PLAXIS

CONNECT Edition 20.04

PLAXIS 3D-Reference Manual



Last Updated: September 01, 2020

Table of Contents

Chapter 1: Introduction 10

Chapter 2: General Information 11

2.1 Units and sign conventions 11
2.1.1 Units 11
2.1.2 Sign convention 12
2.2 File handling 13
2.3 Integrated Structural Modelling (ISM) 14
2.4 PLAXIS Coupling Tool 17
2.5 Help facilities 22

Chapter 3: Input program - General overview 23

3.1 Starting the Input program 23
3.1.1 New project 25
3.1.2 Existing project 28
3.1.3 Packing a project 28
3.2 ProjectWise Integration Commands 30
3.2.1 Add File to ProjectWise 31
3.2.2 Open File stored in ProjectWise 33
3.2.3 Save to ProjectWise 34
3.2.4 To View ProjectWise properties 35
3.3 Bentley Cloud Services 36
3.4 Exporting geometry 37
3.5 Layout of the Input program 38
3.6 Menus in the Menu bar 41
3.6.1 File menu 41
3.6.2 Edit menu 42
3.6.3 View menu 42
3.6.4 Soil menu 43
3.6.5 Structures menu 43
3.6.6 Mesh menu 44
3.6.7 Flow conditions menu 45
3.6.8 Phases menu 45
3.6.9 Options menu 46
3.6.10 Expert menu 46
3.6.11 Help menu 47
3.7 Input program structure - Modes 48
3.8 Model in the drawing area 49
3.8.1 Model view - Zooming 49
3.8.2 Selecting geometry components 49
3.8.3 Model View - Visualisation settings 50
3.8.4 Model View - Orientation 52
3.8.5 Drawing in 3D 53
3.9 General information on command line 56
3.9.1 Indexing in commands 58

3.9.2 Accessing Output commands in Input command line ... 58
3.10 Plaxis Remote Scripting Interface ... 59
3.10.1 Anatomy of a command ... 59
3.10.2 Geometry and calculation modes objects differences ... 61
3.11 Explorers ... 61
3.11.1 Model explorer ... 62
3.11.2 Selection explorer ... 67
3.11.3 Phases explorer ... 67

Chapter 4: Ground modelling - Soil mode ... 69

4.1 Adjustment of the soil contour ... 69
4.2 Creating boreholes ... 70
4.2.1 Creating new boreholes ... 70
4.2.2 Creating multiple boreholes ... 72
4.2.3 Editing boreholes ... 72
4.3 Soil layers ... 73
4.3.1 Creating soil layers in boreholes ... 74
4.3.2 Defining water conditions ... 75
4.3.3 Initial Conditions of Soil ... 77
4.3.4 Preconsolidation ... 78
4.3.5 Surfaces ... 80
4.3.6 Generating soil stratigraphy from CPT logs ... 82
4.4 Importing soil layers ... 86

Chapter 5: Modelling loads and structures - Structures mode ... 90

5.1 Assisting tools ... 90
5.1.1 Selecting geometry components ... 90
5.1.2 Move object ... 91
5.1.3 Rotate object ... 91
5.1.4 Extrude object ... 92
5.1.5 Copy object - Array ... 95
5.2 Geometric entities ... 96
5.2.1 Create point ... 97
5.2.2 Create line ... 98
5.2.3 Create NURBS Curve ... 99
5.2.4 Create polycurve ... 99
5.2.5 Create surface ... 105
5.2.6 Create volume ... 108
5.2.7 Advanced geometric modelling options ... 108
5.3 Importing geometry ... 113
5.4 Loads ... 117
5.4.1 Point load ... 120
5.4.2 Line load ... 121
5.4.3 Surface load ... 121
5.5 Prescribed Displacements ... 122
5.5.3 Point prescribed displacement ... 124
5.5.4 Line prescribed displacement ... 124
5.5.5 Surface prescribed displacement ... 124
5.5.6 Surface contraction ... 125
5.6 Dynamic loading ... 126
5.6.1 Definition of multipliers ... 126

5.6.2 Harmonic signal ... 127
5.6.3 Signal from table ... 128
5.6.4 Input accelerograms ... 131
5.7 Moving loads ... 135
5.7.1 Path ... 135
5.7.2 Definition of movement functions ... 135
5.7.3 Harmonic signal ... 136
5.7.4 Signal from table ... 137
5.7.5 Linear ... 139
5.8 Structural elements ... 139
5.8.1 Fixed-end anchors ... 140
5.8.2 Node-to-node anchors ... 141
5.8.3 Beams ... 142
5.8.4 Embedded beams ... 143
5.8.5 Plates ... 146
5.8.6 Geogrids ... 146
5.8.7 Interfaces ... 147
5.8.8 Rigid bodies ... 151
5.9 Tunnels ... 153
5.9.1 Tunnel designer ... 154
5.9.2 Definition of tunnel cross section geometry ... 156
5.9.3 Import tunnel geometry ... 160
5.9.4 Definition of tunnel trajectory ... 160
5.9.5 Properties ... 162
5.9.6 Tunnel splitting tool ... 167
5.9.7 Sequencing ... 172
5.10 Hydraulic conditions ... 176
5.10.1 Well ... 177
5.10.2 Line drain ... 178
5.10.3 Surface drain ... 179
5.10.4 Surface flow boundary conditions ... 180
5.11 Flow functions ... 185

Chapter 6: Material properties and material database ... 191

6.1 Modelling soil and interface behaviour ... 193
6.1.1 General tabsheet ... 193
6.1.2 Parameters tabsheet ... 201
6.1.3 Hoek-Brown pre-processing tool ... 228
6.1.4 Cyclic accumulation and optimisation tool ... 234
6.1.5 Groundwater tabsheet ... 249
6.1.6 Interfaces tabsheet ... 258
6.1.7 Initial tabsheet ... 265
6.2 Modelling undrained behaviour ... 269
6.2.1 Undrained (A) ... 270
6.2.2 Undrained (B) ... 270
6.2.3 Undrained (C) ... 271
6.2.4 Undrained behaviour for unsaturated soil ... 271
6.3 Simulation of soil lab tests ... 272
6.3.1 Triaxial test ... 276
6.3.2 Cyclic triaxial test ... 277
6.3.3 Oedometer test ... 279

6.3.4 CRS ... 280
6.3.5 DSS ... 280
6.3.6 CDSS ... 281
6.3.7 General ... 283
6.3.8 Results ... 284
6.3.9 Parameter optimisation ... 285
6.4 Material data sets for plates ... 293
6.4.1 Material set ... 293
6.4.2 Mechanical properties ... 294
6.5 Material data sets for geogrids ... 298
6.5.1 Material set ... 298
6.5.2 Mechanical Properties ... 299
6.6 Material data sets for beams ... 303
6.6.1 Material set ... 303
6.6.2 Beam properties ... 304
6.7 Material data sets for embedded beams ... 308
6.7.1 Material set ... 308
6.7.2 Beam properties ... 310
6.7.3 Interaction properties: bearing capacity ... 312
6.8 Material data sets for anchors ... 315
6.8.1 Material set ... 315
6.8.2 Properties ... 316
6.9 Assigning material data sets to geometry components ... 318

Chapter 7: Meshing and calculation ... 320

7.1 Mesh generation - Mesh mode ... 320
7.1.1 Elements ... 321
7.1.2 Global settings ... 322
7.1.3 Local refinement ... 323
7.1.4 Enhanced mesh refinements (EMR) ... 323
7.2 Swept meshing ... 324
7.3 Defining calculation phases ... 325
7.3.1 Phases explorer ... 325
7.3.2 Order of calculation phases ... 327
7.3.3 Phases window ... 327
7.4 Types of analysis ... 328
7.4.1 Initial stress generation ... 329
7.4.2 Plastic calculation ... 334
7.4.3 Consolidation calculation ... 335
7.4.4 Fully coupled flow-deformation analysis ... 336
7.4.5 Safety calculation (phi/c reduction) ... 337
7.4.6 Dynamic calculation ... 340
7.4.7 Dynamic with consolidation calculation ... 342
7.4.8 Plastic nil-phase ... 343
7.4.9 Updated mesh analysis ... 343
7.5 Loading type ... 345
7.5.1 Staged construction ... 345
7.5.2 Minimum excess pore pressure ... 347
7.5.3 Degree of consolidation ... 347
7.5.4 Target ΣMsf ... 348
7.5.5 Incremental multipliers ... 348

7.6 Water pressure calculation 349
7.6.1 Calculation type 349
7.6.2 Global water level 351
7.7 Load stepping procedures 351
7.7.1 Automatic step size procedure 352
7.7.2 Load advancement - Ultimate level 353
7.7.3 Load advancement - Number of steps 354
7.7.4 Automatic time stepping (consolidation or fully coupled analysis) 355
7.7.5 Automatic time stepping 355
7.8 Calculation control parameters 356
7.8.1 General phase parameters 356
7.8.2 Deformation control parameters 358
7.8.3 Numerical control parameters 362
7.8.4 Flow control parameters 372
7.8.5 Dynamic control parameters 373
7.8.6 Reached values 373
7.9 Flow conditions 374
7.9.1 Water levels 375
7.9.2 Assigning water conditions to soil volumes 378
7.9.3 Water pressure generation and previewing 380
7.9.4 Flow boundary conditions 380
7.9.5 Precipitation 381
7.10 Geometry configuration - Staged Construction mode 382
7.10.1 Changing geometry configuration 382
7.10.2 Staged construction procedure in calculations 384
7.10.3 Changing loads 384
7.10.4 Definition of connections 385
7.10.5 Reassigning material data sets 388
7.10.6 Applying volumetric strains in clusters 388
7.10.7 Prestressing of anchors 389
7.10.8 Advancing a tunnel 389
7.10.9 Model conditions 390
7.11 Special option 399
7.12 Starting a calculation 399
7.12.1 Previewing a construction stage 399
7.12.2 Selecting points for curves 400
7.12.3 Execution of the calculation process 400
7.12.4 Aborting a calculation 400
7.12.5 Output during calculations 401
7.12.6 Selecting calculation phases for output 406
7.12.7 Adjustment to input data in between calculations 406
7.12.8 Automatic error checks 407

Chapter 8: Output program - General overview 410

8.1 Layout of the Output program 411
8.2 Menus in the Menu bar 412
8.2.1 File menu 413
8.2.2 View menu 413
8.2.3 Project menu 414
8.2.4 Geometry menu 415
8.2.5 Mesh menu 416

8.2.6 Deformations menu 417
8.2.7 Stresses menu 417
8.2.8 Forces menu 417
8.2.9 Options 417
8.2.10 Tools menu 417
8.2.11 Expert menu 419
8.2.12 Window menu 419
8.2.13 Help menu 420
8.3 General information on command line 420
8.3.1 Indexing in commands 420
8.4 Tools in the Output program 421
8.4.1 Accessing the Output program 421
8.4.2 Exporting output data 422
8.4.3 Curves manager 424
8.4.4 Store the view for reports 424
8.4.5 Zooming the plot 424
8.4.6 Relocation and reorientation of the plot 424
8.4.7 Scaling the displayed results 425
8.4.8 Tables 426
8.4.9 Selection of results 427
8.4.10 Display type of results 427
8.4.11 Select structures 429
8.4.12 Partial geometry 429
8.4.13 Viewing results in cross sections 430
8.4.14 Miscellaneous tools 431
8.5 Display area 434
8.5.1 Legend 434
8.5.2 Modifying the display settings 435
8.6 Views in Output 438
8.7 Report generation 438
8.7.1 Configuration of the document 441
8.8 Creating animations 442
8.9 Paraview Export 443

Chapter 9: Results available in Output program 445

9.1 Connectivity plot 445
9.2 Deformations 446
9.2.2 Deformed mesh 446
9.2.3 Total displacements 447
9.2.4 Phase displacements 447
9.2.5 Sum phase displacements 447
9.2.6 Incremental displacements 448
9.2.7 Velocities 448
9.2.8 Accelerations 448
9.2.9 Accelerations in 'g' 449
9.2.10 Total cartesian strains 449
9.2.11 Phase cartesian strains 449
9.2.12 Incremental cartesian strains 449
9.2.13 Total strains 450
9.2.14 Phase strains 450
9.2.15 Incremental strains 450

9.3 Stresses ... 451
9.3.1 Cartesian effective stresses ... 451
9.3.2 Cartesian total stresses ... 451
9.3.3 Principal effective stresses ... 451
9.3.4 Principal total stresses ... 452
9.3.5 Initial conditions ... 453
9.3.6 State parameters ... 454
9.3.7 Pore pressures ... 456
9.3.8 Groundwater flow ... 458
9.3.9 Plastic points ... 459
9.3.10 Fixed end anchors ... 461
9.3.11 Node to node anchors ... 462
9.3.12 Wells ... 462
9.3.13 Drains ... 462
9.4 Structures and interfaces ... 462
9.4.1 Deformation in structures and interfaces ... 463
9.4.2 Resulting contraction in plates and geogrids ... 463
9.4.3 Resulting forces in beams ... 463
9.4.4 Resulting forces in embedded beams ... 465
9.4.5 Resulting forces in plates ... 465
9.4.6 Resulting forces in geogrids ... 467
9.4.7 Resulting force in anchors ... 467
9.4.8 Resulting stresses in interfaces ... 467
9.4.9 Resulting forces in rigid bodies ... 468
9.4.10 Structural forces in volume piles ... 469

Chapter 10: Curves ... 481

10.1 Selecting points for curves ... 481
10.1.1 Mesh point selection ... 481
10.1.2 Pre-calculation points ... 482
10.1.3 Post-calculation points ... 483
10.2 Generating curves ... 483
10.2.1 Load-displacement curves ... 485
10.2.2 Force-displacement curves ... 486
10.2.3 Displacement-time or force-time curves ... 486
10.2.4 Stress and strain diagrams ... 486
10.2.5 Curves in Dynamic calculations ... 487
10.3 Formatting curves ... 497
10.3.1 Menus for curves ... 498
10.3.2 Editing curve data in table ... 499
10.3.3 Value indication ... 500
10.4 Formatting options ... 500
10.4.1 Chart settings ... 501
10.4.2 Curve settings ... 502
10.5 Regeneration of curves ... 503
10.6 Multiple curves in one chart ... 504

