking the parameter in the left panel and selecting *Use*. It is also possible to simply drag and drop the parameter from the left panel to the right side. In the right panel, there are six columns namely *Type*, *Material*, *Parameter*, *Min*, *Ref* and *Max*. The *Ref* value of the model parameter as specified in the material data base. The user is supposed to enter minimum and maximum values of the parameters in the *Min* and *Max* columns. The *Ref* value does not necessarily need to be in the range of *Min* and *Max* values. For every parameter specified, the *Sensitivity Analysis* is performed for the minimum and maximum values as entered by the user, while reference values are used for the other parameters. Hence, for N parameters, 2N variations are calculated in addition to the reference calculation with all reference values. All variations are listed in the lower panel of the *Sensitivity Analysis* tab sheet. They are stored as separate projects in the sub-folder 'Sensi' of the corresponding PLAXIS 2D project under the corresponding project name added by _##, where ## is the variation number corresponding to the list of variations. For every parameter specified, the *Sensitivity analysis* is performed based on the minimum and maximum values as entered by the user.

To remove a parameter from the analysis, right-click and select *Don't use*.

11.1.2 *Sensitivity analysis*

The orange calculation tabsheet *Sensitivity Analysis* allows the user to add one or more criteria on the basis of which the sensitivity score is calculated. A criterion can be, for example, the absolute displacement of a particular point in the geometry, $|u|$, or the safety factor at the end of a safety analysis, ΣMsf. It can be argued that a stiffness parameter, such as Young's modulus, has a major sensitivity in a *Serviceability Limit State (SLS)* analysis when a criterion is defined based on the displacement of a particular point. A strength parameter, such as the friction angle, may have a minor sensitivity in this case. Reversely, it can be argued that a strength parameter has a major sensitivity in an *Ultimate Limit State (ULS)* analysis when a criterion is defined based on ΣMsf, whilst a stiffness parameter has a minor sensitivity in this case.

- The first step is to add a criterion on which the sensitivity score is calculated. This can be done in the left panel by clicking the *Add criterion* button. It is possible to add multiple criteria for different phases.
- To remove any criterion, click *Remove criterion* button.

Criterion

Displacement	The calculation of the sensitivity score is based on the displacements. The calculations can be specified if it should be done with *Total displacement* $\|U\|$, displacements in x direction U_x or displacements in y direction U_y.
StressStrain	The calculation of the sensitivity score is based on stress or strain criteria. Here is it possible to specify which criterion under *Value type* such as *Cartesian effective stresses*, *Excess pore pressure* or *Total cartesian strains*.
Reached values	The calculation of sensitivity score should be based on the reached values for each phase. The different options for *Value type* are *Reached force* in x or y direction, *ΣMsf*, *ΣMStage*, *Reached total time* or *Reached maximum pore pressure*.

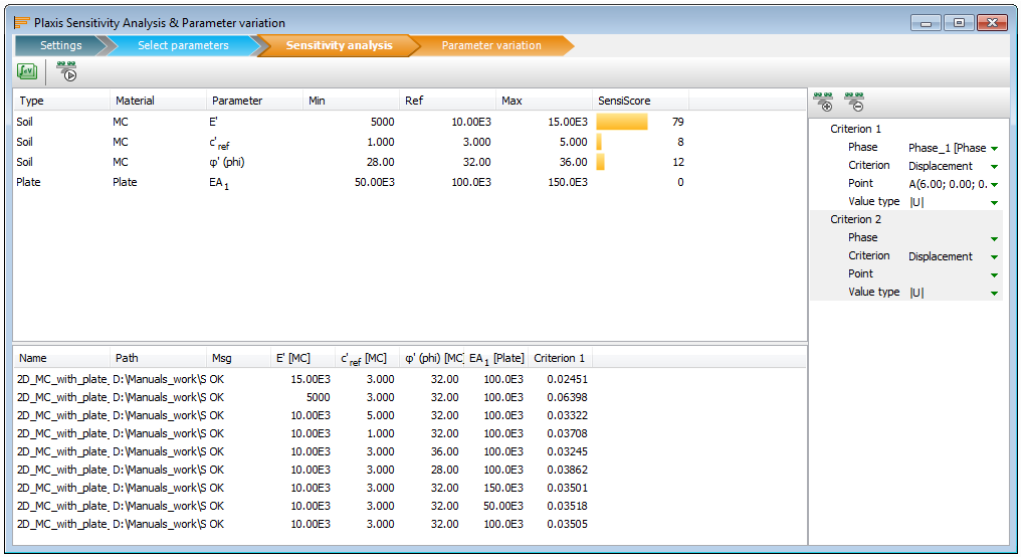


Figure 298: Sensitivity analysis tabsheet

After selecting the different criteria, the next step is to perform the sensitivity analysis by clicking the *Run analysis*. After the calculation is completed, an overview is available in the lower panel regarding all the parameters and whether each step has been successful.

Next step is to calculate the *SensiScore* for the project by clicking the *Recalculate SensiScore* button (Figure 298 (on page 480)). When multiple criteria have been defined, the sensitivity score is based on a weighted average of the scores of all criteria.

For more information, please see Scientific Manual - Theory of sensitivity analysis.

11.1.3 *Parameter variation*

The orange calculation tabsheet *Parameter Variation* allows the user to calculate the upper bound and lower bound results of all combinations of upper bound and lower bound values of the parameters considered in the *Parameter Variation* analysis. For N parameters 2^N (= 2 to the power N) variations are calculated. Hence, the number of calculations required for a *Parameter Variation* analysis is generally larger than the number of calculations required for a *Sensitivity Analysis*. Therefore, it is recommended to first perform a *Sensitivity Analysis* in order to evaluate which parameters have a major influence on the results and then perform a *Parameter Variation* analysis (with a reduced number of parameters compared to the *Sensitivity Analysis*). All variations are listed in the lower panel of the *Parameter Variation* tab sheet. They are stored as separate projects in the sub-folder 'ParVar' of the corresponding PLAXIS 2D project under the corresponding project name added by _## , where ## is the variation number corresponding to the list of variations.

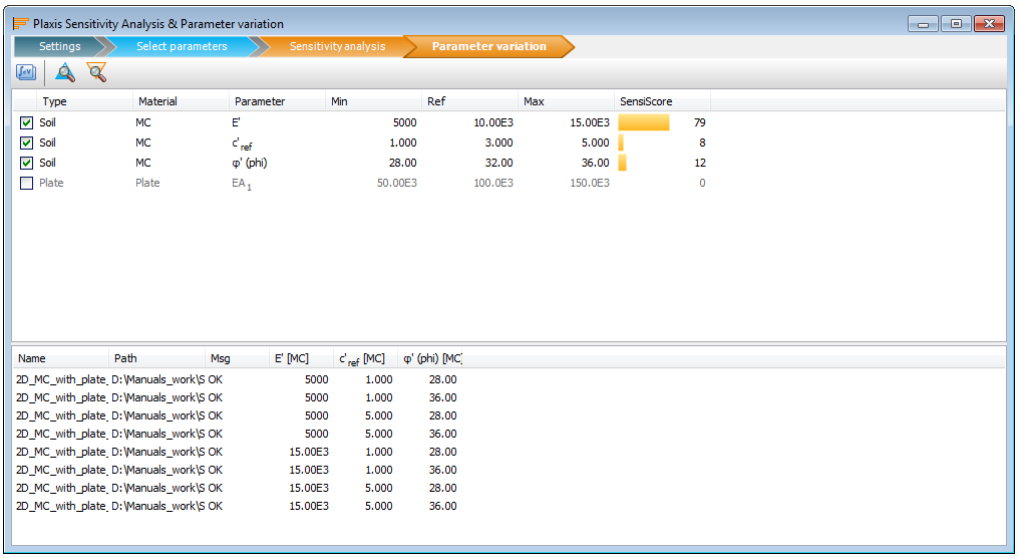


Figure 299: Parameter variation tabsheet

Click on the *Run analysis*, to start the calculation. PLAXIS 2D then starts to recalculate all the modifications. It is then possible to view the results of the calculation.

The results of a *Parameter Variation* analysis are stored in two separate projects in the sub-folder 'ParVar' of the corresponding PLAXIS 2D project: a *<ProjectName>_Max* project containing the maximum values of all numerical results and a *<ProjectName>_Min* project with the minimum values of all numerical results.

Note:

Note that the *Min* and *Max* project contain a collection of results from different parameter variations and do not form a consistent set of results for one particular set of parameters.

The *Min* and *Max* project may be used to evaluate the range of results that can be expected based on the parameter variations considered.

Note:

Note that the range of results depends on the selected *Min* and *Max* values of the parameters considered in the *Parameter Variation* analysis.

11.2 Simulation of soil lab tests [GSE]

The *SoilTest* option is a quick and convenient procedure to simulate basic soil lab tests on the basis of a single point algorithm, i.e. without the need to create a complete finite element model. This option can be used to compare the behaviour as defined by the soil model and the parameters of a soil data set with the results of laboratory test data obtained from a site investigation. It also offers the possibility to optimise model parameters such that a best fit is obtained between the model results and the lab test data. The *SoilTest* facility works for any soil model, both standard soil models as well as user-defined models.

The *SoilTest* option is available from the **Material sets** window if a soil data set is selected (see Figure 300 (on page 482)). Alternatively, the *SoilTest* option can be reached from the **Soil** dialog.

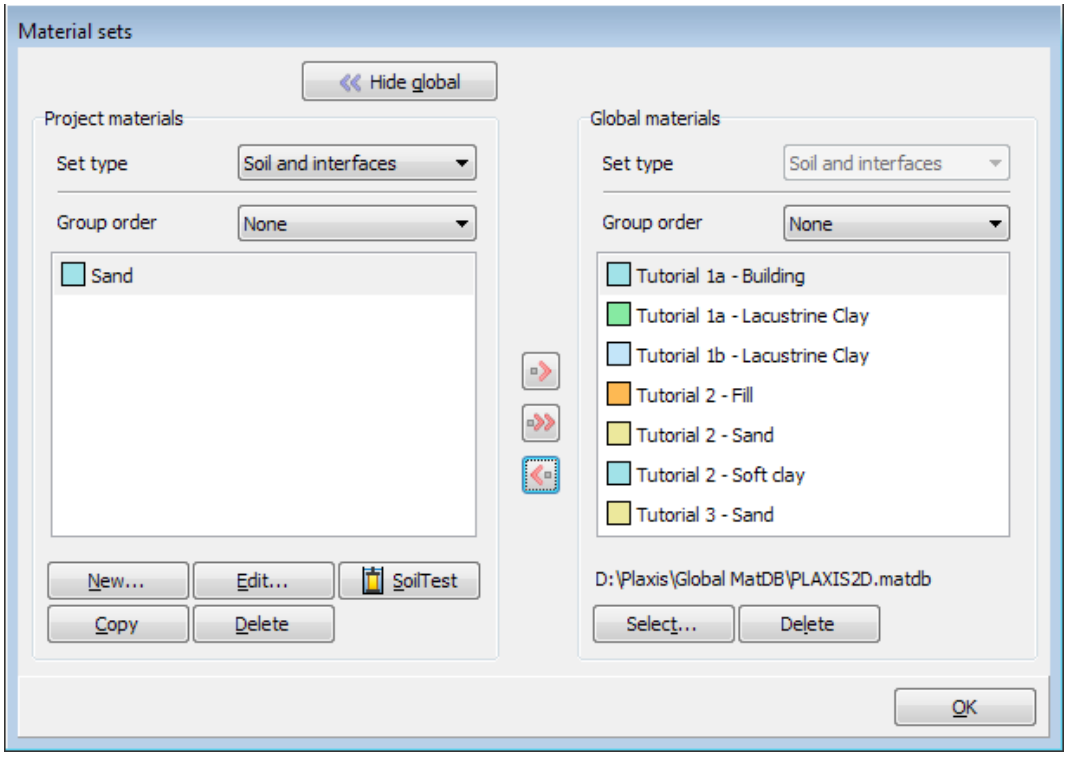


*Figure 300: **Material sets** window showing the project and the global database*

Once the *SoilTest* option has been selected, a separate window will open (Figure 301 (on page 483)). This window contains a menu, a toolbar and several smaller sections. The various items are described in more detail below.

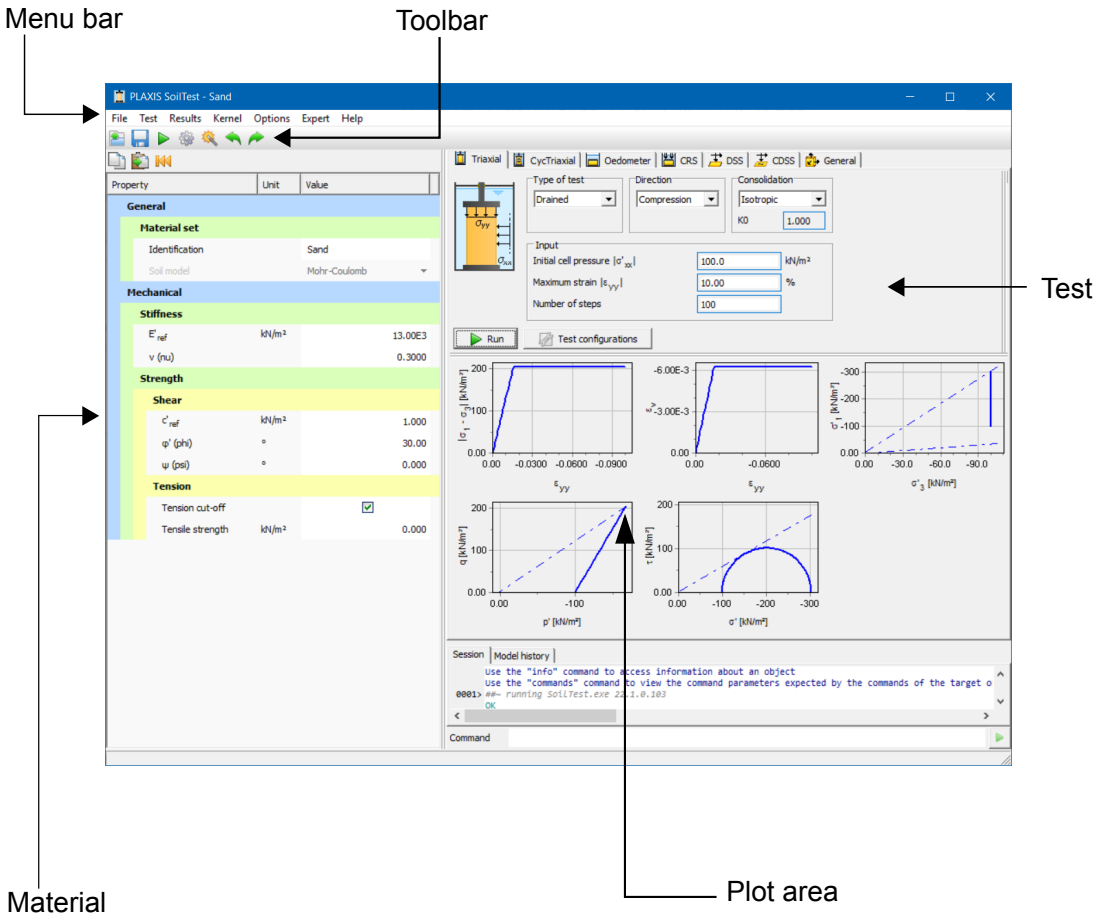


*Figure 301: **SoilTest** window showing drained triaxial test input*

1. ***Main menu***

The menus available in the main window are:

File	To open, save and close a soil test data file (*.vlt).
Test	To select the test that will be simulated. The options available are *Triaxial*, *CycTriaxial*, *Oedometer*, *CRS*, *DSS*, *PMT*,*CDSS*, and *General*.
Results	To select the configuration of diagrams to display.
Kernel	Kernel used for calculation (3D always for PLAXIS 3D and 2D for PLAXIS 2D).

2. ***Toolbar***

The toolbar allows for loading, saving and running of soil test results and opening the **PLAXIS SoilTest - Settings** window to set the configuration of the results. It also contains the parameter optimisation feature (Parameter optimisation (on page 496)). The soil test calculation kernel is a reduced version of the finite element kernel. They, however, share the same constitutive model implementations, i.e. the same stress calculations.

3. ***Material properties***

   The **Material properties** box displays the name, material model and parameters of the currently selected data set. Transferring of material parameters to and from the material database is possible. To copy the modified parameters to the material database:

   - Click the *Copy material* button in the **Material properties** box.
   - In the program open the **Material sets** window and either select the corresponding material set or click *New*.
   - In the **Soil** window click *Paste material* button. The parameters will be copied in the material database. In the same way it is also possible to copy material from material database to soil test.

4. ***Test area***

   The type of test and the testing conditions are defined in the test area. The test options available are *Triaxial*, *CycTriaxial*, *Oedometer*, *CRS*, *DSS*, *CDSS*, PMT and *General*. As one of these options is selected by clicking the corresponding tab, the testing conditions can be defined in the tabsheet. A more detailed description of the tests is given in the following sections.

5. ***Run***

   The *Run* button starts the currently selected test.

6. ***Test configurations***

   The *Test configurations* button can be used to add and manage different soil test configurations. A test configuration contains information about the test type and the values of test input parameters. To save a test configuration select the *Save* option in the menu displayed as the *Test configuration* button is clicked. The *Manage* option can be used to manage the test configurations available. When the *Manage* option is selected, the **Manage configuration** window pops up. Note that the name of the window indicates the test to which the configuration belongs (Figure 302 (on page 484)).

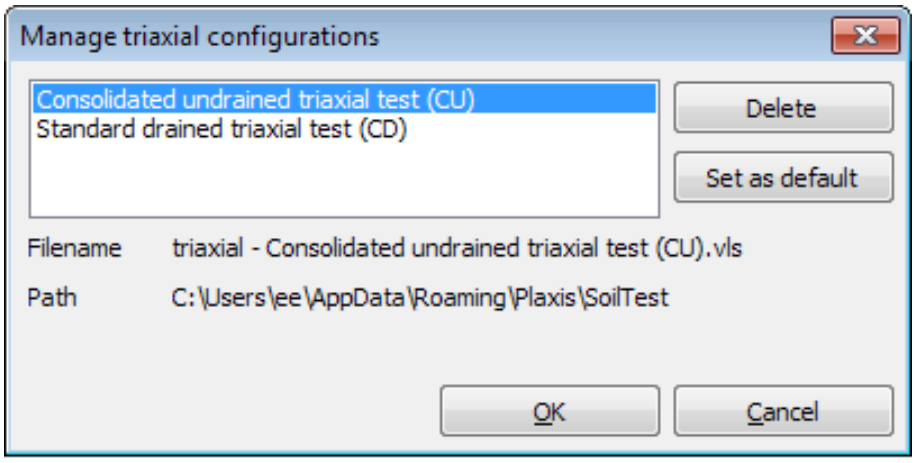


*Figure 302: **Manage configurations** window for triaxial tests*

   The name and the location of the configuration file is indicated in the *Filename* and *Path* respectively in the **Manage configurations** window.

7. ***Set as default***

The *Set as default* button saves the current input parameters as the default parameters. These will be initialised as such the next time the **SoilTest** window is opened.

8. ***Loaded tests***

   When previously saved tests of the current type have been opened from the *File* menu, the **Loaded tests** window lists all these tests within each tabsheet. The results of all loaded tests are shown together with the results of the current test. The *Delete* button can be used to remove the selected test from the list of loaded tests. It does not remove the soil test file (*.vlt) from disk.

9. ***Results***

   The results of the test are displayed in the predefined diagrams in the results area.

11.2.1 *Triaxial test*

The **Triaxial** tabsheet contains facilities to define different types of triaxial tests. Before specifying the test conditions, a selection can be made between different triaxial tests options.

Triaxial test - Options

Drained / undrained triaxial test	In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the *Drainage type* has been set to *Undrained (A)* or *Undrained (B)*, see Modelling undrained behaviour (on page 255)), irrespective of the drainage type setting in the material data set.
Triaxial compression / triaxial extension test	In the former case the axial load is increased; in the latter case the axial load is decreased.
Isotropically consolidated / K_0-consolidated test	In the latter case the K_0-value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

Triaxial test - Conditions

The following test conditions can be defined:

Initial effective stress $\lvert\sigma'_{xx}\rvert$	The absolute value of the isotropic cell pressure at which the sample is consolidated, entered in units of stress. This sets the initial stress state. In the case of a K_0-consolidated test, this value represents the initial lateral stress, σ_{xx}'; the initial vertical stress, σ_{yy}', is defined as σ_{xx}' / K_0
Maximum strain $\lvert\varepsilon_{yy}\rvert$	The absolute value of the axial strain that will be reached in the last calculation step.

Duration	Time increment (only relevant for time-dependent models; consolidation is not considered).
Number of steps	The number of steps that will be used in the calculation.
|vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	The option to activate the mobilized relative shear strength is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This option can be deactivated (isotropic stress state) or deactivated (failure state).

Note:

During a laboratory consolidated undrained triaxial test (CU test) a backpressure is applied to make sure that the sample is fully saturated. Then the sample is consolidated by using a constant cell pressure and back pressure. Note that the value assigned to the *Initial effective stress* in the SoilTest should be the effective stress at the start of the test, which is equal to the cell pressure minus the back pressure at the start of the test.

11.2.2 *Cyclic triaxial test*

The **CycTriaxial** tabsheet contains facilities to define different types of cyclic triaxial tests. Before specifying the test conditions, a selection can be made between different cyclic triaxial tests options.

Cyclic triaxial test - Options

Drained / undrained triaxial test	In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the *Drainage type* has been set to *Undrained (A)* or *Undrained (B)*, see Modelling undrained behaviour (on page 255)), irrespective of the drainage type setting in the material data set.
Isotropically consolidated / K_0-consolidated test	In the latter case the K_0-value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

Cyclic triaxial test - Conditions

The following test conditions can be defined:

|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Initial effective stress $\|\sigma'_{xx}\|$	The absolute value of the isotropic cell pressure at which the sample is consolidated, entered in units of stress. This sets the initial stress state. In the case of a K_0-consolidated test, this value represents the initial lateral stress, σ_{xx}'; the initial vertical stress, σ_{yy}', is defined as σ_{xx}' / K_0
Initial axial stress increment $\Delta\sigma'_{yy}$	After applying the initial cell pressure, it is possible to use an axial stress increment (negative for compression or positive for extension) before performing the cycles (Figure 303 (on page 488)). This is applied in one calculation phase.
Duration	Time increment (only relevant for time-dependent models; consolidation is not considered).
Number of cycles	This defines the number of load cycles for the test. Each cycle gives a total of 3 calculation phases (applying *+1 ×* amplitude, *-2 ×* amplitude, *+1 ×* amplitude).
Number of steps per quarter cycle	This defines the number of steps per quarter cycle. The total number of steps applied in the whole calculation process is: *(1 + 4 ×* number of cycles) *×* number of steps per quarter cycle.
Test control	This option allows the user to define whether the test is stress-controlled or strain-controlled. In the case of strain-control (which is the default), the user needs to specify a strain amplitude, whereas in the case of stress- control, the input of a stress amplitude is required.
Axial strain $\Delta\varepsilon_{yy}$ / Axial stress $\Delta\sigma_{yy}$ amplitude	The user needs to specify a value for the axial strain or axial stress amplitude (Figure 303 (on page 488)), depending on the test control.

Note:

- During a consolidated undrained triaxial test (CU test) in a laboratory, a backpressure is applied to make sure that the sample is fully saturated. Then the sample is consolidated by using a constant cell pressure and back pressure. Note that the value assigned to the *Initial effective stress* in the SoilTest should be the effective stress at the start of the test, which is equal to the cell pressure minus the back pressure at the start of the test.
- Unlike the triaxial test, the option for the load direction (compression or extension) is not available since the stress/strain increment is applied in both directions.

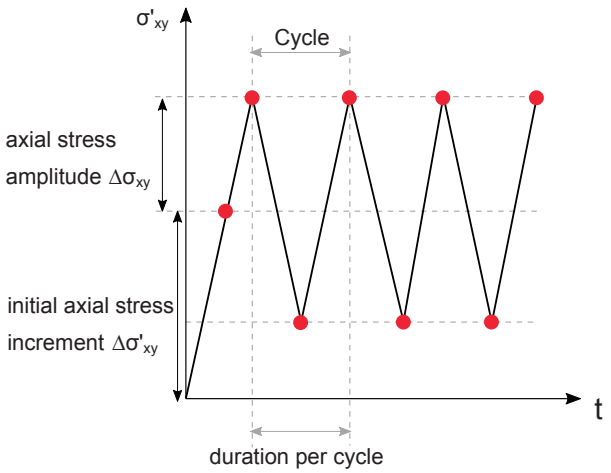


Figure 303: Cyclic triaxial test conditions

11.2.3 *Oedometer test*

The **Oedometer** tabsheet contains facilities to define a one-dimensional compression (oedometer) test. The following settings can be defined:

|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Phases	Lists the different phases of the oedometer test. Each phase is defined by a *Duration* (in units of time), a vertical *Stress increment* (in units of stress) and a *Number of steps*. The initial state is always assumed to be stress free. The given stress increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative stress increment implies additional compression, whereas a positive stress increment implies unloading or tension. If a period of constant load is desired, enter the desired duration with a zero stress increment.
Add	Adds a new phase to the end of the **Phases** list.
Insert	Inserts a new phase before the currently selected phase.
Remove	Removes the currently selected phase from the **Phases** list.

11.2.4 *CRS*

The **CRS** tabsheet contains facilities to define a constant rate-of-strain compression test. The following settings can be defined:

|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Phases	Lists the different phases of the CRS test. Each phase is defined by a *Duration* (in units of time), a vertical *Strain increment* (in %) and a *Number of steps*. The initial state is always assumed to be stress free. The given strain increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative strain increment implies additional compression, whereas a positive strain increment implies unloading or tension. If a period of zero strain is desired, enter the desired duration with a zero strain increment.
Add	Adds a new phase to the end of the **Phases** list.
Insert	Inserts a new phase before the currently selected phase.
Remove	Removes the currently selected phase from the **Phases** list.

11.2.5 *DSS*

The **DSS** tabsheet contains facilities to define a direct simple-shear test. Before specifying the test conditions, a selection can be made between different test options.

DSS - Options

Drained / undrained triaxial test	In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the *Drainage type* has been set to *Undrained (A)* or *Undrained (B)*, see Modelling undrained behaviour (on page 255)), irrespective of the drainage type setting in the material data set.
Isotropically consolidated / K_0-consolidated test	In the latter case the K_0-value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

DSS - Conditions

The following settings can be defined:

|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Initial stress $\|\sigma_{yy}\|$	The absolute value of the initial vertical stress at which the sample is consolidated, entered in units of stress. In the case of an isotropically consolidated test, the initial lateral stress is equal to the initial vertical stress. In the case of a K_0-consolidated test, the initial lateral stress is equal to $K_0\sigma_{yy}$.
Duration	Time increment (only relevant for time-dependent models; consolidation is not considered).
Number of steps	The number of steps that will be used in the calculation.
Maximum shear strain $\|\gamma_{xy}\|$	The maximum value of shear strain (entered in reached in the last calculation step.

11.2.6 *CDSS*

The **CDSS** tabsheet contains facilities to define a cyclic direct simple-shear test. Before specifying the test conditions, a selection can be made between different test options.

CDSS - Options

Drained / undrained triaxial test	In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the *Drainage type* has been set to *Undrained (A)* or *Undrained (B)*, see Modelling undrained behaviour (on page 255)), irrespective of the drainage type setting in the material data set.
Isotropically consolidated / K_0-consolidated test	In the latter case the K_0-value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

CDSS - Conditions

The following settings can be defined:

|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Initial stress $\|\sigma_{yy}\|$	The absolute value of the initial vertical stress at which the sample is consolidated, entered in units of stress. In the case of an isotropically consolidated test, the initial lateral stress is equal to the initial vertical stress. In the case of a K_0-consolidated test, the initial lateral stress is equal to $K_0\sigma_{yy}$.
Initial static shear σ_{xy}	The initial static shear defines an initial shear stress component, leading to an initial principal stress rotation, which will form an offset to the shear stress increments that result from cyclic loading. The initial static shear is applied as a first calculation phase in the test. Irrespective of the test option (drained or undrained), this first phase is always applied under drained conditions.
Duration	Time increment Δt as indicated in Figure 304 (on page 492) (only relevant for time-dependent models; consolidation is not considered).
Number of cycles	This defines the number of load cycles for the test. Each cycle gives a total of 3 calculation phases (applying *+1 ×* amplitude, *-2 ×* amplitude, *+1 ×* amplitude, Figure 304 (on page 492)).
Number of steps per quarter cycle	This defines the number of steps per quarter cycle. The total number of steps applied in the whole calculation process is: *(1 + 4 ×* number of cycles*)* × number of steps per quarter cycle.
Duration per cycle	The duration per cycle expressed in days.
Test control	This option allows the user to define whether the test is stress-controlled or strain-controlled. In the case of strain-control (which is the default), the user needs to specify a strain amplitude, whereas in the case of stress- control, the input of a stress amplitude is required.
Shear strain $\Delta\gamma_{xy}$ / Shear stress $\Delta\sigma_{xy}$ amplitude	The user needs to specify a value for the shear strain or shear stress amplitude, depending on the test control. In order to apply a particular cyclic stress ratio, $CSR = \Delta\sigma_{xy} / \sigma_{yy}$, the user needs to define the appropriate *Initial stress* σ_{yy} , set the *Test control* to *Stress*, and input the *Shear stress* amplitude as $\Delta\sigma_{xy} = CSR\ \sigma_{yy}$.

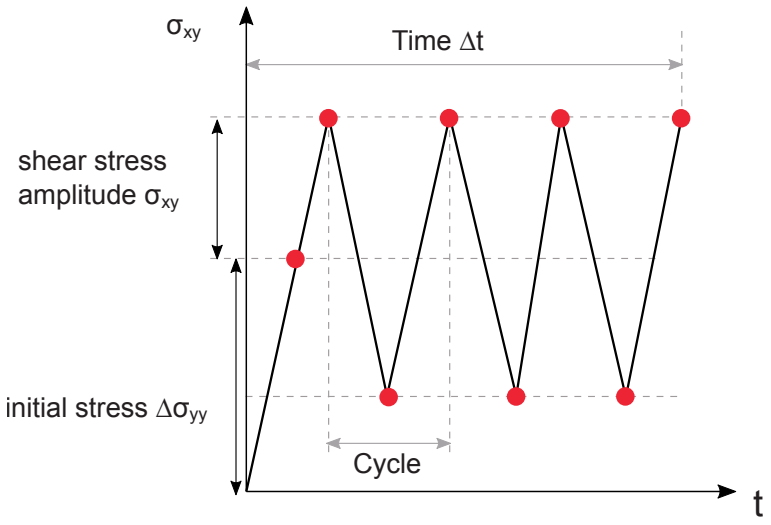


Figure 304: Cyclic DSS test conditions

Note:

Note that cycle boundaries and phase boundaries do not match except when $t = 0$ and $t = \Delta t$

11.2.7 *PMT*

The **PMT** tabsheet contains facilities to define a *Pressuremeter test*. Rather than simulating a 'real' pressuremeter test using a full finite element model, the *SoilTest* kernel performs a one-dimensional (1D) cylindrical cavity expansion simulation, and the results are transformed as if they were obtained from a finite element *Pressuremeter test* simulation. The calculation is based on an updated mesh (large deformations) analysis.

Note:

- The *Pressuremeter test* is only available for the Hardening Soil model in drained conditions.
- The test may be used to simulate the pressuremeter response considering volumetric strain up to 40%.

Before specifying the test conditions, a selection can be made between different test options.

PMT - Options

Isotropic initial stress	In this case, isotropic initial stress conditions are assumed.
K_0 initial stress	In this case, the K_0-value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

PMT - Options

The following settings can be defined:

Initial stress $\|\sigma_{yy}\|$	This defines the initial radius of the pressuremeter device.
Number of steps	This defines the number of steps for the test.
Test control	This option allows the user to define whether the test is stress-controlled or strain-controlled. In the case of strain-control (which is the default), the user needs to specify a strain amplitude, whereas in the case of stress- control, the input of a stress amplitude is required.
Maximum radial strain $\|\varepsilon_{xx}\|$	The maximum value of radial strain (entered in %) that will be reached in the last calculation step.
|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path (see the Material Models Manual - *Preliminaries on material modelling - The initial preconsolidation stress in advanced models*). This option is only available for the advanced soil models.
Apply Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Duration	Time increment (only relevant for time-dependent models; consolidation is not considered).

PMT - Transformation formulae

The cylindrical cavity expansion is simulated by means of an axisymmetric 1D finite element modelling approach. The obtained results are post-processed in the *SoilTest* calculation kernel and transformed to *Pressuremeter test* results. The adopted approach reads:

- The 1D cylindrical cavity expansion calculation is performed.
- The cavity secant stiffness is calculated as:

$$K_{\sec,cav} = (P_{cav} - P_{cav,0}) / (\Delta V / V_0) \qquad \textit{Eq. [129]}$$

where

$p_{cav,0}$ = Initial lateral stress, equal to $K_0 \cdot \sigma'_{yy}$.
p_{cav} = Cavity pressure at each calculation step and $\Delta V/V_0$ is the volumetric strain.