Chapter 11: Engineering Applications ... 505

11.1 Modelling vacuum consolidation ... 505
11.1.1 Vacuum consolidation in a one-dimensional soil column ... 505
11.1.2 Vacuum consolidation in a 2D or 3D model ... 505

11.1.3 Other requirements ... 506
11.1.4 Calculation options ... 506
11.1.5 Switching-off vacuum ... 506
11.2 Modelling ground anchors ... 507
11.2.1 Geometric modelling of ground anchors ... 507
11.2.2 Properties of ground anchors ... 507
11.2.3 Activation and pre-stressing of ground anchors ... 508
11.3 Transverse forces in embedded beams ... 508
11.3.1 Abutment pile ... 509
11.3.2 Quay wall with relieving platform ... 509

Chapter 12: References ... 510

Appendices ... 513

Appendix A: Program and data file structure ... 514
Appendix B: The format required for the files to import CPT data ... 517
Appendix C: Cyclic accumulation Tool - Contour File Formats ... 519
Appendix D: Summary of the creation of General Water Levels ... 524
Appendix E: Conventional and unsaturated soil behaviour in PLAXIS ... 526
Appendix F: Shortcuts Input program ... 537
Appendix G: Shortcuts Output Program ... 539
Appendix H: Python HTTP REST API wrapper ... 542
H.1 Python HTTP REST API wrapper ... 542
H.2 Starting the scripting server ... 542
H.3 Connecting to a scripting server ... 543
H.4 Creating standalone scripts ... 544
H.5 Calling new_server without arguments ... 545
H.6 Disabling the user interface ... 545
H.7 Identifying the type and version of PLAXIS ... 546
H.8 Creating a simple project ... 547
H.9 Performance and logging ... 549
H.10 External manipulations during script execution ... 549
H.11 Error handling mode ... 549
H.12 Installing third-party Python modules ... 550
H.13 Tokenizer ... 551
H.14 Sample Applications ... 552
Appendix I: HTTP REST API ... 568
Appendix J: Calculation warning and errors in PLAXIS ... 586

1 Introduction

PLAXIS 3D is a special purpose three-dimensional finite element program used to perform deformation, stability and flow analysis for various types of geotechnical applications.

The program uses a convenient graphical user interface that enables users to quickly generate a geometry model and finite element mesh.

Users need to be familiar with the Windows environment, and should preferably (but not necessarily) have some experience with PLAXIS 2D. 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 three-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 (https://communities.bentley.com/products/geotechnical1/w/wiki/41123/plaxis).

Some functions are only available in some specific modules. If so, the symbol of the module appears in front of the function's name:

 PLAXIS GSE features available only to users with Geotechnical SELECT subscription .

 Available in Dynamics module only.

 Available in PlaxFlow (Flow module) only.

 A feature that is available as Tech Preview.

2 General Information

Before describing the specific features in the different parts of the PLAXIS 3D 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 and time. These basic units are defined in the **General Settings** tabsheet of the **Project properties** window in the Input program. The default units are meters [m] for length, kilonewtons [kN] for force and days [day] for time, seconds [s] for dynamic time. 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 system of units and the output data should be interpreted in terms of the same 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.

Length	Conversion	Force	Conversion	Time	Conversion
mm	0.001 m	N	0.001 kN	s (sec)	1/86400 day
cm	0.01 m	[kN]	1 kN	min	1/1440 day
[m]	1 m	MN	1000 kN	h	1/24 day
km	1000 m	lbf (pounds force)	0.0044482 kN	[day]	1 day
in (inch)	0.0254 m	kip (kilo pound)	4.4482 kN		
ft (feed)	0.3048 m				
yd (yard)	0.9144 m				

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]

Units are generally only used as a reference for the user but, to some extent, changing the basic units in the **Project properties** window will automatically convert existing input values to the new units. This applies to parameters in material data sets and other material properties in the Input program. It does not apply to geometry related input values like geometry data, loads, prescribed displacements or phreatic levels or to any value outside the Input program. If it is the user's intention to use a different system of units in an existing project, the user has to modify all geometrical data manually and redo all calculations.

The generation of a 3D finite element model in PLAXIS 3D is based on the creation of a geometry model. The geometry model involves a composition of volumes , 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*

Stresses computed in PLAXIS 3D are based on the Cartesian coordinate system shown in Figure 1 (on page 13).

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 13) shows the positive stress directions.

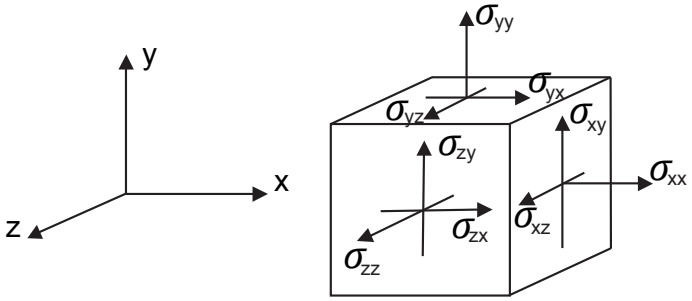


Figure 1: Coordinate system and indication of positive stress components

2.2 File handling

All file handling in PLAXIS 3D is done using a modified version of the general Windows® file requester Figure 2 (on page 13).

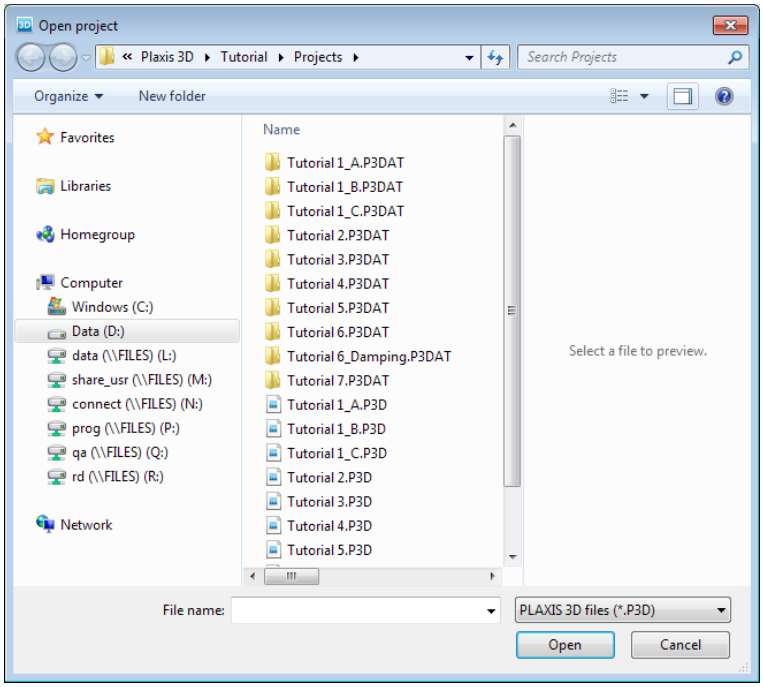


Figure 2: PLAXIS 3D file requester

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 3D project has a structured format and is named <project>.P3D, where <project> is the project title. Besides this file, additional data is stored in multiple files in the sub-folder <project>.P3DAT. It is generally not necessary to enter such a folder because it is not possible to read individual files in this folder.

2.3 Integrated Structural Modelling (ISM) GSE

Note:

This feature is provided as a Technical Preview

Bentley's Integrated Structural Model (ISM) is a technology for sharing structural engineering project information among structural modelling, analysis, design, drafting, and detailing applications. ISM is similar to Building Information Modelling (BIM), but focuses on the information that is important in the design, construction, and modification of the load bearing components of buildings, bridges, and other structures.

PLAXIS 3D can exchange structure information with Bentley Integrated Structural Model (ISM) repositories.

2.3.1 *Purpose of ISM*

There are two related purposes for ISM:

- The transfer of structural information between applications.
- The coordination of structural information between applications.

To provide for the first purpose (transferring information), ISM provides a means of defining, storing, reading and querying ISM models.

To provide for the second purpose (coordination of information), ISM additionally provides capabilities to detect differences between ISM models and to selectively (based on user selection) update either an ISM repository or an application's data to provide a user-controlled level of consistency between the two data sets.

2.3.2 *ISM and Application Data*

ISM is not intended to store all of the information that all of its client applications contain. Rather, it is intended to store and communicate a consensus view of data that is common to two or more of its client applications, such as PLAXIS 3D.

PLAXIS 3D continues to hold and maintain its own private copy of project data. Some of the application data will duplicate that of the associated ISM repository. The application data may even conflict with that in the ISM repository. PLAXIS 3D (or the user) may decide that a conflict gives the best data for the different uses in PLAXIS 3D versus ISM.

The following lists show which information is supported for use with ISM.

Supported PLAXIS 3D object types:

- Beam (not curved)
- Point (only those belonging to a beam)
- Surface (not curved)
- Plate (flat, may contain holes, but no curved edges like arcs)
- Materials (linear-elastic only) for the above

Supported ISM object types:

- CurveMember (Not curved)
- SurfaceMember (Not curved)
- Node
- ParametricSection
- TableSection
- CustomSection
- Concrete
- Steel

Supported ISM object properties:

- Id
- ShortId
- Location
- Material
- Area
- Diameter
- Thickness
- Depth
- Width
- Orientation
- Section
- SectionName
- StandardsOrganization
- Manufacturer
- Revision
- Nodes
- Members
- Openings
- ParametricShapeKind
- PlacementSurface
- PoissonsRatio
- UnitMass
- ElasticModulus
- SpecifiedElasticModulus

2.3.3 *ISM synchronizer tools*

PLAXIS 3D can send structural data to and from an ISM repository through a set of ISM Syncing tools. These tools allow both creation and updating of PLAXIS 3D models as well as ISM repositories. Further, these flexible tools allow you to begin models and move data as your workflow dictates.

These tools are available from the **File** > **ISM** menu:

-  Create ISM Repository. Transfers the current model opened in PLAXIS 3D and generates a new ISM repository. This is the most common way in which an ISM repository is initially created.

- New From ISM Repository. Creates a new PLAXIS 3D model from an existing ISM repository. This is used to transfer model data from other tools used in the workflow.
- Update ISM Repository. Will coordinate changes made to the model in PLAXIS 3D and coordinate some or all of those changes with an existing ISM repository.
- Update From ISM Repository. Allows the user to update a PLAXIS 3D model with some or all of the changes which have been made to the ISM repository.

2.3.4 *ISM synchronizer*

Before using the ISM tools, you have to start the Remote Scripting Server first, by going to the menu **Expert** > **Configure Remote Scripting server** and starting the server.

When using one of the synchronizer tools, the ISM synchronizer is opened, see Figure 3 (on page 16) .

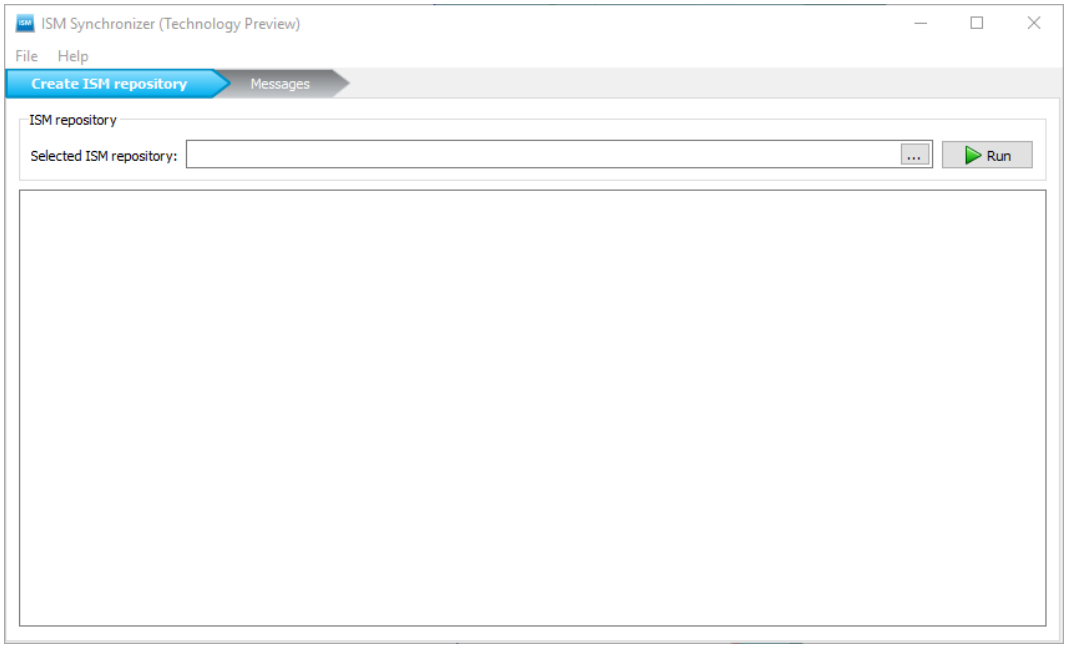


Figure 3: ISM Synchronizer

To choose the repository, click the ellipsis button (...) and browse for the folder and file.

Once the repository is set, click the *Run* button to create or update a repository. Whenever necessary, the Structural Synchronizer will open in a new window.

2.3.5 *Structural Synchronizer*

Note: Structural Synchronizer needs to be installed. The software can be downloaded from Bentley's Connect Center.

When running a tool from the ISM synchronizer, the Structural Synchronizer will be opened.

The program provided by ISM for accepting or rejecting model data changes is called "Structural Synchronizer". Structural Synchronizer provides the user with a powerful set of tools for moving data between the applications used in a daily workflow. Even relatively small structural models have enormous amounts of data and ISM allows this data to be re-used with ease. Care must be taken that only desired data is transferred between applications.

When accepting changes made by client applications to an ISM Repository, some attention must be paid to what changes are actually being made. A small change in a client application can have unintended repercussions, if accepted. A repository is intended to represent the data that is common to all the client applications. Some client application models will use only a subset of the repository data but changes made to them can affect the entire repository if these changes are accepted when a repository update action is performed.

Note: This brief overview is not intended as a sufficient description on using the full capabilities of Structural Synchronizer. Please refer to the Structural Synchronizer product documentation for more detailed information.

2.4 PLAXIS Coupling Tool

The PLAXIS Coupling Tool enables users to integrate the structural FEM package STAAD into PLAXIS 3D.

Mostly, the superstructure (above ground level) is modeled in structural FEM software by structural engineers and the substructure (below ground level) is modeled by PLAXIS 3D, and that way the flow of information between systems is decoupled.

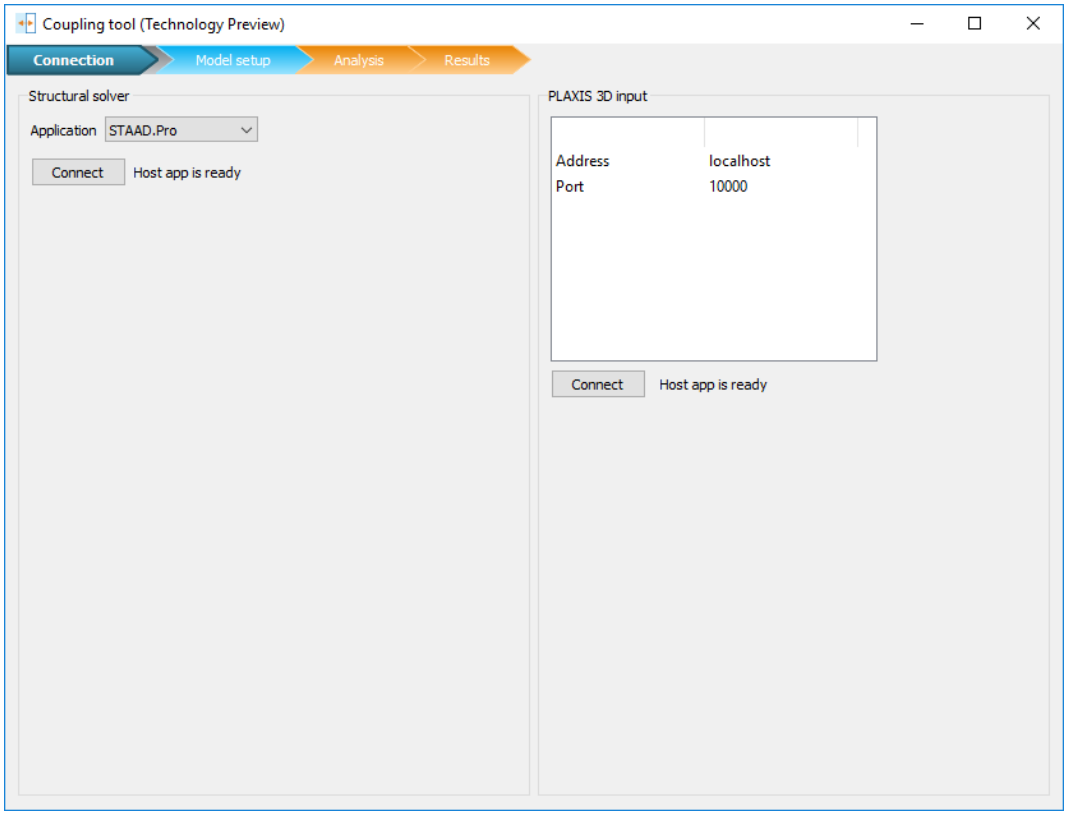
As a consequence, simplifying assumptions are made to be able to handle the interaction calculations between super- and substructure. While this works for simple cases, more complicated cases require a coupled analysis.

The PLAXIS Coupling Tool performs an iterative analysis, which will converge to a solution that satisfies boundary conditions in both models (super- and substructure). The assumption for this type of analysis is that the user has modeled the superstructure in a structural FEM package (STAAD.Pro) which provides an API, and the substructure is modeled in PLAXIS 3D.

2.4.1 *Coupling tool - Connection*

To start working with the Coupling tool and connect PLAXIS 3D and STAAD.Pro do the following:

1. Start STAAD and open a project.
2. Start and open a project.
3. In PLAXIS 3D go to the *Expert > PLAXIS Coupling tool* menu to open the Coupling Tool.
4. The tool needs a running remote scripting server. If there is no server started yet, the **Current state** server window will open. Set the *Port* and *Password* and click the *Start server* button. Once the server has started, click the *Close* button to close the window.
5. The **Coupling tool** opens on the **Connection** tab, see below. The **Connection** tab shows the details about the Structural solver and the PLAXIS 3D input.
6. Click the *Connect* button to connect to the Structual solver you have chosen in *Application*.



2.4.2 *Coupling Tool - Model setup*

The **Model setup** tab (see Figure 4 (on page 19)) is used to prepare both models (Structual solver and PLAXIS 3D) for the analysis.

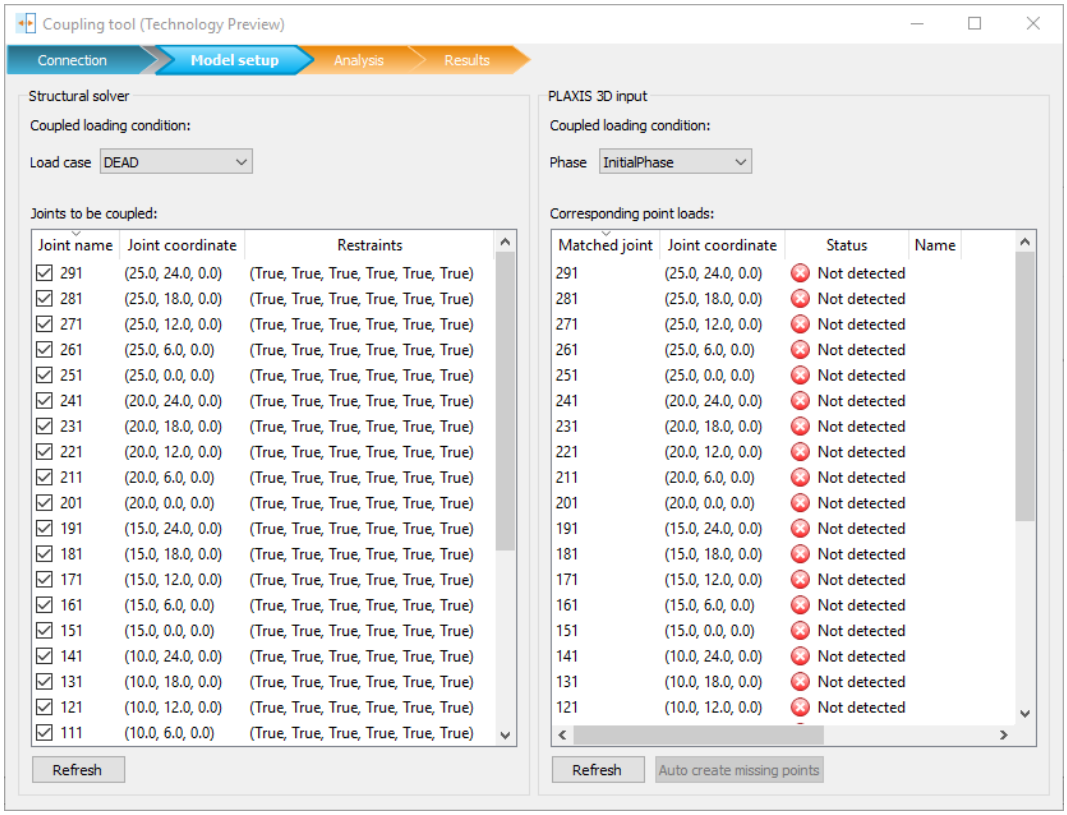


Figure 4: Model Setup Start

2.4.3 *Load condition*

First the load condition needs to be set. For the structural solver the *Load case* has to be selected and for PLAXIS 3D the phase has to be selected.

The phase for PLAXIS 3D is the stage in which the foundation of the model is included. For example, the user has created the following 3 stages for a simple structure foundation construction system:

1. Initial phase
2. Foundation construction phase
3. Foundation construction phase and Structure built

2.4.4 *Interaction elements*

The second step is the definition of the interaction elements, by mapping a structural joint with a restraint (structural model) to a point load (PLAXIS 3D model), within a certain coordinate tolerance.

The requirement is that, each joint must have a corresponding point load. PLAXIS 3D will detect all the joints in the structural model that have a restraint defined, and present them in the left table.

In this table, you can choose the joints to include by setting a check mark. The table also allows you to detect if joints are missing. In that case, go back and modify the structural model (in the structural solver) and click the *Refresh* button, which will re-detect all of the joints. After determing the structural joints to be coupled are

determined, you have to check if each of them has a corresponding point load. The point loads are shown in the table on the right.

The tool will detect all of the point loads in the stages mode of PLAXIS 3D. If there are missing point loads, these point loads with their coordinates will be presented in the table on the right, and there are two options. You can create the missing point loads in PLAXIS 3D and click the *Refresh* button. Alternatively let the Coupling tool create them by clicking the *Auto create* button.

Note that, using the first option, you have control over the mesh. For the second option, PLAXIS 3D automatically add the points and mesh the model again.

At any point during the model setup, if you try to proceed to the **Analysis** tab, the tool will check whether the load conditions and interaction elements are complete. If they are not complete, a dialog with information on missing settings is shown and progressing to the **Analysis** tab is prevented. If the checks are ok, the tool will automatically launch PLAXIS 3D output and proceed to the analysis tab.

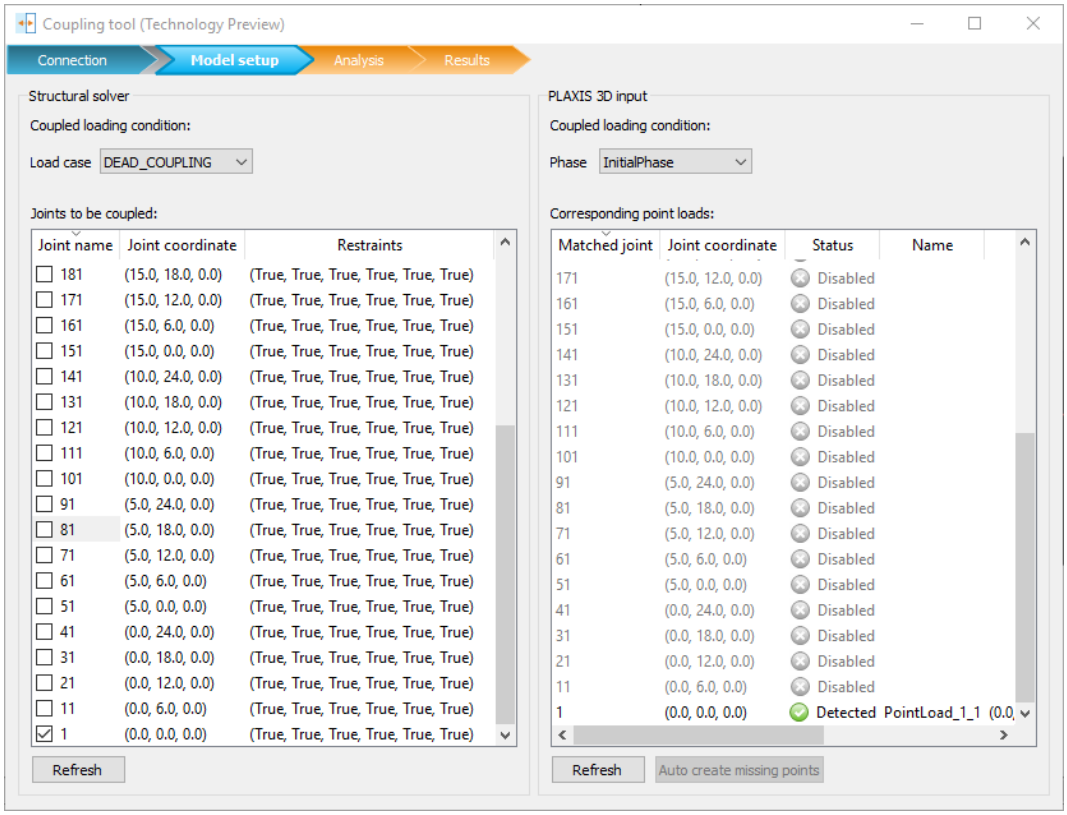


Figure 5: Model Setup after creating point loads

2.4.5 *Coupling Tool - Analysis*

The **Analysis** tab shows information on the analysis control parameters and convergence criteria.

Click the *Run* button to start the iteration. You can watch the progress of the iterations on the middle section, including status messages. Clicking the corresponding buttons, you can stop or pause the iterations in case you estimate the error is enough, or see the analysis diverge, even though there are predetermined convergence criteria.

Note: The pause or stop button will not stop calculations immediately, but wait until the iteration ends. This could take a significant amount of time.

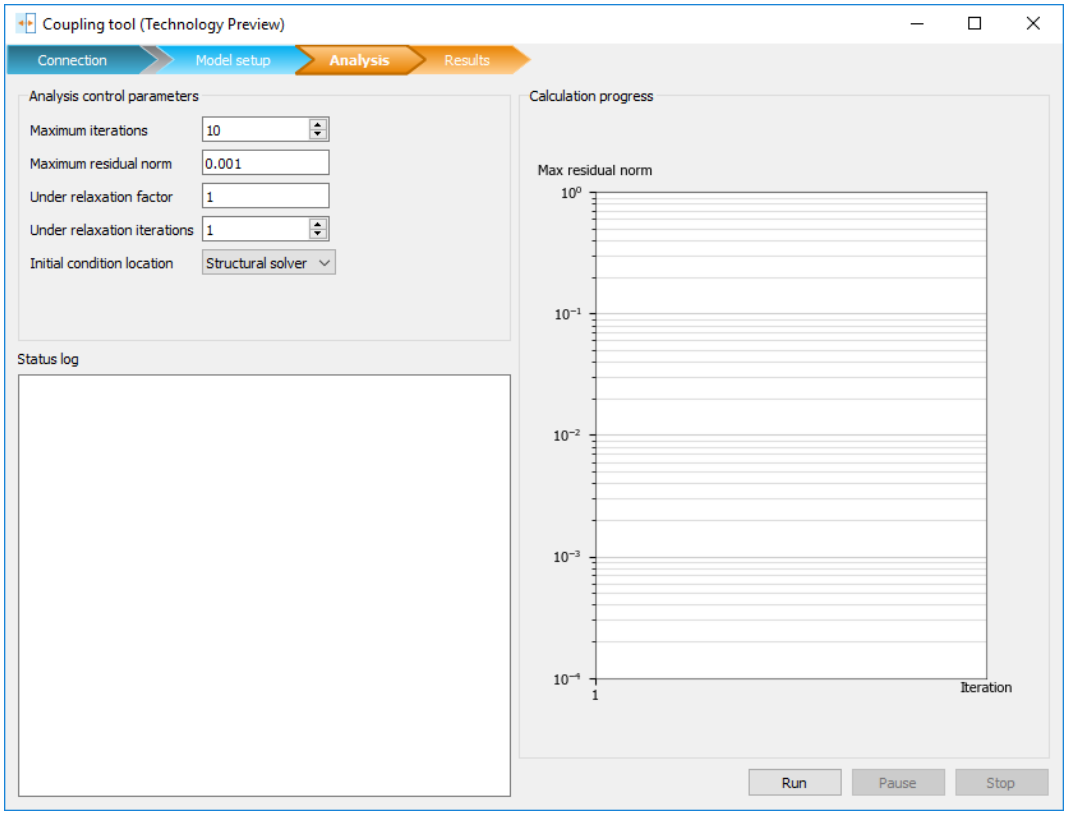


Figure 6: Coupling tool -Analysis

2.4.6 *Coupling Tool - Results*

On the *Results* tab, you see a table with the iteration results. It shows the results for the interaction elements, which are only the coupling related results.