- The PMT pressure is calculated as:

$$P_{PMT} = P_{cav,0} + (P_{cav} - P_{cav,0})\left[A(\Delta V / V0) + B\right] \qquad \textit{Eq. [130]}$$

- The coefficients A and B are derived as follows:

$$A = 0.1575537(Z^{0.4905724}) \qquad \textit{Eq. [131]}$$

$$B = 0.0008217Z + 0.9702063 \qquad \textit{where} \quad 1.0 \le B \le 1.25$$

- The parameter Z is defined as:

$$Z = E_{oed} \cdot m / \left(p_{cav} + c' / \tan(\phi)\right) \qquad \textit{Eq. [132]}$$

where

E_{oed} = Oedometer Young's modulus at a stress level equal to p_{cav}, and m, c', φ are model parameters of the Hardening Soil model.

PMT - Results

The *Pressuremeter test* results are presented in two graphs:

- The first graph shows the PMT pressure p_{PMT} as a function of the radial strain $\varepsilon_{xx} = u_x / R_0$.
- The second graph illustrates the PMT pressure p_{PMT} as a function of the volumetric strain $\Delta V/V_0 = (R^2 - R_0^2) / R_0^2$, where $\Delta V = h\ t\ \pi\ (R^2-R_0^2)$, $V_0 = h\ \pi\ R_0^2$, and R is the increased radius of the pressuremeter device when the pressure is applied.

11.2.8 *General*

The **General** tabsheet contains facilities to define arbitrary stress and strain conditions. The following settings can be defined:

Type of test	The type of the test, whether *Drained* or *Undrained* can be specified.
|Vertical precons. stress|	The vertical preconsolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. From the vertical preconsolidation stress the program calculates the isotropic preconsolidation stress based on the K_0^{nc} loading path of the Material Models Manual - *Preliminaries on material modelling - The initial pre-consolidation stress in advanced models*. This option is only available for the advanced soil models.
Mobilized relative shear strength	This option is only available for the Hardening Soil model and Hardening Soil model with small-strain stiffness to set the initial shear hardening contour. This value must be between 0 (isotropic stress state) and 1 (failure state).
Phases	Lists the initial stress conditions and the stress/strain conditions in the subsequent phases of the test. In the initial phase it should be indicated for each direction whether a stress increment or a strain increment is defined for that direction (applies to all phases). Each phase is defined by a *Duration* (in units of time) and a *Number of steps*, followed by the applied stress or strain increments. The given stress or strain increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative stress or strain increment implies additional compression, whereas a positive stress or strain increment implies unloading or tension.
Add	Adds a new phase to the end of the **Phases** list.
Insert	Inserts a new phase before the currently selected phase.
Remove	Removes the currently selected phase from the **Phases** list.

11.2.9 *Results*

The **Results** window shows several predefined typical diagrams to display the results of the current test in terms of stress and strain quantities. Results for a PMT test are different than for other tests (see PMT - Results). Double-clicking one of the graphs opens the selected diagram in a larger window (Figure 305 (on page 495)). This window shows the selected diagram, the table of the data points that are used to plot this diagram as well as the tangent and the secant values of the plot. Note that, the point to be taken into consideration for the calculation of the tangent and the secant values can be determined by clicking on the plot. The diagram or the data can be copied to the clipboard by selecting the corresponding option in the drop-down menu displayed when the *Copy* button is clicked.

Note:

- The failure line is indicated by a dashed line in the plot.
- In plots where deviatoric stress q is considered, the failure line is always shown for the compression point.

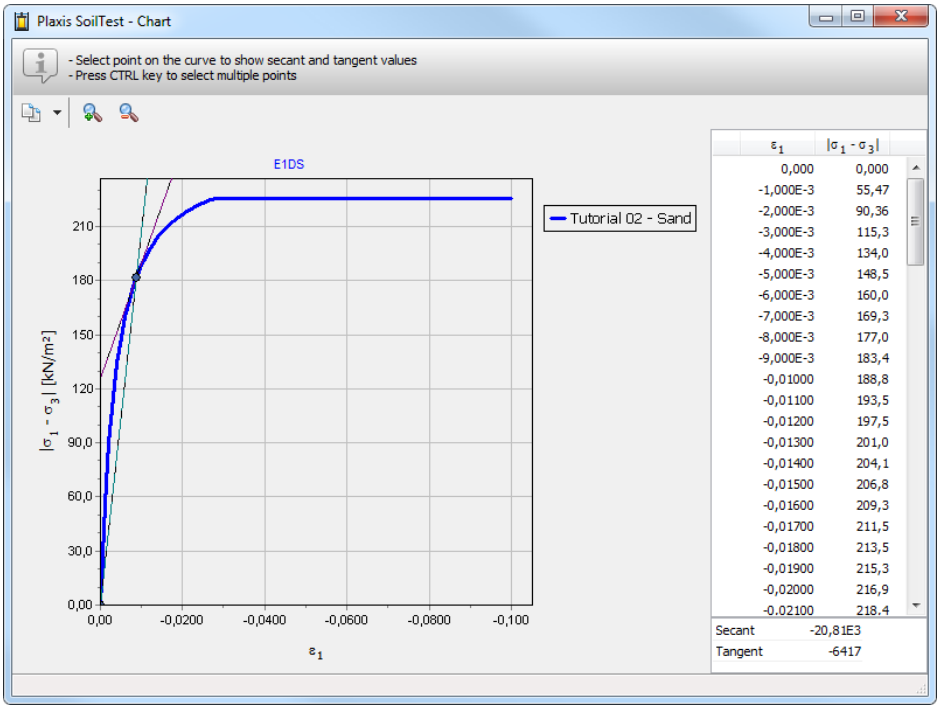


Figure 305: Results diagram

The diagram can be zoomed in or out using the mouse by first clicking and holding the left mouse button in the diagram area and then moving the mouse to a second location and releasing the mouse button. Moving the mouse from the left upper corner to the right lower corner zooms the diagram to the selected area, whereas moving the mouse from the right lower corner to the left upper corner resets the view. The zoom action can also be undone using the *Zoom out* option on the toolbar. The wheel button of the mouse can be used for panning: click and hold the mouse wheel down and move the diagram to the desired position.

The secant and the tangent are useful for the back-calculation of stiffness parameters from stress-strain diagrams. The corresponding secant and tangent values are indicated below the table. The secant and the tangent can be computed interactively on the chart:

- By clicking one point, a line is drawn from the origin of the curve to the point.
- By pressing *Ctrl* and clicking a second point: a line is drawn between the first and second point. The secant determines the inclination of the line. The tangent is determined with respect to the second point.
  - By keeping *Ctrl* pressed, the user clicks the third point: the first point is deselected and a new line is drawn between the second and the third point.
  - By pressing *Shift* and clicking the third point: the second point is deselected and a new line is drawn between the first and the third point.
- By clicking any point in the proximity of the curve without pressing *Ctrl*, any selected point is deselected and a new line is drawn according to the current behaviour (from the origin to the point).

11.2.10 *Parameter optimisation*

The soil test facility can be used to optimise model parameters such that a best fit is obtained between the model results and the results of real soil lab tests. This option can be selected from the toolbar.

Click the *Parameter optimisation* button in the toolbar. The **Parameter optimisation** window will appear, showing different colour tabs according to the various steps to follow in the parameter optimisation process (Figure 306 (on page 496)). The first tab (*Select parameters*) is active.

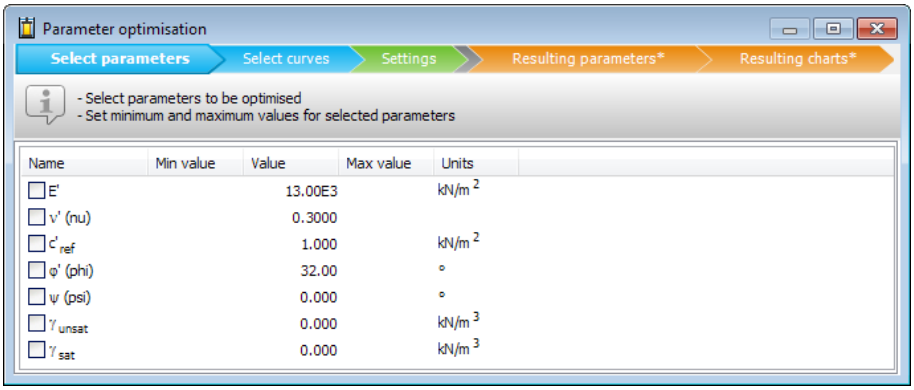


Figure 306: ***Parameter optimisation*** *window*

Note:

Note that it is possible to optimise parameters for both native and User-defined Soil Model.

Select parameters

The *Select parameters* tab shows the parameters of the selected material data set that could participate in the optimisation process. Click on the square in front of the parameter(s) that need(s) to be optimised (Figure 307 (on page 497)). The more parameters are selected, the more time the optimisation process will take. For the selected parameters, minimum and maximum values need to be specified. The optimisation algorithm will search for optimum values within this range. If the optimised value turns out to be equal to the minimum or maximum value, it might be that the best value lies outside the specified range.

Note that parameters may influence only specific parts of a test. For example, consider an oedometer test with a loading and unloading part, modelled with the Hardening Soil model. The loading part will be dominated by the parameter E_{oed}^{ref}, whereas the unloading part will be dominated by the parameter E_{ur}^{ref}. In order to obtain a best

fit, the optimisation may be performed in separate runs; one to optimise E_{oed}^{ref} based on the loading part and one to optimise E_{ur}^{ref} based on the unloading part of the test.

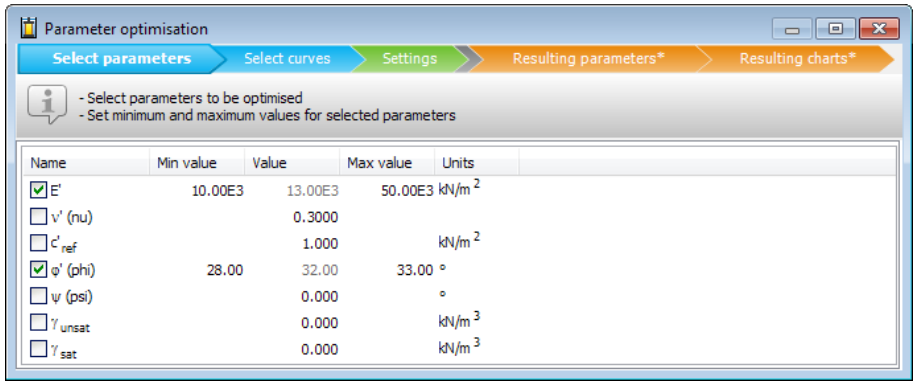


*Figure 307: Selection of the parameters in the **Select parameters** tabsheet*

Select curves

The *Select curves* tab enables selection and uploading of real soil lab test data and corresponding test conditions. Alternatively, synthetic test data may be used in the form of other PLAXIS 2D soil test results. In this way it is possible to optimise, for example, parameters of the Mohr-Coulomb model against simulated tests using the Hardening Soil model.

Initially, the window shows a tree with the five standard test types (*Triaxial*, *CycTriaxial*, *Oedometer*, *CRS*, *DSS*, *CDSS*, and *General*). For each test type, different test conditions can be defined, which can be taken into account in the optimisation process. By default, the *Current model test* is available as test conditions for each test type. The *Current model test* contains the test conditions as previously defined for that test (Figure 308 (on page 497)).

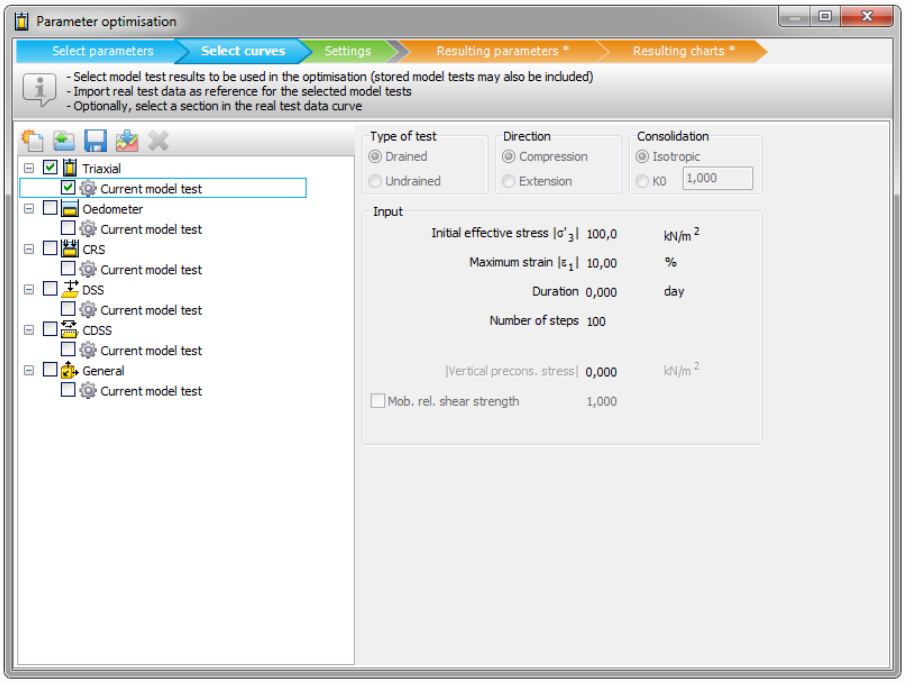


*Figure 308: Selection of the test curves in the **Select curves** tabsheet*

- New test conditions can be defined by selecting the *New test configuration* option from the tool bar. This will introduce a new test under the selected test, for which the test conditions can be defined in the right-hand panel (Figure 309 (on page 498)).
- All test data can be saved, by clicking on *Save to file* button. The test data will be saved to an XML file with a default file extension of **.vlo.xml*.
- In both cases (*Current model test* and a new test) corresponding test data need to be selected and uploaded using the *Import curve* option. Another possibility is to upload test conditions together with the test data in case it is stored in the format of a PLAXIS 2Dsoil test project (<test>.vlt).

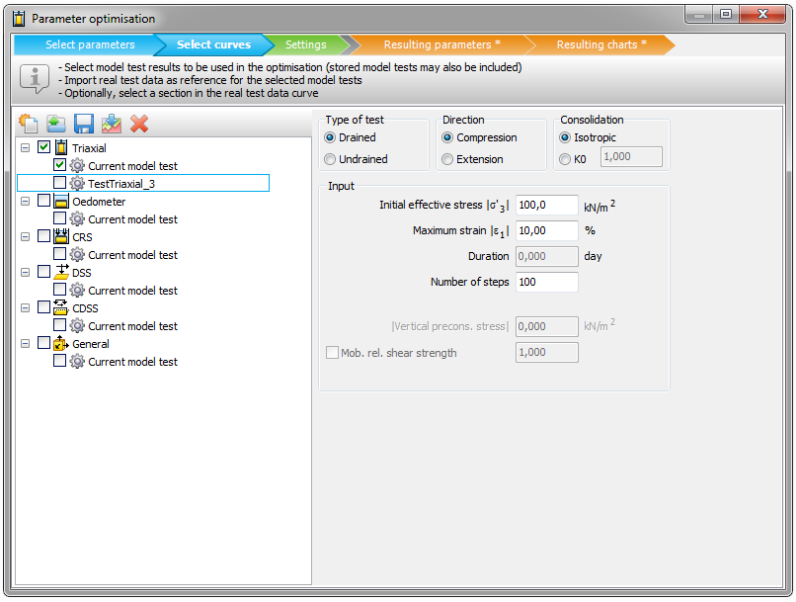


*Figure 309: Custom test definition in the **Select curves** tabsheet*

Hence, there are different ways to define test conditions and to select the external test data. The possibilities are summarized below:

- If the test data corresponds to one of the *Current model test* conditions, the corresponding line should be selected in the tree and the *Import curve* option should be used to upload the test data (Figure 310 (on page 499)). The test data are assumed to be stored in a text file (<data>.txt) and should contain two columns, separated by a *Space*, *Tab*, *Comma*, *Colon (:)*, *Semicolon (;)* or arbitrary character. The separator is to be indicated at the top of the **Import test data** window. The meaning of the values in each column has to be selected from the drop down list below the column. Here, a selection can be made amongst various stress and strain quantities. Moreover, the basic units of the test data quantities need to be selected from the drop down lists in the *Units* group. By pressing *OK* the data is read and visualised in a diagram, and the curve is listed in the tree under the *Current model* (test) conditions.

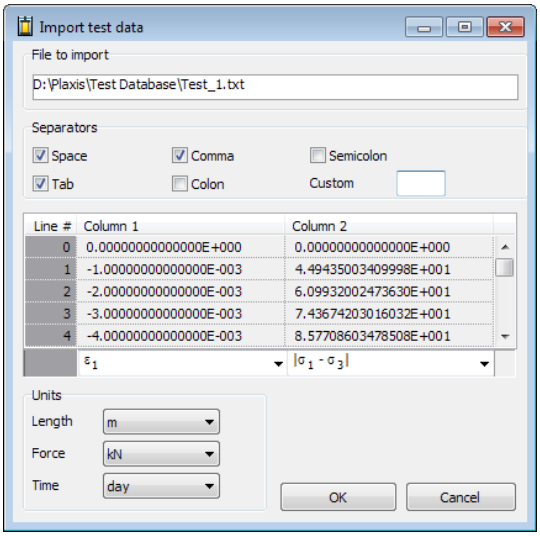


*Figure 310: **Import test data** window*

- If the test data corresponds to other than one of the current model test conditions, first new test conditions need to be defined. Select the appropriate test type and click the *New test configuration* button. The test conditions of the data to be uploaded can be defined in the right-hand panel. Subsequently, the *Import curve* option should be used to upload the test data. The test data are assumed to be stored in a text file (<data>.txt) and should contain two columns (see explanation before). The meaning of the values in each column has to be selected from the drop down list below the column. Moreover, the basic units of the test quantities need to be selected from the drop down lists in the *Units* group. By pressing *OK* the data is read and a visualised in a diagram, and the curve is listed in the tree under the *Custom* (test) conditions (Figure 311 (on page 499)).

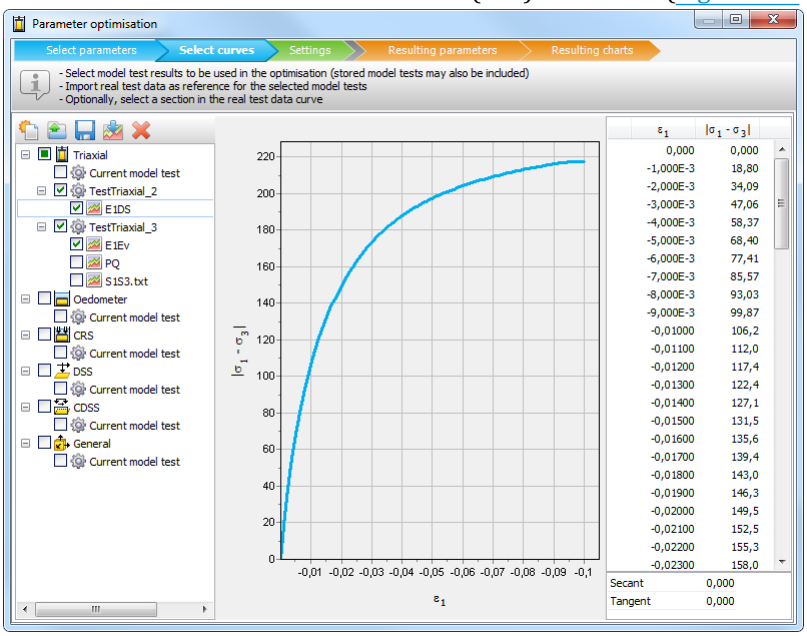


Figure 311: Display of the imported test curves

- If the test data together with the test conditions are stored in the format of a PLAXIS soil test project (<test>.vlt), the *Open file* option should be used. After selection of a valid PLAXIS soil test project, the test conditions are listed under the corresponding test type in the tree, and the available test data curves are listed under the test conditions (Figure 312 (on page 500)). This option should typically be used to fit

current model parameters to synthetic data previously produced in the PLAXIS soil test facility and stored in <test>.vlt format.

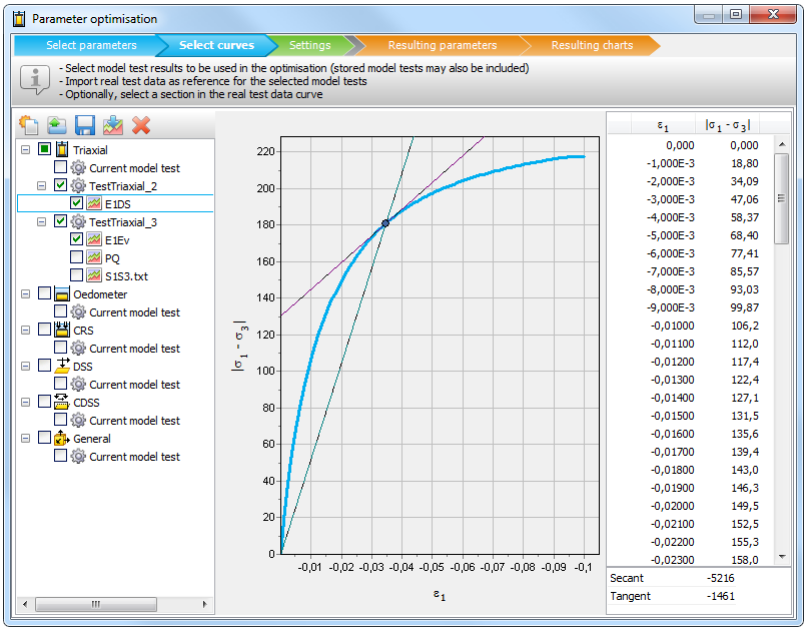


Figure 312: Importing data from SoilTest

Note:

- When a line representing test conditions is selected in the tree, the corresponding test conditions are shown on the panel at the right-hand side.
- When a line representing test data is selected in the tree, the corresponding curve is visualised in the diagram, and a table of corresponding data points is shown at the right-hand side of the diagram.
- A sub-set of test data to be used in the optimisation process can be selected in the table at the right-hand side by clicking on the corresponding cells, using the standard multi-select convention (using *Shift* for ranges and *Ctrl* for individual values). The selected values are indicated as 'thick' lines in the curve whereas non-selected values are indicated as 'thin' lines.
- Always select *all data* corresponding to a *complete phase* of the test (for example, if it is the user's intention to optimise parameters based on the unloading phase in an oedometer test that involves both loading and unloading, select the *full range* of unloading test data).
- A line in the tree (either test conditions or test data) can be removed by selecting that line and clicking the red cross in the toolbar.

All test data to be used in the optimisation process need to be selected in the tree by clicking the square in front of the corresponding line (if not already selected). The corresponding test conditions are automatically selected.

Multiple phases

In the case of an *Oedometer*, *CRS* or *General* test, the *SoilTest* facility allows for multiple phases. However, the parameter optimisation facility can only deal with one complete phase at a time. Therefore, after importing the test data, the desired calculation phase needs to be selected from the drop down list above the test data curve, together with the corresponding part of the test data in the column at the right-hand side. Note that the *full range of data* corresponding to this test phase is selected. In this way it is possible, for example, to optimise a

primary loading stiffness against the first (loading) phase in an oedometer test and the unloading stiffness against the second (unloading) phase. Note that the test data needs to be selected for each phase.

Settings

The *Settings* tab enables the accuracy selection of the optimisation process (Figure 313 (on page 501)). Three levels of search intensity are available: *Coarse and quick*, *Moderate*, *Thorough*. In addition, the relative tolerance of the search algorithm can be selected. The default value is 1E-3. Note that a more rigorous optimisation may give more accurate results, but also requires more calculation time. The calculation time also depends on the number of parameters to be optimised, as selected in the first tab. The optimisation process is based on a 'particle swarm' algorithm. This algorithm can overcome local minima in the optimisation process.

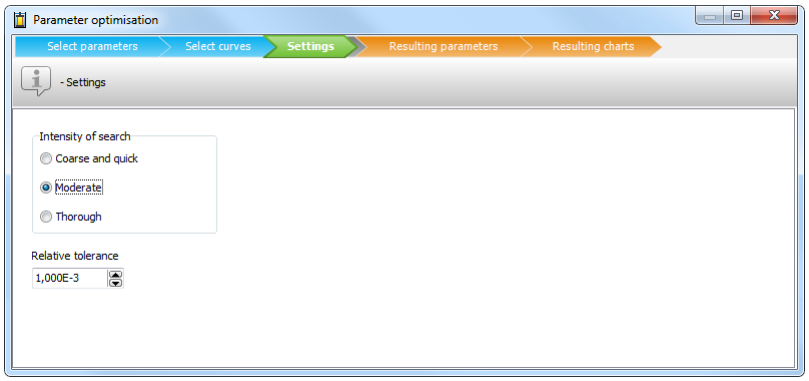


Figure 313: Settings window

Resulting parameters

The *Resulting parameters* tab shows the optimum values of the parameters used to obtain the best fit to the selected test data in addition to the minimum and maximum values and the reference values in the material data set (Figure 314 (on page 501)). If the optimum value is equal to the minimum or maximum value, it might be that the best value lies outside the specified range. Finally, the table shows the sensitivity of the selected parameters. A sensitivity of 100% means that the parameter has a high influence on the simulated test results, whereas a low sensitivity means that the parameter has a low influence on the simulated test results. Note that a low sensitivity also means that the test may not be suitable to optimise that parameter and, as a result, the suggested optimum value may not be accurate. Therefore it is better to do separate optimisations for different parameters based on relevant sections of test data curves rather than one optimisation with multiple parameters based on the full data curves.

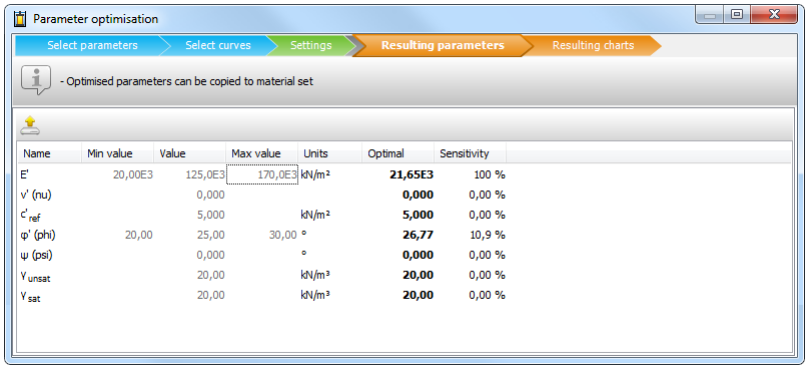


Figure 314: Resulting parameters window

A button is available to copy the optimised parameters to the material data set. This should only be done after it has been properly validated that the optimised parameters are indeed better than the original parameters, considering the use of the material data set in the finite element model.

Note that the parameters optimised for soil lab tests may not be the best parameters for the practical application as considered in the finite element model.

Resulting charts

The *Resulting charts* tab shows the results of the selected tests (Figure 315 (on page 502)).

For each test, three curves are visible:

Optimisation target	This curve represents the uploaded test data.
Optimisation results	This curve represents the simulated test with optimised parameters.
Reference simulation	This curve represents the simulated test with original parameters. It has no meaning in the optimisation process, but just shows how good or bad the existing material data set would fit the uploaded test data for the selected test conditions without optimisation.

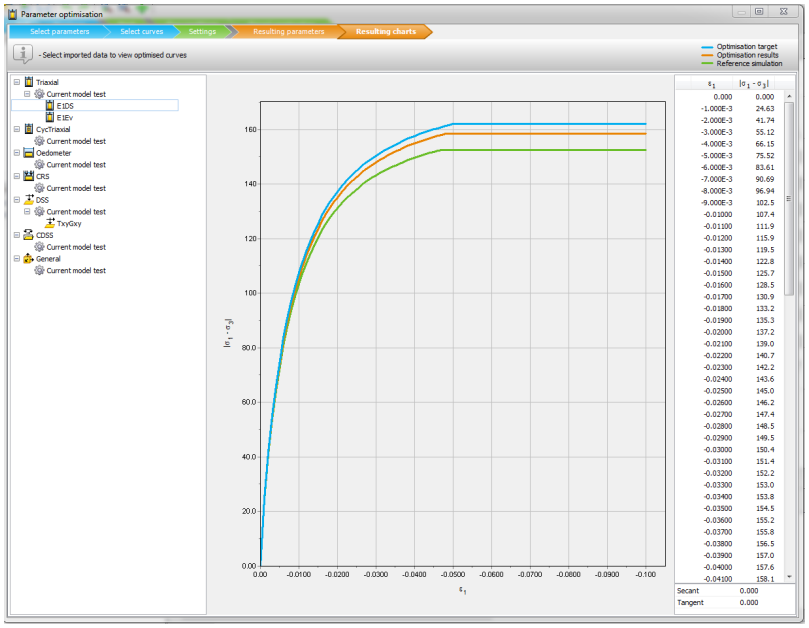


Figure 315: Resulting charts window

Limitations

The *Parameter optimisation* facility should be used with care. Note that parameters optimised for soil lab tests may not be the best parameters for the practical application as considered in the finite element model. This is because the application may involve stress levels, stress paths and strain levels which might be significantly different from the ones that occur in the soil lab tests.

Furthermore, the parameter optimisation facility has the following limitations:

- The experimental curve should be given in the order where the first column is a prescribed value, and the second is the computed one.
- It is not possible to automatically optimise test data curves that consist of multiple phases (for example loading and unloading phases). Such curves need to be uploaded for each phase and the corresponding part of the curves (phase) needs to be selected in order to perform the optimisation.
- The optimisation process itself is a numerical procedure which may involve numerical errors. The user remains responsible for validating the outcome of the optimisation process and the use of optimised model parameters in applications. The parameter optimisation is based on curve fitting. It is not possible to fit envelopes such as the failure line in a p-q diagram. As a consequence, it is not possible to find the optimised friction angle or cohesion from a p-q diagram.

11.3 Calculation using design approaches [ADV]

A design approach can be applied to a calculation phase by selecting the appropriate design approach of the phase in the *General* subtree in the **Phases** window (Figure 316 (on page 503)).

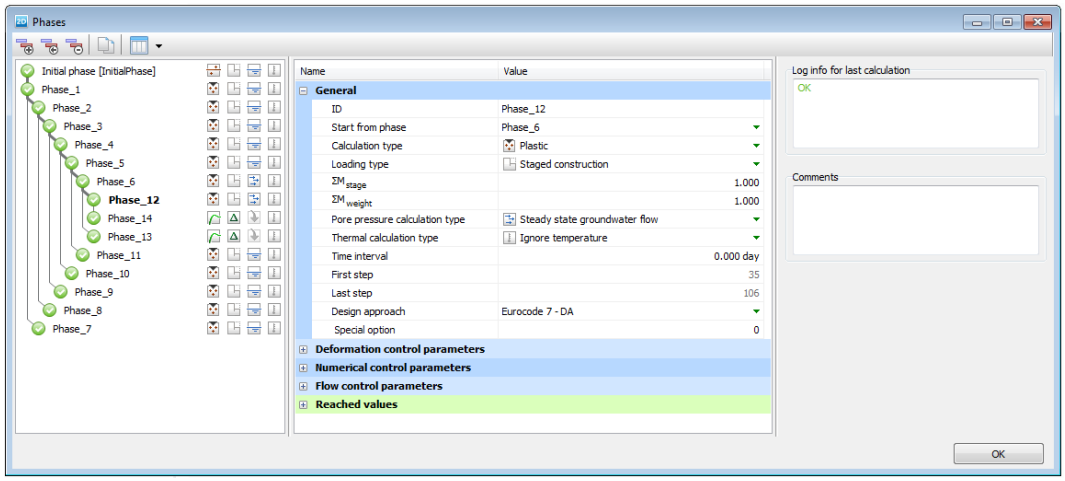


*Figure 316: Selection of design approach in the **Phases** window*

The partial factors for materials and loads are already defined in the Input program. However, in the **Staged construction** window, besides the activations of the loads, the corresponding labels should be assigned as well by selecting the corresponding option in the *Object explorers* (Figure 317 (on page 504)).

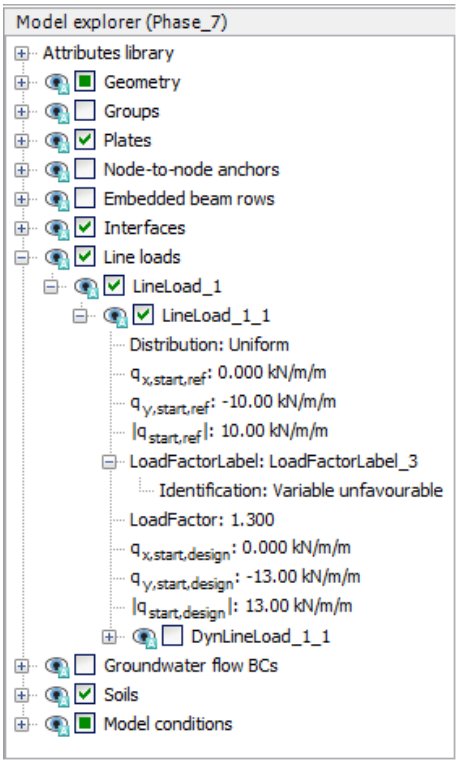


Figure 317: Selection of load label in the Selection explorer

11.4 Plaxis Remote Scripting Interface [GSE]

In PLAXIS 2D it is possible to design the model using Python scripting. This approach allows saving time when many calculations of similar applications are needed. The user can use command line available in Input and Output to run multiple commands in the Command runner, but it is also possible to use the remote scripting interface and its Python wrapper. Both command runner and remote scripting allow the user to use commands that are not exposed by the UI tools, as well as store and run macros using the extensive help (manuals and references). The full list of commands with parameters information is available in PLAXIS software under **Help** > **Command Reference**.

11.4.1 *Anatomy of a command*

Each command has a name and a target. Some commands may also require a parameter.

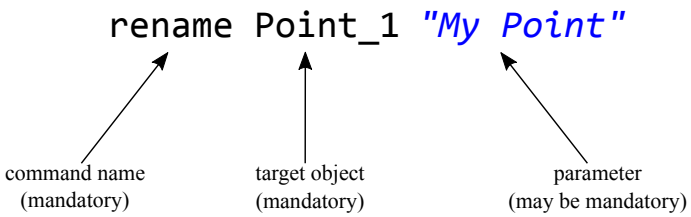


Figure 318: PLAXIS command

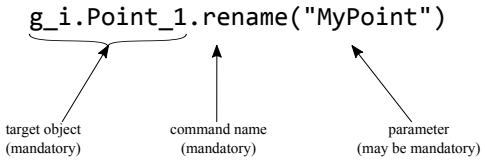


Figure 319: Python equivalent in PLAXIS

A few examples of PLAXIS commands and their equivalents in Python:

Command: `well Line_1`

Python: `g_i.well(g_i.line_1)`

Command: `tabulate (Line_1 Line_2) "First Second" "First=Point_1" }`

Python: `g_i.tabulate(g_i.Line_1, g_i.Line_2, "First Second", "First=Point_1")`

Command: `beam Line_1 "Material" BeamMat_1`

Python: `g_i.beam((g_i.Line_1, g_i.Line_2), "Material", g_i.Beammat_1)`

Command: `activate (Plate_1 Plate_2) (Phase_4 Phase_5)`

Python: `g_i.activate((g_i.Plate_1, g_i.Plate_2), (g_i.Phase_4, g_i.Phase_5))`

Command: `soil (Volume_2 Volume_3) "Material" SoilMat_1`

Python: `g_i.soil((g_i.Volume_2, g_i.Volume_3), "Material", g_i.SoilMat_1)`

Command: `rotateline (Point_1 Points[-2]) Line_1 45`

Python: `g_i.rotateline((g_1.Point_1, g_i.Points[-2]), g_i.Line_1, 45)`

- ***Command name***

  A command represents the name of an action that is to be executed, and it is given as first word in the command line as a global command (e.g. after g_i.) or as a specific target command (after some object) in the scripting. It is a method that is to be implemented.

  Python examples:

`g_i.gotomesh()`	`#gotomesh is the command name`
`g_i.borehole(0, 0)`	`#borehole is the command name`
`Point_1.echo()`	`#echo is the command name`

- ***Target***

  The target is the object for which the method is to be implemented. It can be an object (group of objects, list), global object, etc.

  Python examples:

`g_i.gotomesh()`	`#the target is a global environment object`
`g_i.Point_1.setproperties("x", 1, "y", 2)`	`#the target is a global environment object`
`polycurve_g.close()`	`#the target is polycurve_g`

- ***Parameter***

  Parameters represent the extra information passed to a command when it is called. In programming terms, they are arguments to a method. In the Command line the parameters are listed after the target. In Python scripting, parameters are supplied between the parentheses after the command's name. Some commands may not need a parameter, and some may require one or more parameters.

  Python examples:

`g_i.gotoflow()`	`#gotoflow command called without parameters`
`g_i.pointload(1, 0, "Fx", 10)`	`#pointload command uses several parameters to create a point load on the point (1 0 2) and sets its force to 10`

11.4.2 *Geometry and calculation modes objects differences*

Each object (line, line load, plate, etc.) created in the *Geometry* mode (*Soil*, *Structures*) is named and stored according to the general indexing rule (for more information see Indexing in commands (on page 46)). However, when the mode is changed to the *Calculation* mode (*Mesh*, *Water conditions*, *Staged construction*), the entity gets replaced by a different entity which may have fewer or more other properties and is stored differently. For example, the line with name `Polycurve_1` will get a new name in *Calculation* modes, which will now be `Polycurve_1_1`, `Polycurve_1_2`, etc. This is done due to intersection: when changing the modes, the geometric entity might be intersected by other objects, so the program will automatically intersect this entity, splitting it into sub-objects.

11.5 PLAXIS 2D to 3D Converter [GSE]

The PLAXIS 2D to 3D Converter tool converts a PLAXIS 2D geometry and its soil properties into an extruded model in PLAXIS 3D.

In general in this conversion:

- Points are extruded as lines, lines as surfaces and polygons become volumes.
- Depending on its type, a structural element will be extruded or not (e.g. plates get converted into surfaces while anchors remain as line objects after the conversion to PLAXIS 3D).

Note: Some components of the PLAXIS 2D might be excluded, in those occasions the user needs to pay attention to the warning messages displayed by the software (see Results (on page 511)).

11.5.1 Remote scripting server configuration [GSE]

Before using the PLAXIS 2D to 3D convertor is necessary to setup the connection between the PLAXIS 2D and PLAXIS 3D products through the *Remote scripting server* feature. This can be done as follows:

1. Open the PLAXIS 2D model to be converted and created as a new project within PLAXIS 3D.
2. While in PLAXIS 2D Input program click on **Expert menu** > **Configure remote scripting server...**
3. From the emerging window, after configuring communication *Port* and *Password*, press on **Start Server** and the port will be activated. See Figure 320 (on page 507).

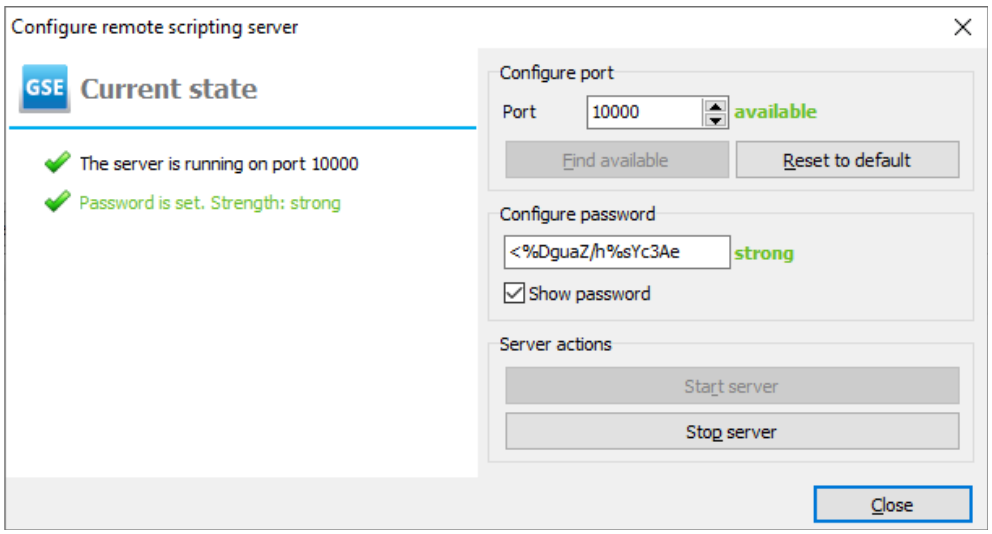


Figure 320: Configuration of Remote scripting server for PLAXIS 2D

4. On PLAXIS 3D Input program repeat similar process as in step 2.
5. Once the remote scripting server emerging window appears go to **Configure port** > **Find available** > **Start server** as shown in Figure 321 (on page 508).

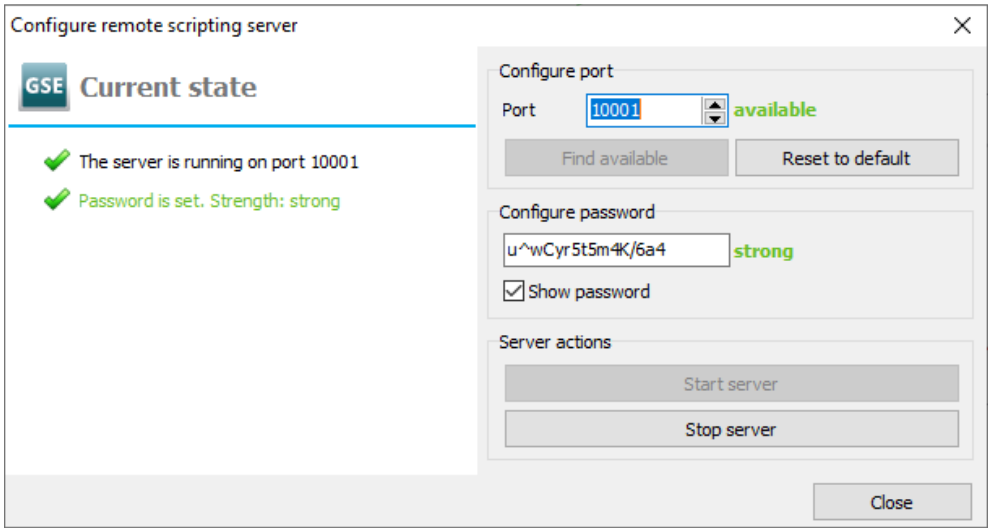


Figure 321: Configuration of Remote scripting server for PLAXIS 3D

Note: In the most recent PLAXIS versions, only the configuration of the *remote scripting server* within PLAXIS 2D is required (steps 1 to 3). As a consequence, by starting the converter (see Running the PLAXIS 2D to 3D converter tool (on page 508)) PLAXIS 3D and the *remote scripting server* are automatically launched and set, respectively.

11.5.2 Running the PLAXIS 2D to 3D converter tool

After ensuring that a PLAXIS 2D model is in place (inside the PLAXIS 2D *Input program*), the converter can be started by clicking on **Expert menu** > **PLAXIS 2D to 3D Converter** . After that, the tool window will be displayed (see Figure 322 (on page 509)).

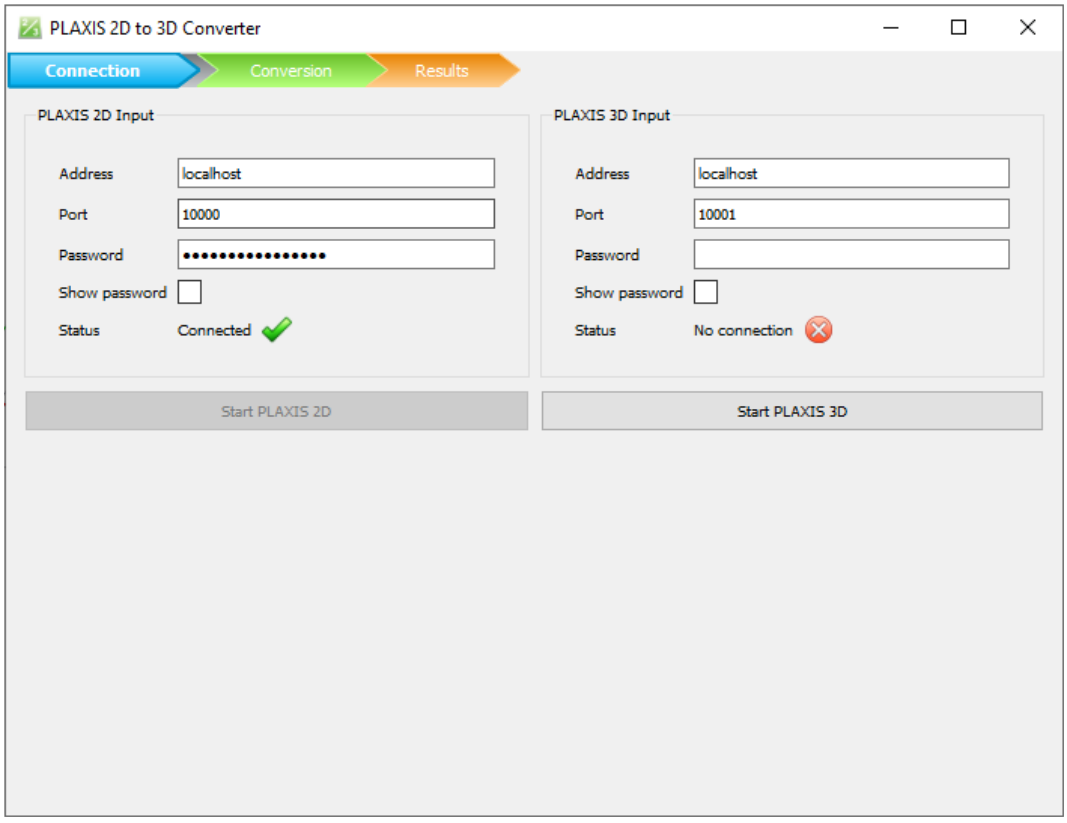


Figure 322: PLAXIS 2D to 3D converter tool modules

Note: The conversion tool will appear just if the Remote scripting server configuration [GSE] (on page 507) was executed on PLAXIS 2D.

The **PLAXIS 2D to 3D converter** contains the following modules for the extrusion process: **Connection**, **Conversion** and **Results**.

Connection

The **Connection** module allows to create the link between PLAXIS 2D and PLAXIS 3D.

To establish the connection, in this module by default most of the information is filled (as shown in Figure 322 (on page 509)). However, the *Password* for PLAXIS 3D needs to be completed. This password can be retrieved from PLAXIS 3D - *Input program* > **Expert menu** > **Configure remote remote scripting server.** A successful connection will be displayed as shown in Figure 323 (on page 510).

Note: If different *Address*, *Port* or *Password* are defined, the users need to fill the information that matches their own servers configuration.

Note: Alternatively, especially for the latest PLAXIS versions, the converter tool in PLAXIS 2D will launch automatically PLAXIS 3D and it will fill the required parameters for the connection.

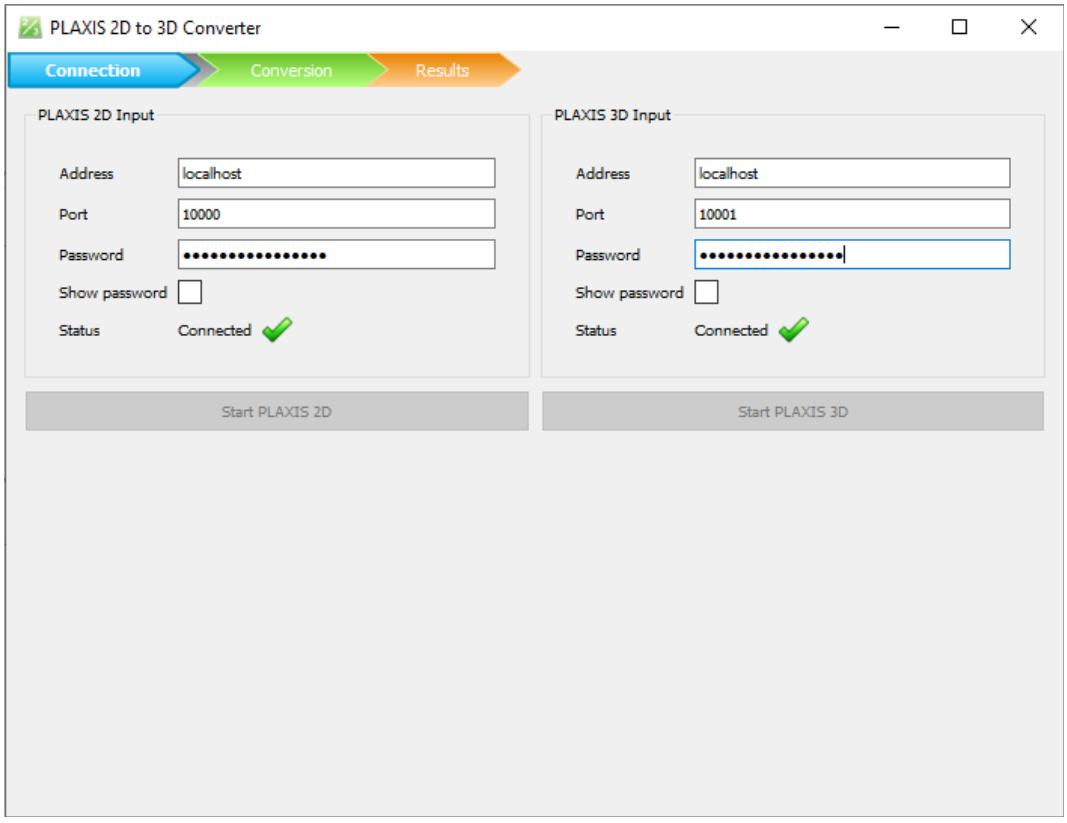


Figure 323: Successful connection between PLAXIS 2D and PLAXIS 3D

Conversion

The **Conversion** module allows to define the general extrusion settings (see Figure 324 (on page 511)). In particular, it is necessary that users select the reference axes for the project (*Base Plane in 3D*) and the *extrusion depth*.

To start the conversion please click on *Run conversion*.

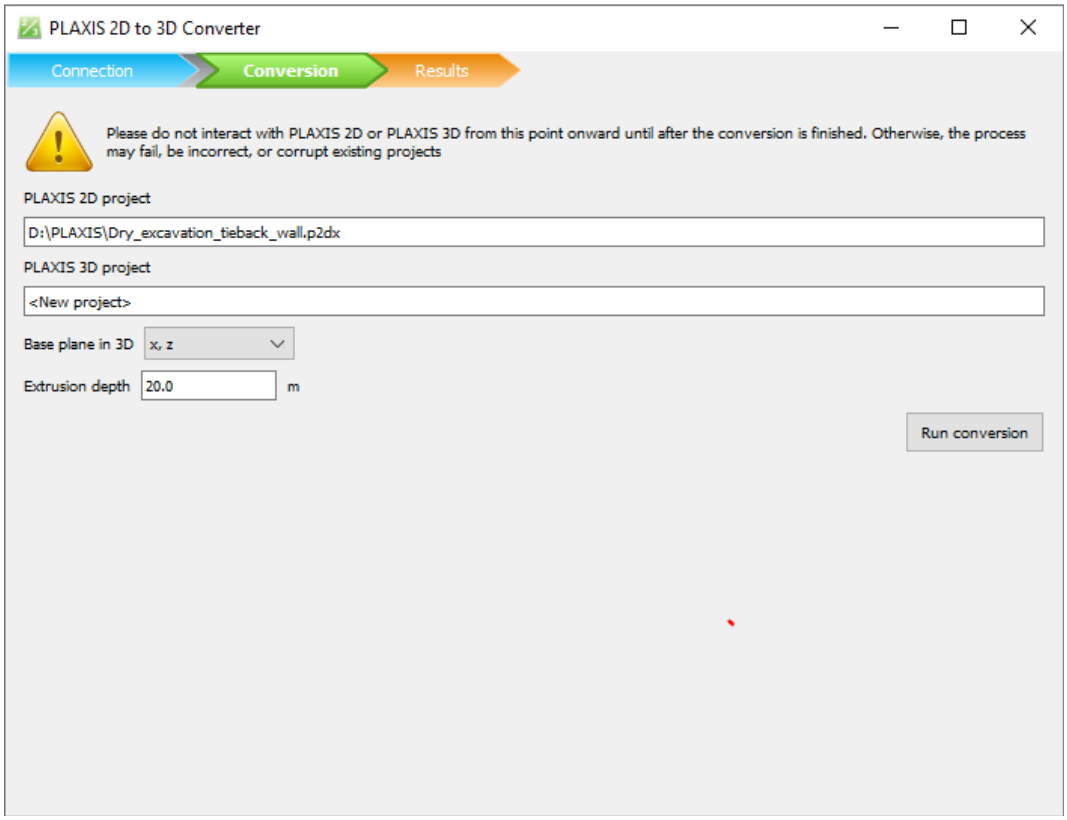


Figure 324: Conversion module settings

Results

Once the conversion is run, the **Results module** displays the progress of the conversion using the *Status* and *Remarks* sections. *Status* shows the objects that are getting extruded into PLAXIS 3D. *Remarks* presents different warnings about the objects and properties that, respectively, were not extruded or assigned to the 3D model (e.g see Figure 325 (on page 512)). Each warning displays a possible way to address it manually on PLAXIS 3D.

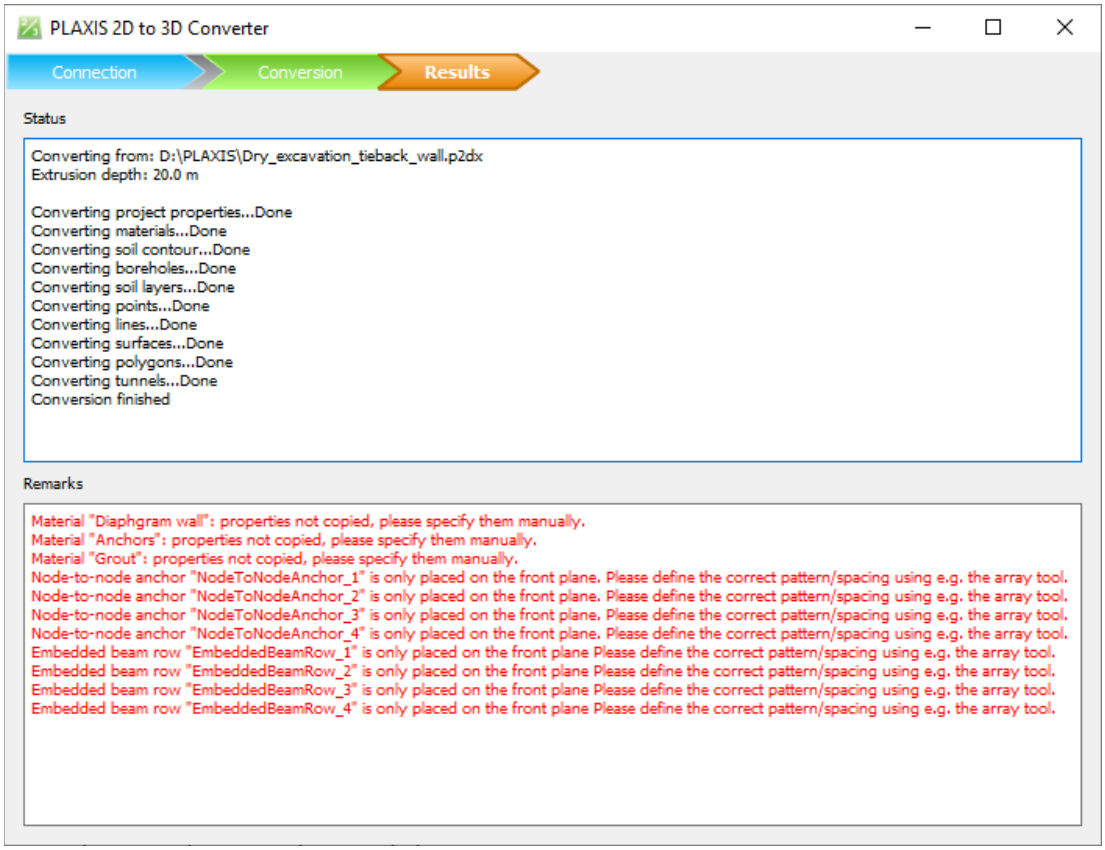


Figure 325: Results module - Status and Remarks after conversion

11.5.3 PLAXIS 3D model outcome

After the conversion of the PLAXIS 2D model (e.g. see Figure 326 (on page 513)), the transformed elements will be displayed into PLAXIS 3D (e.g. see Figure 327 (on page 513)). As mentioned in Results (on page 511), the non-processed elements will require manual editing on PLAXIS 3D Input program both in the Ground modelling - Soil mode (on page 48) and Modelling loads and structures - Structures mode (on page 68).

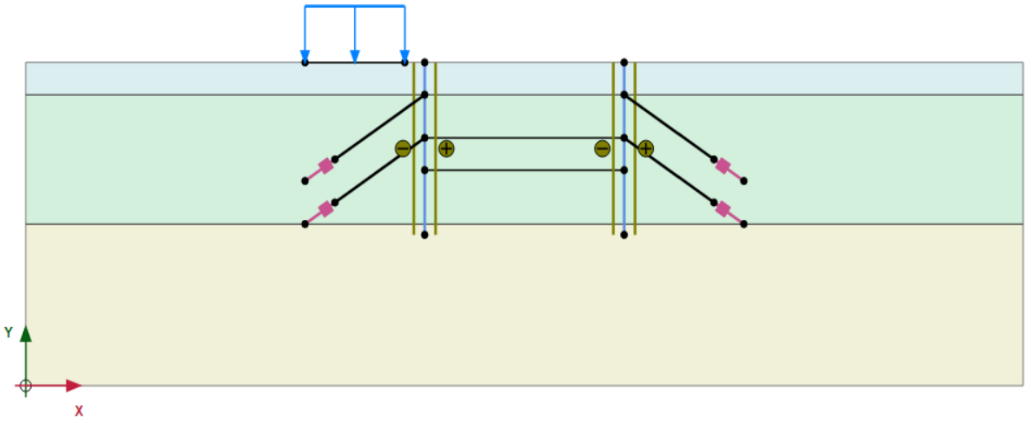


Figure 326: Example PLAXIS 2D model

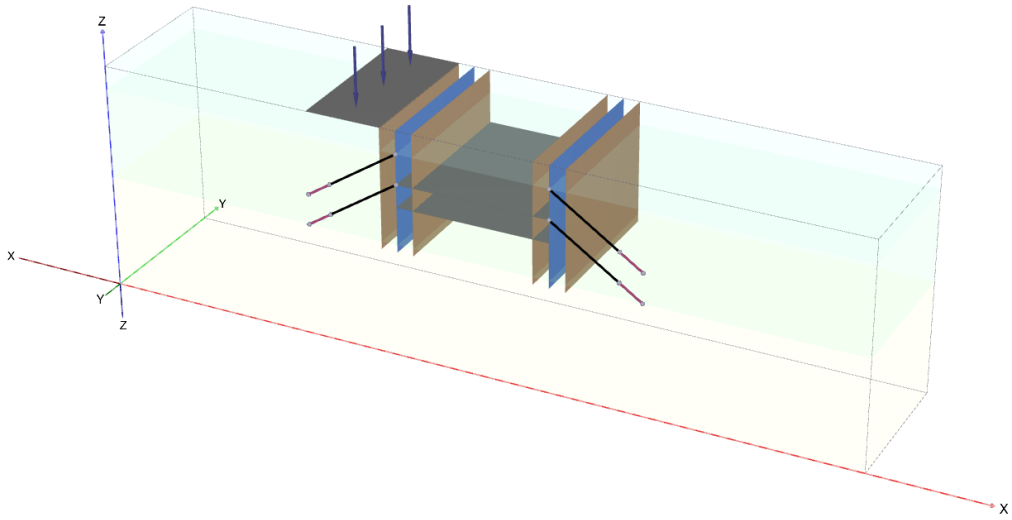


Figure 327: Example PLAXIS 3D model after extrusion

Tip:

- One example of elements that will need manual editing on PLAXIS 3D Input program are structural elements such as anchors (see Figure 327 (on page 513)). As expected, anchors will remain as linear elements (not extruded as surfaces) and also they will be placed only in front the plane. In this circumstance, the user is advised to edit the model using *Assisting tools* such as Copy object - Array (on page 69) to replicate those elements into the model.
- For more information on how to edit a PLAXIS 3D model after using the PLAXIS 2D to 3D converter tool please check Bentley communities.

11.6 *Packing a project*

The created project can be compressed using the **Pack project** application (Figure 328 (on page 514)), which is available in the *File* menu of the Input program. This application can be executed directly from the PLAXIS 2D AE installation folder by double clicking the corresponding file (PackProject.exe). A shortcut to the application can be created as well.

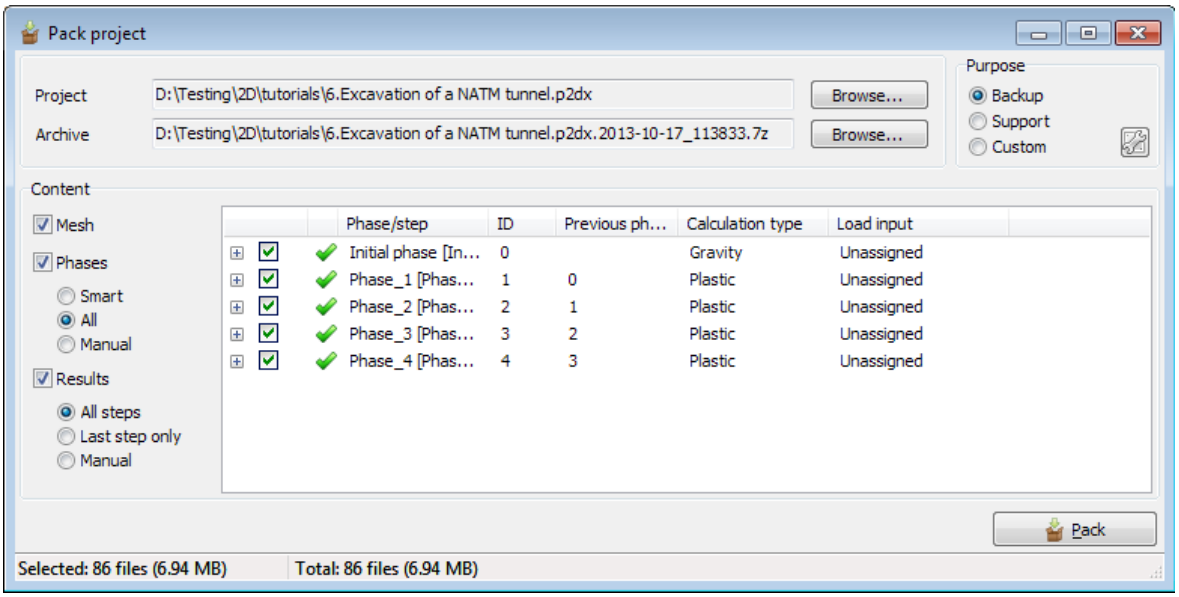


*Figure 328: **Pack project** window*

The project to be compressed and the archive can be located using the **Browse** button. The options available in the **Purpose** box are:

Backup	All the files in the project are included in the compressed project as well as the mesh information, phase specification and the results of all the saved calculation steps. The extension of the project file, indicating in which program it was created, and the archiving date are included in the archive name.
Support	Selecting this option enables including all the information required to give support for the project at hand. Note that support is only provided to GSE users.
Custom	The user can define the information to be included in the archive.

The options for compression and volume size are available in the **Archive options** window (Figure 329 (on page 515)), displayed by clicking the button in the *Purpose* box.

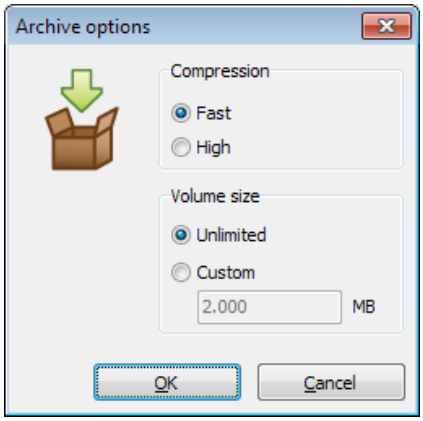


*Figure 329: **Archive options** window*

The **Content** box displays the options for the information to be included in the archive is shown. The options available are:

Mesh	The information related to geometry is included when the *Mesh* option is selected.
Phases	The options available are: • **Smart:** When a phase is selected in the tree, the parent phase is selected automatically in order to provide a consistent chain of phases. • **All:** All the phases available in the project are selected. • **Manual:** Specific phases can be selected by the user.
Results	The results to be included in the archive can be selected. The options available are: • **All steps:** The results of all the calculation steps are included in the archive. • **Last step only:** The results of only the last calculation step of each phase are included in the archive. • **Manual:** The results of specific calculation steps can be selected by the user.

Note that when the *Backup* or the *Support* option is selected, the *Content* options are automatically selected by the program.

11.7 Exporting geometry [GSE]

11.7.1 *STEP and DXF format*

PLAXIS 2D allows to export the geometry of the model in different formats. This is available in the **File** menu of the input program (Figure 330 (on page 516)) in the **Geometry** modes (**Soil** and **Structures**).

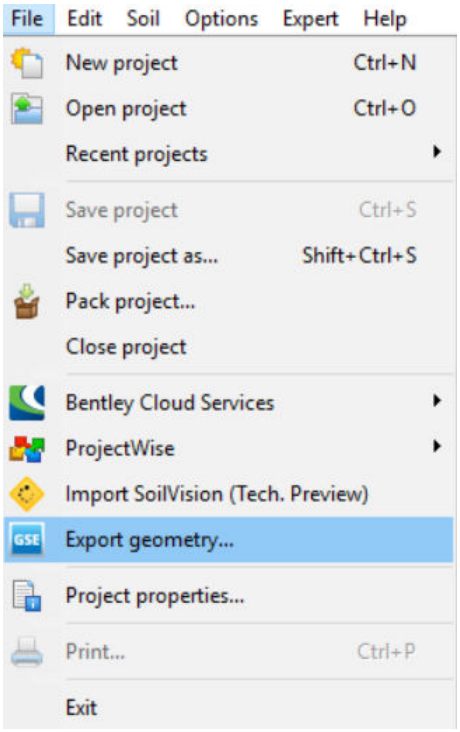


Figure 330: Export geometry

The file can be saved as a **.step* (Standard for the Exchange of Product model data) or **.dxf* (Drawing Interchange Format) file. The file contains the entire *generated geometry* in one file (not the cut geometry). This means items such as boreholes, top soil surface and tunnel designer data are excluded, only the entities generated from those are included (e.g. generated soil volumes, generated tunnel slices, lining, rock bolts lines). The data is purely geometric, not annotated with features, and not divided in layers.