The table shows the following values: error, displacement values, forces.

Model related results are in PLAXIS 3D Output and the structural solver if you wish to check them.

You can also calculate the linear equivalent spring stiffness in z direction per joint by clicking on the calculate spring constants button, and they will be presented in the table in a column. After checking and if necessary modifying the stiffness values, you will be able to import the springs to the structural solver by clicking the *Import springs* button. The springs cannot be imported into the model that is used for analysis. You have to save it in a different model.

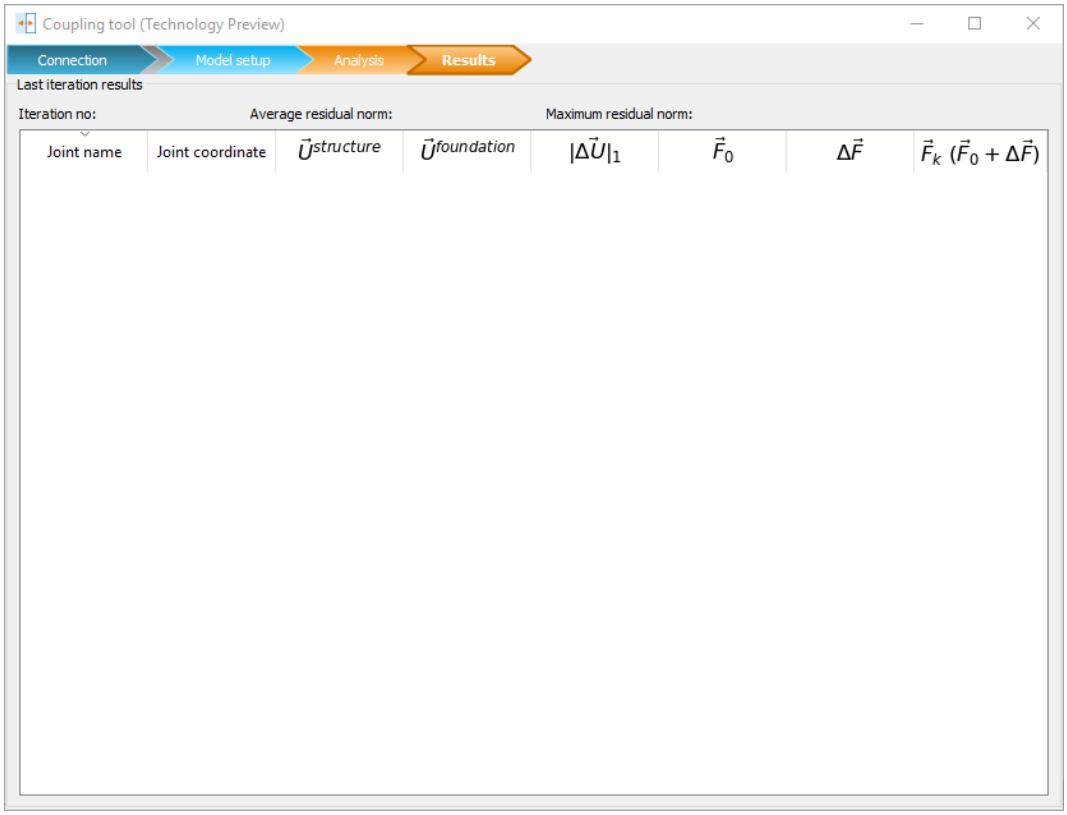


Figure 7: Coupling tool results

2.5 Help facilities

To inform the user about the various program options and features, PLAXIS 3D 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 47) and Help menu (on page 420) 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. Feedback from the program on the user actions is given in the command line area. 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 3D program, the user has to create a 3D geometry model composed of points, lines, surfaces, volumes, and other components 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 23 (on page 39)). 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 23 (on page 39)).

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 3D by double clicking the shortcut on your desktop or clicking the entry in the **Windows** > **Bentley Geotechnical** menu.

PLAXIS 3D will first check if you are entitled to use the software and present you with the available options and licences, see Figure 8 (on page 24).

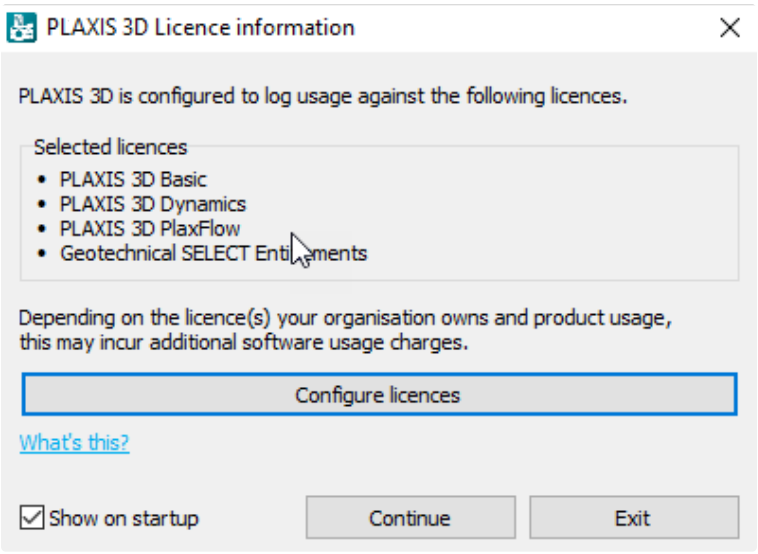


Figure 8: Licence check

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

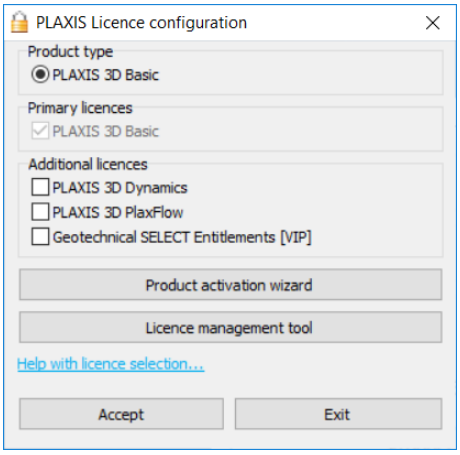


Figure 9: Licence configuration

In the configuration dialog, see Figure 9 (on page 24), 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 3D is open, you have to restart PLAXIS 3D 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 3D.

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 10 (on page 25)).

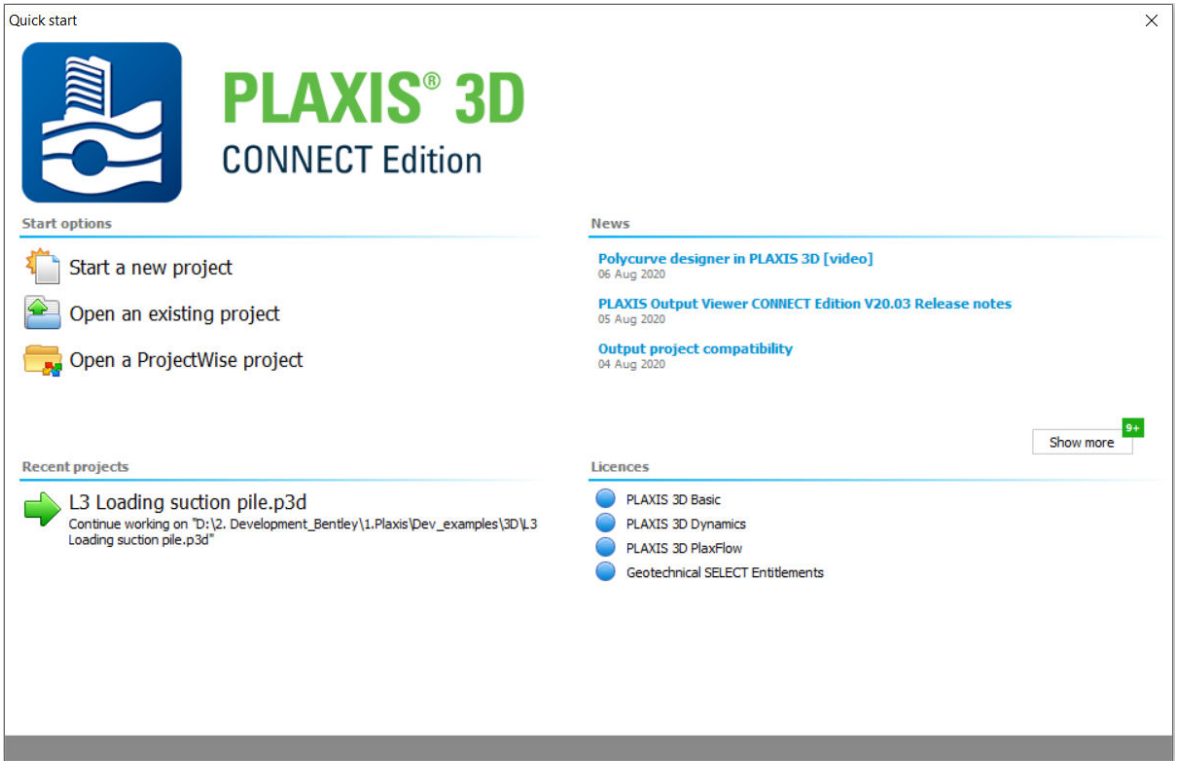


Figure 10: Quick start window

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 11 (on page 26)) contains the project name and description and offers the possibility to select a company logo.

The **Model** tabsheet (Figure 12 (on page 27) contains the basic units for length, force and time (see Units (on page 11)), the direction and the value of gravity, 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

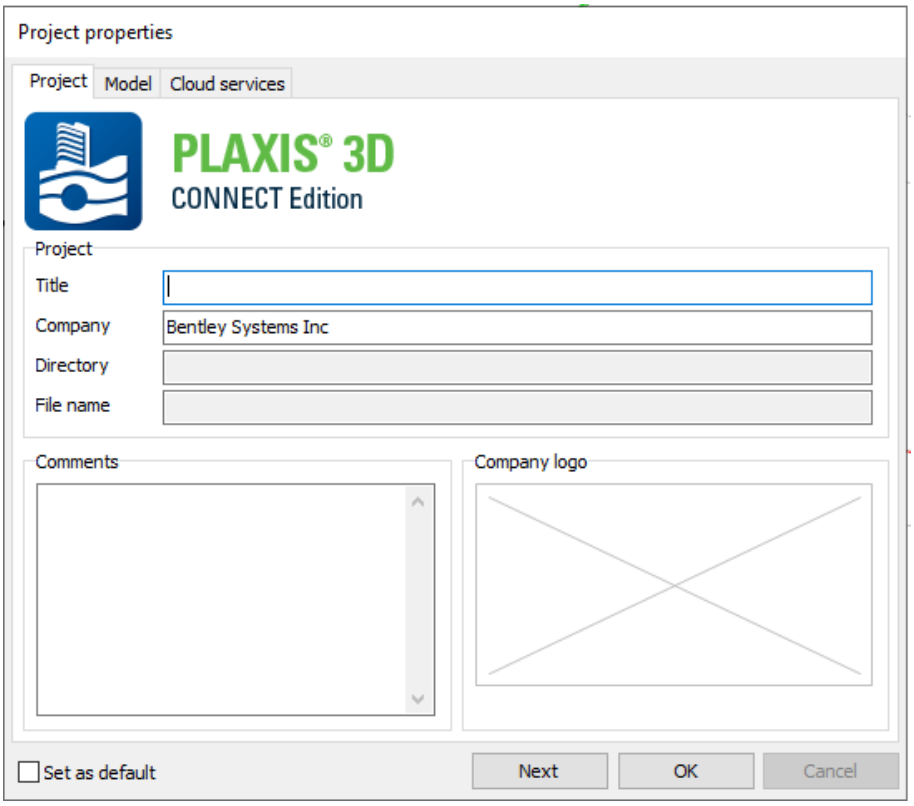


Figure 11: Project properties window (Project tabsheet)

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.