11.8 PLAXIS 2D LE Model Importer [GSE]

The PLAXIS 2D LE Model Importer tool allows to import PLAXIS LE (.svm), SVSLOPE and SVSOLID model geometry, material definitions and their supported parameters to an equivalent PLAXIS 2D model. This is available in the **File** menu of the input program in the Geometry modes (Soil and Structures) (Figure 331 (on page 517)).

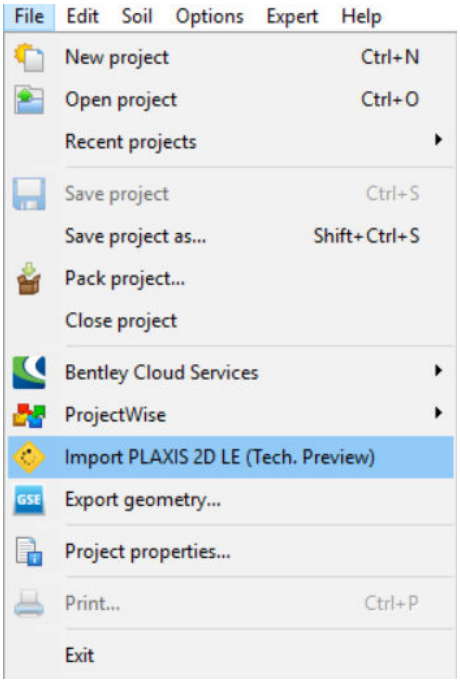


Figure 331: PLAXIS 2D LE Model Importer

File >*Import PLAXIS 2D LE* prompts the user to configure the remote scripting server – set a password and configure an available port (see Starting the remote scripting server (on page 558)). This opens the **PLAXIS 2D LE model importer** (Figure 332 (on page 518)), where the user can select the PLAXIS LE model to be imported and click on the import button.

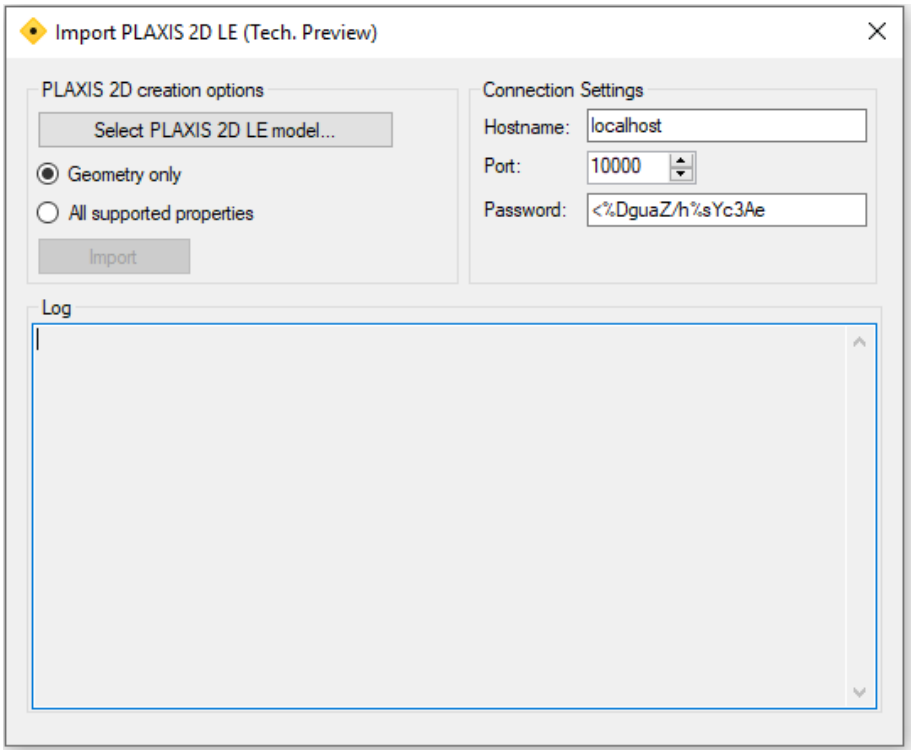


Figure 332: PLAXIS 2D LE model importer window

Note: After the import the new PLAXIS 2D model will inherit the units of the original PLAXIS 2D LE model.

11.9 Bentley Cloud Services [GSE]

Bentley Cloud Services can be accessed from the **File** menu, or from the icons in the menu bar. This menu contains the following four options:

- **Associate project**: this opens a separate window listing the available Bentley projects. The user can select any of them. This will store into "Project.ConnectedProjectInfo" (available from the command line as read-only properties) the GUID and the name of this cloud project. This property is not reset when using "Save as", i.e. a "saved as" project will also carry this project association.
- **Disassociate project**: when there is a project association, this item removes the association between the current PLAXIS project and the Bentley project (i.e. it resets the content of "Project.ConnectedProjectInfo").
- **Personal portal**: This can be used at any time to open the personal Bentley portal of the user in the default web browser.
- **Project portal**: When a Bentley project is associated to the current PLAXIS project, this opens the project portal in the default web browser.

Except the **Disassociate project** functionality, the other three requires the user to be logged in with the **Connection Client**.

11.10 ProjectWise Integration Commands [GSE]

When ProjectWise has been installed, four specific Integration commands are available in PLAXIS 2D.

- **Open...**
- **Add...**
- **Save...**
- **Properties...**

These commands are accessible through the **File** > **ProjectWise** menu and the toolbar.

11.10.1 Add File to ProjectWise

To add the current PLAXIS 2D to aProjectWise repository, use the following procedure:

You must have a PLAXIS project open.

Either select **File** > **ProjectWise** > **Add to ProjectWise** or select the **Add File to ProjectWise** icon in the toolbar .

If you have not previously logged into a ProjectWise datasource, then the **ProjectWise Log in** dialog opens. Provide the Datasource and required user credentials and then click **Log in**.

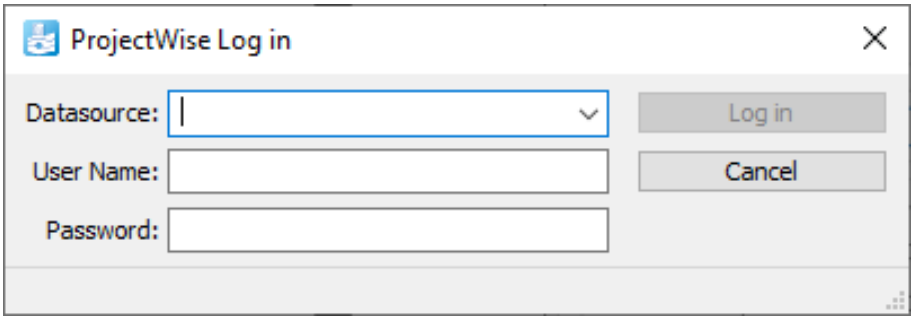


Figure 333: ProjectWise Log in dialog

Select a Wizard dialog appears, as shown in Figure 334 (on page 520). Select any of the options - No Wizard or Advanced Wizard.

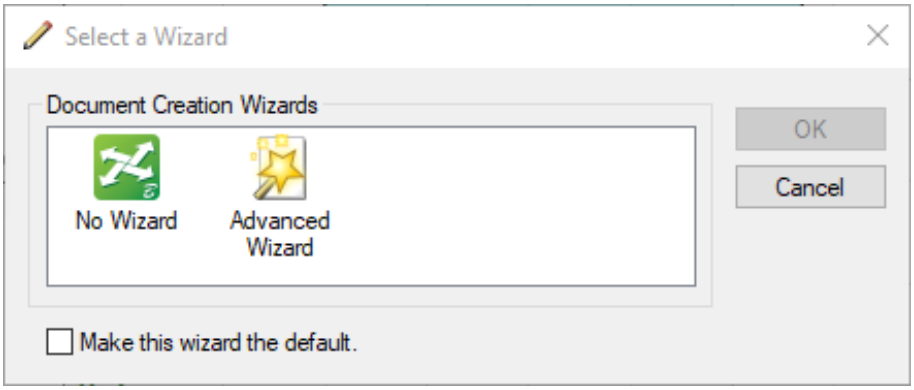


Figure 334: Select a Wizard dialog

Selecting **No Wizard** opens **Save Document As** window (Figure 335 (on page 520)). Select the target folder. Fill in Document name and Description. Format type should be Packed PLAXIS 2D Project.

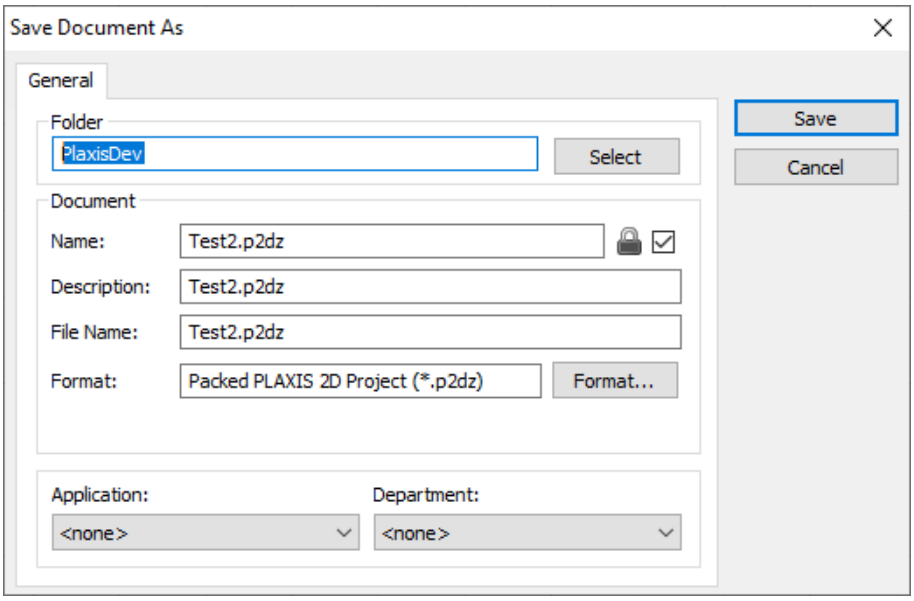


Figure 335: Save Document As dialog

On selecting Advanced Wizard, **Advanced document creation** window appears. Click **Next**, **Select Target folder** dialog appears. Select folder, and click **Next**. **Document Properties** window appears. Fill in Document name, Description, Version. **Save as Type** format should be Packed PLAXIS 2D Project. Click Next, **Create a Document** window appears. Confirm the document properties and click **Next**. Provide the Datasource and required user credentials and then click **Log in**. In the next dialog enter comments describing the document or later the changes you have made.

Note: These comments will be visible in the Audit Trail tab in the **ProjectWise Properties** window.

A progress bar is displayed during the data transfer indicating the file content transfer status.

Add to ProjectWise dialog appears (Figure 336 (on page 521)), where the user can choose the corresponding files in the project to be compressed and uploaded to ProjectWise .

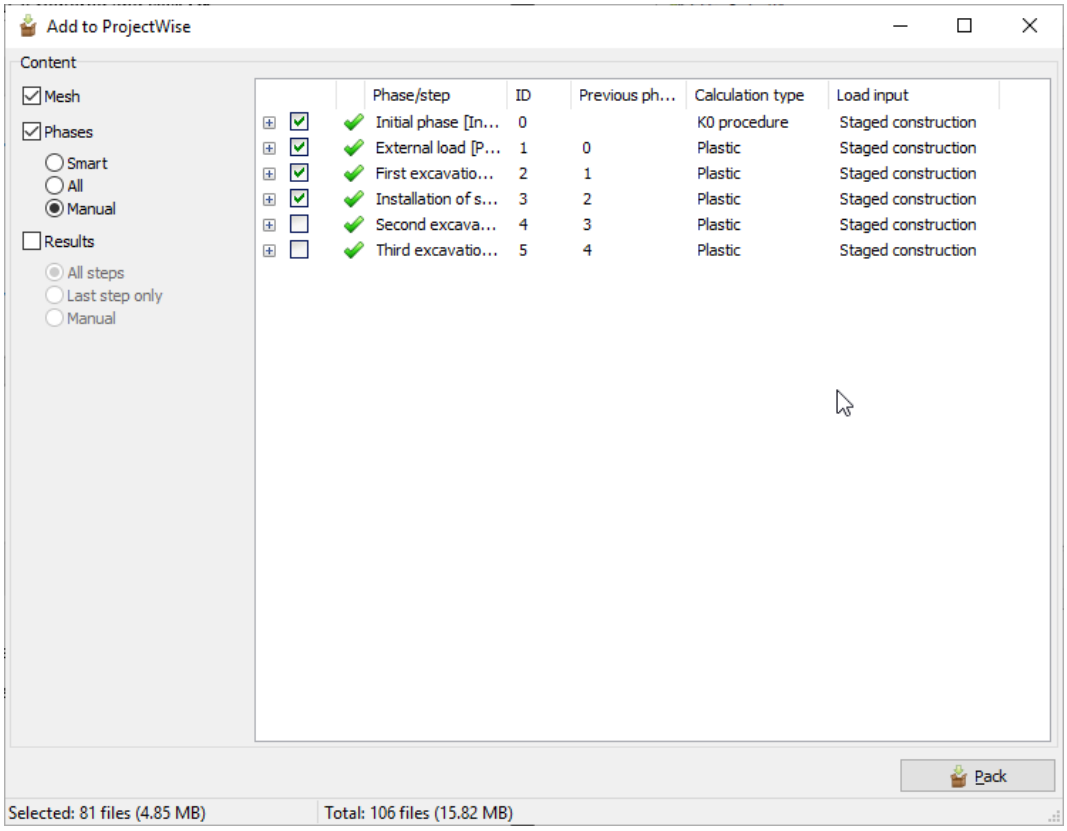


Figure 336: Add to ProjectWise

The uploaded PLAXIS project is now checked out from the ProjectWise datasource. This means that you have exclusive access to the project. Others can only open it as 'read-only'. PLAXIS now recognizes that the current model is linked to ProjectWise with a '[PW:Checked Out]' text appearing in the project title, and allows the **File** > **ProjectWise** > **Properties** commands to become active.

11.10.2 Open File stored in ProjectWise

To open a PLAXIS file stored in a ProjectWise repository, use the following procedure:

1. Either select **File** > **ProjectWise** > **Open from ProjectWise** or select the **Open from ProjectWise** tool from the toolbar. If you have not previously logged into a ProjectWise datasource, then the **ProjectWise Log in** dialog opens, allowing you to log into a **ProjectWise** datasource. After logging in to the datasource, the **Open from ProjectWise** dialog (Figure 337 (on page 522)) is displayed, allowing the user to browse/ search the datasource contents and select the document to open.

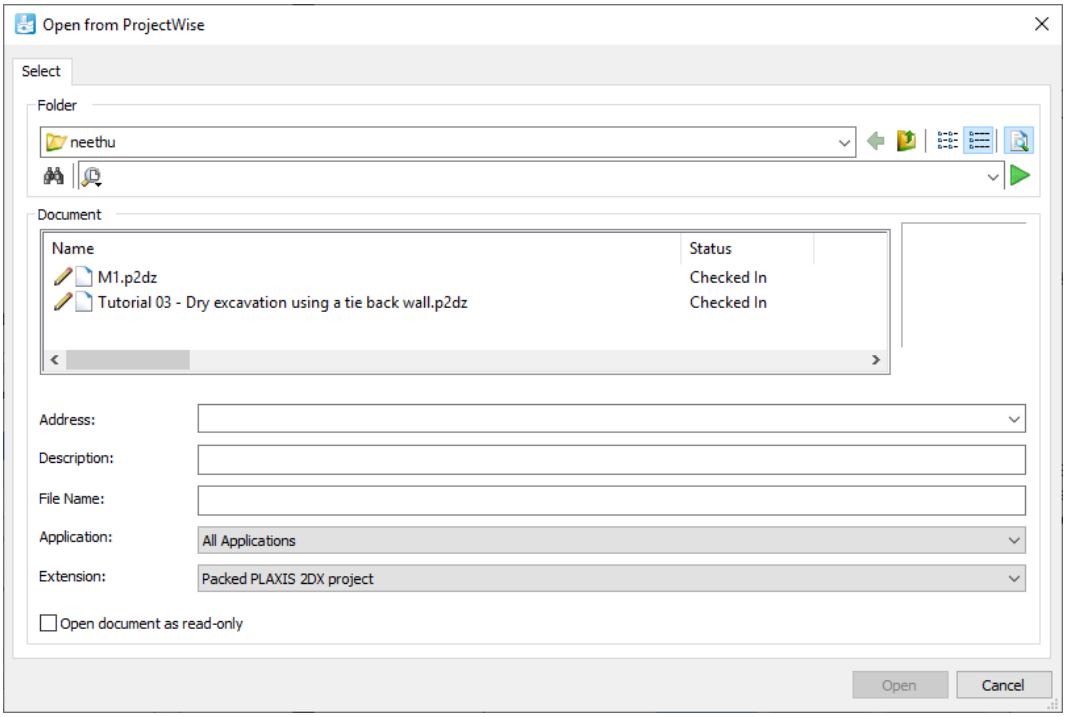


Figure 337: Open from ProjectWise

2. Select a project within the ProjectWise datasource.
3. Click **Open**.
   a. If the project has never been opened before, then PLAXIS asks to specify a local folder where the project can be downloaded.
   b. If the project has been opened before, PLAXIS will check if the local version is up-to-date, and if not, it will be updated. If there are local changes then, there is an option to overwrite them or select a different folder.

The ProjectWise project is checked out from the ProjectWise datasource, copied to local disk, and opened by PLAXIS. If the project is checked out by another user then it can only be opened on 'read-only' mode. This means that if changes are made to the local model it cannot be uploaded to the server. If the other user finishes working on the model and checks in the project, your local copy will have conflicts with the server copy. The next time when you open the project you will get a notification about this conflict. Then you can either overwrite your local changes with the server copy or you can open the project from the server into a new folder.

PLAXIS recognizes that the current model is linked to ProjectWise and allows the **File** > **ProjectWise** > **Save to ProjectWise** and the **File** > **ProjectWise** > **Properties** commands to become active.

11.10.3 Save to ProjectWise

To update the ProjectWise server copy of the file with the latest changes that have been made to the local copy of the file, use the following procedure.

When a PLAXIS model is either opened from a ProjectWise datasource or has been added to a ProjectWise datasource , this functionality is active.

1. Select **File** > **ProjectWise** > **Save to ProjectWise** or the Save to ProjectWise icon from the toolbar.

   The model that is currently open in PLAXIS is checked for changes. If the model has been edited since the last save, a message is displayed asking if these changes should be saved before updating the file server copy.

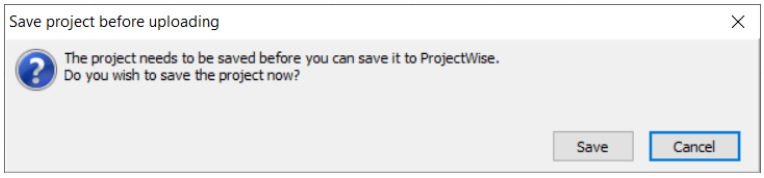


Figure 338: Save project dialog

2. Click **Save**.
3. Refer to Add File to ProjectWise (on page 519) for further steps.

A Progress bar is displayed during the data transfer indicating file content transfer status.

11.10.4 To View ProjectWise properties

To review or edit the ProjectWise properties and metadata associated with the current PLAXIS model, use the following procedure.

1. Select **File** > **ProjectWise** > **Properties**.

   The **ProjectWise Document Properties** dialog opens.

2. Edit the ProjectWise properties and metadata as necessary.

   Note: The 'Audit Trail' tab shows the edit history and the comments entered while saving changes.

3. Click **Save**.

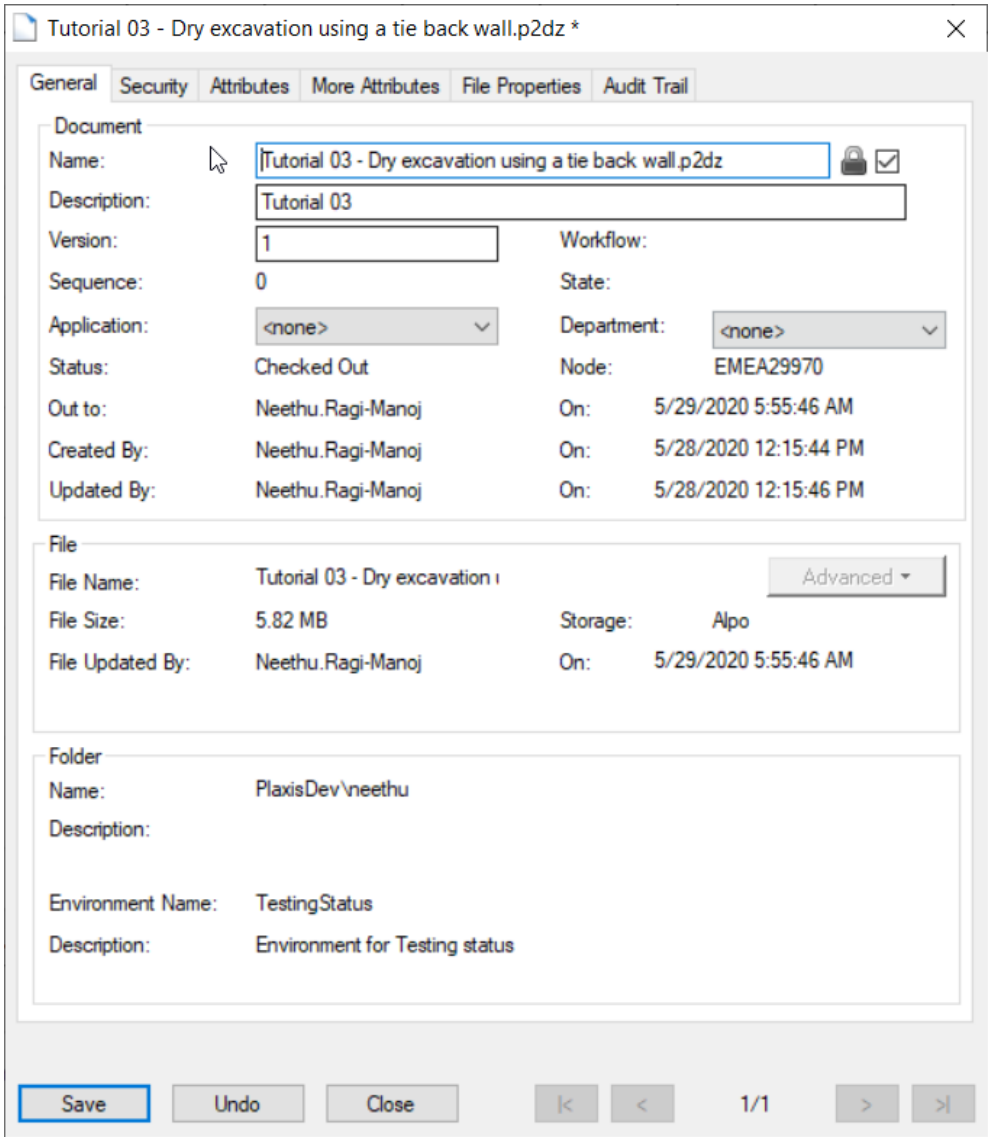


Figure 339: ProjectWise Document Properties

A Progress bar is displayed during the data transfer indicating file content transfer status.

12 Engineering Applications

12.1 Modelling ground anchors

Ground anchors are used to restrain soil retaining walls to prevent excessive forward movement or to restrain excavation bottom plates to prevent uplift under high pore pressure conditions or to restrain other foundation structures that are subjected to tensile forces. This section explains how ground anchors can be modelled in PLAXIS 2D.

There are various ways to create ground anchors in the real world, but they can be all modelled in a similar way in PLAXIS 2D. Most methods in reality are using a steel tube or anchor bar or a bunch of steel wires that are drilled into the ground through an opening in the structure to be restrained. At the end part of the anchor in the ground, grout (cement) is injected at high pressure, forming a grout body around the anchor bar with a diameter that is significantly larger than the diameter of the anchor bar. This grout body has a strong interaction with the surrounding soil, whereas the non-grouted part in between the grouted part and the wall (the 'free length') has hardly any interaction with the soil. When the anchor is installed, it is usually pre-stressed up to 60-80% of its designed working load. After that, the anchor force will further evolve as a result of the construction process.

12.1.1 *Geometric modelling of ground anchors*

The stress state around the grouted part of a ground anchor is highly three-dimensional. This makes it difficult to model a ground anchor realistically in a 2D model. Nevertheless, an approximation is possible that enables a realistic analysis of deformation and overall stability of soil retaining structures, both in 2D and 3D.

A ground anchor can be created by a combination of a node-to-node anchor and an embedded beam row, in which the embedded beam row represents the grouted part of the anchor and the node-to-node anchor represents the anchor free length. The behaviour of the embedded beam row is set to 'Grout body'. The connection type is internally set to 'Free'. This is required since otherwise the embedded beam row will also be directly connected to the corresponding soil element, which will lead to an unrealistic loss of axial force in the connection point. Setting the connection to 'Free' will ensure that the axial force in the node-to-node anchor is more or less equal to the axial force at the top of the embedded beam row.

12.1.2 *Properties of ground anchors*

The stiffness of a ground anchor (i.e. the resistance against elongation) is formed by the axial stiffness of the anchor bar, *EA*. This applies to the anchor free length as well as the grouted part of the anchor, since upon loading of the anchor, tensile cracks will occur in the grout body, and hence, the grout body itself will not contribute to the axial stiffness.

The axial stiffness, *EA*, can be specified directly as input parameter in a material data set for anchors, as used for the node-to-node anchor. In 2D, the anchor spacing in out-of-plane direction, l_s required. Internally, will divide the axial stiffness by the spacing in order to calculate the axial stiffness per unit of width in a plane strain model. Optionally, a maximum anchor tensile force $F_{max,tens}$ force $F_{max,comp}$ should be set to 0). It is also possible to specify the *Material type* as *Elastoplastic with residual strength*, which enables the modelling of anchor breakage ($F_{residual,tens}$ = 0).

The properties of the embedded beam row, as used for the grouted part of the anchor, involve the stiffness properties as well as the interaction properties with the surrounding soil. Rather than specifying *EA* directly as the axial stiffness, there is a separate input of Young's modulus, *E*, and the anchor cross section geometry. Here, it should be considered that the anchor geometry is dominated by the diameter of the grout body, D_g, whereas the axial stiffness is dominated by *EA* of the anchor bar. Hence, when selecting *Predefined* as *Pile type* and *Massive circular pile* as *Predefined pile type* (it is a ground anchor rather than a pile here), the *Diameter* should be set to the approximate diameter of the grout body, D_g. It is important to select the diameter of the grout body (and not the diameter of the anchor bar) to set the right size of the elastic zone around the embedded beam such that the specified anchor bearing capacity can be obtained. Now, Young's modulus must be selected such that the product of *E* and *A* (= $\pi D_g^2/4$) is equal to *EA* of the anchor bar, as used in the material data set for the node-to-node anchor. Note that this leads to a fictitious value of Young's modulus which is different than a realistic Young's modulus of steel. In 2D, the stiffness properties also involve the out-of-plane anchor spacing, l_s, which is the same value as entered for the node-to-node anchor.

The interaction properties involve only skin friction. Here, the distribution can be assumed uniform by setting $T_{skin,start,max} = T_{skin,end,max}$ equal to the total designed ground anchor bearing capacity (in the unit of force) divided by the length of the embedded beam row. Ground anchors do not have a base resistance, so F_{max} = 0. The interface stiffness factors can be left to their default values.

12.1.3 *Activation and pre-stressing of ground anchors*

The activation of ground anchors in a calculation phase as part of the staged construction settings is done by activating both the node-to-node anchor part and the embedded beam row part. Usually, when ground anchors are installed, they are pre-stressed. The pre-stress force can be specified by selecting the node-to-node anchor part and checking the *Adjust prestress* option in the *Selection explorer* and specifying the right pre-stress force $F_{prestress}$. Note that in all models (2D and 3D) this is a force PER ANCHOR (not per unit of width out-of-plane).

In the calculation phase in which the ground anchor is pre-stressed, the resulting anchor force will be exactly equal to the pre-stress force. In subsequent calculation phases, however, the anchor force is supposed to further evolve as a result of the construction process. Therefore, the anchor should NOT be pre-stressed again in subsequent calculation phases, unless this is also done in reality ('post tensioning' of ground anchors).

12.2 Transverse forces in embedded beams

In addition to displacement differences and shear forces in the axial direction, a pile can undergo transverse forces, $t_{\perp}$, due to lateral displacements. The lateral displacements can be induced by a transverse force applied at the top of the pile or as a consequence of the transverse distributed load caused by the lateral displacement field of the soil surrounding the pile. In the first case, the overall behaviour may not show realistic results. In the second case, although embedded beam row elements are not meant to be used as laterally loaded piles, they show reasonable results and the overall behaviour is realistic. The second case, moreover, represents a lot of engineering cases and this section explains the most common applications in PLAXIS 2D.

12.2.1 *Abutment pile*

An abutment is a structure at the ends of an arch (e.g. dams) or span (e.g. bridges) meant to transfer vertical or horizontal loads to the ground or other foundation elements. The abutment is a crucial substructure in a lot of geotechnical applications: dams, bridges, slope stability, embankments and retaining walls. When the abutment structure is not sufficient to bear the external loads or guarantee compatible displacements, piles could be added to improve the performance.

The abutment piles are subjected to axial as well as transverse forces through the displacement field of the soil. $T_{Lateral}$ forces can be induced by: moving loads (vehicles or trains on bridges and embankments), arching effect (dams), landslides and ground flow (slope stability). The embedded beam row elements of PLAXIS 2D can realistically model the piles in these engineering applications.

The user should refer to Embedded Beam Rows (on page 100) and General properties (on page 275). No further information is required for the modelling techniques embedded beam row elements in abutment pile applications.

12.2.2 *Quay wall with relieving platform*

A quay wall is an earth retaining structure used to moor ships. It bears vertical loads caused by storage and handling equipment and horizontal loads from the wind, soil pressure and ship impacts. Quay walls might differ in type and magnitude of the terminal. The key requirement for ships is the retaining height; when handling the freight, it is necessary to provide sufficient area and bearing capacity. For larger quay walls, using relieve platforms frees the quay walls from the high soil pressure brought by the high retaining height and the vertical loads induced by massive storage. The presence of the relieving platform, generally coupled with bearing piles, reduces the horizontal loads on the quay wall and, consequently, the bending moments. The structure could comprise an anchorage at the connection point of the quay wall and the relieving platform.

The bearing piles of the relieving platform take a part of the horizontal load that is typically taken by the retaining wall. For this reason, these piles are subjected to transverse forces induced by the lateral displacement field of the soil. The embedded beam row elements of PLAXIS 2D can realistically deal with this application. Furthermore, if the quay has a bulk-storage load, there are horizontal components difficult to model with other approaches (e.g. sub-grade reaction models). PLAXIS 2D can easily deal with this complex loading conditions.

The user should refer to Embedded beams and General properties (on page 275). No further information is required for the modelling techniques embedded beam row elements in quay wall with relieving platform applications.

12.3 Modelling vacuum consolidation [ADV]

Vacuum consolidation is a technique to apply preloading on a construction site by creating an 'under-pressure' in the ground and thus using the external atmospheric pressure as preloading. In this way, the stability of the sub-soil is increased and settlements during and after the construction are reduced. This technique is usually applied on near-saturated soils with a high water table. This section explains the details of modelling vacuum consolidation in PLAXIS 2D.

There are various methods of vacuum consolidation in the real world, but they are all modelled in a similar way in PLAXIS 2D. Most methods in reality are using vertical drains, which are somehow connected at the top to an air pump that reduces the air pressure in the drains until a near-vacuum exists. In practice, a complete vacuum (100 kN/m^2 pressure) is not achievable, but an effective under-pressure of 60 - 90 kN/m^2 is feasible.

Since PLAXIS 2D does not take air pressure into account (atmospheric pressure is assumed to be the zero reference pressure level), a reduction of the groundwater head is used instead to simulate vacuum consolidation. This means that the way vacuum consolidation is modelled leads to negative pore stresses (suction), which are not there in reality.

12.3.1 *Vacuum consolidation in a one-dimensional soil column*

In the simplified case of a one-dimensional soil column, vacuum consolidation can be modelled by performing a groundwater flow calculation or a fully coupled flow-deformation analysis with hydraulic conditions at the model boundaries such that in the vacuum area the groundwater head is prescribed at a level that is 10 m (or less) lower than the vertical coordinate of the global phreatic level. A reduction of the groundwater head of 10 m is equivalent to an under-pressure of 100 kN/m^2 (i.e. complete vacuum).

12.3.2 *Vacuum consolidation in a 2D or 3D model*

In a 2D or 3D numerical model of a realistic project, vacuum consolidation can be modelled by performing a groundwater flow calculation or a fully coupled flow-deformation analysis with vacuum drains (Drain (on page 133)) in which the head specified in those drains is 10 m (or less) lower than the vertical coordinate of the global phreatic level. A reduction of the groundwater head of 10 m is equivalent to an under-pressure of 100 kN/m^2 (i.e. complete vacuum). The distance between the vacuum drains in the model is arbitrary, but should be selected such that the difference in groundwater head in the vacuum area is limited. In general, a distance between the drains less than a quarter of the drain length seems appropriate.