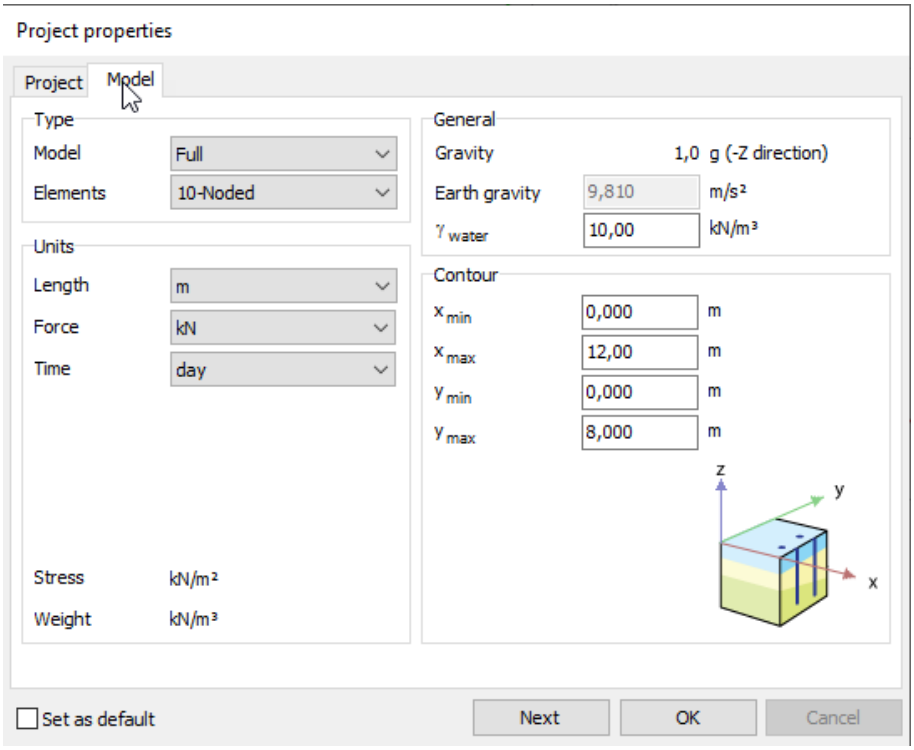


Figure 12: Project properties window (Model tabsheet)

1. **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 z-axis. Gravity is implicitly included in the unit weights given by the user (Material properties and material database (on page 191)). In this way, the gravity is controlled by the total load multiplier for weights of materials, $\Sigma Mweight$ (General phase parameters (on page 356)).

   In dynamic calculations, the value of the gravity acceleration, g, is used to calculate the material density, ρ, from the unit of weight, γ ($\rho=\gamma/g$).

2. **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 *Model* 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.

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 and day for time. The corresponding units for stress and unit weights are listed in the box below the basic units.

In a dynamic analysis, the time is usually measured in [seconds] rather than the default unit [days]. In PLAXIS 3D *Time* and *Dynamic time* are different parameters. The time interval in a dynamic analysis is always the dynamic time and PLAXIS 3D always uses seconds [s] as the unit of *Dynamic time*.

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 horizontal 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. The contour of the geometry can also be modified in the drawing area when the *Soil* mode is selected.

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 13) pops up. It enables the user to browse through all available folders and to select the desired PLAXIS 3D project file (*.P3D). After the selection of an existing project, the corresponding geometry is presented in the main window.

An existing PLAXIS 3D 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 3D files *.P3D'.

3.1.3 *Packing a project*

 The created project can be compressed using the **Pack project** application (Figure 13 (on page 29)), which is available in the *File* menu of the Input program. This application can be executed directly from the PLAXIS 3D installation folder by double clicking the corresponding file (PackProject.exe). A shortcut to the application can be created as well.

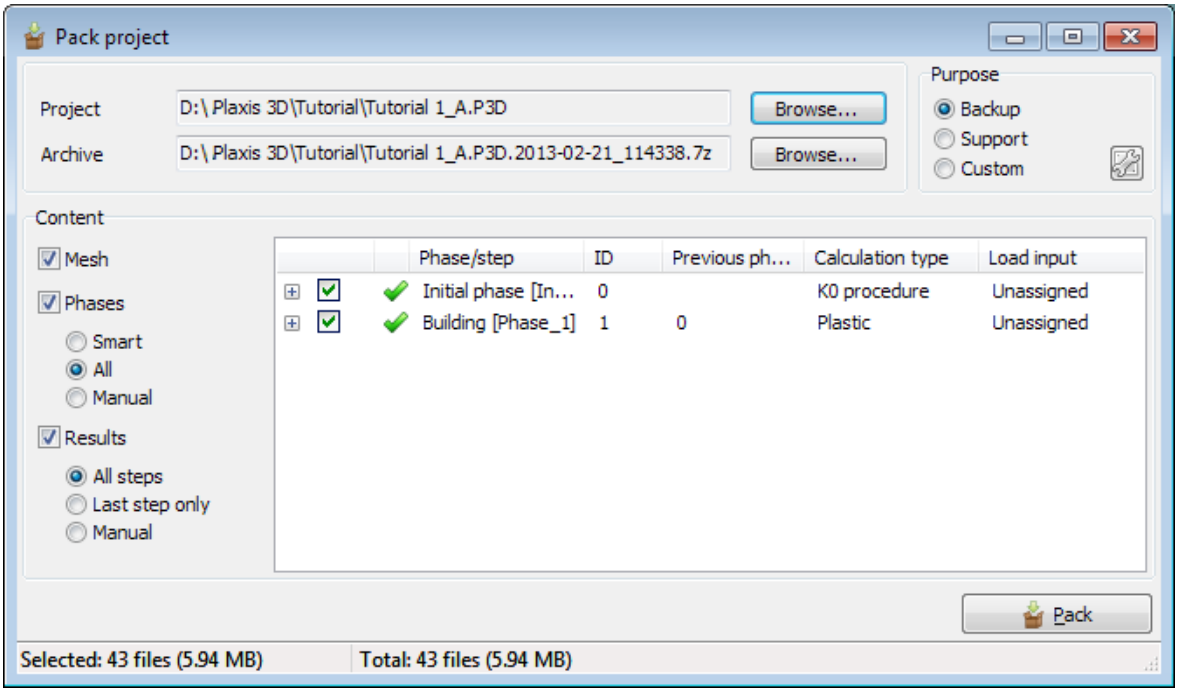


*Figure 13: **Pack project** window*

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 14 (on page 30)), displayed by clicking the button in the *Purpose* box.

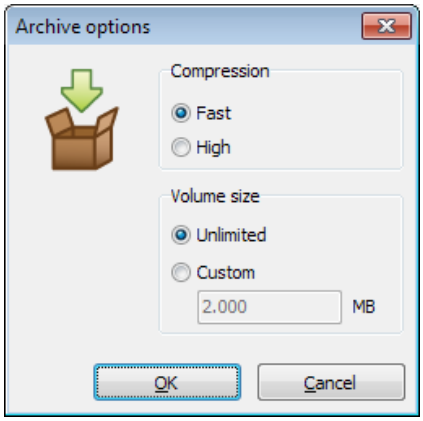


*Figure 14: **Archive options** window*

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.

3.2 ProjectWise Integration Commands GSE

When ProjectWise has been installed, four specific Integration commands are available in PLAXIS 3D.

- **Open...**
- **Add...**
- **Save...**
- **Properties...**

These commands are accessible through the **File** > **ProjectWise** menu and the toolbar.

3.2.1 Add File to ProjectWise

To add the current PLAXIS 3D 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**.

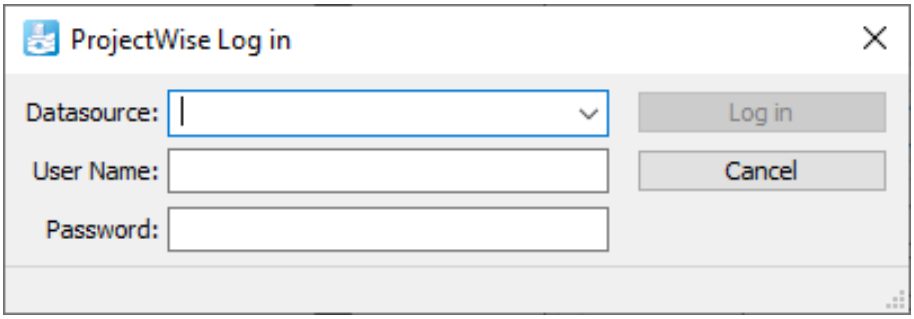


Figure 15: ProjectWise Log in dialog

Select a Wizard dialog appears, as shown in Figure 16 (on page 31). Select any of the options - No Wizard or Advanced Wizard.

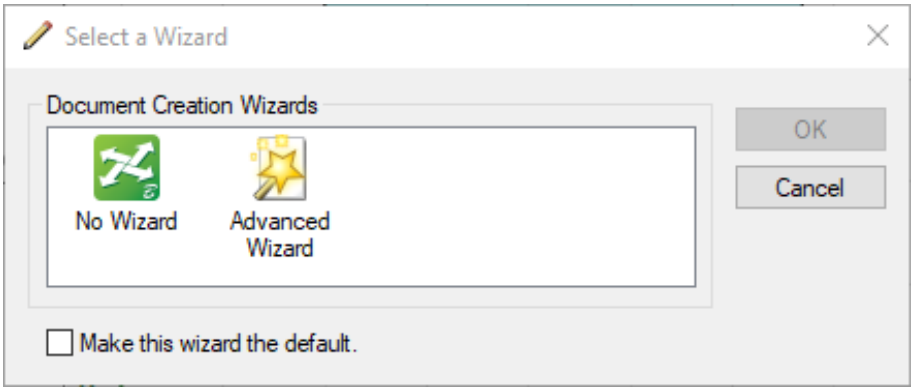


Figure 16: Select a Wizard dialog

Selecting **No Wizard** opens **Save Document As** window (Figure 17 (on page 32)). Select the target folder. Fill in Document name and Description. Format type should be Packed PLAXIS 3D Project.

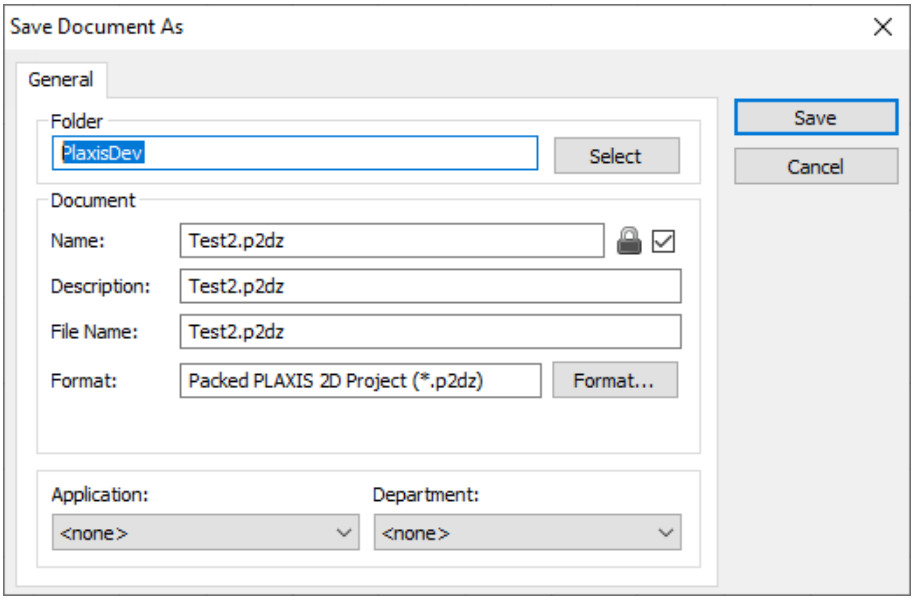


Figure 17: Save Document As dialog

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 3D 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 18 (on page 33)), where the user can choose the corresponding files in the project to be compressed and uploaded to ProjectWise .

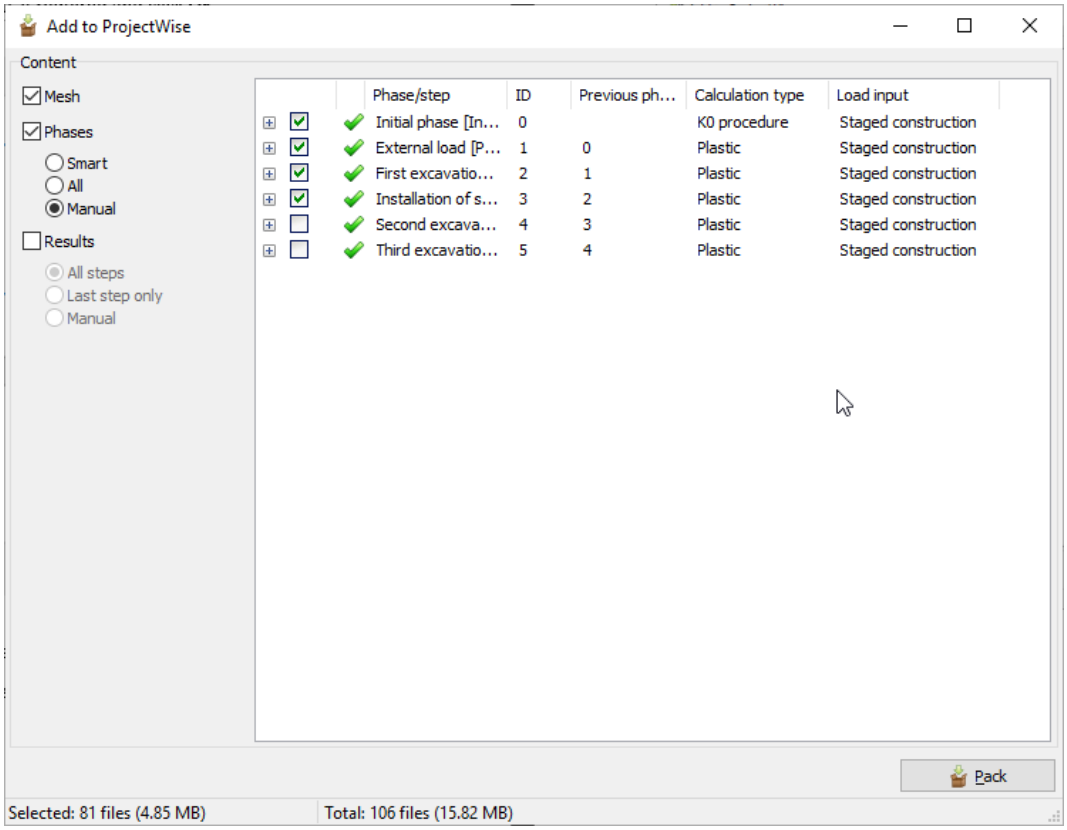


Figure 18: Add to ProjectWise

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.

3.2.2 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 19 (on page 34)) is displayed, allowing the user to browse/search the datasource contents and select the document to open.

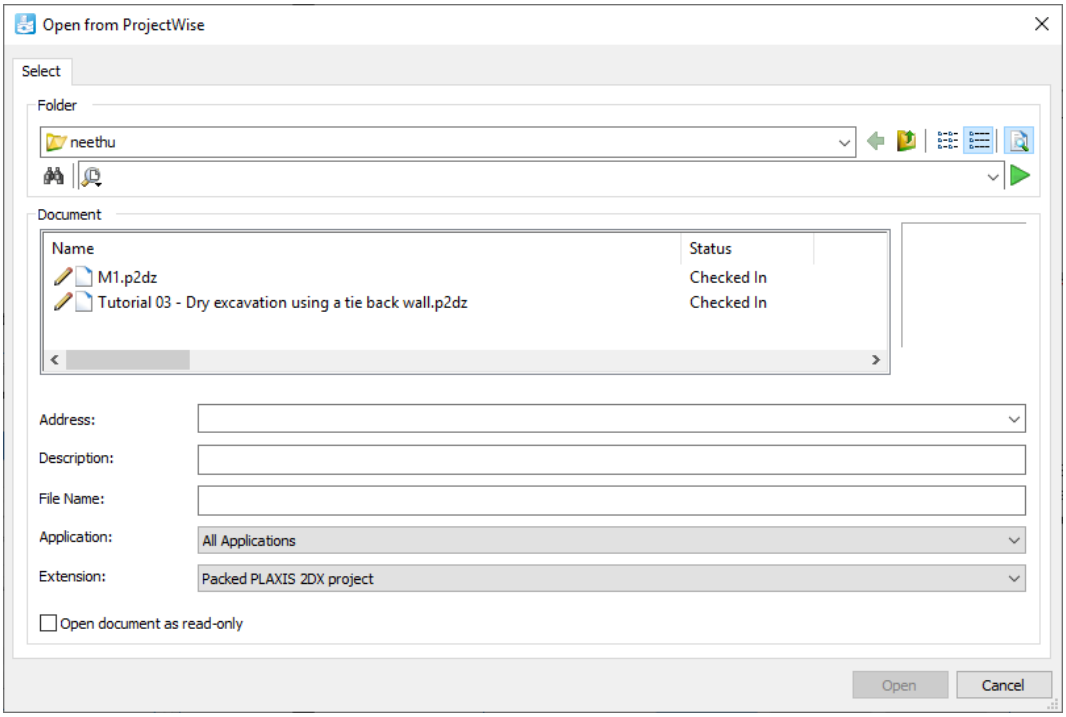


Figure 19: Open from ProjectWise

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.

3.2.3 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.

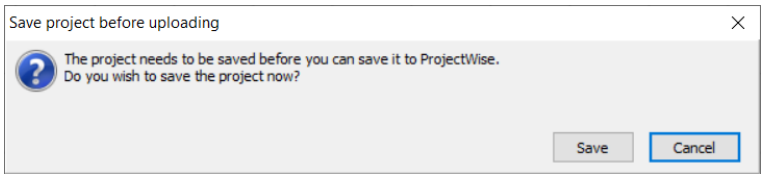


Figure 20: Save project dialog

2. Click **Save**.
3. Refer to Add File to ProjectWise (on page 31) for further steps.

A Progress bar is displayed during the data transfer indicating file content transfer status.

3.2.4 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**.

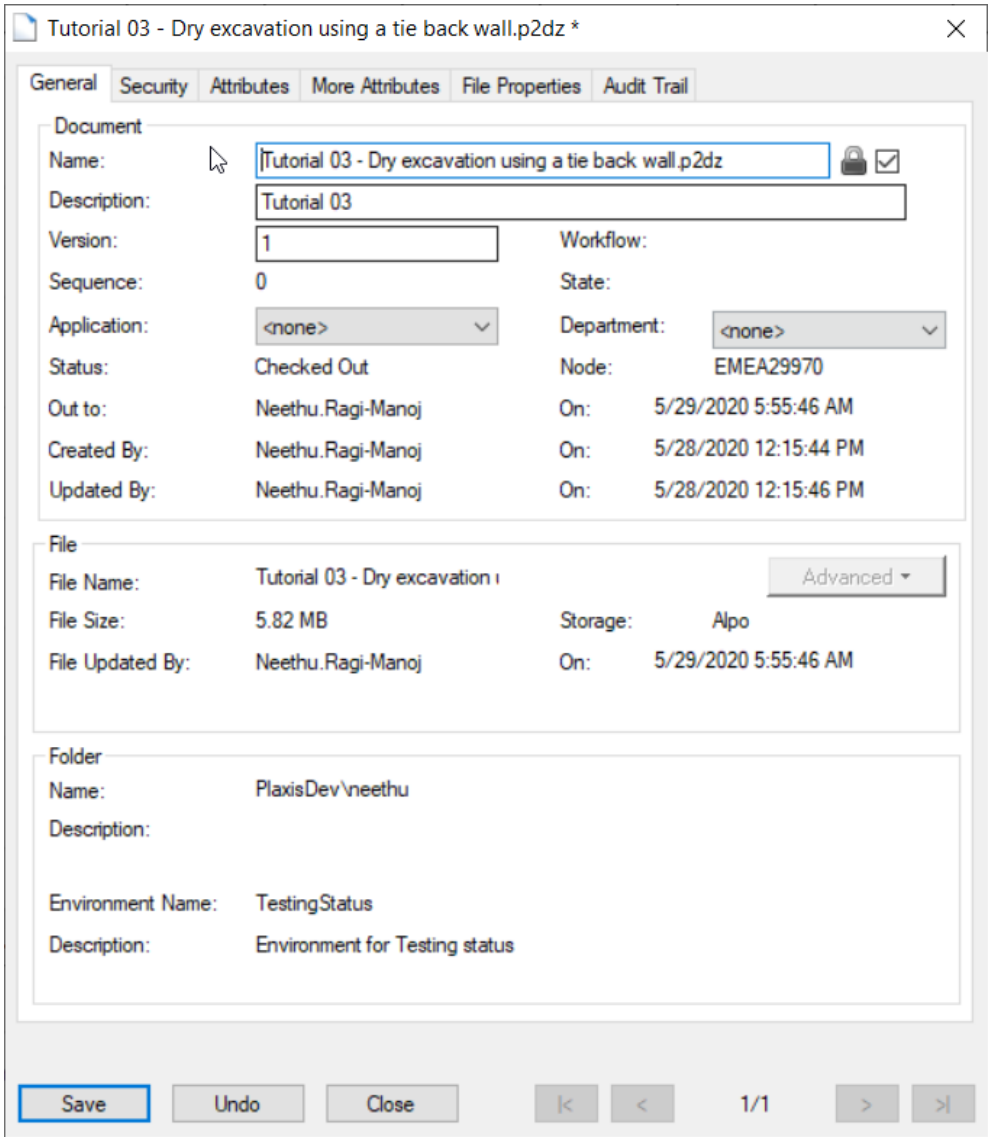


Figure 21: ProjectWise Document Properties

A Progress bar is displayed during the data transfer indicating file content transfer status.

3.3 Bentley Cloud Services

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**.

3.4 Exporting geometry GSE

3.4.1 *STEP and DXF format*

PLAXIS 3D allows to export the geometry of the model in different formats. This is available in the **File** menu of the input program (Figure 22 (on page 37)) .

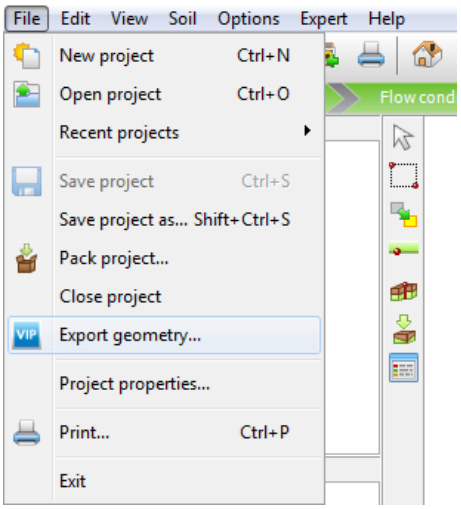


Figure 22: Export geometry

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.

3.4.2 *STL format*

PLAXIS 3D also allows for storing volume and surface objects in *STL format*. This enables, for example, the visualisation of a PLAXIS 3D model using external 3D visualisation software, or to create a 3D print of a PLAXIS 3D model.

In order to create STL files of your PLAXIS 3D model, the following command can be used:

__saveobjects 'stl' '<folder>' (Note the double underscore!) where <folder> is the name of the folder on the computer or network drive where the STL-files are to be stored (for example 'c:/temp'). Each volume and surface object is stored in a separate STL-file.

Note that for 3D printing only volumetric objects can be used.

3.5 Layout of the Input program

The general layout of the Input program for a new project is shown in Figure 23 (on page 39). The main window of the Input program contains the following items:

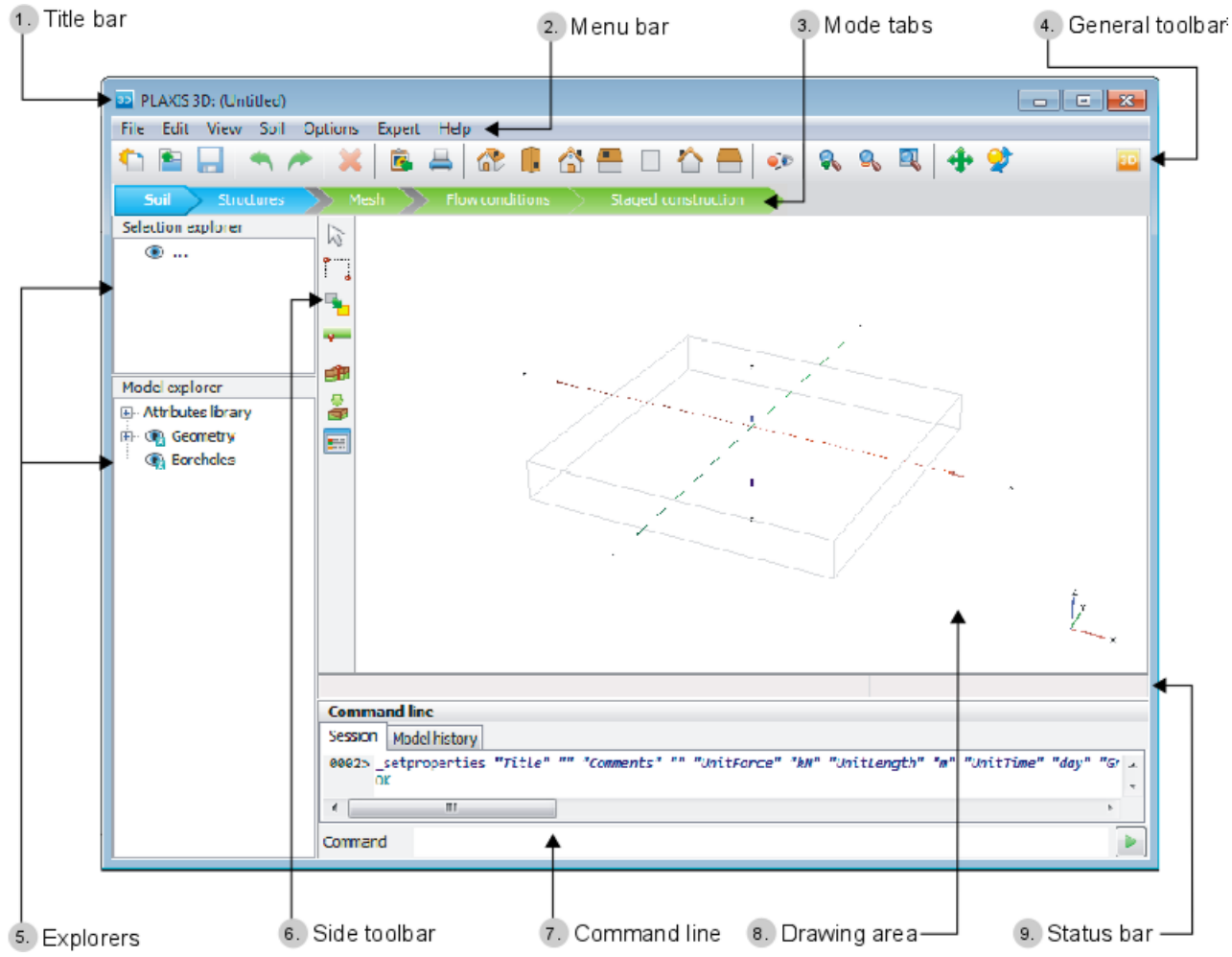


Figure 23: Layout of the Input program

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	The soil stratigraphy is defined.
Structures	The structural components of the model are defined.
Mesh	The mesh properties are defined and the geometry model is discretised and transformed to a finite element model.

Flow conditions	Water levels are defined.
Staged construction	The calculation settings are defined and the project is calculated.

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 48).

4. ***General toolbar***

   The general toolbar contains buttons for general actions such as file operations, printing 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	It displays the properties of the selected geometry objects and the features assigned to them and gives the possibility to hide/show, activate/deactivate these objects (Selection explorer (on page 67)).
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 (Model explorer (on page 62)).
Phases explorer	It shows the list of the calculation phases and enables adding, inserting, deleting or modifying a calculation phase as well as the dependencies of all calculation phases (Phases explorer (on page 67)).

   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 3D 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 49).

9. ***Status bar***

   The status bar displays information about the location of the mouse cursor in the drawing area, the model object it is located on and a hint on snapping when a drawing tool is selected.

3.6 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 48)). The menus available in the Input program are:

3.6.1 *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.
Pack project	To compress the current project.
Close project	To close the current project.
Bentley cloud services	To access to the different cloud services.
ProjectWise	To integrate PLAXIS projects with ProjectWise for infrastructure design collaboration.

ISM GSE	To create or update a (new) ISM repository (Integrated Structural Modelling (ISM) (on page 14)).
Export geometry GSE	To export the current project. Available in **Soil** and **Structures** modes only.
Project properties	To activate the **Project properties** window (Project properties (on page 26)).
Print	To print the geometry model on a selected printer.
Exit	To leave the Input program.

3.6.2 *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.