12.3.3 *Other requirements*

A reduction of the groundwater head implies that the soil in the vacuum area becomes unsaturated, whilst this soil volume is supposed to be fully saturated. The user must arrange additionally that saturated conditions apply to this volume. This requires the following settings to be made in the corresponding material data sets:

- The unsaturated unit weight, γ_{unsat} (**General** tabsheet of the **Material data set**), must be set equal to the saturated unit weight, γ_{sat}.
- The hydraulic model must be set to *Saturated* after selecting *User-defined* as hydraulic data set (*Model* group in **Groundwater** tabsheet).

If these settings are not made, the unit weight of the soil will change from saturated to unsaturated as soon as the phreatic level drops as a result of the reduction of the groundwater head in the vacuum drains. Moreover, the soil permeability will reduce according to the reduced relative permeability in the unsaturated zone, depending on the selected hydraulic data set (by default *Fine material*). Both effects are not realistic and can be overcome by making the aforementioned changes in the corresponding material data sets.

12.3.4 *Calculation options*

Vacuum consolidation (using reduced groundwater head boundary conditions or reduced heads in vacuum drains) can be applied in the following calculation types:

- Plastic (select *Steady-state groundwater flow* as *Pore pressure calculation type*)
- Consolidation (select *Steady-state groundwater flow* as *Pore pressure calculation type*)
- *Fully-coupled flow-deformation analysis*

This means that all input requirements for a groundwater flow calculation have to be met, i.e:

- All material data sets must have non-zero permeabilities
- Hydraulic boundary conditions (groundwater head and closed flow boundaries, if applicable) must have been specified

Moreover, it is required to de-select the *Ignore suction* option in the **Deformation control parameters** section of the **Phases** window.

Note that only vacuum drains allow a groundwater head to be specified below the actual drain level, which leads to tensile pore stresses (suction). Normal drains do not allow for suction. Also note that, if vacuum drains are used in a *Consolidation* calculation whilst the pore pressure calculation type is set to *Phreatic*, the drains will work as normal drains rather than vacuum drains. This means that they only affect the consolidation of excess pore pressures, whilst the steady-state pore pressure is fully determined by the global water level and local cluster settings.

12.3.5 *Switching-off vacuum*

If the vacuum is to be 'switched-off' in subsequent calculations while the drains are still supposed to be active for consolidation purposes, the corresponding head in the drains needs to be changed from the reduced head level to the original global water level. This leads to the situation that the new pressure head in the drain is higher than the groundwater head in the area around the drain. As a result, one might expect that water will be flowing from the drain into the ground, which is an artefact of the numerical modelling of vacuum consolidation.

13 References

1. Amorosi, A., Boldini, D., Elia, G. (2010). Parametric study on seismic ground response by finite element modelling. Computers and Geotechnics, 515–528.
2. Andersen, K.A., Puech, A.A., Jardine, R.J. (2013). Cyclic resistant geotechnical design and parameter selection for offshore engineering and other applications. In 18th ICSMGE, Design for cyclic loading: piles and other foundations. Paris, 9–44.
3. Andersen, K.H. (1976). Behaviour of clay subjected to undrained cyclic loading. In Intern. Conf. on the Behaviour of Offshore Structures. Trondheim, Norwen, 392–403.
4. Andersen, K.H. (1991). Foundation design of offshore gravity structures. Blackie & Son Ltd., Glasgow & London.
5. Andersen, K.H. (2004). Cyclic clay data for foundation design of structures subjected to wave loading. International Conf. on Cyclic Behaviour of Soils and Liquefaction Phenomena, 371–387.
6. Andersen, K.H. (2007). Bearing capacity under cyclic loading - offshore, along the coast and on land. In The 21st Bjerrum Lecture. Oslo, Norwen.
7. Andersen, K.H. (2015). Cyclic soil parameters for offshore foundation design. Frontiers in Offshore Geotechnics III, 5–82.
8. Andersen, K.H., Kleven, A., Heien, D. (1988). Cyclic soil data for design of gravity structures. Journal of Geotechnical engineering, 114, 517–539.
9. Aukenthaler, M., Brinkgreve, R.B.J., Haxaire, A. (2016). A practical approach to obtain the soil freezing characteristic curve and the freezing/melting point of a soil-water system. GeoVancouver 2016.
10. Bakker, K.J., Brinkgreve, R.B.J. (1990). The use of hybrid beam elements to model sheet-pile behaviour in two dimensional deformation analysis. Proc. 2nd European Specialty Conference on Numerical Methods in Geotechnical Engineering, 559–572.
11. Bathe, K.J. (1982). Finite element analysis in engineering analysis. Prentice-Hall, New Jersey.
12. Bathe, K.J. (1996). Finite Element Procedures. Prentice Hall.
13. Bauduin, C., Vos, M.D., Simpson, B. (2000). Some considerations on the use of finite element methods in ultimate limit state design. In LSC 2000: International workshop on Limit State Design in Geotechnical Engineering. Melbourne, Australia.
14. Benz, T., Schwab, R., Vermeer, P.A., Kauther, R.A. (2007). A Hoek-Brown criterion with intrinsic material strength factorization. Int. J. of Rock Mechanics and Mining Sci., 45(2), 210–222.
15. Bolton, M.D. (1986). The strength and dilatancy of sands. Geotechnique, 36(1), 65–78.
16. Brinkgreve, R.B.J., Bakker, H.L. (1991). Non-linear finite element analysis of safety factors. In Proc. 7th Int. Conf. on Comp. Methods and Advances in Geomechanics. Cairns, Australia, 1117–1122.
17. Brinkgreve, R.B.J., Kappert, M.H., Bonnier, P.G. (2007). Hysteretic damping in small-strain stiffness model. In Proc. 10th Int. Conf. on Comp. Methods and Advances in Geomechanics. Rhodes, Greece, 737–742.
18. Brinkgreve, R.B.J., Laera, A. (2015). Site response analysis and liquefaction evaluation. Knowledge Base, PLAXIS.
19. Brinkgreve, R.B.J., Post, M. (2013). On the use of finite element models for geotechnical design. Bernessen mit numerischen methoden Workshop(Veroffentlichungen des Institutes, 27), 111–222.

References

20. Burd, H.J., Houlsby, G.T. (1989). Numerical modelling of reinforced unpaved roads. Proc. 3rd Int. Symp. on Numerical Models in Geomechanics, 699–706.

21. Carsel, R.F., Parrish, R.S. (1988). Developing joint probability distributions of soil water retention characteristics. Water Resources Research, 24(5), 755–769.

22. Chopra, A.K. (1995). Dynamics of structures. Theory and application to earthquake engineering.

23. CUR (2004). Geotechnical exchange format for cpt-data. Technical report, CUR.

24. Das, B.M. (1995). Fundamentals of soil dynamics. Elsevier.

25. de Borst, R., Vermeer, P.A. (1984). Possibilities and limitations of finite elements for limit analysis. Geotechnique, 34(20), 199–210.

26. Galavi, V. (2010). Groundwater flow, fully coupled flow deformation and undrained analyses in PLAXIS 2D and 3D. Technical report, Plaxis BV.

27. Goodman, R.E., Taylor, R.L., Brekke, T.L. (1968). A model for mechanics of jointed rock. Journal of the Soil Mechanics and Foundations Division, 94, 19–43.

28. Hashash, Y., Park, D. (2002). Viscous damping formulation and high frequency motion propagation in non-linear site response analysis. Soil dynamic earthquake engineering, 22 (7), 611–624.

29. Hird, C.C., Kwok, C.M. (1989). Finite element studies of interface behaviour in reinforced embankments on soft grounds. Computers and Geotechnics, 8, 111–131.

30. Hoek, E. (2007). Practical Rock Engineering. Rocscience.

31. Hoek, E., Carranza-Torres, C., Corkum, B. (2002). Hoek-brown failure criterion. In Proc. 5th North American Symposium. NARMS-TAC, Toronto, Canada.

32. Hoek, E., Diederichs, M.S. (2006). Empirical estimation of rock mass modulus. International Journal of Rock Mechanics and Mining Sciences, 43(2), 203–215.

33. Hoek, E., Karzulovic, A. (2000). Rock mass properties for surface mines. In W.A. Hustralid, M.K.M.D.J.A. van Zyl (eds.), Slope Stability in Surface Mining, Society of Mining, Metallurgy, and Exploration, Inc., chapter 6.

34. Hudson, M., Idriss, I., Beirkae, M. (1994). QUAD4M User's manual.

35. ISRM (International Society for Rock Mechanics) (1978) Suggested Methods for the Quantitative Description of Discontinuities in Rock Masses. International Journal of Rock Mechanics and Mining Sciences & Geomechanics Abstracts, 15, 319-368.

36. Jostad, H.P., Torgerstrud, Engin, H. (2015). A fe procedure for calculation of fixity of jack-up foundations with skirts using cyclic strain contour diagrams. In The 15th International Conference: The Jack-up Platform. London.

37. Joyner, W.B., Chen, A.T.F. (1975). Calculation of non linear ground response in earthquake. Bulletin of Seismological Society of America, 65, 1315–1336.

38. Lysmer, J., Kuhlmeyer, R.L. (1969). Finite dynamic model for infinite media. Engineering mechanics division ASCE, 95, 859–877.

39. Marinos, P., Hoek, E. (2001). Estimating the mechanical properties of heterogeneous rock masses such as flysh. Bulletin of Engineering Geology, 92,60–85.

40. Matsuishi, M., Endo, T. (1968). Fatigue of metals subjected to varying stress. Japan Society of Mechanical Engineers, Fukuoka, Japan, 68(2), 37–40.

41. Nagtegaal, J.C., Parks, D.M., Rice, J.R. (1974). On numerically accurate finite element solutions in the fully plastic range. Comp. Meth. Appl. Mech. Engng., 4, 153–177.

42. Noren-Cosgriff, K., Jostad, H.P., Madshus, C. (2015). Idealized load composition for determination of cyclic undrained degradation of soils. In Proc., Int. Symp. on Frontiers in Offshore Geotechnics.

43. Owen, D.R.J., Hinton, E. (1982). Finite Elements in Plasticity. Pineridge Press Limited, Swansea.

44. Priest, S.D. (1993) Discontinuity Analysis for Rock Engineering. Chapman & Hall, London.

45. Rheinholdt, W.C., Riks, E. (1986). Solution techniques for non-linear finite element equations. In A.K. Noor, W.D. Pilkey (eds.), State-of-the-art Surveys on Finite Element Techniques, chapter 7.

46. Robertson, P.K., Campanella, R.G., Gillespie, D., Greig, J. (1986). Use of piezometer cone data. In A.S. of Engineers (ASCE) (ed.), Proceedings of the ASCE Specialty Conference In Situ '86: Use of In Situ Tests in Geotechnical Engineering. 1263–80.

47. Rowe, R.K., Ho, S.K. (1988). Application of finite element techniques to the analysis of reinforced soil walls. In P.M. Jarett, A. McGown (eds.), The Application of Polymeric Reinforcement in Soil Retaining Structures. 541–553.

48. Scott MH, Fenves GL. (2010). Krylov Subspace Accelerated Newton Algorithm: Application to Dynamic Progressive Collapse Simulation of Frames. Journal of Structural Engineering 2010; 136(5): 473-480. (17) (PDF) Development of A Robust Nonlinear Solution Algorithm for Structural Analysis.

49. Schank, O., Gärtner, K. (2006). On fast factorization pivoting methods for symmetric indefinite systems. Electronic Transactions on Numerical Analysis, 23, 158–179.

50. Schank, O., Wächter, A., Hagemann, M. (2007). Matching based preprocessing algorithms to the solution of saddle□point problems in large□scale nonconvex interior point optimization. Computational Optimization and Applications, 36 (2-3), 321–341.

51. Schikora, K., Fink, T. (1982). Berechnungsmethoden moderner bergmännischer bauweisen beim u-bahn-bau. Bauingenieur, 57, 193–198.

52. Sloan, S.W. (1981). Numerical analysis of incompressible and plastic solids using finite elements. Ph.D. thesis, University of Cambridge, U.K.

53. Sloan, S.W., Randolph, M.F. (1982). Numerical prediction of collapse loads using finite element methods. Int. J. Num. Analyt. Meth. in Geomech., 6, 47–76.

54. Sluis, J. (2012). Validation of embedded pile row in PLAXIS 2D. Master's thesis, Delft University of Technology.

55. Smith, I.M. (1982). Programming the finite element method with application to geomechanics. John Wiley & Sons, Chichester.

56. Song, E.X. (1990). Elasto-plastic consolidation under steady and cyclic loads. Ph.D. thesis, Delft University of Technology, The Netherlands.

57. van Genuchten, M.T. (1980). A closed-form equation for predicting the hydraulic conductivity of unsaturated soils. Soil science society of America journal, 44(5), 892–898.

58. van Langen, H. (1991). Numerical analysis of soil structure interaction. Ph.D. thesis, Delft University of Technology, The Netherlands.

59. van Langen, H., Vermeer, P.A. (1990). Automatic step size correction for non-associated plasticity problems. Int. J. Num. Meth. Eng., 29, 579–598.

60. van Langen, H., Vermeer, P.A. (1991). Interface elements for singular plasticity points. Int. J. Num. Analyt. Meth. in Geomech., 15, 301–315.

61. Vermeer, P.A., van Langen, H. (1989). Soil collapse computations with finite elements. In Ingenieur-Archive 59. 221–236.

62. Vermeer, P.A., Verruijt, A. (1981). An accuracy condition for consolidation by finite elements. Int. J. for Num. Anal. Met. in Geom., 5, 1–14.

63. Wösten, J.H.M., Lilly, A., Nemes, A., Bas, C.L. (1999). Development and use of a database of hydraulic properties of european soils. Geoderma, 90(3), 169–185.

64. Wösten, J.H.M., Veerman, G.J., DeGroot, W.J.M., Stolte, J. (2001). Soil moisture retention and hydraulic conductivity characteristics of top- and sub-soils in the netherlands: the staring series. Rapport, Alterra, Wageningen.

65. Wyllie, D.C., Mah, C. (2004). Rock slope engineering. CRC Press, Boca Raton.

66. Zienkiewicz, O.C. (1977). The Finite Element Method. McGraw-Hill, London.

67. Zienkiewicz, O.C., Bicanic, N., Shen, F.Q. (1988). Earthquake input definition and the transmitting boundary conditions. Advances in Computational Nonlinear Mechanics I, 109–138.

Appendices

A Import CPT data: required file format [GSE]

A.1 ASCII files

PLAXIS supports CPT data in CPT format CUR (2004) (on page 531). In case the data is not provided in CPT format, PLAXIS enables importing them with the condition that they are saved in properly formatted ASCII files.

The file must contain a small **header** specifying the contained data. If it is not available, the field data cannot be recognised and the importing of the file will be aborted. The header must be the first content in the file in order to prevent that arbitrary data of the file is misinterpreted. The header must have the following format:

```
X[m]                    # optional comment
Y                       # optional comment
Z                       # optional comment
d[m] q f x x u          # optional comment
```

X, Y, Z denote a number specifying the location and elevation of the starting point of the CPT. Missing coordinates are set automatically to zero. Note that PLAXIS does not use this information for modelling purposes.

Following the coordinate specification, specifiers for the column content must be specified. The following specifiers are supported:

d	penetration depth
q	tip resistance
f	sleeve friction
u	pore pressure
x	columns of data to be skipped

Note:

- Except for the 'x' character, column specifiers can occur only once. If a column specifier occurs more than once, the double specifier and all successive specifiers are skipped.
- Units can be introduced inside square brackets ([<unit>]) next to the column specifier. The user can use any combination of units as displayed in Unit conventions (on page 535).
- If units of specifiers are not inserted in the header, PLAXIS will assume that the input units are the same as project units.
- The delimiter used to separate the data columns should be consistent with the one used in the header.

- The first number containing a decimal separator determines which decimal separator is used throughout the import.
  1. 1. or .23 or 1.23 or 1.23E-5 renders using the decimal point for decimal separator.
  2. 1, or ,23 or 1,23 or 1,23E-5 renders using the comma as decimal separator.
  3. 1E-5 or 125 assumes the decimal point for decimal separator, the next number containing a decimal separator decides on the format settings.
  4. Thousand separators are not recognised.
- Comments can be added after '#'

A.2 Unit conventions

If the user define the units of specifiers using brackets (as mentioned in ASCII files (on page 534)), the following units can be adopted:

1. For **Pressure:** [Pa], [kPa], [MPa], [psi], or
2. Any combination of **force/lenght^2**, where force can be [N, kN, MN, lbf, kip] and length can be [mm, cc, m, km, in, ft, yd].

Note: For combined units, the text file must have the following units format: [force/length2]. For instance, [kN/m2].

After the file is loaded PLAXIS will automatically convert the input units into the project units.

A.3 CPT format sample

Based on the specifications showed in the previous sections, a sample of the file format is presented below:

```
200 [m]
500
100
# some comments
d[m]    q[MPa]  f[MPa]  u[Mpa]   #Header
0.01    0.24    0       0.001
0.02    0.53    0       0.001
0.03    0.53    0       0.001
0.04    0.63    0       0.001
0.05    0.68    0       0.001
0.06    0.64    0.009   0.001
0.07    0.74    0.013   0.001
...
.
.
```

B

Cyclic accumulation Tool - Contour File Formats [ADV]

When using the Cyclic accumulation and optimisation tool, custom contours may be uploaded for the contour diagram data. This is the case for both the *Cyclic accumulation* and *Stress-strain curves* (two tabsheets in the tool).

The files that are loaded need to have the format as described below. A number of conditions apply.

B.1 Cyclic accumulation

In the first tabsheet of the *Cyclic accumulation* tool it is required to load contour diagrams, either from the example files provided by NGI or from user-defined files.

The cyclic contours are charts with N_{cycles} on the x-axis and τ_{cyc} / s_u^c on the y-axis. The chart consists of a certain number of curves. Each contour line represents the locus of points characterized by the same shear deformation γ.

1. The first row is always disregarded. It is usually a title.
2. The minimum number of contours is 2. One is always defined at $\gamma = 0\ \%$ for $\tau_{cyc} / s_u^c = 0$ and the other one is defined for another shear strain level γ (which corresponds to the shear strain at failure).
3. The maximum number of contours is 150.
4. The minimum number of points for each contour is 2.
5. The maximum value of N_{cycles} is 10.000.
6. Each contour is determined by a series of data points that are interpolated using linear interpolation.
7. The contours should be digitized along constant N_{cycles} values.
8. The contours data should be presented with increasing N_{cycles} values, decreasing τ_{cyc} / s_u^c ratios and for increasing cyclic shear strain levels.
9. The contours are limited by four curves:
   a. The lower limit is given by the horizontal contour at the base for $\tau_{cyc} / s_u^c = 0$ and $\gamma = 0\ \%$
   b. The upper limit is given by the contour for γ equal to the shear deformation at failure.
   c. All the contours start with a point for N_{cycles} = 1 and an ending point for N_{cycles} equal to the number of cycles above which all the contours become horizontal (i.e. τ is constant). This leads to two vertical boundaries for N_{cycles} = 1 and $N_{cycles} = N_{max}$.

An example of a custom contour file is shown below.

```
Cyclic shear strain contours: DSS                    Title
0.00000   5          The first contour line corresponds to gamma = 0% (first item) and defined by 5 points
0.00000   1
0.00000   10         First contour line:
0.00000   100        - first column = tau/su (y-axis in the contour diagram in the GUI)
0.00000   1000       - second column = Ncycles (x-axis in the contour diagram)
0.00000   10000
0.10000   5          The second contour line corresponds to gamma = 0.1% and is defined by 5 points
0.14732   1
0.14242   10         Second contour line
0.13134   100
0.12337   1000
0.12590   10000
0.25000   5
0.26267   1
0.25280   10         Thirs contour line
0.23073   100
0.21099   1000
0.18705   10000
0.50000   5
0.37270   1
0.36139   10         Fourth contour
0.31770   100        line
0.26993   1000
0.21583   10000
1.00000   5
0.50405   1
0.46822   10
0.37094   100
0.27919   1000
0.22662   10000
3.00000   5
0.68507   1
0.51629   10
0.38513   100
0.28845   1000
0.23022   10000
15.0000   5
0.83060   1
0.55011   10         Last / failure contour line
0.40110   100
0.30079   1000
0.23741   10000
```

Figure 340: Example of a custom contour file

B.2 Stress - Strain Curves

In the second tab of the *Cyclic accumulation and optimisation* tool, it is required to load a contour diagram. If custom contours are loaded, the conditions below need to be met.

Each diagram has the following characteristics:

1. The contours data should be presented with increasing cyclic contour number.
2. The values are given as stress ratio, i.e. the cyclic and average shear strength τ_{cyc} and τ_a are divided by s_u^c both for triaxial and DSS tests.
3. The contours are defined by at least 4 curves for DSS and 5 for TX:
   a. The first cyclic contours should always be given at $\gamma_{cyc} = 0\ \%$ and $\tau_{cyc} / s_u^c = 0$ for both triaxial and DSS diagrams.
   b. For a DSS diagram, the first average contours is given at $\gamma_a = 0\ \%$ and $\tau_{cyc} / s_u^c = 0$
   c. For a DSS diagram, the maximum cyclic and the maximum average contours should be defined for the maximum shear deformation at failure.
   d. For a triaxial diagram, the maximum cyclic contour is defined for γ equal to the cyclic shear deformation at failure, while for the average contours the maximum is defined for the maximum shear deformation at failure for triaxial compression test, and the minimum (negative value) for the maximum shear deformation at failure for triaxial extension.
   e. For TX, the contour corresponding to $\gamma_a = 0\ \%$ should always be given (as the starting point of the path line is at the intersection between $\gamma_{cyc} = 0\ \%$ and $\gamma_a = 0\ \%$)
4. If the behaviour is Anisotropic, define at least 4 contour diagrams:
   a. One TX and one DSS for $N < N_{eq}$
   b. One TX and one DSS for $N > N_{eq}$
5. If the behaviour is Isotropic, the TX contours can be neglected and at least 2 contour diagrams should be given:
   a. One DSS with $N < N_{eq}$
   b. One DSS with $N > N_{eq}$

The structure of the file for the given Drammen clay contours is the following and example charts for DSS and TX tests are shown:

Table 21: File structure of 3D custom contours (Stress-strain curves tab). Example based on Drammen clay.

Line	Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Example	Comments
1	Number of contour diagrams	-	-	-	-	-	8	8 diagrams are defined
2	N-cross section	-	-	-	-	-	1	The following diagram is defined at the cross section of $N_{cycle} = 1$

Line	Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Example	Comments
3	Contour number	Number of average shear strain contours	Number of cyclic shear strain contours	-	-	-	1 11 7	If an odd number is assigned to the contour number, it describes a triaxial diagram, otherwise it refers to a DSS diagram. 11 contour lines are defined for different gamma average values 7 contour lines are defined for different cyclic shear strains
4	Number of cyclic shear strain contours	Number of the average shear contour	Cyclic shear strain (%)	Average shear strain (%)	$\tau_a \,/\, s_u^c$	$\tau_{cyc} \,/\, s_u^c$	1 1 0 -15 -0.5362 0.0000	1 is the number of the cyclic contour and is constant for the next 11 lines. 1 is the number of the average contour and it progresses from 1 to 11. For this block -0.5362 represents the value of $\tau_a \,/\, s_u^c$ at the intersection with the cyclic contour at $\gamma_{cyc} = 0\ \%$.
5							1 2 0 -5 -0.43312 0.0000	
.	.	.	.	.	.	.	.	Lines from 6 to 13
14							1 11 0 15 1.0000 0.0000	This is the last line for the cyclic contour at $\gamma_{cyc} = 0\ \%$. For the triaxial diagram, the maximum $\tau_a \,/\, s_u^c$ is always 1 (since it represents the static strength, i.e. $\tau_a = s_u^c$)

Line	Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Example	Comments
15	Number of the cyclic shear contour	Number of the average shear contour	Cyclic shear strain (%)	Average shear strain (%)	$\tau_a \,/\, s_u^c$	$\tau_{cyc} \,/\, s_u^c$	2 1 0.1 -15 -0.49731 0.13065	This is the first line of the second cyclic contour at $\gamma_{cyc} = 0$ %
after 3 + 11x7 lines, a new block starts to define the DSS test								
81	N-cross section	-	-	-	-	-	1	The following diagram is defined at the cross section of $N_{cycle} = 1$ as before but the following lines contain an even contour number so it describes a DSS chart
	Contour number	Number of average shear strain contours	Number of cyclic shear strain contours				2 7 7	The first item indicates that it is a DSS chart (even number). The second and third item indicate the number of average and cyclic shear strain contours, respectively.
	Number of the cyclic shear contour	Number of the average shear contour	Cyclic shear strain(%)	Average shear strain (%)	$\tau_a \,/\, s_u^c$	$\tau_{cyc} \,/\, s_u^c$	1 1 0 0 0.0000 0.0000	

C

Summary of the creation of General Water Levels

No. of boreholes	Head in boreholes	No. of layers	Layer condition	No. of water levels	Description
1	Value specified	1	Head	1	horizontal water level at the level defined by Head
1	Value specified	1	Hydrostatic[a)]	2	horizontal water level; 1 at the level defined by Head; 1 at the level resulting from p_{top}
1	Value specified	1	Interpolate	1	horizontal water level at the level defined by Head
1	Value specified	1	Dry	1	horizontal water level at the level defined by Head
1	Value specified	1	User-defined	2	horizontal water level; 1 at the level defined by Head; 1 at the level resulting from p_{top} and p_{bottom}
>1	Different values per borehole	1	Head	1	non-horizontal water level
>1	Different values per borehole	1	Hydrostatic	2	1 non-horizontal water level due to specified heads; 1 horizontal layer due to the hydrostatic condition of the soil layer
>1	Different values per borehole	1	Interpolate	1	non-horizontal water level
>1	Different values per borehole	1	Dry	1	non-horizontal water level
>1	Different values per borehole	1[b)]	User-defined	2	1 non-horizontal water level due to specified heads, the water level due to soil layer condition can be either horizontal or non-horizontal depending on the defined conditions.

a) For hydrostatic cases, the situations different from the Head are considered. If the phreatic level in hydrostatic condition is equal to Head, no extra water level will be generated.

b)The number of generated water levels for cases where more than one soil layer is available can be derived accordingly.

D

Conventional and unsaturated soil behaviour in PLAXIS

PLAXIS handles soil as a two-phase material. The classical way of doing this is by considering Terzaghi's definition of effective stress, simply being the total stress minor the pore pressure. The phreatic level separates the saturated soil below from the unsaturated (or 'dry') soil above it. In the classical approach, neither suction nor the degree of saturation are considered in the unsaturated zone. This classical approach can be used in PLAXIS by selecting the calculation option 'Ignore suction', which is by default selected in all calculation types (except in a fully coupled flow-deformation analysis).

Alternatively, PLAXIS may be used to deal with most features of unsaturated soil mechanics. In that case, the option 'Ignore suction' must be disabled. PLAXIS will then consider Bishop's definition of effective stress and it will include suction (negative pore stress) and the degree of saturation in the calculation. In this case, the selection of appropriate hydraulic models for the relationship between suction and saturation as well as the relationship between suction and relative permeability becomes essential.

Before defining the possibilities and limitations of both options, a definition is given of the various stresses and related quantities involved. Note that compression is negative and tension (or suction) is positive.

D.1 *Definition of stresses and related quantities*

Total stresses are divided into effective stresses and pore pressures. Pore pressures are composed of steady-state pore pressures and excess pore pressures. Since decades, PLAXIS 2D uses the term 'active pore pressure' to denote the contribution of the pore stresses in the total stresses. In the context of unsaturated soil behaviour, active pore pressure is defined as:

Active pore pressure: $p_{active} = S_{eff}\,(p_{steady} + p_{excess}) = S_{eff}\,p_{water}$

Pore water pressure: $p_{water} = p_{steady} + p_{excess}$

Note that for fully saturated soil the 'active pore pressure' is equal to the 'pore water pressure', but there is a clear difference in unsaturated zone.

Steady-state pore pressure: p_{steady}

- Input for a deformation analysis
- Direct generation based on phreatic levels and cluster-related pore pressure distribution
- Or, calculated from steady-state groundwater flow calculation

Excess pore pressure: p_{excess}

- Result from undrained behaviour (*Undrained (A)* or *(B)*) and K_w / n from Table 23 (on page 551)and Table 23 (on page 551)

- Affected by loading, a (sudden) change in hydraulic conditions and consolidation.
- In the case of a fully coupled flow-deformation analysis, p_{steady} is calculated from a preliminary steady-state groundwater flow calculation using the hydraulic boundary conditions at the end of the calculation phase. This enables the calculation and output of p_{excess} for a fully coupled analysis in all steps. $p_{excess} = p_{water} - p_{steady}$.

Effective degree of saturation: $S_{eff} = (S-S_{res}) / (S_{sat}-S_{res})$

Degree of saturation: S

Residual degree of saturation: S_{res}

Saturated degree of saturation: S_{sat} (usually 1.0)

- There is a relationship between the (effective) degree of saturation (S or S_{eff}) and the suction in the unsaturated zone, which depends on the selected soil-water retention curve in the material data set.

The term 'suction' is used to denote any positive value the difference between pore air and water pressures: $p_a - p_{water}$.

In PLAXIS, a widely accepted hypothesis of constant atmospheric air pressure is assumed. The air pressure is therefore taken zero since the atmospheric pressure can be taken as a reference value.

Suction is the tension component in p_{water}: $Suction = Max(p_{water}, 0)$

Atmospheric pressure: p_a (100 kPa, but in PLAXIS 2D taken as the zero reference level)

Effective suction: $Suction_{eff} = S_{eff} Suction$

- When multiplied by the tangent of the friction angle, this quantity gives a kind of 'artificial cohesion' in the soil.

Effective stress: σ' (stress in the soil skeleton)

Total stress: $\sigma = \sigma' + p_{active}$

Terzaghi's effective stress: $\sigma' = \sigma - p_{active}$

- Terzaghi's definition of effective stress applies to saturated soils

Bishop' s stress: $\sigma' = \sigma - p_a + \chi (p_a - p_{water})$

Effective stress coefficient: χ

In PLAXIS 2D, χ is supposed to be equal to S_{eff}, so Bishop' s stress in PLAXIS 2D:

$\sigma' = \sigma - S_{eff} \cdot (p_{steady} + p_{excess}) = \sigma - p_{active}$

- When the soil reaches full saturation ($S_{eff} = 1$), Bishop's stress coincides with Terzaghi's effective stress.
- Using Bishop's effective stress as a basis for all calculations, distinction can still be made between conventional calculations (fully saturated or dry materials) and calculations in which unsaturated soil behaviour is taken into account
- The aforementioned distinction can be made by ignoring suction or allowing suction.

In the following, the consequences of ignoring or allowing suction are described.

D.2 *Ignore suction*

When the calculation option *Ignore suction* is selected, soil below the phreatic level is considered to be fully saturated ($S = 1$), whereas soil above the phreatic level is considered to be ideally unsaturated ($S = 0$). The

phreatic level itself is defined directly by the user or calculated as a result of a groundwater flow calculation or a fully coupled flow-deformation analysis. Positive steady-state pore stresses will be set to zero. However, any excess pore pressure above and below the phreatic level, both positive and negative, will be taken into account. This requires S_{eff} to be set to 1, regardless of S, S_{res} and S_{sat}.

When selecting the calculation option *Ignore suction*, the following conditions apply:

- Any positive value of p_{steady} is cut at 0.
- On or below the phreatic level $(p_{steady} \leq 0)$: $S = 1$
- In drained and undrained (A, B) materials: $S_{eff} = 1$ (over-rules previous value)

With this, the stresses and related quantities as defined previously fully apply. In principle, almost all calculation types (*Plastic*, *Consolidation*, *Dynamic*, *Safety*, except Fully coupled flow-deformation), are available, but some specific features or combinations are not available. Limitations are described below:

Note that it is still possible to have suction (pore water tension) as a result of unloading in *Undrained (A,B)* materials. Hence, the option *Ignore suction* will NOT ignore suction as a result of excess pore tension.

As mentioned in Undrained behaviour for unsaturated soil (on page 258), IgnoreSuction, in some cases, may underestimate shear strength and overestimate volume change behaviour. In general, it is difficult to conclude whether this leads to a more or less conservative design. Obviously, the use of Ignoresuction may require less material parameters (especially Soil water-retention curve parameters), but on the other hand, the analysis is less realistic than when suction is considered. The user is advised to keep this in mind with utmost care.

D.3 *Allow suction (i.e. Ignore suction not selected)*

When the calculation option Ignore suction is NOT selected, suction is allowed and included in the active pore pressure and pore water pressure. The saturation of the soil depends on the soil-water retention curve as defined in the corresponding material data sets.

With this, the stresses and related quantities as defined previously fully apply. In principle, all calculation types (*Plastic*, *Consolidation*, *Fully coupled flow-deformation*, *Dynamic*s, *Safety*) are available, but some specific features or combinations are not available. Limitations are described below. (see Possibilities and limitations (on page 546)).