3.6.3 *View menu*

Pan camera	To move the view in the drawing area. Note that the model is not relocated.
Rotate camera	To rotate the view in the drawing area. Note that the model is not reoriented.
Zoom in	To zoom into a rectangular area for a more detailed view. After selection, click on the location to zoom in, or indicate the zoom area 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. Alternatively, the mouse scroll wheel may be used for zooming.
Zoom out	To zoom out of the current view, proportionally to the drawn square size.

Reset zoom	To restore the full drawing area.
Default views	To change the orientation of the model to the views defined by the program.
Explode	To virtually disassemble the components of the project geometry. This option is available only in the **Calculation** modes.
Implode	To virtually re-assemble the components of the project geometry. This option is available only in the **Calculation** modes.

3.6.4 *Soil menu*

Modify soil layers	To modify the soil layers in the model using the borehole facility.
Select	To select boreholes and volumes in the model (Selecting geometry components (on page 90)).
Select multiple objects	To select boreholes and volumes in the model located in the region defined using the tool (Selecting geometry components (on page 90)).
Move object	To relocate boreholes in the model.
Adjust soil contour	To modify the soil contour.
Create borehole	To create a new borehole.
Import soil 	To import predefined soil volumes.
Show material	To open the material data base with material data sets.

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

3.6.5 *Structures menu*

Show dynamic multipliers 	To open the **Dynamic multipliers** window.
Show movement functions	To open the **Movement functions** window, see Moving loads (on page 135) for more info.

Select	To select entities in the model. The feature is described in Selecting geometry components (on page 90).
Select multiple objects	To select entities in the model located in the region defined using the tool Selecting geometry components (on page 90).
Move object	To move or rotate structural entities in the model (Move object (on page 91) and Rotate object (on page 91)).
Extrude object	To extrude a selection of entities (Extrude object (on page 92)).
Create array	To create multiple copies of a selection of entities (Copy object - Array (on page 95)).
Create point	To create a point in the model (Create point (on page 97)).
Create line	To create a line in the model (Create line (on page 98)).
Create surface	To create a surface in the model (Create surface (on page 105)).
Start designer	Has sub-menus: the menu **Creat polycurve** to create a single continuous curve composed of lines and arcs in the model (Definition of tunnel cross section geometry (on page 156)) and the menu **Creat tunnel** to start Tunnel designer (Tunnels (on page 153)).
Create load	To create a load in the model (Loads (on page 117)).
Create prescribed displacement	To create a prescribed displacement in the model (Prescribed Displacements (on page 122)).
Create hydraulic conditions	To create hydraulic conditions in the model (Hydraulic conditions (on page 176)).
Import 	Sub-menus to import predefined structures composed of volumes and surfaces or to import point clouds.
Show materials	To open the material data base with material data sets.
Check geometry	To check for geometric inconsistencies in the model.

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

3.6.6 *Mesh menu*

Select	To select entities in the model Selecting geometry components (on page 90).

Select multiple objects	To select entities in the model located in the region defined using the tool Selecting geometry components (on page 90).
Check geometry	To check for geometric inconsistencies in the model.
Refine mesh	To refine the mesh locally.
Coarsen mesh	To coarsen the mesh locally.
Reset local coarseness	To reset the fineness factor of the mesh locally to the default value.
Check geometry	To check the geometry consistency
Generate mesh	To generate the mesh for the defined geometry.
View mesh	To display the generated mesh for the defined geometry.
Select points for curves	To select points to be considered in the plots.

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

3.6.7 *Flow conditions menu*

Select	To select entities in the model Selecting geometry components (on page 90).
Select multiple objects	To select entities in the model located in the region defined using the tool (Selecting geometry components (on page 90)).
Move object	To relocate a user water level in the model (Move object (on page 91)).
Create water level	To create a water level in the model.
Import water level 	To import predefined water levels. More information on importing geometry is available in Importing geometry (on page 113).
Preview phase	To preview the phase selected in the **Phases** explorer.

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

3.6.8 *Phases menu*

Edit phases	To open the **Phases** dialog window.
Show materials	To open the material data base with material data sets.
Show dynamic multipliers 	To open the **Dynamic multipliers** window.
Show flow functions	To open the **Flow functions** window.

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

3.6.9 *Options menu*

Snap to grid	To enable or disable snapping to grid.
Show grid and ruler	To show or hide grid and ruler.
Show cursor location	To show the cursor location in the drawing area.
Show local axes	To show or hide local axes. Note that different colours are used to differentiate the local axes. Red, green and blue indicate the local axis 1, 2 and 3 respectively.
Visualization settings	To modify visualisation settings. The settings of the visualization of the program may be changed in the **Visualization settings** window.
Display numbers using	To select the *Significant digits* for the project.
Autocomplete 	To activate or deactivate the autocomplete function in the command line.
Licencing information	To switch on or off showing the licence information at startup.

3.6.10 *Expert menu* 

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 (on page 542) and HTTP REST API (on page 568)).
Run Python tool	To configure remote scripting server and run python scripts (Appendices Python HTTP REST API wrapper (on page 542) and HTTP REST API (on page 568)).
Python	To **Configure Python Interpreter**, open and use **Interpreter**, **Editor**, **Command prompt**, **Jupyter notebook** and to run python scripts (Appendices G and H).
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 (on page 542) and HTTP REST API (on page 568)).
Calculation manager	To easily run unattended calculations on projects (Typically less than 10 projects).
Plaxis Coupling Tool 	To start the coupling tool to integrate the structural FEM package STAAD.Pro into PLAXIS 3D.
View files	To display the contents of the (binary) files used in the current project.

3.6.11 *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 product page.
News Feed	To launch the PLAXIS latest news
Disclaimer	The complete disclaimer text is displayed.
About	Information about the program version and licence are displayed.

3.7 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.

3.7.1 *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 191). Note that the model contour and 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, beams), interfaces or loads, can be assigned to geometric entities only in the **Structures** mode.

3.7.2 *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	Besides water levels generated from the water conditions defined in the **Soil** mode, user-defined water levels can be specified and modified in this 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.

3.8 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.

3.8.1 *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. Other options for zooming are available in the *View* menu or in the toolbar.

-  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 zoom out from the current view, click the *Zoom out* button or select the corresponding option in the *View* menu.
-  To restore the original view, click the *Reset zoom* button.

3.8.2 *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 *Ctrl* 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 , beams 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 56)).

3.8.3 *Model View - Visualisation settings*

The general settings of the view are arranged in the **Visualization settings** window, which consists of two tabsheets where the view options and the default visibilities can be defined.

Note: The **Visualization settings** window is displayed as the corresponding option is selected in the *Options* menu.

View options

The display of the supporting tools (symbol, grid, global axes, etc.) in the view and their presence in the drawing area is defined in the **Visualization settings** window in the *Options* menu.

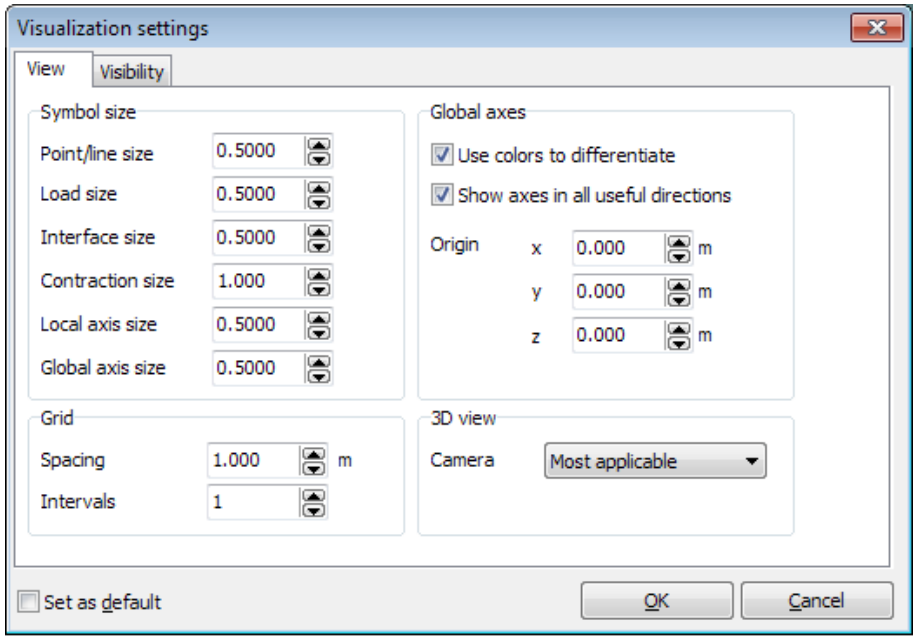


Figure 24: View options for drawing area

Symbol size: The size of the symbols is defined in the **Symbol size** dialog box. The default values are as shown in Figure 24 (on page 51). These values can be used as a reference to get the preferred display.

Grid: The grid spacing and the number of intervals are available in the **Grid** dialog box. To display the grid and ruler select the corresponding option in the *Options* menu. Points indicating the grid are shown in the rulers which are displayed at the mouse cursor when a drawing tool is selected.

Global axes: The availability, the display and the origin of the global axis in the drawing area are defined in the *Global axes* box.

3D view: The view of the model can be arranged according to the selected option in the **Visualization settings** window. The options available are:

Most applicable	The perspective view of the model is displayed only when the corresponding option is selected as default orientation (Model View - Orientation (on page 52)). For the other default view options the objective view of the model is displayed.
Always perspective	The perspective view of the model is displayed regardless of the options selected as default orientation (Model View - Orientation (on page 52)). The perspective view mimics real life such as closer objects appear larger than far away objects, and parallel lines vanish toward the horizon.
Always orthogonal	The model will always be displayed such that two same-size objects will appear the same size no matter how far apart they are from each other or from the camera.

Default visibilities: The default visibility of the objects in the model can be defined in the **Visibility** tabsheet (Figure 25 (on page 52)).

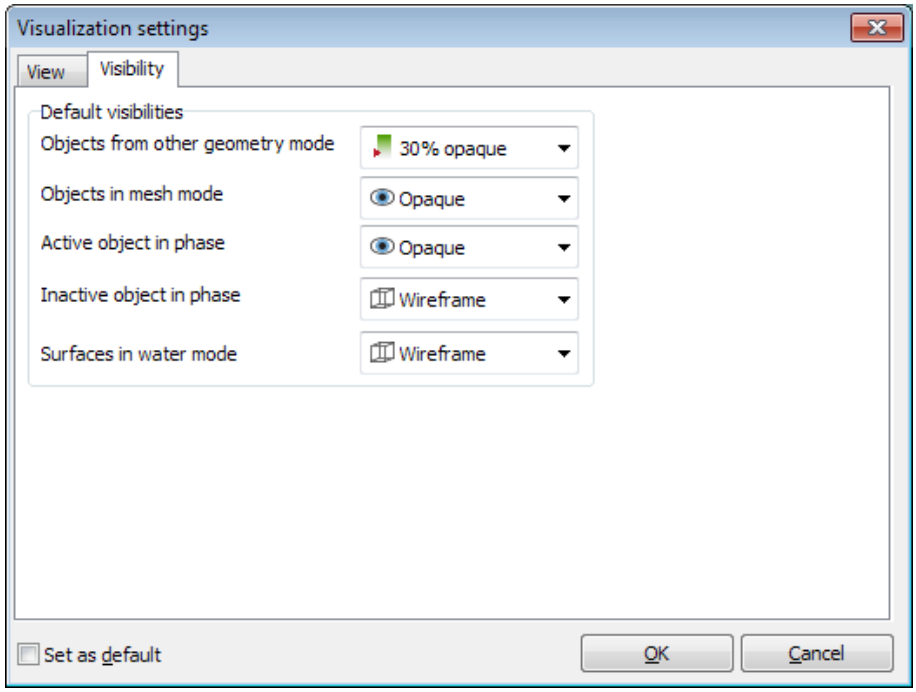


Figure 25: Visibility of objects

The available options are:

- The objects are displayed in full opacity. For individual objects in the model, right-click the object then click *Show*.
- The objects are displayed in *70* percent opacity (*30* percent transparent). For individual objects in the model, right click the object, point to *Custom* and then click *Show 70%* .
- The objects are displayed in *30* percent opacity (*70* percent transparent). For individual objects in the model, right click the object, point to *Custom* and then click *Show 30%* .
- The objects are displayed as wireframes. For individual objects in the model, right click the object, point to *Custom* and then click *Show as wireframe* .
- The objects are made completely invisible (hidden). For individual objects in the model, right click the object then click *Hide*.

The default visibility of an object (or group of objects) is indicated by the letter 'A' in the explorers. The default values for visualisation settings are shown in Figure 24 (on page 51) and Figure 25 (on page 52).

The settings defined in this tabsheet will be the default for the entities in the drawing area. However, the settings can be modified by right-clicking on the entity either in the drawing area or in **Object explorers** (Explorers (on page 61)). The view can be shown or hidden by selecting the corresponding option. More options on the view are available under the *Custom* option in the right mouse button menu.

3.8.4 *Model View - Orientation*

The orientation of the view of the model can be changed to achieve better examination by either using the default view options or by panning or rotating the view.

· *Default view options*

Default views are available both in the *View* menu and in the general toolbar.



Figure 26: Default view options

The **Movement limitation** window pops up as one of the default views, except the first (*Perspective view*), is selected. The movement limitation simplifies defining three dimensional geometry. The movement limitation is cancelled by clicking on the *Perspective view* button. Definition of geometry is described in detail in Drawing in 3D (on page 53).

· *View states...*

The user can save the current camera orientation under the specified name.

- Click the *View states...* button to save the state view or to see the saved views.

· *Non-default views*

The model in the drawing area can be panned or rotated using the left and the middle mouse buttons. The options are available either as buttons in the general toolbar or as options in the *View* menu.

- Click the *Pan camera* button to pan holding the left mouse button pressed. To rotate the view, keep the *Ctrl* key pressed while dragging the view.
- Click the *Rotate camera* button to rotate holding the left mouse button pressed. To pan the view, keep the *Ctrl* key pressed while dragging the view.

3.8.5 *Drawing in 3D*

Defining 3D geometry using mouse input is a difficult task considering the fact that the location in the direction perpendicular to the view cannot be precisely defined. This part of the manual introduces the tools used in PLAXIS 3D to enable drawing in 3D. The drawing procedure is illustrated with an example.