D.4 *Cavitation cut-off*

Positive pore stresses may also occur as a result of unloading in undrained materials. However, there is a limit to the amount of positive pore stresses (pore water tension) that can occur, which is the cavitation stress. This limit can be taken into account in PLAXIS 2D by selecting the *Cavitation cut-off* option and setting the cavitation stress. As a result, the pore water pressure, p_{water}, cannot be higher than the cavitation stress, $p_{cavitation}$. As soon as the cavitation stress is reached, the excess pore pressure is manipulated:

At cavitation: $p_{excess} = p_{cavitation} - p_{steady}$

D.5 *Unit weight of soil*

Although a proper definition of the unit weight of soil would involve the degree of saturation, PLAXIS 2D applies a unit weight based on the current position of the phreatic level in all types of calculations.

- On or below the phreatic level $(p_{steady} \leq 0)$: $\gamma = \gamma_{sat}$

- Above the phreatic level *($p_{steady} > 0$): $\gamma = \gamma_{unsat}$*

For a fully coupled flow-deformation analysis the evaluation of the unit weight is based on p_{water} instead of p_{steady}. This means that during a fully coupled flow-deformation analysis the position of the phreatic level and hence the material weight can change.

D.6 *Possibilities and limitations*

There are limitations in the use of specific features in the various calculations types in PLAXIS 2D depending on whether suction is ignored or allowed. An overview of the limitations is given below.

D.7 *Steady-state groundwater flow* [ADV]

- To generate steady-state pore pressures as input for *Gravity loading*, *Plastic* calculation, *Consolidation* analysis or without deformation analysis.

Drainage type	Zero pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*
Suction	Possible when suction is allowed or in the case of large excess pore tension. Included in active pore pressures, pore water pressures, saturation and relative permeability.
Cavitation cut-off	Not applicable

The following warning will appear when starting the calculation:

Zero permeabilities in some soil clusters	Material set with zero permeability found, but not allowed in this type of calculation.

D.8 *Transient groundwater flow* [ULT]

- To calculate time-dependent pore pressures without deformation analysis.

Drainage type	Zero pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*
Suction	Possible when suction is allowed or in case of large excess pore tension. Included in active pore pressures, pore water pressures, saturation and relative permeability.
Cavitation cut-off	Not applicable

The following warning will appear when starting the calculation:

Zero permeability in some soil clusters	Material set with zero permeability found, but not allowed in this type of calculation.

Zero time interval	Zero time interval

Note: *Transient groundwater flow* is ONLY possible in case of *Flow only* calculation.

D.9 *K0-procedure*

- Only possible as first (initial) calculation phase
- Does not consider external loads; only material weight
- Recommended for situations involving a horizontal ground surface and horizontal soil layering without excavations or structures.

Drainage type	No pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*; no generation of excess pore pressures.
Steady-state groundwater flow [ADV]	Possible, but not recommended since non-hydrostatic pore pressures may lead to non-equilibrium.
Suction	Included in steady-state pore pressures when suction is allowed
Cavitation cut-off	Not possible, but also not relevant; undrained behaviour is ignored
Updated mesh	Not possible (deformations are not considered)
Updated water pressures	Not possible (deformations are not considered)

D.10 *Gravity loading*

- Only possible as first (initial) calculation phase.
- Does not consider *OCR* or *POP*. Initial overconsolidation needs to be 'simulated' using external loads (apply and remove in next phase) or increased *ΣMweight* (reset to 1 in next phase).

Drainage type	No pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*; no generation of excess pore pressures.
Steady-state groundwater flow [ADV]	Possible
Suction	Included in steady-state pore pressures when suction is allowed
Cavitation cut-off	Not possible, but also not relevant; undrained behaviour is ignored
Updated mesh	Not possible
Updated water pressures	Not possible

D.11 *Plastic calculation*

Not possible as first (initial) calculation phase.

Drainage type	No pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*. Excess pore pressures generated in soil clusters of the types *Undrained (A)* and *(B)*, except when using *Ignore undrained behaviour (A,B)*
Steady-state groundwater flow [ADV]	Possible (semi-coupled analysis). Note that in *Undrained (A,B)* soil clusters a change in steady-state pore pressures gives an opposite change in excess pore pressures.
Suction	Possible when suction is allowed or in case of large excess pore tension. Included in active pore pressures and pore water pressures.
Cavitation cut-off	Possible
Updated mesh	Possible when starting from initial phase or when parent phase is also *Updated mesh*
Updated water pressures	Possible when Updated mesh selected (only in 2D)

The following warning will appear when starting the calculation:

Pore pressures are changed and undrained materials are present while *Ignore undrained behaviour (A,B)* is NOT selected	Note that a change in steady-state pore pressures will introduce extra excess pore pressures in *Undrained (A,B)* materials

D.12 *Safety analysis*

Not possible as first (initial) calculation phase.

Drainage type	No pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*. Excess pore pressures generated in soil clusters of the types *Undrained (A)* and *(B)*, except when using *Ignore undrained behaviour (A,B)*
Steady-state groundwater flow [ADV]	Not possible (steady-state pore pressures taken from parent phase)
Suction	Possible when suction is allowed or in case of large excess pore tension. Included in active pore pressures and pore water pressures.
Cavitation cut-off	Possible
Updated mesh	Possible when parent phase is also *Updated mesh*. Note that only deformations from the parent phase are considered and that the mesh is not further updated during the *Safety* analysis.

Updated water pressures	Possible when *Updated mesh* selected. Note that only updated water pressures from the parent phase are considered and that water pressures are not further updated during the *Safety* analysis.

The following warning will appear when starting the calculation:

Using *Updated mesh*	Note that the mesh is NOT further updated during a *Safety* analysis

D.13 *Consolidation analysis* [ADV]

- Not possible as first (initial) calculation phase.
- Considers only consolidation of excess pore pressures.
- Requires non-zero permeabilities.
- Requires non-zero time interval.
- Suction is included in active and pore water pressures but not in permeability, namely relative permeability is always equal to unity.

Drainage type	No consolidation in clusters of the type *Non-porous*; other drainage types have no effect on consolidation; drainage properties are defined by permeabilities, stiffness and time.
Steady-state groundwater flow	Possible (redefines steady-state pore pressures). Note that a change in steady-state pore pressures may initially lead to an opposite change in excess pore pressures, depending on the drainage properties.
Suction	Possible when suction is allowed or in case of large excess pore tension. Included in active pore pressures, pore water pressures, saturation but not in relative permeability. In other words, relative permeability is always taken as unity in Consolidation Analysis. This may induce a quicker dissipation of pore pressure. Numerical convergence is also expected to be faster.
Cavitation cut-off	Not possible
Updated mesh	Possible when parent phase is also *Updated mesh*
Updated water pressures	Possible when *Updated mesh* selected

The following warnings will appear when starting the calculation:

Steady-state pore pressures are changed	Note that a change in steady-state pore pressures will introduce extra excess pore pressures in *Undrained (A,B)* materials
Zero time interval	Zero time interval
Some permeability zero	Zero permeability in some soil clusters

D.14 *Fully coupled flow-deformation analysis* [ULT]

- Not possible as first (initial) calculation phase.
- Considers full coupling between flow and deformation, i.e. 'consolidation' based on total pore water pressures
- Requires non-zero time interval
- Requires non-zero permeabilities
- Requires hydraulic boundary conditions (groundwater head, closed flow)

Drainage type	No flow in clusters of the type Non-porous; other drainage types have no effect on flow; drainage properties are defined by permeabilities, stiffness and time.
Steady-state groundwater flow	Not possible (groundwater flow is automatically included). Steady-state pore pressures at the end of the phase are automatically calculated based on the hydraulic boundary conditions.
Suction	Suction is always allowed. Included in active pore pressures, pore water pressures, water saturation as well as relative permeability. The dissipation of water pressure is expected to be slower than in Consolidation Analysis.
Cavitation cut-off	Not possible
Updated mesh	Not possible. Note that parent phase should NOT be *Updated mesh*
Updated water pressures	Not possible

The following warnings will appear when starting the calculation:

Zero time interval	Zero time interval
Some permeability zero	Zero permeability in some soil clusters

D.15 *Dynamics analysis* [ULT]

- Not possible as first (initial) calculation phase.
- Requires non-zero time interval.

Drainage type	No pore pressures generated in soil clusters of the types *Undrained (C)* and *Non-porous*. Excess pore pressures generated in soil clusters of the types *Undrained (A)* and *(B)*, except when using *Ignore undrained behaviour (A,B)*
Steady-state groundwater flow	Not possible (steady-state pore pressures taken from parent phase)
Suction	Included in steady-state pore pressures when suction is allowed
Cavitation cut-off	Possible

Updated mesh	Possible
Updated water pressures	Not possible

D.16 *Dynamics analysis with Consolidation* [ULT]

Table 22: Bulk modulus of water in Deformation analysis

Type of material	Plastic (ignore undrained)	Plastic
Undrained ($p_{water} \leq 0$)	$K_w = 0$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Undrained ($p_{water} > 0$)	$K_w = 0$	$K_w^{unsat} = \frac{K_w}{1 + \frac{K_w}{S_w}\left(-\frac{\partial}{\partial p_w} S_w\right)}$ *
Drained ($p_{water} \leq 0$)	$K_w = 0$	$K_w = 0$
Drained ($p_{water} > 0$)	$K_w = 0$	$K_w = 0$
Cluster just been activated	$K_w = 0$	$K_w = 0$
Non-porous or dry cluster	$K_w = 0$	$K_w = 0$

(*) $\frac{\partial}{\partial p_w} S_w$ is the derivative of saturation w.r.t pore water pressure (suction) i.e. the slope of water retention curve. As saturation decreases when suction increases, this slope is always negative and equal to zero at full saturation.

Table 23: Bulk modulus of water in Deformation analysis

Type of material	Consolidation and FC ($0 < S < 1$)	Safety or dynamics (drained)	Safety or dynamics (undrained)
Undrained ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$K_w = 0$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Undrained ($p_{water} > 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$ * *	$K_w = 0$	$K_w^{unsat} = \frac{K_w}{1 + \frac{K_w}{S_w}\left(-\frac{\partial}{\partial p_w} S_w\right)}$ *
Drained ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$K_w = 0$	$K_w = 0$

Type of material	Consolidation and FC ($0 < S < 1$)	Safety or dynamics (drained)	Safety or dynamics (undrained)
Drained ($p_{water} > 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$K_w = 0$	$K_w = 0$
Cluster just activated and option 'force drained behaviour newly activated clusters' selected	FE Formulation $K_w = K_w^{sat} \cdot 10^{-8}$	Not relevant	Not relevant
Non-porous or dry cluster	Not relevant	$K_w = 0$	$K_w = 0$

(**)Effect of S taken into account directly in the FE Formulation (Galavi (2010) (on page 531).

Type of Material	Steady - state	Transient
Undrained ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Undrained ($p_{water} > 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Drained ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Drained ($p_{water} > 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$
Cluster just been activated	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$K_w = K_w^{sat} \cdot 10^{-8}$
Non-porous or dry cluster	$K_w = 0$	$K_w = 0$

E

Shortcuts Input program

Key	Action
Ctrl - A	Select all
Ctrl - E	Explode
Ctrl - L	Modify soil layers (only in **Soil** mode)
Ctrl - M	Show materials
Ctrl - N	New Project
Ctrl - O	Open Project
Ctrl - R	Snap to grid
Ctrl - S	Save Project
Ctrl - Y	Redo
Ctrl - Z	Undo
Ctrl - F9	Run commands (GSE)
Shift - Alt - C	Change soil colour intensity
Shift - Ctrl - E	Implode
Shift - Ctrl - S	Save project as
Shift+Ctrl+F9	Opens Command runner, pastes Windows clipboard in commands runner and executes them directly
Shift + Ctrl + N	Starts a new project, opens Command runner, pastes Windows clipboard in command runner and executes them directly
Shift +Ctrl + Alt + <number>	Run an added Macros
Tab	Snap to object while drawing

Shortcuts Input program

Key	Action
Delete	Delete selection
F1	Manuals
F10	Visualisation settings
F12	Toggle the display in the **Phases** window

F

Shortcuts Output Program

Key	Action	View
Ctrl + A	Select all structures of selected type	Model, Structure, Cross section
Ctrl + C	Copy	All
Ctrl + D	Display prescribed displacements	All
Ctrl + E	Explode	Model
Ctrl + F	Display fixities	All
Ctrl + H	Display phreatic level	All
Ctrl + I	Stress points	All
Ctrl + L	Display loads	All
Ctrl + M	Materials	All
Ctrl + N	Nodes	All
Ctrl + O	Open project	All
Ctrl + P	Print	All
Ctrl + R	Reset view	All
Ctrl + S	Save view	All
Ctrl + T	Table	All
Ctrl + W	Shrink	Model
Ctrl + F4	Close window	All
Ctrl + 0	Connectivity plot	All
Ctrl + 1	Deformed mesh	All
Ctrl + 2	Total displacements	All

Shortcuts Output Program

Key	Action	View
Ctrl + 3	Incremental displacements	All
Ctrl + 4	Total strains	All
Ctrl + 5	Incremental strains	All
Ctrl + 6	Plastic points	All
Ctrl + 7	Pore pressures	All
Ctrl + (+)	Move Cross section forward 1/100 of the model size	Cross section
Ctrl + (-)	Move Cross section backward 1/100 of the model size	Cross section
Ctrl + Alt + C	Change soil colour intensity	Model, Cross section
Ctrl + Shift + A	Show all soil elements	Model
Ctrl + Shift + M	Create animation	Model, Cross section, Structure
Ctrl + Shift + N	Hide all soil elements	Model
Ctrl + Shift + Enter	Goes to structure view with selected materials	Model, Cross section, Structure
Ctrl + Shift + (+)	Move Cross section 1/1000 of the model size	Cross section
Ctrl + Shift + (-)	Move Cross section 1/1000 of the model size	Cross section
Escape	Clear selected structures	Model, Cross section, Structure
Shift + Ctrl + E	Implode	Model
Shift + Ctrl + W	Expand	Model
F1	Manuals	All
F2	Curves manager	All
F10	Settings	All

Shortcuts Output Program

Key	Action
Ctrl + A	Select all
Ctrl + F	Find value
Ctrl + M	Jump to maximum value in column
Ctrl + N	Jump to minimum value in column

Key	Action	View
Wheel move	Zoom in/out	All
Press Select structures + Click on structure	Select small group the structure belongs to	Model, Cross section, Structure - Resets current selection
Press Select structures + *Shift* + Click on structure	Toggles selection of small group the structure belongs to	Model, Cross section, Structure - Selection of other structure type will be cleared.
Press Select structures + *Ctrl* + Click on structure	Toggles selection of structure	Model, Cross section, Structure - Selection of other structure type will be cleared
Press Select structures + *Alt* + Click on structure	Toggles selection of large group the structure belongs to	Model, Cross section, Structure - Selection of other structure type will be cleared
Press Hide soil + *Ctrl* + Click on soil	Hides soil element	Model, Forces - Won't work if structure is selected
Press Hide soil + *Ctrl + Shift* + Click on soil	Hides soil cluster	Model, Forces - Won't work if structure is selected

G

Python HTTP REST API wrapper [GSE]

The Python wrapper for the PLAXIS HTTP REST API hides the technicalities of dealing with the HTTP communications behind a comfortable object-oriented wrapper.

G.1 Python HTTP REST API wrapper

Prerequisites

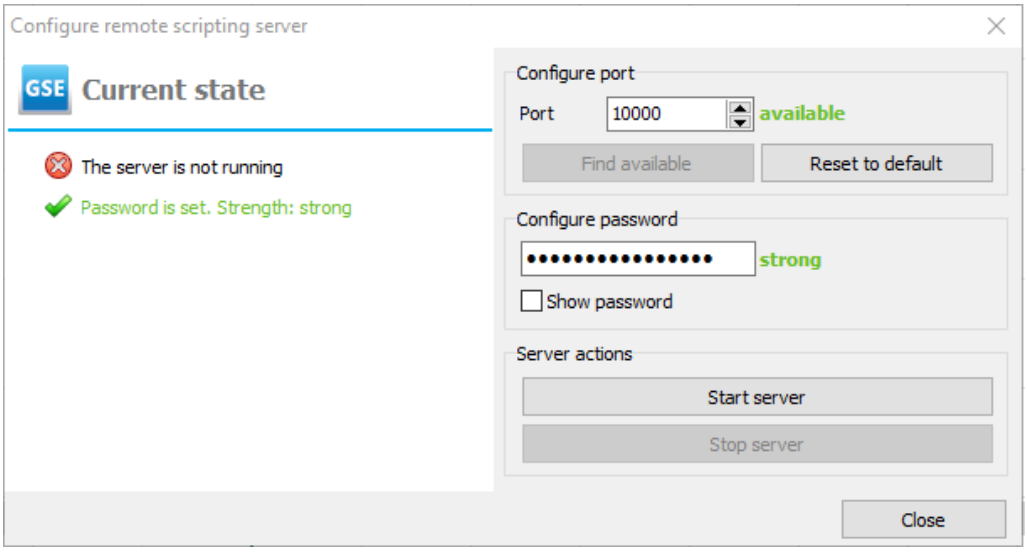
1. An editor for writing Python scripts in. This documentation assumes that the SciTE editor will be used that comes with the PLAXIS installation.
2. At least a very rudimentary knowledge of the Python language is recommended, as this document makes no attempt to teach it. Good resources for this purpose are:
   a. Dive Into Python 3 (Seehttp://diveintopython3.problemsolving.io/)
   b. After Hours Programming http://www.afterhoursprogramming.com/tutorial/Python/Overview/
   c. Think Python: How to Think Like a Computer Scientist http://greenteapress.com/thinkpython/html/index.html
3. Your firewall must not block the PLAXIS application from accessing the internet, nor must it block other applications (in particular the python.exe executable) from talking to the remote scripting server embedded inside the PLAXIS application.

Note: Note that when using Python and other modules, users must do so under the licensing conditions of those modules. The user can check the Python version delivered by PLAXIS by launching the interactive interpreter from the **Expert** menu.

G.2 Starting the remote scripting server

1. Start your PLAXIS application.
2. Start a new project.

3. Go to the **Expert** menu and select the Configure remote scripting server option. The corresponding window pops up. Depending on your machine, the window may look slightly different.



4. Ensure you find an available port and then start the server.
5. A default password will have been generated to ensure a secure connection between the server and the remote scripting interface. You can change this password if you want to, just be aware that leaving the text box empty disables the encryption.
6. Information about ports currently in use can be retrieved in a Windows Command Prompt, with the following commands (the second one may require administrator privileges):

```
netstat -a
netstat -ab
```

When using the remote scripting in an automated process, you can start the server without manual interaction by launching your PLAXIS application with the AppServerPort command line parameter. For example in order to start the server in PLAXIS 2D on port 21403:

```
C:\Program Files\Bentley\Geotechnical\PLAXIS 2D CONNECT Edition V20
--AppServerPassword=mypassword --AppServerPort=21403
```

Changing the password can be done by using the `AppServerPassword` command line parameter. Passing the password this way is optional and it will default to the generated one.

When the server is running, the main window of your PLAXIS application will reflect this information:

G.3 Connecting to a scripting server

Local server (using SciTE)

- Start SciTE from the Windows Start menu. It will be called 'Python editor for PLAXIS 2D . This will present you with an empty document. Let us write a script that will start a new project.
- First we must import the scripting library:

```
from plxscripting.easy import*
```

- Connect to the PLAXIS application (21403 is the port number, substitute yours as applicable):

  ```
  s_i, g_i = new_server('localhost', 21403, password='yourpassword')
  ```

  The variable s_i is bound to an object representing the PLAXIS application. It is now possible to control that from the interactive Python prompt.

- Start a new project as follows:

  ```
  >>> s_i.new()
  ```

  Save this script somewhere on your hard drive. Make sure it ends with .py so SciTE correctly identifies it as a Python script. Now we are able to run this script by going to Tools and then clicking on Go or by pressing F5
- A new project should now be visible in the PLAXIS application. If instead an error is returned, check the following for the PLAXIS application you are trying to control:
  - Is the application running?
  - Does its title bar indicate that the server is active?
  - Is the server active on the port you have specified when calling `new_server`? You can also try a different port.
  - Is your firewall blocking communications to the application?
  - Is the password given to `new_server` the same as the one in the 'configure remote scripting server' dialog?

Remote PLAXIS application

Connecting to a PLAXIS application running on a computer other than your own requires you to know the name or IP address of that machine. The IP address can be found by typing in a Windows command prompt the following:

```
ipconfig | find "IPv4"
```

The result will give a set of four numbers separated by dots. Example:

```
IPv4 Address. . . . . . . . . . . : 10.4.2.178
```

In order to connect to this machine from within the same network, you can simply replace `'localhost'` in the boilerplate

connection code with the IP address you found, such as `'10.4.2.178'`. For example:

```
>>> s_i, g_i = new_server('10.4.2.178', 21403, password='yourpassword')
```

If you want to connect to the same machine over the internet, you will need to set up appropriate port forwarding rules in your router. Note that opening up your computer to access from the network or even the internet carries significant security risks - do so only if you have considered these risks.

G.4 Creating standalone scripts

In the examples above, we have looked at possibilities for using the API interactively from a Python prompt. While this is very useful for testing and finding out how things work, soon you will need the ability to save code and run it again without having to retype it.

In the SciTE editor make a new text file by going to File > New or by pressing Ctrl+N. After that, you can type the code, save and run it using Tools >Go. A Python script equivalent to the interactive session presented above looks as follows:

```
from plxscripting.easy import *

s_i, g_i = new_server('localhost', 21403, password='yourpassword')
s_i.new()

g_i.SoilContour.initializerectangular(0, 0, 10, 10)
g_i.borehole(0)
g_i.soillayer(10)
material = g_i.soilmat()
material.setproperties("SoilModel", 1, "gammaUnsat", 16,
"gammaSat", 20, "Gref", 10000)
g_i.Soils[0].Material = material

g_i.gotostructures()
g_i.lineload((3, 0), (7, 0))

g_i.gotomesh()
g_i.mesh(0.2)

output_port = g_i.selectmeshpoints()
s_o, g_o = new_server('localhost', output_port, password='yourpassword')
g_o.addcurvepoint('node', g_o.Soil_1_1, (5, 0))
g_o.update()

g_i.gotostages()
phase1 = g_i.phase(g_i.Phases[0])
g_i.LineLoads[0].Active[phase1] = True
g_i.calculate()

output_port = g_i.view(phase1)
s_o, g_o = new_server('localhost', output_port, password='yourpassword')
#In newer versions of the scripting layer it is possible to just write:
g_o.CurvePoints.Nodes[0]
utot = g_o.getcurveresults(g_o.CurvePoints.Nodes.value[0], g_o.Phases[1],
g_o.ResultTypes.Soil.Utot)

print(utot)

g_i.save(r'c:\data\scripting_sample')
```

G.5 Calling new_server without arguments

It is possible to call `new_server` without any arguments, but in that case a script can only be executed from the Expert menu in PLAXIS. Either choose Expert > Python > Run Script > Open to run your script or put it in one of the pytools folders and choose Expert > Python > Run Script > Tools.

G.6 Disabling the user interface

For some projects the overhead of updating the user interface may slow down the execution of the script quite significantly. If you want maximum execution speed, launch your PLAXISapplication with the command parameter `--NO_CONTROLLERS`. For example:

```
Plaxis2DXInput.exe --AppServerPort=21403 --NO_CONTROLLERS
```

The downside of this approach is that you will be unable to monitor progress or diagnose problems visually. Instead, you will have to:

1. Stop the running script
2. Save the current project
3. Restart the application and open the saved project

G.7 Identifying the type and version of PLAXIS

If there is a need to write codes that are usable under both programs PLAXIS 2D and PLAXIS 3D, it is possible to check the type and version of PLAXIS software. With `s` being a server-type object as returned by the `new_server(...)` method, the following code must be valid:

```
>>> print(s.name, s.major_version, s.minor_version, s.is_2d, s.is_3d)
PLAXIS 2D 2018 0 True False
>>> if s.major_version + s.minor_version / 10 > 2020: # i.e. both are numbers
...    print('We come from the future')
We come from the future
>>> from .const import PLAXIS_2D, PLAXIS_3D
>>> if s.name == PLAXIS_2D:
...     print('This is PLAXIS 2D')
... elif s.is_3d: # indicates whether the application is 2D or 3D *capable*
                  # (not whether it's the 2D or 3D *product*; 2D MPM will have
is_2d=True!)
...     print('This is 3D or 3D MPM')
... else:
...     print('Unknown application!')
This is PLAXIS 2D
```

The following also work:

```
from easy import * # must implicitly import the app type constants
s_i, g_i = new_server('localhost', 10000)
if s_i.name == PLAXIS_3D:
   print('Is PLAXIS 3D')
```

Drawing a rectangular/prismatic soil cluster in either 2D or 3D

```
from plxscripting.easy import new_server
from math import isnan, nan

s_i, g_i = new_server() # works when launched from Expert menu
```

```
def get_number(msg):
   nr = nan
   while isnan(nr):
       try:
           nr = float(input(msg + ': '))
       except:
           pass
   return nr

width = get_number('Width')
height = get_number('Height')

if s_i.is_3d:
    # get the depth of the cluster
    depth = get_number('Depth')
    make_coords = lambda x, z: (x, 0, z)
else:
    make_coords = lambda x, y: (x, y)


coords = [(0, 0), (width, 0), (width, height), (0, height)]
result = g_i.polygon(*[make_coords(*pos) for pos in coords])[0]

if s_i.is_3d: # perform the extrusion and remove original polygon
    volume = g_i.extrude(result, (0, depth, 0))[0]
    g_i.delete(result)
    result = volume

print('Created cluster: {}'.format(result.Name))
```

G.8 Creating a simple project

All commands that are available in the PLAXIS command line are also available via the Python wrapper. For more information, see the commands reference provided with your PLAXIS application. Note that private commands in the application (i.e. commands starting with a double underscore, __) are spelled without the double underscore when used from Python. Commands that conflict with Python keywords should be spelled with an extra trailing underscore in Python (e.g. `import` becomes `import_`). Identifiers containing characters that are not part of the ASCII character set will be stripped of those characters (e.g. °C becomes C).

- Start by connecting from Python to PLAXIS 2D as described above.
- Make sure that the folder where the scripting sample will be saved does exist (`c:\data\`).
- Execute the code displayed below. For the sake of completeness, the replies returned by the different statements are included, but some of them are session-specific and will be different for you. It is helpful while executing the Python code to keep an eye on the command line in the PLAXIS application, in order to gain an understanding of how Python code maps to commands. Keep also in mind that Python, unlike the PLAXIS command line, is case sensitive.

```
>>> s_i.new()
'OK'
>>> g_i.SoilContour.initializerectangular(0, 0, 10, 10)
'OK'
>>> g_i.borehole(0)
<Borehole {BB449552-2ABF-408D-8293-897B5939D764}>
```

```
>>> g_i.soillayer(10)
<GeneratedSoilVolume {859D7DB5-585E-4452-8C38-E700C398EF9C}>
>>> material = g_i.soilmat()
>>> material.setproperties("SoilModel", 1, "gammaUnsat", 16,
"gammaSat", 20, "Gref", 10000)
'Edited SoilMat_1'
>>> g_i.Soils[0].Material = material
>>> g_i.gotostructures()
'OK'
>>> g_i.lineload((3, 0), (7, 0))
[<Point {AD9DB344-A82E-4ACD-93BB-A3C02369556B}>, <Point {9E140226-A968-46A1-
A2EA-5125EE83968F}>, <Line {F1E1AE01-C72A-44CD-A647-8EC39CA024D7}>, <LineLoad
{B3A5244F-3097-44CC-8E3E-2290AC4508B0}>]
>>> g_i.gotomesh()
'OK'
>>> g_i.mesh(0.2)
Generated 181 elements, 374 nodes
>>> output_port = g_i.selectmeshpoints()
>>> s_o, g_o = new_server('localhost' , output_port, password="yourpassword")
>>> g_o.addcurvepoint('node' , g_o.Soil_1_1, (5, 0))
'OK'
>>> g_o.update()
'OK'
>>> g_i.gotostages()
'OK'
>>> phase1 = g_i.phase(g_i.Phases[0])
'OK'
>>> g_i.LineLoads[0].Active[phase1] = True
>>> g_i.calculate()
'OK'
>>> output_port = g_i.view(phase1)
>>> s_o, g_o = new_server ('localhost', output_port, password="yourpassword")
>>> utot = g_o.getcurveresults(g_o.Curvepoints.Nodes.value[0],
g_o.Phases[1],g_o.ResultTypes.Soil.Utot)
>>> print(utot)
8.76955286493468E-5
>>> g_i.save(r'c:\data\scripting_sample')
'Project saved as: c:\\data\\scripting_sample.p2dx'
```

- The command line history on the application side looks as follows (excluding the commands sent to Output):

```
0001> initializerectangular SoilContour 0 0 10 10
0002> borehole 0
0003> soillayer 10
0004> soilmat
0005> setproperties SoilMat_1 "SoilModel" 1 "gammaUnsat" 16
"gammaSat" 20 "Gref" 10000
0006> set Soil_1.Material SoilMat_1
0007> gotostructures
0008> lineload (3 0) (7 0)
0009> gotomesh
0010> mesh 0.2
0011> selectmeshpoints
0012> gotostages
0013> phase InitialPhase
0014> set LineLoad_1_1.Active Phase_1 True
0015> calculate
0016> view Phase_1
0017> save "c:\data\scripting_sample"
```

G.9 Performance and logging

By default the server object (typically referred to as `s` in these examples) implements a caching system to reduce the number of calls by the script to the PLAXIS application, thereby improving the overall performance of the system. It can be switched off using:

```
s_i.allow_caching = False
```

Additionally, a logging system is available in order to monitor the communications between the Python script and the PLAXIS application. It can be toggled on or off using the `enable_logging()` and `disable_logging()` methods of the server object:

```
s_i.enable_logging() # will write to TEMP\PlaxisScriptLogs\
s_i.disable_logging()
```

G.10 External manipulations during script execution

While using Python scripts to manipulate the state of your PLAXIS application, it is technically possible to interact with the same application in other ways as well (e.g. by changing a project manually, or firing commands separately in an interpreter while the main script is paused). This is very useful for troubleshooting.

In particular situations it may however lead to problems, in that internal data structures built on the Python side have become obsolete due to actions outside of the script. The main culprit is (re)opening a project while the script already has some internal state/variables. Errors can look like this:

```
plxscripting.plx_scripting_exceptions.PlxScriptingError: Unsuccessful command:
GUID does not refer to object in registry: {F00F9FB0-B903-42BE-AE1F-C7B8E200885D}
```

If you encounter such an error, simply stop the execution of your script and restart it.

G.11 Error handling mode

When the server is started, it is possible to optionally set an error handling mode by using the `error_mode` option. The `error_mode` consists of a behaviour and optionally one or more modifiers.

The `error_mode` option influences the behaviour of in-request exceptions, as well as out-of-request exceptions. In-request exceptions are raised during execution of a request and out-of-request exceptions occur outside an HTTP request. An example of the latter is if repainting the UI leads to a crash after the command has been executed and returned to the scripting client.

The `error_mode` is always a tuple consisting of none, one or more of:

1. Behaviour **RAISE** [default; empty error_mode means RAISE]: raises an exception.
2. Behaviour **INTERPRETER**: upon exception, fires an interactive Python interpreter for troubleshooting, if the server is not currently in an interactive interpreter mode already.
3. Modifier **RETRY**: tries the same API call again with a delay of 200 ms.

4. Modifier **PRECONDITION**: checks (and clears) the exception stack prior to making another API call.
5. Modifier **NOCLEAR**: ensures that when an out-of-request exception is retrieved, it is not cleared from the server.

If the user specifies `RAISE, INTERPRETER` as combo behaviour in the `new_server` call, `INTERPRETER` wins.

Examples

1. Setting the error handling mode:

```
s_i, g_i = new_server('localhost', 10000,
          error_mode=(INTERPRETER, RETRY))
```

2. Manual API calls

   It's useful to be able to manually request information about out-of-request failure state, particularly when using `NOCLEAR` or using the interactive interpreter to troubleshoot issues.

```
# raise a Python exception including the cleaned madExcept info, because this is an
in-request failure
        s_i.raise_()

        # triggers out-of-request exception, retrievable using get_error, but does
not crash immediately
        s.raiseasync()

        # give the raiseasync time to work
        import time; time.sleep(0.5)

        # None if no exception, exception string otherwise
        s.get_error(clear=False)

        # as above, but does clear the exception information from the server
        s.get_error(clear=True)

        # equivalent to clear=True
        s.get_error()
```

G.12 Installing third-party Python modules

When the user wants to install a third-party package that is not already available in the PLAXIS Python distribution we deliver, then you can do so by opening the command prompt from **Expert** > **Python** > **Command prompt** in PLAXIS Input or Output program.

As this requires administrator rights on your PC, you first need to launch PLAXIS Input or Output as Administrator. This can be done by click with RMB on the PLAXIS shortcut and selecting Run as Administrator.

In the Command prompt, you can install new packages by using pip. For example, if you want to install the dropbox package you would type:

For example, if you want to install the dropbox package you would type:

```
python -m pip install dropbox
```

If you want to install the cffi (C foreign function interface) package you would type:

```
python -m pip install cffi
```

G.13 Tokenizer

The Python API allows parsing a command line string into the separate tokens.

- The server object has a `tokenize` method. This method returns a `Tokenizer` object. This object may be in a fully valid state, or a partially valid state. It always contains one list of `Token` objects, but depending on the success of the action, this list may be available only as `Tokenizer.partial_tokens` or as both `Tokenizer.tokens`and `Tokenizer.partial_tokens` simultaneously - see the example below for more information.

The list of tokens contains `Token` objects.

`partial_tokens` contains the successfully parsed tokens, which may or may not be all tokens in the string.

- There are subclasses of the `Token` class to match the types of tokens from the HTTP REST API, see HTTP REST API [GSE] (on page 584)
- The `Token` classes have properties as specified for the HTTP REST API (see HTTP REST API [GSE] (on page 584)). The only exception is the type property: this one will be absent in Python, since it is implicitly encoded in the `Token` subclass.

Examples

```
from plxscripting.easy import *
s, g = new_server("localhost", 10000)

tokenizer = s.tokenize("addpoint Polygon_2 0 4 0 # comment")
print(tokenizer.success) # prints True
print(tokenizer.error_position) # prints -1
print(tokenizer.tokens == tokenizer.partial_tokens) # prints True

tokens = tokenizer.tokens

print(len(tokens)) # prints 6

token = tokens[0]
print(token.position) # prints 0
print(token.length) # prints 8
print(token.end_position) # prints 7

print(token.__class__) # prints <class 'plxscripting.tokenizer.TokenIdentifier'>
                       # The various Token classes mirror the ones of HTTP REST API
print(token.value) # prints 'addpoint' (w/o quotes)
print(token) # prints plxscripting.tokenizer.TokenIdentifier
             # ({'type': 'identifier', 'value': 'addpoint', 'length': 8, 'position':
1})

print(tokens[2].__class__) # prints <class plxscripting.tokenizer.TokenInteger'>
print(tokens[2].value) # prints 0

print(tokens[-1].value) # prints '# comment'
print(tokens[-1].content) # prints ' comment' (with leading space, w/o quotes)

for token in tokens:
```

```
    print(token) # prints all the tokens

# error conditions
tokenizer = s.tokenize("abc ?")
tokens = tokenizer.tokens # raises an exception about Unrecognized token
print(tokenizer.success) # prints False
print(tokenizer.error) # prints 'Unrecognized token' (w/o quotes)
print(tokenizer.error_position) # prints 4
print(len(tokenizer.partial_tokens)) # prints 1, since the first token is OK
```

G.14 Sample Applications

This section contains a few simple tasks that highlight different aspects of the Python scripting wrapper. It is assumed PLAXIS 2D is running and the necessary connection boilerplate code has been executed already.

Opening a project, removing all lines containing plates and saving again

```
s_i.open(r'c:\data\lines_with_plates.p2dx')
lines = [b.Parent for b in g_i.Plates]
g_i.delete(*lines)
# alternatively (look at the difference in the command line):
# g_i.delete(lines)
g_i.save()
```

Reporting all points that are not used by lines and that have no features

```
points_in_lines = set()
for line in g_i.Lines[:]:
    for point in line:
        points_in_lines.add(point)
        points = set(g_i.Points)
        points_no_lines = points - points_in_lines
        points_to_report = []
        for p in points_no_lines:
            if len(p.UserFeatures.value) == 0:
                points_to_report.append(p)
        for p in points_to_report:
            print(p.Name)
```

Creating an array, storing the results and performing actions on the diagonal

```
# Code will work in any project, as it operates only on its own local results,
regardless of what else is present in the project.
g_i.gotostructures()
point = g_i.point(0, 0)
added_points = g_i.arrayr(point, 6, (1, 0), 6, (0, 1))
diagonal_points = []
for p in added_points:
    if abs(p.x - p.y) < 1E-4
        diagonal_points.append(p)
```

```
g_i.pointload(diagonal_points)
```

Lengthening all the embedded beam rows in a model

```
# Code below can be used to quickly generate a test model
## res = g_i.embeddedbeam((0, 0), (0, -15))
## line = [item for item in res if item._plx_type=='Line'][0]
## g_i.arrayr(line, 5, (1, 0))

def get_bottom_point(line):
        first, second = line.First, line.Second
        delta = second.y - first.y
    if abs(delta) < 1E-4: # horizontal line
        return None
    if delta > 0: # Second point is higher -> move First down
        return first
    else: # First point is higher -> move Second down
        return second

def lengthen_embedded_beams(extra_depth):
    # start by selecting the lines which have embedded beam rows
    pile_lines = [pile.Parent for pile in g_i.EmbeddedBeams[:]]

    # we don't know whether the line is drawn top-to-bottom
    # or bottom-to-top, so find out which point is lowest
    points_to_move = []
    for line in pile_lines:
        bottom_point = get_bottom_point(line)
        if bottom_point:
            points_to_move.append(bottom_point)

    g_i.move(points_to_move, (0, -extra_depth))

lengthen_embedded_beams(2.5)
```

Making all plates in all phases have identical settings in staged construction

```
    refplate = g_i.Plates[0]
    otherplates = g_i.Plates[1:]
    for phase in g_i.Phases:
        if refplate.Active[phase] is None: # phase not yet initialized
            continue
        for otherplate in otherplates:
            otherplate.Material[phase] = refplate.Material[phase]
            otherplate.Active[phase] = refplate.Active[phase]
```

Create safety phase after every consolidation phase

```
for phase in g_i.Phases[:]:
    if phase.DeformCalcType == phase.DeformCalcType.consolidation:
        newphase = g_i.phase(phase)
        newphase.DeformCalcType = newphase.DeformCalcType.safety
```

Send an email using Gmail after calculations are finished

```
# A Gmail account is needed for this. Google Apps for Business accounts
# should work as well.
import smtplib
from email.mime.multipart import MIMEMultipart
from email.mime.text import MIMEText

def opensession_gmail(username, password):
    # Courtesy of http://segfault.in/2010/12/sending-gmail-from-python/
    # Other email providers may require a different way of connecting.
    # It may be necessary to enable access for less secure apps when
    # using GMail, see https://www.google.com/settings/security/lesssecureapps
    session = smtplib.SMTP('smtp.gmail.com', 587)
    session.ehlo()
    session.starttls()
    session.ehlo()
    session.login(username, password)
    return session

def sendmail(smtp_session, sender_address,
             to_address, subject, body):
    msg = MIMEMultipart()
    msg['From'] = sender_address
    msg['To'] = to_address
    msg['Subject'] = subject
    part = MIMEText('text', 'plain')
    part.set_payload(body)
    msg.attach(part)
    smtp_session.sendmail(sender_address, to_address, msg.as_string())

def report_phases_by_kind(target_report, phases,
                          kind, report_func):
    if phases:
        target_report.append(kind)
        target_report.append('=' * len(kind))
        for phase in phases:
            target_report.append(report_func(phase))
        target_report.append('')

def phase_to_string(phase):
    return "{} [{}]".format(phase.Identification, phase.Name)

def phase_to_string_with_error_code(phase):
    return "{} [{}]. LogInfo: {}".format(
        phase.Identification, phase.Name, phase.LogInfo)

def report_phases(phases_list):
    successful_phases = []
    failed_phases = []
    uncalculated_phases = []
    for phase in phases_list:
        if phase.ShouldCalculate:
            uncalculated_phases.append(phase)
        elif phase.LogInfo == '0':
            successful_phases.append(phase)
```

```
        else:
            failed_phases.append(phase)
    report_lines = []
    report_phases_by_kind(report_lines, successful_phases,
                          'Successful', phase_to_string)
    report_phases_by_kind(report_lines, failed_phases,
                          'Failed', phase_to_string_with_error_code)
    report_phases_by_kind(report_lines, uncalculated_phases,
                          'Skipped', phase_to_string)

    return '\n'.join(report_lines)

def calculate_with_report():
    phases_to_calculate = []

    for phase in g_i.Phases[:]:
        if phase.ShouldCalculate:
            phases_to_calculate.append(phase)

    res = g_i.calculate()
    report = [
        'Title: {}'.format(g_i.Project.Title),
        'Location: {}'.format(g_i.Project.Filename),
        '',
        report_phases(phases_to_calculate),
        '',
        'Command feedback:',
        res]

    return '\n'.join(report)

message = calculate_with_report()

# change the account information below for your own account
username = 'your.name@gmail.com'
password = 'abc123'
to_address = username # sending to ourselves, but can specify something else too

session = opensession_gmail(username, password)
sendmail(session, username, to_address,
        'Plaxis calculation report',
        message)
```

Executing commands specified in a file

```
# For debugging particular problems it may be interesting to execute
# commands directly (effectively bypassing the Python object wrapper)
# and parse feedback on demand. It may also have some performance
# benefits as the communication with the server is reduced. This way
# of communicating with the application also allows you to do your
# own error handling. The code below demonstrates several approaches
# to this task, examine the data that's printed out to better
# understand the differences.

# Code below can be used to quickly generate a test model
## from tempfile import NamedTemporaryFile
```

```
## import os
## f = NamedTemporaryFile(mode='w', delete=False)
## f.write('\n'.join(['bh 0', 'sl 2', 'plate 0 0 1 1']))
## filename = f.name

f = open(filename, 'r') # filename should be defined in advance
commands = [line.rstrip() for line in f.readlines()]
f.close()

# If you used the quickly generated test model, also enable
# the cleaning below:
## os.remove(filename)

# Run one command at a time and have the wrapper handle the response
# for you - not much extra control.
for command in commands:
    print(s_i.call_and_handle_command(command))

# Run multiple commands at a time and have the wrapper handle
# the responses for you. There is less communication with the
# server than when executing one command at a time so it's
# potentially faster.
print(s_i.call_and_handle_commands(*commands))

# Run one command at a time and handle the response yourself. The
# response now consists of Python primitives representing the
# JSON communication rather than comfortable objects.
for command in commands:
    print(s_i.call_commands(command))

# Run multiple commands at a time and handle the responses yourself.
print(s_i.call_commands(*commands))

# call_commands will not lead to an exception in case of invalid
# commands, unlike the other method calls presented.
print(s_i.call_commands('this is an invalid command'))
```

Creating a polar array of inclined piles

```
from math import radians, sin, cos

# input parameters
nr_pilerows = 4 # must be > 1
total_angle_deg = 180 # over which to distribute the piles, must be > 0 and <= 360
pilerow_length = 10
pilerow_inclination_deg = 15
array_center = (5, 0)
array_radius = 10

# calculations
def rel_coords_to_absolute(rel_coords, radius, offset):
    coords = [radius * coord for coord in rel_coords]
    return [p[0] + p[1] for p in zip(coords, offset)]

if total_angle_deg == 360: # fencepost problem -> decrease angle
    total_angle_deg = total_angle_deg / (nr_pilerows) * (nr_pilerows - 1)
```

```
pilerow_inclination_rad = radians(pilerow_inclination_deg)
array_bottom_radius = array_radius + \
                      pilerow_length * sin(pilerow_inclination_rad)
array_bottom_center = (array_center[0], array_center[1] - pilerow_length *
cos(pilerow_inclination_rad))
pilerow_coords = []
for i in range(nr_pilerows):
    angle = radians(total_angle_deg * i / (nr_pilerows - 1))
    rel_coords = [cos(angle), sin(angle), 0]
    top_coords = rel_coords_to_absolute(
                rel_coords, array_radius, array_center)
    bottom_coords = rel_coords_to_absolute(
                rel_coords, array_bottom_radius, array_bottom_center)
    pilerow_coords.append((top_coords, bottom_coords))

    for coords_pair in pilerow_coords:
        g_i.embeddedpilerow(*coords_pair)
```

Convert 2D to 3D

```
# This is a limited converter that will only work for a small subset of potential 2D
geometry configurations. We need to be careful about which environment variables
# live in (3D will not be able to accept as input objects that live in the 2D
environment and the other way around).
# For this reason we will append a 2 or 3 to each variable in order to indicate where
the object lives. Keep in mind that for this case both 2D and 3D need to
# be running simultaneously, and they must open different ports for the scripting
engine. Depending on which versions are in use, it may also happen that the scripting
# API is incompatible between the two, so additional effort will need to be undertaken
to load different versions of the API for 2D and 3D.

# connect to 2D
s2, g2 = new_server('localhost', 10002, password='yourpassword')

# Code below can be used to quickly build a 2D project that
# we can use for conversion
## s2.new()
## g2.gotostructures()
## g2.plate((2, 5), (2, 2))
## g2.plate((6, 5), (6, 2))
## g2.polygon((0, 0), (8, 0), (8, 5), (0, 5))

# connect to 3D
s3, g3 = new_server('localhost', 10003, password='yourpassword')

# In 3D we need to specify a particular slice thickness
while True:
    try:
        depth = float(input('Specify non-zero 3D slice thickness> '))
        if depth != 0.0:
            break
        else:
            print('Must be non-zero!')
    except:
        print('Invalid input, try again!')
```

```
        pass

s3.new()
g3.gotostructures()

# convert each line to a polygon (map y to z!)
for line2 in g2.Lines:
# Need to dereference the values, because the intrinsic
# properties live in 2D, cannot send them over to 3D!
pointcoords = [
    (line2.First.x.value, 0.0, line2.First.y.value),
    (line2.Second.x.value, 0.0, line2.Second.y.value),
    (line2.First.x.value, depth, line2.First.y.value),
    (line2.Second.x.value, depth, line2.Second.y.value)
    ]
surface3 = g3.surface(
    pointcoords[0], pointcoords[1],
    pointcoords[3], pointcoords[2])
# Limitations
# - not all 2D line features become surface
#   feature in 3D (e.g. N2N anchors).
# - features may depend on normal orientation
#   (e.g. interfaces)
for uf2 in line2.UserFeatures.value:
    # shorthand: assume that the command == type name,
    # which is not necessarily true for all user features
    ufconstructor 3 = g3.__getattr__(
        uf2._plx_type.lower())
    # add the feature
    ufconstructor3(surface3)

# convert each polygon to a volume
for polygon2 in g2.Polygons:
    # the addition operation on numerical intrinsic properties
    # automatically dereferences their value
    pointcoords = [(polygon2.x + p2.x, 0.0, polygon2.y + p2.y)
                   for p2 in polygon2.Points.value]
    surface3 = g3.surface(*pointcoords)
    # use extrusion to create a volume
    volume3 = g3.extrude(surface3, (0.0, depth, 0.0))
    # remove the original surface
    g3.delete(surface3)
```

Using both Input and Output at the same time

```
# Sometimes you also want to automate the selection of history points
# or the retrival of results in Output.
s_i, g_i = new_server('localhost', 10000, password='yourpassword')
s_i.open(r'C:\Users\x\Document\PlaxisProjects\some_existing_project.p2dx')

# Calling the view command while Input has a running server will also start
# Output with a server and return the port that the new server uses.
# The commands preview, view and viewmesh will behave similarly.
output_port = g_i.selectmeshpoints()
# Connect to the Output server the same way we connect to the Input one.
s_o, g_o = new_server('localhost', output_port, password='yourpassword')
```

```
# Select a history point and close Output with the update command.
g_o.addcurvepoint('node', g_o.Soil_1_1, 0, 0)
g_o.update()
```

Computing a derived result from existing result types

This example shows how to compute a derived result starting from standard result types. This code requires existing results to be processed.

```
import math

from plxscripting.easy import new_server

s_i, g_i = new_server('localhost', 10000, password='yourpassword')

last_phase = g_i.Phases[-1]
ux = g_i.getresults(last_phase, g_i.ResultTypes.Soil.Ux, 'node')
uy = g_i.getresults(last_phase, g_i.ResultTypes.Soil.Uy, 'node')

displacement_angles = [math.atan2(y, x) for x, y in zip(ux, uy)]

print(displacement_angles)
```

Finding the local maximum displacement from existing result types

This example shows how to find the local maximum displacement using the *getsingleresult* command. This code requires existing results to be processed.

```
import math

from plxscripting.easy import new_server

s_i, g_i = new_server('localhost', 10000, password='yourpassword')

step = 0.1
last_phase = g_i.Phases[-1]
result_type = g_i.ResultTypes.Soil.Utot

def get_displacement(coordinate):
    result = g_i.getsingleresult(last_phase, result_type, *coordinate)
    if result == 'not found':
        return float('nan')
    else:
        return float(result)

#Use simple hill climbing to get to the coordinate with the highest
#displacement.
def get_walk_direction(coordinate, utot):
    coordinate_dx = (coordinate[0] + step, coordinate[1])
    result_dx = get_displacement(coordinate_dx)
    if math.isnan(result_dx):
        return (0.0, 0.0, True)

    coordinate_dy = (coordinate[0], coordinate[1] + step)
```

```
    result_dy = get_displacement(coordinate_dy)
    if math.isnan(result_dy):
        return (0.0, 0.0, True)

max_utot = 0.0
coordinate_at_max = (0.0, 0.0)

coordinate = (0.0, 0.0)
done = False
while not done:
    utot = get_displacement(coordinate)
    if math.isnan(utot):
        break

    max_utot = max(utot, max_utot)
    coordinate_at_max = coordinate

    dutot_dx, dutot_dy, done = get_walk_direction(coordinate, utot)
    #Get new coordinate with sign of partial derivatives.
    coordinate = (
        coordinate[0] + math.copysign(step, dutot_dx),
        coordinate[1] + math.copysign(step, dutot_dy))

unit_length = g_i.GeneralInfo.UnitLength
print("Found coordinate ({0} {3}, {1} {3}). "
        "With a displacement of: {2} {3}".format(
        coordinate_at_max[0], coordinate_at_max[1], max_utot, unit_length))
```

Receiving an image in the scripting layer created by Output

```
from plxscripting.easy import new_server

s_o, g_o = new_server('localhost', 10000, password='yourpassword')

newest_plot = g_o.Plots[-1]
newest_plot.ResultType = g_o.ResultTypes.Soil.Utot
newest_plot.PlotType = 'shadings'
newest_plot.Phase = g_o.Phases[-1]

#image_wrapper is an object that can save the created
#image or, if Pillow is installed, you can get the internal
#Pillow.Image object and use that.
image_wrapper = newest_plot.export(1600, 1200)

try:
    from PIL import ImageFilter
    pil_image = image_wrapper.image
    new_image = pil_image.filter(ImageFilter.BLUR)
    new_image.save("test.png")
except ImportError:
    #Just save if we don't have Pillow
    image_wrapper.save("test.png")
```

A class that can be used to quickly create simple projects

The following code demonstrates a possible approach to building a model template class, that can be tweaked in subclasses in order to create model variants without duplicating large amounts of code. Other examples in this document will build upon this class. Make sure you save it as "simple_project.py" in the same directory where you save the examples that need it.

```
imoprt os
import subprocess

INPUT_SERVER_PORT = 10000
PLAXIS_PATH = r"C:\Program Files\Plaxis\PLAXIS 2D"
```

```
class SimpleProject(object):
    """
    Class that provides a way to quickly setup a project for example purposes.
    """
    def __init__(self, g_i):
        from plxscripting.easy import new_server
        self._new_server = new_server

        args = [os.path.join(plaxis_path, "Plaxis2DXInput.exe"),
        "--AppServerPort={}".format(INPUT_SERVER_PORT)]
        self._input_process = subprocess.Popen(args)

        self._s_i, self._g_i = self._new_server(
        'localhost', 10000, password='yourpassword', timeout=10.0)

    def gather_results(self):
        raise NotImplementedError("Override gather_results in subclass.")

    def output_results(self):
        raise NotImplementedError("Override output_results in subclass.")

    def close_input(self):
        self._input_process.kill()

    @property
    def g_i(self):
        return self._g_i

    def add_soil_layers(self):
    raise NotImplementedError("Override add_soil_layers in subclass.")

    def apply_soil_material(self):
        SAND_PARAMETERS = [
            ('MaterialName', 'Sand'),
            ('Colour', 10676870),
            ('SoilModel', 3), # Hardening soil
            ('DrainageType', 'Drained'),
            ('gammaUnsat', 17),
            ('gammaSat', 20),
            ('E50ref', 43000),
            ('EoedRef', 28000),
            ('EurRef', 129000),
            ('powerm', 0.5),
            ('cref', 1),
            ('phi', 34.0),
            ('psi', 4.0),
```

```
            ('nu', 0.2),
            ('Rinter', 0.7),
            ('KONC', 0.5),
            ('OCR', 1.0),
            ('POP', 0.0)
        ]
        sand = self._g_i.soilmat(*SAND_PARAMETERS)

        for soil_layer in self._g_i.SoilLayers:
            self._g_i.setmaterial(soil_layer, sand)

    def add_structures(self):
        pass # Not adding any plates is fine too.

    def apply_plate_materials(self):
        DIAPHRAGM_WALL_PARAMETERS = [
            ('MaterialName', 'Wall'),
            ('Colour', 16711680),
            ('Elasticity', 0), # Elastic
            ('IsIsotropic', True),
            ('IsEndBearing', True),
            ('EA', 12000000),
            ('EI', 120000),
            ('nu', 0.15),
            ('d', 0.34641),
            ('w', 8.3),
            ('Mp', 1000000000000000.0),
            ('Np', 10000000000.0),
            ('Np2', 10000000000.0),
            ('RayleighAlpha', 0),
            ('RayleighBeta', 0),
            ('Gref', 15061311)
        ]
        diaphragm_wall_material = self._g_i.platemat(*DIAPHRAGM_WALL_PARAMETERS)

        for plate in self._g_i.Plates:
            self._g_i.setmaterial(plate, diaphragm_wall_material)

    def mesh(self):
        self._g_i.gotomesh()
        self._g_i.mesh(0.06)

    def select_curve_points(self):
        pass # Not selecting any curve-points is fine too.

    def configure_phases(self):
        raise NotImplementedError("Override configure_phases in subclass.")

    def make_project(self):
        self.add_soil_layers()
        self.apply_soil_material()
        self.add_structures()
        self.apply_plate_material()
        self.mesh()
        self.select_curve_points()
        self.configure_phases()
        self._g_i.calculate()
```

```
def run(project_class):
    # Replace with the path to your PLAXIS installation.
    project = project_class(r"c:\Program Files (x86)\Plaxis\PLAXIS 2DX")
    project.make_project()
    project.gather_results()
    project.output_results()
    project.close_input()
```

Create a csv file from curve results

This example must be saved in the same directory as the "simple_project.py" module created in a previous paragraph.

```
import sys
import csv

from simple_project import SimpleProject, run
```

```
class ExportCSVProject(SimpleProject):
    def add_soil_layers(self):
        self._g_i.SoilContour.initializerectangular(0, 0, 100, 30)

        borehole = self._g_i.borehole(0)
        self._g_i.soillayer(0)
        self._g_i.setsoillayerlevel(borehole, 0, 30)
        self._g_i.setsoillayerlevel(borehole, 1, 27)
        self._g_i.soillayer(0)
        self._g_i.setsoillayerlevel(borehole, 2, 15)
        self._g_i.soillayer(0)
        self._g_i.setsoillayerlevel(borehole, 3, 0)
        borehole.Head = 23

    def add_structures(self):
        self._g_i.gotostructures()
        self._g_i.plate(40, 30, 40, 14)
        self._g_i.plate(60, 30, 60, 14)

        for plate in self._g_i.Plates:
            self._g_i.posinterface(plate.Parent)
            self._g_i.neginterface(plate.Parent)

        self._g_i.lineload(28, 30, 38, 30)

    def select_curve_points(self):
        output_port = self._g_i.selectmeshpoints()
        s_o, g_o = self._new_server('localhost', output_port, password='yourpassword')
        g_o.addcurvepoint('node', 40, 30, -1, 0)
        g_o.update()

    def configure_phases(self):
        self._g_i.gotostages()

        self._g_i.activate(self._g_i.LineLoads, self._g_i.InitialPhase)
        self._g_i.activate(self._g_i.Plates, self._g_i.InitialPhase)
```

```
        self._g_i.activate(self._g_i.Interfaces, self._g_i.InitialPhase)
        self._g_i.deactivate(
            self._g_i.SoilPolygons[1], self._g_i.InitialPhase)

        line_load = self._g_i.LineLoads[0]

        current_phase = self._g_i.InitialPhase
        line_load.qy_start[current_phase] = 0.0

        for i in range(1, 10):
            current_phase = self._g_i.phase(current_phase)
            self._g_i.setcurrentphase(current_phase)
            line_load.qy_start[current_phase] = -10.0 * i

    def gather_results(self):
        output_port = self._g_i.view(self._g_i.InitialPhase)
        s_o, g_o = self._new_server('localhost', output_port, password='yourpassword')

        self._results = []

        # phaseID is only correct because we know the phases are in order!
        for phaseID, phase in enumerate(g_o.Phases):
            if phaseID == 0:
                # Skip the initial phase and just write 0, because getcurveresults
            # won't return any results for it.
            self._results.append({
                    'phaseID': 0,
                    'displacement': 0
                })
            else:
                displacement_string = g_o.getcurveresults(g_o.Nodes[0],
                    phase, g_o.ResultTypes.Soil.Utot)
            self._results.append({
                    'phaseID': phaseID,
                    'displacement': displacement_string
                })

            g_o.update()

    def output_results(self):
    # Prevent excess newlines.
    if sys.version_info.major == 2:
        file_kwargs = { 'mode': 'wb' }
    else:
        file_kwargs = { 'mode': 'w', 'newline': '' }

        # By default the csv writers will separate values with a comma and the
        # period will be used as the decimal separator.
        # However, this might be problematic when trying to import a csv file
        # on a system that uses comma as the decimal separator.
        # In these cases do the following:
        # - import the locale modules and call: locale.setlocale(locale.LC_ALL, '')
        # - get the correct separator with: locale.localeconv()['decimal_point']
        # - replace the period in the output results with the correct separator
        # - use a different csv dialect (see Python docs) so the comma isn't used
        #   as the column separator.
        with open("results.csv", **file_kwargs) as csv_file:
            writer = csv.DictWriter(csv_file, fieldnames=['phaseID', 'displacement'])
```

```
            for row in self._results:
                writer.writerow(row)


if __name__ == '__main__':
    run(ExportCSVProject)
```

Create a combined image of the mesh for two different phases

This example must be saved in the same directory as the simple_project.py module created in a previous paragraph.

```
from simple_project import SimpleProject, run


from PIL import Image, ImageFont, ImageDraw
```

```
class ExportPlotProject(SimpleProject):
    def add_soil_layers(self):
        self._g_i.borehole(0)
        self._g_i.soillayer(4)

    def add_structures(self):
        self._g_i.gotostructures()
        self._g_i.plate(6, 0, 6, -2)

        self._g_i.pointload(self._g_i.Points[0])
        self._g_i.PointLoads[0].Fy = -10

    def configure_phases(self):
        self._g_i.gotostages()
        self._g_i.activate(self._g_i.Plates, self._g_i.InitialPhase)

        next_phase = self._g_i.phase(self._g_i.InitialPhase)
        self._g_i.activate(self._g_i.PointLoads, next_phase)

    def gather_results(self):
                    width, height = 800, 600

        output_port = self._g_i.view(self._g_i.InitialPhase)
        s_o, g_o = self._new_server('localhost', output_port, password='yourpassword')

        # Hide everything except the mesh itself.
        g_o.Plots[0].DrawFrame = False
        g_o.Plots[0].DrawTitle = False
        g_o.Plots[0].DrawLegend = False
        g_o.Plots[0].DrawAxes = False

        g_o.Plots[0].Phase = g_o.InitialPhase
        self._initial_phase_image = g_o.Plots[0].export(width, height)

        g_o.Plots[0].Phase = g_o.Phase_1
        self._phase_1_image = g_o.Plots[0].export(width, height)

        g_o.close()
```

```
    def output_results(self):
        # Get Pillow.Image objects.
        inner_initial_image = self._initial_phase_image.image
        inner_phase_1_image = self._phase_1_image.image

        assert inner_initial_image.size == inner_phase_1_image.size

        cropping_left = (0, 0,
                         inner_initial_image.width // 2, inner_initial_image.height)
        left_image = inner_initial_image.crop(cropping_left)

        cropping_right = (inner_phase_1_image.width // 2, 0,
                         inner_phase_1_image.width, inner_phase_1_image.height)
        right_image = inner_phase_1_image.crop(cropping_right)

        result_image = Image.new(
                         inner_initial_image.mode, inner_initial_image.size)
        result_image.paste(left_image, cropping_left)
        result_image.paste(right_image, cropping_right)

        default_font = ImageFont.load_default()
        drawing_context = ImageDraw.Draw(result_image)

        initial_phase_ID = self._g_i.Phases[0].Identification.value
        drawing_context.text((200, 160), initial_phase_ID,
            font=default_font, fill=(0, 0, 0, 255))

        phase_1_ID = self._g_i.Phases[1].Identification.value
        after_text_size = default_font.getsize(phase_1_ID)
        drawing_context.text((600 - after_text_size[0], 160),
            phase_1_ID, font=default_font, fill=(0, 0, 0, 255))

        result_image.save("plot_initial_vs_phase_1.png")


if __name__ == '__main__':
    run(ExportPlotProject)
```

Create a matplotlib plot from a line cross-section

This example must be saved in the same directory as the simple_project.py module created in a previous paragraph.

```
from simple_project import SimpleProject, run

from matplotlib import pyplot
```

```
SAMPLE_COUNT = 16


class CrossSectionProject(SimpleProject):
    def add_soil_layers(self):
        self._g_i.borehole(0)
        self._g_i.soillayer(4)
```

```
    def add_structures(self):
        self._g_i.gotostructures()
        self._g_i.lineload(5, 0, 7, 0)

    def configure_phases(self):
        self._g_i.gotostages()
        self._g_i.activate(self._g_i.Plates, self._g_i.InitialPhase)

        next_phase = self._g_i.phase(self._g_i.InitialPhase)
        self._g_i.activate(self._g_i.PointLoads, next_phase)

    def gather_results(self):
        start = (0.0, -4.0)
        end = (6.0, 0.0)

        output_port = self._g_i.view(self._g_i.Phases[-1])
        s_o, g_o = self._new_server('localhost', output_port, password='yourpassword')

        # Assumes that start and end contain floats, will not work correctly
        # with integers in Python 2.
        step = [(e - s) / (SAMPLE_COUNT - 1) for e, s in zip(end, start)]

        self._results = []
        for i in range(SAMPLE_COUNT):
            position = (start[0] + i * step[0], start[1] + i * step[1])
            result_string = g_o.getsingleresult(g_o.Phases[-1],
                g_o.ResultTypes.Soil.Utot, position)

            # Check if position lies outside the mesh, it might make more sense
            # to use 0 as the result for other result types.
            if result_string == "not found":
                raise Exception("Used getsingleresult for point outside mesh.")

            self._results.append(float(result_string))

        g_o.update()

    def output_results(self):
        pyplot.plot(range(len(self._results)), self._results, 'r')
        pyplot.grid(True)
        pyplot.show()


if __name__ == '__main__':
    run(CrossSectionProject)
```

H HTTP REST API [GSE]

H.1 Overview

PLAXIS provides an HTTP based API. This API allows a developer or user to do normal PLAXIS tasks such as creating projects, firing regular commands as found on the existing command line, performing meshing and calculations. In addition, unique representations of Plaxis objects can be retrieved, kept, and then used in later operations, such as the fetching of their properties and methods, as well as calling methods. This provides additional flexibility and power to the existing command line.

There are several categories of actions that can be performed via the HTTP API. Each of these have their own URL path, as follows:

/environment	Start, close, recover or open projects
/commands	Fire regular PLAXIS command line commands and fetch any created objects
/members	Fetch the members of objects
/propertyvalues	Fetch the property values of objects
/namedobjects	Fetch existing objects by their command line names
/list	Fetch items from listable objects
/enumeration	Fetch all possible values of enumeration objects
/exceptions	Get information about exceptions that occurred outside the server requests
/tokenizer	Offers tokenization services

All request data must be encoded using UTF-8.

H.2 Object representations

Resources which create or retrieve objects return representations of those objects. These representations comprise a Globally Unique Identifier (GUID), a type name, and a flag indicating whether the object is listable or not. The JSON structure for this representation is as follows:

```
"guid": <string>
"islistable": <boolean>
```

```
"ownerguid": <string>
"type": <string>
```

The server can recognise and retrieve information about objects contained within the PLAXIS project based on that object's GUID. This means that, for example, a particular PLAXIS object can be created using the */commands* resource, and then later the values of that object's properties can be retrieved by sending that object's GUID in a request to the */propertyvalues* resource. The same GUID can be used in the creation of additional objects by sending it in a request to the */commands* resource. If the object is listable, sending the guid as part of a request to the */list* resource will enable access to the object's properties by index, and so on. The *ownerguid* is only returned for objects that are intrinsic properties. It is the guid of the owning object.

H.3 Using a web browser to access the API

It is possible to perform a subset of the actions that are described below by using a browser address bar. This can be useful for quickly viewing the data representations that are returned from the API.

There is a limitation to the complexity of queries that can be made in this mode, which means that certain resources and actions are not possible. This includes the */list* resource, and also the querying of specific named properties in the */propertyvalues* resource (however it is still possible to query all properties for an object using a browser address bar).

Each resource description below includes an example action using the address bar.

H.4 HTTP API Resources

/environment

This resource is for managing projects. Creating, restoring, closing or opening a project are permitted. If a project is opened, then a filename string is supplied for locating an existing project locally, or otherwise becomes the name of a new project.

Note that it is not possible to save projects via this resource. Instead, fire a save command using the */commands* resource.

Method: POST

Data parameters

```
{
"action":
{
"name": *1 <string> from {"new", "open", "close", "restore"}*,
"filename": *1 <string>*
}
}
```

Data parameters example

```
{
"action":
{
"name": "open",
"filename": "my_plaxis_project.p2dx"
}
}
{
```

```
"action":
{
"name": "close",
"filename": ""
}
}
```

Success responses

Code	Reason
200	OK
404 STATUS NOT FOUND	"Incorrect or unspecified action: [action-name]"
404 STATUS NOT FOUND	"Incorrect or unspecified file name, or not a valid project: [filename]"
400 BAD REQUEST	"Syntax of request could not be understood by the server."
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Example JSON request

```
payload = {"action": {"name": "new", "filename": ""}}
headers = {"content-type": "application/json"}
response = requests.post("/environment", data=json.dumps(payload), headers=headers)
```

Example browser request

```
http://localhost:8001/environment?action.name=new
http://localhost:8001/environment?
action.name=open&action.filename=my_plaxis_project.p2dx
```

/commands

This resource allows all standard commands (see command reference pages) to be run on an active PLAXIS project. Any objects created as the result of commands are represented in the response.

The server can recognise and "translate" GUIDs that refer to existing objects, as part of the command string. This means that it is possible to use GUIDs in the place of regular command line names to refer to other objects. This is beneficial, because the names of objects may be changed over the lifetime of a project (e.g. using the *rename* command), but the GUID remains fixed.

For example, it is possible to create a line between two existing points by referring to the GUIDs of those points in the command string of a request:

```
 line <guid1> <guid2>
```

Note that a command that fails on the command line will not result in an error response from the server. These are treated as successful requests to the API. The server response indicates whether a command failed or not in the 'success' field. (See example below.) Whether the command succeeded or not, the ' additionalinfo' field indicates the response from that action as seen on the regular command line.

Method: POST

Data parameters

```
{
"action":
{
"commands":
[
*{1, ...}[<string>]*
]
}
}
```

Data parameters example

```
{
"action":
{
"commands":
[
"point 1 2"
]
}
}
{
"action":
{
"commands":
[
"embeddedbeamrow 4 4 {860C1FC5-BDE1-4887-98A2-8FE20B61FCD5}"
]
}
}
{
"action":
{
"commands":
[
"bad"
]
}
}
```

Code	Reason
200	OK

Data

```
{
"commands":
[
{
"feedback":
{
"extrainfo": "Added Point_1",
"returnedobjects":
[
{
"islistable": true,
```

```
"type": "Point",
"guid": "{860C1FC5-BDE1-4887-98A2-8FE20B61FCD5}"
}
],
"debuginfo": "",
"success": true,
"errorpos": -1
},
"command": "point 1 2"
}
]
}
{
"commands": [
{
"feedback": {
"extrainfo": "Added Point_3\r\nAdded Line_1\r\nAdded EmbeddedBeamRow_1",
"returnedobjects": [
{
"islistable": true,
"type": "Point",
"guid": "{92851C68-561A-4C21-9ACD-46F50905FD4E}"
},{
"islistable": true,
"type": "Line",
"guid": "{7DB7DF8E-1FE4-4624-A3D2-F682DB9C8F18}"
},{
"islistable": false,
"type": "EmbeddedBeamRow",
"guid": "{A678F396-C8F3-41EE-9A8E-5798BB0B6054}"
}],
"debuginfo": "",
"success": true,
"errorpos": -1
},
"command": "embeddedbeamrow 4 4 Point_1"
}]
}
```


Code	Reason
200	OK

Data


```
{
"commands":
[
{
"feedback":
{
"extrainfo": "Command \"bad\" is not recognized as a global command.
Check the spelling or try specifying a target object.",
"debuginfo": "",
"success": false,
"errorpos": 1
},
"command": "bad"
```

```
}
]
}
```

Here the command returned an associative dictionary containing the 'ContentType'field. This will indicate what the rest of the dictionary will contain. In this particular case where the 'ContentType'is 'image/png'the dictionary will contain an additional 'data'field whose value is a base64 encoded PNG image.

The 'json'field does not necessarily have to be a dictionary, it can also be an integer or any other type allowed in JSON.

Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
404 STATUS NOT FOUND	"Commands not specified"
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Example JSON request

```
payload = {"action": {"commands": ["point 1 2"]}
headers = {"content-type": "application/json"}
response = requests.post("/commands", data=json.dumps(payload), headers=headers)
```

Example browser request

```
http://localhost:8001/commands?action.commands=gotostages
```

/members

This resource fetches the members (properties and methods) of an existing object within PLAXIS, identified by its GUID.

Property members are represented as objects.The type names of these objects are simplified in comparison to their type names within PLAXIS 2D. This means that all 'Text' subtypes for instance will be simply described as having a type of 'Text'. The same principle applies to numerical, boolean, enumeration and object types.

Method members are represented as names only.

Method: POST

Data parameters

```
{
"action":
{
"members":
[
*{1,...}[GUID string]*
]
}
}
```

Data parameters example

```
{
"action":
{
"members":
[
"{68A2C1A7-924B-46C3-89DF-F597DAADC359}"
]
}
}
{
"action":
{
"members":
[
"bad"
]
}
}
```

Code	Reason
200	OK

Data

```
{
"queries": {
"{68A2C1A7-924B-46C3-89DF-F597DAADC359}": {
"extrainfo": "",
"success": true,
"properties": {
"z": {
"islistable": false,
"ownerguid": "{68A2C1A7-924B-46C3-89DF-F597DAADC359}",
"type": "Number",
"guid": "{5FCCF1CC-7576-4565-B931-DE54241EA21B}"
},
"y": {
"islistable": false,
"ownerguid": "{68A2C1A7-924B-46C3-89DF-F597DAADC359}",
"type": "Number",
"guid": "{D52CBED3-8C59-4967-A5BA-89ECE6E9BD94}"
},
"x": {
"islistable": false,
"ownerguid": "{68A2C1A7-924B-46C3-89DF-F597DAADC359}",
"type": "Number",
"guid": "{25DA6A75-4F25-42F3-9654-3672A93FCE1F}"
},
"Name": {
"islistable": false,
"ownerguid": "{68A2C1A7-924B-46C3-89DF-F597DAADC359}",
"type": "Text",
"guid": "{A4209590-9487-4B98-9966-34A6A58DBE2D}"
},
"Comments": {
```

```
"islistable": false,
"ownerguid": "{68A2C1A7-924B-46C3-89DF-F597DAADC359}",
"type": "Text",
"guid": "{9430592C-7D5D-4735-B572-2B29C87FBF1F}"
}
},
"commands": [
"echo","__dump","commands","rename","set","info","__observers","setproperties","move","
rotate","rotateline"],
"commandlinename": "Point_1"
}
}
}
{
"queries": {
"{0D04D0CB-86BF-4430-882C-FA56E45DF7AB}": {
"extrainfo": "GUID does not refer to object in registry: {0D04D0CB-86BF-4430-882C-
FA56E45DF7AB}",
"success": false
}
}
}
{
"queries": {
"bad": {
"extrainfo": "Supplied string is not a valid GUID: bad",
"success": false
}
}
}
```


Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
404 STATUS NOT FOUND	"No GUIDs specified"
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Example JSON request

```
payload = {"action": {"members": ["{9C5875EE-ABA4-4DC2-9E81-1FF839215232}"]}}
headers = {"content-type": "application/json"}
response = requests.post("/members", data=json.dumps(payload), headers=headers)
```

Example browser request

```
http://localhost:8001/members?action.members={0D04D0CB-86BF-4430-882C-FA56E45DF7AB}
```

/propertyvalues

This resource provides the current values of particular properties of PLAXIS objects. If a property is itself an object (such as the point properties of a line), then it is represented as a full object. If a property is a primitive value type, then the relevant primitive value is returned.

There are two ways of calling this resource. One is to supply just a guid of the owner object, in which case all properties of this object are represented in the response. The other way is to supply not just the owner guid, but also a property name and an optional phase parameter. Such a structure is treated as a request for one single property. If that property is staged, and the supplied phase guid is valid, then the value for that property and phase is returned.

Method: POST

Data parameters

```
{
"action":
{
"propertyvalues":
[*{1} GUID string*]
}
}
{
"action":
{
"propertyvalues":
{
"owner": *{1} GUID string*
("phaseguid: *{1} GUID string*)
}
}
}
```

Data parameters example

```
{
"action":
{
"propertyvalues":
["{D4981EA8-EC07-471D-A8C3-7510B4F74F25}"]
}
}
{
"action":
{
"propertyvalues":
{
"owner": "{D4981EA8-EC07-471D-A8C3-7510B4F74F25}"
"propertyname": "Active"
"phaseguid": "{C3581EA8-FB07-5344-A8C3-569359320603}"
}
}
```

Success responses

```
{
"queries": {
"{D4981EA8-EC07-471D-A8C3-7510B4F74F25}": {
"extrainfo": "",
"success": true,
"properties": {
```

```
"Second": {
"islistable": true,
"type": "Point",
"guid": "{A67959C9-C12D-41D6-8136-52C1E5711165}"
},
"Length": 6.92820323027551,
"First": {
"islistable": true,
"type": "Point",
"guid": "{4B889C3F-7F5C-4611-8D31-F4A39C71B11A}"
},
"Name": "Line_1",
"Comments": ""
}
}
}
}
{
"queries":
{
"{A10835D7-AE38-449C-BF67-6C9CE22705BD}":
{
"extrainfo": "",
"properties": {"Comments": ""},
"success": true
}
}
}
```


Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
404 STATUS NOT FOUND	"No GUIDs specified"
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Sample JSON request

```
payload = {"action": {"propertyvalues": ["{9C5875EE-ABA4-4DC2-9E81-1FF839215232}"]}}
headers = {"content-type": "application/json"}
response = requests.post("/propertyvalues", data=json.dumps(payload), headers=headers)
```

Sample browser request

```
http://localhost:8001/propertyvalues?action.propertyvalues={0D04D0CB-86BF-4430-882C-
FA56E45DF7AB}
```

/namedobjects

Supplies a representation of objects within PLAXIS 2D which are identified by their command line name. This is useful when accessing internal lists of objects within Plaxis with known names, such as lists of particular user

features or geometric objects. This resource can also be used to retrieve unique representations of objects for which the command line name is known, such as 'Point$_1$'.

Method: POST

Data parameters

Example Request:

```
{
"action":
{
"namedobjects":
[
"namedobjects": ["Points"]
]
}
}
{
"action":
{
"namedobjects":
[
"namedobjects": ["Plate_1"]
]
}
}
```

Code	Reason
200	OK

Data

```
{
"namedobjects": {
"Points": {
"extrainfo": "",
"success": true,
"returnedobject": {
"islistable": true,
"type": "ModelGroup",
"guid": "{32871FF1-809C-470A-94B0-4B56B28BC67B}"
}
}
}
}
```

Code	Reason
200	OK

Data

```
{
"namedobjects": {
"Plate_1": {
```

```
"extrainfo": "",
"success": true,
"returnedobject": {
"islistable": false,
"type": "Plate",
"guid": "{C0C4EDA6-DFF7-4593-A280-A1EC20D5EADD}"
}
}
}
}
```


Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
404 STATUS NOT FOUND	"No object names specified"
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Sample browser request

```
http://localhost:8001/namedobjects?action.namedobjects=Lines
http://localhost:8001/namedobjects?action.namedobjects=Phases
```

/list

Provides the ability to perform read operations on lists within PLAXIS 2D, such as getting values at particular indices, finding the number of objects in the list, and filtering.

Method: POST

Data parameters

Example request:

```
{"action": {"listqueries": [{"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"method": "sublist",
"startindex": 0,
"stopindex": 1]}}
{"action": {"listqueries": [{"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"method": "count"]}}
{"action": {"listqueries": [{"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"method": "index",
"startindex": 0]}}
```

Success responses


```
{"listqueries": [{"extrainfo": "",
"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"methodname": "sublist",
"outputdata": [{"guid": "{81A754C8-1393-48BA-AD6E-5F6953F9FBE7}",
"islistable": true,
"type": "Point"}],
"startindex": 0,
"stopindex": 1,
"success": true}]}
{"listqueries": [{"extrainfo": "",
```

```
"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"methodname": "count",
"outputdata": 5,
"success": true}]}
{"listqueries": [{"extrainfo": "",
"guid": "{CF1DECEB-A28D-4609-B38C-8D3FF5E573A1}",
"methodname": "index",
"outputdata": [{"guid": "{81B754C8-1353-48BA-AD6E-5A6953F9CBE5}",
"islistable": true,
"type": "Point"}],
"success": true}]}
```


Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Sample browser request

Not currently possible to call this resource via the browser address bar.

/enumeration

Provides a list of all possible enumeration values of one or more enumeration objects, when supplied with their GUIDs.

Success responses

Code	Reason
200	OK

Data


```
{
"queries": {
"{31AFFD41-0EB4-4A67-94F4-298642E17507}": {
"extrainfo": "",
"success": true,
"enumvalues": {
"hinged": 1,
"free": 2,
"rigid": 0
}
}
}
}
```


Code	Reason
200	OK

Data

```
{
"queries": {
"{0B4774CE-3307-4B86-B644-BFDE82CF3AAA}": {
"extrainfo": "GUID does not refer to object in registry:
{0B4774CE-3307-4B86-B644-BFDE82CF3AAA}",
"success": false
}
}
}
```

Code	Reason
200	OK

Data

```
{
"queries": {
"{C896C54E-1CBA-4B0F-9987-E1FF57A9131E}": {
"extrainfo": "Guid {C896C54E-1CBA-4B0F-9987-E1FF57A9131E} does not refer to
an enumeration intrinsic property type.",
"success": false
}
}
}
```

Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
405 METHOD NOT ALLOWED	"No active project"
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

/exceptions

Exceptions that occur during execution of a server request are automatically caught and returned as error result in the response of that request. Exceptions however may also happen independently of the requests, e.g. because a problem during repainting. In such situations, an exception message box may be displayed by the application. This resource allows access to information about this type of exceptions.

The getlast variant will clear the last exception message on the server, while the peeklast variant will leave it in place.

Method: POST

Data parameters

```
{
"action":
{
"name": *1 <string> from {"getlast", "peeklast"}*
```

```
}
}
```

Data parameters example

```
{
"action":
{
"name": "getlast"
}
}{
"action":
{
"name": "peeklast"
}
}
```

Success responses

Code	Reason
200	OK

Data

```
{
"exceptions": [
"date/time : 2015-04-17, 16:09:46, 750ms\r\ncomputer name : PC075\r\nwts client
name : ...etc..."
]
}
```

Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

payload	"action": "name": "getlast"
headers	"content-type": "application/json"
response	requests.post("/exceptions", data=json.dumps(payload)
headers	headers)

Example browser request

http://localhost:8001/exceptions?action.name=getlast

/tokenizer

Allows parsing a command line string into the separate tokens according to the PLAXIS command line language. A token is a sequence of one or more characters that together form a basic indivisible entity of source code.

Each token has the following properties:

- type: indicates what type of token it is (see list below)
- value: which is the parsed representation of the item (e.g. the token consisting of the characters 10 will resolve to be of integer type with value 10).
- position: indicates the start position of the token in the original string (0-based)
- length: the number of characters the token consumed from the original string

The following types of tokens exist:

- identifier: something that will act either as command or as object identifier
- comment: a piece of comment, i.e. a sequence of characters starting with # up to the end of the string. The *value* of the token includes the starting # sign.
  - content: e.g. `running` in the case of `#running`
- externalinterpreter: a line that should be executed by an external interpreter, e.g. /output echo Points. The *value* of the token includes the starting / sign. It has three additional properties:
  - interpretername: e.g. output in the case of `/output echo Points`
  - externalcommand: e.g. `echo Points` in the same case
  - content: e.g. `output echo Points` in the case of `output echo Points`
- operand tokens:
  - bracket: identifies a bracket type. Bracket tokens have additional properties:
    - brackettype: can be *round*, *square*, *curly* for `(), [], {}` respectively
    - bracketstate: can be *open* or *close* for `([{respectively )]}`
  - member: used to separate an object from its members (e.g. the . in `Line\_1.Beam`)
  - comma, plus, minus, multiplier, divider, assign: currently unused, but they are the ,, +,-,*,/,=operands respectively
- numerical tokens:
  - integer: represents a number that can be represented by a 32-bit signed integer
  - float: represents a number that can be represented as a floating point value

Method: POST

Data parameters

```
{
"action":
{
"tokenize":
[
*{1,...}[string]*
]
}
}
```

Data parameters example

```
{
"action":
{
```

```
"tokenize":
[
"point 10"
]
}
}
{
"action":
{
"tokenize":
[
"point 10", "point 8"
]
}
}
{
"action":
{
"tokenize":
[
"a ?"
]
}
}
```

Success responses

Code	Reason
200	OK

```
{
"tokenize":
[
{
"tokens": [
{"position": 0,
"length": 5,
"type": "identifier",
"value": "point"},
{"position": 6,
"length": 2,
"type": "integer",
"value": 10},
],
"success": true,
"extrainfo": "",
"errorpos": -1
},
"tokenize": "point 10"
}
]
}
```

Data

```
{
"tokenize":
```

```
[
{
"tokens": [
{"position": 0,
"length": 1,
"type": "identifier",
"value": "a"},
],
"success": false,
"extrainfo": "Unrecognized token",
"errorpos": 2
},
"tokenize": "a ?"
}
]
}
```

Error responses

Code	Reason
400 BAD REQUEST	"Syntax of request could not be understood by the server."
415 UNSUPPORTED MEDIA TYPE	"Unsupported content-encoding of [encoding type]"

Example JSON request

```
payload = {"action": {"name": "getlast"}
headers = {"content-type": "application/json"}
response = requests.post("/exceptions", data=json.dumps(payload), headers=headers)
```

Example browser request

http://localhost:8001/tokenizer?action.tokenize=point%201

Calculation warning and errors in PLAXIS

The errors that may occur while using PLAXIS.

Table 24: Warnings

Error code	Message	Hint
10	PICOS solver was used instead of PARDISO due to lack of memory.	Install more RAM in your computer
9	Different stress type was used in the previous phase.	In the previous phase, a different stress type (Terzaghi stress or Bishop stress) was used. Please check your input.
8	Internally changes in setting: Updated mesh is selected.	Due to settings of previous phase(s), the updated mesh option was selected in the calculation.
7	Free field cannot be used in an axisymmetric geometry	Free field boundaries cannot be used in axisymmetric models (2D).
6	Dynamic loads applied on free field base elements are moved to main domain by kernel	Dynamic loads that where applied on a free field boundary will be applied directly on the elements inside the model.
5	One or more Tension of Mohr-Coulomb points found	Inspect Output for the Initial phase (K0 procedure) and check the K0 values for the materials in which the plastic points occur as well
4	Static loads have been changed in a dynamics analysis	The value for a static load has been changed in a dynamics analysis. Please re-check your load values if this was unintentional.
3	Prescribed ultimate state fully reached. Non-zero static prescribed displacements are ignored	Non-zero static prescribed displacements were ignored.

Error code	Message	Hint
2	Prescribed ultimate state fully reached. Unfinished construction stage. Next calculation must be staged construction	For this phase a Σ *Mstage* < 1 was defined. The next phase should be staged construction.

Table 25: Errors

Error code	Message	Hint
0	OK.	Calculation succeeded and converged
1, 8, 9	Stiffness matrix too big for reserved RAM memory	Decrease model size. If the problem persists, install more RAM in your computer
2, ..., 7	Problem too big for reserved RAM memory	Decrease model size. If the problem persists, install more RAM in your computer
10	Deformation not compatible in STRESBL	Sometimes in Updated mesh calculations it may happen that elements are turned 'inside out'. Inspect the results. It might help to increase the stiffness in the problem area.
11, 16	Determinant is zero or nearly zero.	The determinant is zero or nearly zero. Causes for this could be: • Bad mesh quality. (elements with low quality). This can be reviewed in the generated mesh (*Mesh ->Quality*) • Missing boundary conditions. Review the boundary conditions for this phase • Floating (unconnected) clusters/ volumes
12	Material set not found.	Check if all assigned materials (soils and structural elements) are available and defined
13	Water weight is zero	Check the project's water weight
14	Deformation not compatible in GLBMB.	A (very) large deformation occurred in a (2D) beam, resulting in a too large curvature. Try to force smaller steps by using smaller load increments and/or a tighter tolerance (or use smaller DSC max value)

Error code	Message	Hint
15	Jacobian matrix is less than or nearly zero.	This is usually the result of one of these two causes (or a combination): • Badly shaped elements: good elements are those with the same size for width, length and height of the element. Please inspect the mesh quality. • Large stiffness differences. For numerical reasons, the stiffness values in the model may not differ too much (e.g. a ratio larger than 10^6 between the lowest stiffness and the highest stiffness should be avoided).
17, 19, 20	Stiffness matrix is nearly singular and cannot be solved.	Stiffness matrix is nearly singular and cannot be solved. & Causes for this could be: • Bad mesh quality. (elements with low quality). This can be reviewed in the generated mesh (*Mesh ->Quality*). • Large stiffness differences. For numerical reasons, the stiffness values in the model may not differ too much (e.g. a ratio larger than 10^6 between the lowest stiffness and the highest stiffness) should be avoided. • A combination of the two above. • Missing boundary conditions. Review the boundary conditions for this phase. • Floating/unconnected clusters/ volumes.
24	Material set with zero permeability found, but not allowed in this type of calculation.	Groundwater flow and consolidation calculations require non-zero permeability in all elements. Check materials and add permeability value: $k_i > 0$
25	Groundwater weight is zero.	Check the project's water weight
26	Unable to load dll. <param1>	Check the presence of user defined soil models dll files in the program folder. <param1> is the name of the file.
27	File not found. <param1>	A file needed for the calculation was not found. Regenerating the mesh could solve it. If not, please send it to support. <param1> is the name of the missing file.

Error code	Message	Hint
28	Peak plastic shear strain is too low.	NGI-ADP model, UDCAM-S model: γ_{peak} value(s) nearly zero. Please check the input values. Note that the value is expressed in percent (%).
29	Error factorization matrix	Check for floating clusters or insufficient boundary conditions.
30	Number of dynamic sub-steps is ZERO.	The defined value for dynamic substeps is zero. To solve it, increase the number of dynamic substeps (see iterative settings).
31	Error backsub matrix.	Internal error. Please send the PLAXIS project to support.
32	Memory allocation error.	There is not enough memory available.
33	Internal error. Please, pack this project using *Pack project* and send to support	An internal error occurred. It could be one of these causes: • Zero length of embedded beam row in 2D. Update version and remesh. • An error with free field boundaries. Please check the input For any other cause, please send the PLAXIS 2D project to support
34	Ultimate state not reached in GW-Flow analysis.	The groundwater flow calculation did not converge. Please check the input parameters and the phase settings.
35	No critical time step found; no draining boundaries	Check boundary conditions. It could be that all boundaries are closed.
36	NaN found in an element stiffness matrix.	(NaN = Not a Number) Causes could be elements with a zero area or bad material parameters
37	DeAllocation memory error.	Internal error. Please send the PLAXIS project to support.
38	Invalid plate material table.	Check non-linear structural materials datasets.
39	NaN found during calculation, probably severe divergence.	(NaN = Not a Number). This could be caused by bad input parameters. If the problem persists, please send the PLAXIS project to support.

Error code	Message	Hint
40	Severe divergence.	The global error grew too large in the calculation ((> 10^6)). This could be related to bad input for soil parameters. Please check the model in the last step to identify the problem area. Otherwise, please send the PLAXIS project to support.
41	$K_{nc}^{0} \simeq 0$ in Sekiguchi-Ohta model, please correct.	Re-evaluate Sekiguchi-Ohta model material datasets.
42	Water file does not exist. <param1>	2D2011/2D2012: Please regenerate water conditions file by opening staged construction, generate water pressures and then press update. If more than one phase has this problem, regenerating the mesh might solve it faster for all phases.
43	Not proper Newmark coefficients.	Re-evaluate Newmark coefficients.
44	No drift correction for unequal time steps.	Drift correction is based on equal time steps. Create a table of equal time steps or switch off drift correction
45	At least 2 data points are needed for drift correction.	Drift correction needs at least 2 data points. Increase the number of data points or switch off drift correction.
46	Time must be increasing in a tabular multiplier function.	Re-evaluate dynamic multiplier tables: time must be increasing each time step
47	Invalid material parameter set. <param1>	Re-evaluate the mentioned material parameter set.
48	Error in reading flow communication file. <param1>	File <param1> not found, probably something went wrong in flow calculation or flow calculation is very slow. Usually this is solved by decreasing differences in permeabilities.
49	non-zero dynamic load without multiplier.	Please assign a dynamic multiplier function to all dynamic loads
50	non-zero dynamic displacement without multiplier.	Please assign a dynamic multiplier function to all dynamic loads
51	Gravitational acceleration (g) is zero.	Re-evaluate the gravitational acceleration in the project properties

Error code	Message	Hint
52	Both left and right hand side free field boundaries must be active.	Activate both sides of the free field boundaries
53	Number of nodes on lateral boundaries must be equal to tie their degrees of freedom.	Regenerate the mesh to match this criterion. Make sure the left and right side of the geometry are identical.
54	Parent phase uses updated mesh while this type of calculation does not support updated mesh.	Redefine an Updated mesh state for prior phases, or reset displacements to zero for this phase. Note that displacements cannot be reset to zero after an updated mesh analysis.
101	Soil body collapses	The program has detected soil failure. Please evaluate the calculation results in Output to show why it fails
102	Not enough load steps.	Increase the value assigned to the Max steps parameter.
103	Load advancement procedure fails.	The program has detected numerical problems. Please inspect the input data and the calculation results in Output to evaluate why it fails.
104	Prescribed ultimate time not reached.	Prescribed time interval in a consolidation analysis is not reached. Probably due to soil failure. Please inspect the input data and the calculation results in Output to evaluate why it fails.
105	Prescribed minimum excess pore pressure not reached.	Prescribed minimum excess pore pressure in a consolidation analysis is not reached. Probably due to soil failure. Please inspect the input data and the calculation results in Output to evaluate why it fails.
107	Prescribed ultimate level Σ *Msf*sf not reached	Increase the value assigned to the Max steps parameter.
110	Accuracy condition not reached in last step.	This could be solved by increasing the value assigned to Max steps.
111	Soil body collapses. Accuracy condition not reached in last step.	Combination of 101 and 110.
112	Not enough load steps. Accuracy condition not reached in last step.	Combination of 102 and 110.

Error code	Message	Hint
113	Not enough load steps. Load advancement procedure fails. Accuracy condition not reached in last step.	Combination of 103 and 110.
114	Prescribed ultimate time not reached. Accuracy condition not reached in last step.	Combination of 104 and 110.
115	Prescribed minimum excess pore pressure not reached. Accuracy condition not reached in last step.	Combination of 105 and 110.
117	Prescribed ultimate level Σ *Msf*sf not reached. Accuracy condition not reached in last step.	Combination of 107 and 110.
201	Picos_Dec: matrix type differs from initial type	Internal error. Please send the PLAXIS project to support.
202	Picos_Back: matrix type differs from initial type.	Internal error. Please send the PLAXIS project to support.
203	Picos_Prep: iDof > nDof.	Internal error. Please send the PLAXIS project to support.
241	Unexpected problem. Please contact Plaxis.	Internal error. Please send your project to support.
244	Unexpected problem. Please contact Plaxis.	Internal error. Please send your project to support.
245	Unexpected problem. Please contact Plaxis.	Internal error. Please send your project to support.
246	Programmer abort request. For more details see the .Ixx or .Dxx file. <Params>	See the details in these files, or send your project to support.
247	Complex error. For more details see the .Ixx or .Dxx file. <Params>	See the details in these files, or send to support.
248	Reference to non-existing material set. Redefined staged construction phase.	Redefine staged construction for this phase and check material assignments.
249	Not enough (virtual) memory	Decrease model size. If the problem persists, install more RAM in your computer
250	Mesh files does not exist.	Missing mesh data. Regenerate the mesh.

Error code	Message	Hint
251	Use dt > 0 for dynamics.	Make sure that the time interval is larger than 0: dt>0
252	Calculation results of previous phase do not match the current mesh. Recalculate the previous phase.	Recalculate the previous phase(s) to solve it.
253	Disk (nearly) full.	Disk is full: clear some disk space or move the calculation to a larger disk. Note: since 2012, all calculations are performed in the Windows TEMP folder. That means that there should be enough space available on the drive where your TEMP folder is installed.
254	Cancelled by user.	The user stopped the calculation.
255	Abort request from USER-subroutine.	This is caused by a programmer error in de UDSM (user-defined soil model) code.
256	Run-time-error in kernel.	This is caused by an undetermined error/ crash of the kernel (in the newer versions). Please send the PLAXIS project to support.
501	Beam material set: ratio I3/I2 too large	Check beam material datasets for the ratio I3/I2.
501	Beam material set: ratio I2/I3 too large	check beam material datasets for the ratio I2/I3
1000	The results from the previous phase are missing. Please recalculate the previous phase.	Recalculate the previous phase.

I.1 Meshing errors

Errors such as invalid surface mesh or invalid element(s) found etc, that may occur during meshing can be solved by taking into account the hint/solution provided in the command line.

I.2 Cyclic accumulation and optimization tool errors

The errors that may occur in the *Cyclic accumulation* tab:

Table 26: Errors that may occur in the Stress-strain curves tab

Error message	Hint
Too many strain contours, maximum is 150	It may occur when the user uploads custom contour data for a number of contour lines larger than 150
The contour diagrams should be defined up to the same maximum number of cycles	It indicates that the last point of each line of a custom contour data should be defined at the same number of cycles.
Interpolation problem, shear stress level =	The cyclic accumulation procedure could not be successfully performed for a combination of wrong load parcels data or custom contours.
Infinite slope, cannot interpolate	the cyclic accumulation procedure could not be successfully performed for a combination of wrong load parcels data or custom contours.

The errors that may occur in the *Parameter optimisation* tab:

Table 27: Exit codes, message printed in Optimization_results.txt and message shown in the GUI

Exit code	Error message
1	Internal error, model is invalid
2	Optimisation file not indicated
3	Cannot load "hscapitr.dll"
4	"hscapitr.dll" is corrupt
5	Unknown search intensity is given
6	Cannot allocate memory
7	Internal error, PSWARM
8	Could not start optimisation. Check parameter ranges
9	Cannot find feasible points
10	Cannot find curve file
11	The curve phase information is incompatible with the test configuration
12	Cannot find optimisation input file
13	Material parameter index is outside of the range
14	Material parameter bounds are outside of the range

Exit code	Error message
15	Cannot find VLC input file
16	Error in VLC input file
17	Wrong material parameter in curve file
18	Aborted by user
19	Small overlap curves

For the errors mentioned in table above, please send your project to support.

J Overview Features and Licence levels -

PLAXIS 2D offer an extensive number of features for the modelling of diverse geotechnical problems; for this reason, according to our user needs these features are offered under different licencing levels (see General Information Manual for more information about licencing).

In the following table an overview of some of the offered software features is included. Users can use this table to have an initial idea of of the capabilities at each licencing level. Since not all PLAXIS 2D features can be listed in the table, users are encouraged to review in detail the present manual where the features are identified according their licence level group (please check Features and Licence levels (on page 11)).

Feature/Tool	Licence Level		
	PLAXIS 2D	PLAXIS 2D Advanced [ADV]	PLAXIS 2D Ultimate [ULT]
Feature			
Soil, structures, mesh, flow, staged construction calculation modes	✓	✓	✓
Initial stress from K_0, Field stress, gravity loading calculation type	✓	✓	✓
General deformation analysis (Plastic calculation type)	✓	✓	✓
Factor of Safety (Safety calculation type)	✓	✓	✓
Pore pressures from phreatic levels	✓	✓	✓
Blend surfaces and Loft polycurve	✓	✓	✓
Polar and rectangular array	✓	✓	✓
Automatic centerline extraction tools	✓	✓	✓

Overview Features and Licence levels -

Feature/Tool	Licence Level		
	PLAXIS 2D	PLAXIS 2D Advanced [ADV]	PLAXIS 2D Ultimate [ULT]
Tunnel designer	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Consolidation analysis (Consolidation calculation type)	-	✓	✓
Pseudostatic analysis	-	✓	✓
Steady-state groundwater flow	-	✓	✓
Steady-state thermal flow	-	✓	✓
Multicore calculation (no limit of cores)	-	✓	✓
Flow only analysis	-	✓	✓
Transient groundwater flow	-	-	✓
Dynamics and seismic analysis	-	-	✓
Transient thermal flow	-	-	✓
Fully coupled flow deformation analysis	-	-	✓
Site response analysis	-	-	✓ + [GSE]
Tools			
Sensitivity analysis and parameter analysis	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Design approaches	-	✓	✓
Soil test and inverse analysis	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Generate Stratigraphy from imported CPT Logs	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Remote scripting (Input, output, macros and Soiltest)	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]

Overview Features and Licence levels -

Feature/Tool	Licence Level		
	PLAXIS 2D	PLAXIS 2D Advanced [ADV]	PLAXIS 2D Ultimate [ULT]
Scripting reference	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
CAD importer and exporter	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
PLAXIS 2D LE importer	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
2D to 3D converter	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Calculation manager	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Autocomplete	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]
Export to Paraview	✓ + [GSE]	✓ + [GSE]	✓ + [GSE]

Note: Soil models available in PLAXIS and their relation with licencing levels can be seen in the Appendices of the Material Models Manual .