Note: Instruction movies describing definition of geometry in 3D are available on https://www.youtube.com/c/BentleyGeotechnicalEngineering.

The drawing tools available in the side toolbar of the **Geometry** modes enable defining the geometry of the model. When one of the drawing tools is selected, the location of the mouse cursor in the drawing area is displayed in the status bar.

Default movement limitation

While defining geometry, it is possible to move either in a plane or in the direction of an axis perpendicular to the plane.

When a new project is created, the program, by default, enables movement in the *xy*-plane located at *z=0*. Note that only the *x*- and *y*-coordinates change in the *Cursor location indicator*.

Holding the *Shift* key pressed while moving the mouse, relocates the cursor in the *z*- direction. While doing this, the *x*- and *y*- coordinates remain the same, only the *z*- coordinate changes in the *Cursor location indicator*.

Defining movement limitation

Default views can be used as an assisting tool to define geometry in 3D. When one of the default views is selected, the model view is re-oriented and the **Movement limitation** window pops up.

In the **Movement limitation** window, one of the options available in the drop-down menu can be selected as movement plane. There are two options of movement in the direction perpendicular to the movement plane.

(x/y/z)-coordinate is fixed	The location of the plane in the axis perpendicular to it should be defined in the *Movement plane* box. As the value is defined, the coordinate will be fixed at that value while moving in the drawing area. The coordinate is added in the *Available* drop-down menu. More than one location in the perpendicular axis can be defined.
Shift moves in the x/y/z-direction	The movement will be restricted to the perpendicular direction when the *Shift* key is pressed while moving in the drawing area.

The movement limitation can be reset to default by clicking the *Free movement (default)* button in the **Movement limitation** window or by clicking the *Perspective view* button in the general toolbar.

The drawing process in 3D is illustrated in the following example:

- Create a new project.
- Activate the **Structures** mode by clicking the corresponding tab in the *Modes* menu.
-  Click the **Create line** button in the side toolbar of the *Structures* mode.
- Click at (0 0 0) in the drawing area to define the first point.
- Move the mouse cursor in the drawing area. Note that only the *x* and *y* coordinates change in the cursor position indicator. Click at (0 10 0) to define the second point.
-  Click the **Back view** button in the general toolbar. The model view will be re-oriented showing the back side of the model. The **Movement limitation** window pops (Figure 27 (on page 55)). Note that the y-coordinate is fixed to 10.
- Click at (0 10 4) to define the third point.

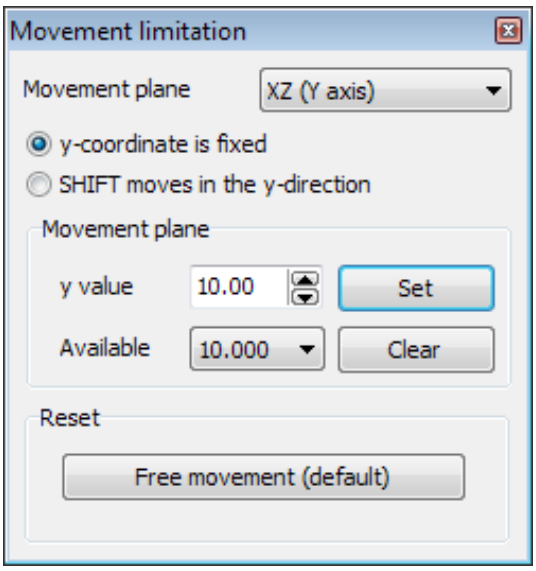


*Figure 27: **Movement limitation** window for Back view*

- Click the **Left view** button in the general toolbar. The model view will be oriented showing the left side of the model. The **Movement limitation** window pops. Select the **SHIFT moves in x-direction** option. Keep the *Shift* key pressed while moving in the drawing area. The values corresponding to *y* and *z* do not change.
- Release the *Shift* key at x = 12 and click at (12 0 4).
- Click the **Perspective view** button in the general toolbar. The **Movement limitation** window is closed.
- Click at the commandline. Note that the first point in the current line segment is already defined. Type 0 0 0 to define the last point and press *Enter*.
- Right-click to finalize the process. The resulting geometry is shown in Figure 28 (on page 55).

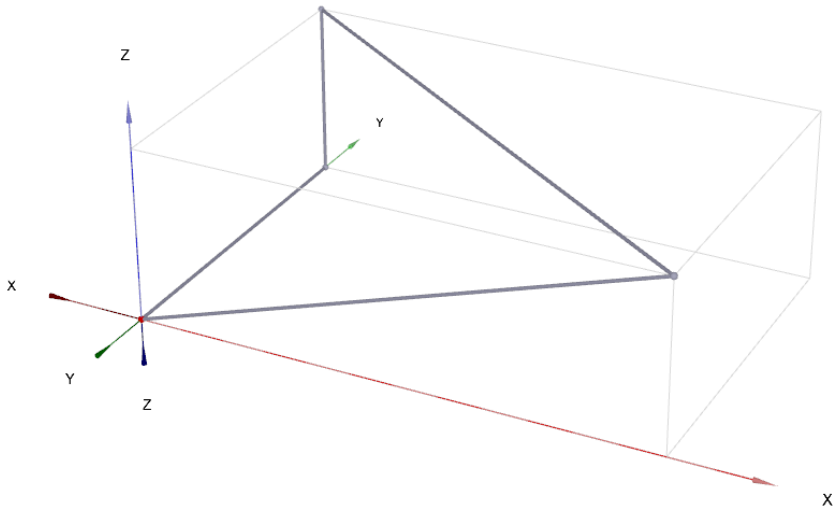


Figure 28: Resulting geometry of the illustration example

Note: The **Movement limitation** window can be closed by selecting the *Perspective view* option of the default views.

3.9 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 3` or as `point (1 2 3)`. 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 3D, 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 3D.

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 Volume_1 Volume_2`
- `delete (Volume_1 Volume_2)`

Another example is the `cylinder` command:

```
cylinder (cyl)
    Number' Number'
    Number' Number' <Coords: Number' Number' Number'>'
    Number' Number' <Coords: Number' Number' Number'>' <Coords: Number' Number'
Number'>'
```

If we take a closer look at the bottommost line, the signature consists of two numbers and two sets of coordinates. The first two parameters represent the radius and height parameters, respectively followed by the position of the cylinder, and a vector representing the orientation of the cylinder.

An example of the `cylinder` command:

- `cylinder 3 12 (0 0 0) (0 0 1)`
- `cylinder 3 12 (point_1.x point_1.y point_1.z) (0 0 1)`

An extended overview and description of all available commands is accessible via the *Help* menu in the user interface of the PLAXIS 3D Input program.

3.9.1 *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 0)(5 1 0)(5 3 0)(4 7 0)`	`#Create points with coordinates specified in the brackets and lines connecting them`
`>tabulate Points "x y z"`	`#List the created points and their coordinates`

`Object x y z`
`Point_1 1 2 0`
`Point_2 5 1 0`
`Point_3 5 3 0`
`Point_4 4 7 0`	

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

3.9.2 *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 5)`	`#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.`

3.10 Plaxis Remote Scripting Interface

In PLAXIS 3D 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 the Command Reference under the **Help** menu.

3.10.1 *Anatomy of a command*

Each command has a name and a target. Some commands may also require a parameter.

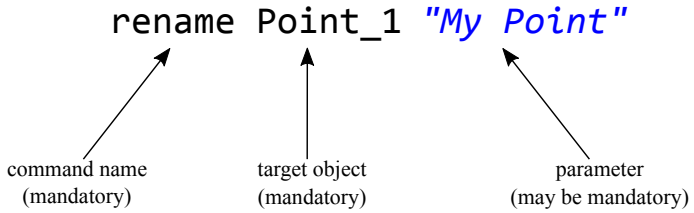


Figure 29: PLAXIS command

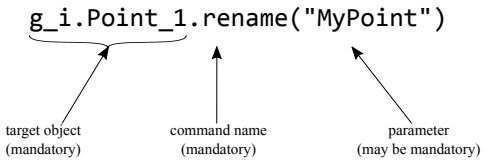


Figure 30: Python equivalent in PLAXIS

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, 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, "z", 0)`	`#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, 2, "Fx", 10)`	`#pointload command uses several parameters to create a point load on the point (1 0 2) and sets its force to 10`

3.10.2 *Geometry and calculation modes objects differences*

Each object (line, line load, plate, beam, 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 58)). 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.

3.11 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 48). However, as phases are defined only in the *Phase definition* modes (see notes of Calculation modes (on page 48)), it is greyed out in the **Mesh** mode.

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

3.11.1 *Model explorer*

Model explorer displays information related to the physical entities composing the model. It is automatically updated as the model is modified (Figure 31 (on page 62)).

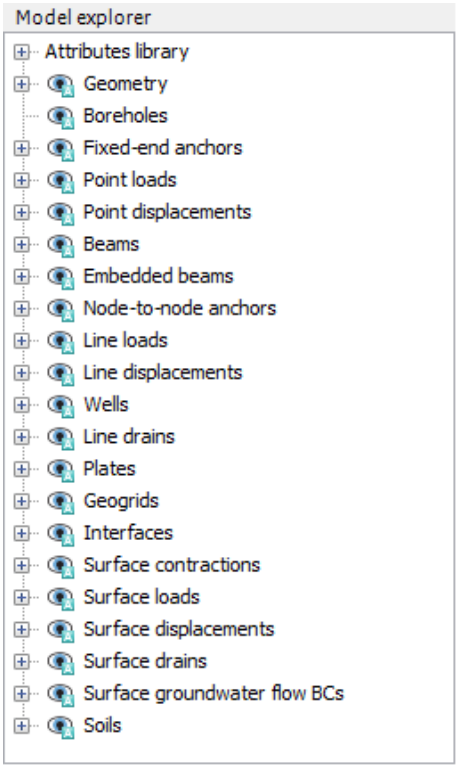


Figure 31: Layout of the Model explorer in different modes: Structures Mode

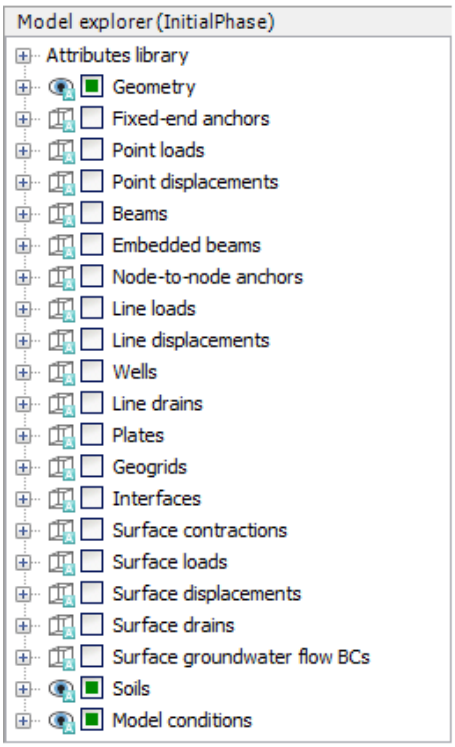


Figure 32: Layout of the Model explorer in different modes: Staged construction Mode

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 *Dynamic 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.

Dynamic multipliers 	All the displacement and load dynamic multipliers defined in the project are listed under *Dynamic multipliers*.
Flow functions	All the flow functions (*Head functions* and *Discharge functions*) defined in the project are listed under *Flow 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.

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*. For each borehole, the *x* and *y* coordinates of each borehole and the *Head* level are given.

Information about boreholes is available in the *Model explorer* only in the *Geometry* modes. The values of coordinates and *Head* can be modified in the corresponding cells.

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 97).
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 98).
Surfaces	The surfaces created in the model are listed. For each surface the coordinates of its reference point, the local coordinate axes, the features assigned and their properties are given. The flat surfaces created by either using the *Create surface* tool or by decomposing imported volumes or surfaces are listed as polygons. The imported surfaces are listed as surfaces. Features such as plates, surface loads or surface prescribed displacements can be assigned to surfaces. More information on surfaces is given in Create surface (on page 105).
Volumes 	The volumes created or imported in the *Structures* mode are listed. For each volume the coordinates of its reference point and the material data set assigned to it is given. Note that the *Import structural volume* option is available only for GSE. More information on volumes is given in Create volume (on page 108).
Soil volumes 	The soil volumes generated according to the soil layers in boreholes or imported in the *Soil* mode (Material properties and material database (on page 191)) and the material data sets assigned are given. Note that the *Import soil volume* option is available only for GSE.

Polycurves	The polycurves created in the model are listed. For each polycurve the sections composing it and their properties as well as the components of the vectors defining the *Direction axes* of the polycurve are given. More information on polycurves is given in Create polycurve (on page 99).

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 48)).

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 in the *Staged construction* mode are listed. This option is available in the *Model explorer* only in the *Phase definition* modes.

Beams: All the beams created in the model are listed. Information about the assigned material is given for each beam.

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

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.

Surface loads: All the surface loads created in the model are listed. The distribution and the values of the load components can be defined.

Contractions: All the contractions created in the model are listed. The distribution and the values of the contractions can be defined.

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

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

Line 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.

Surface groundwater flow BC's: All the surface 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.

Wells: 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.

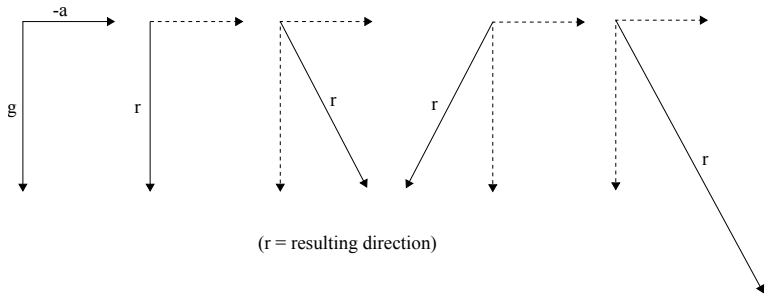
Surface drains: All the surface drains created in the model are listed. The head can be defined for each surface drain.

Line drains: All the line 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:

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 390). 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 	The model conditions for a dynamic analysis can be defined at the extreme boundaries of the model. The options available are *None* and *Viscous*. 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 *Dynamic* calculations see Model conditions (on page 390).
GroundwaterFlow	The model conditions for a groundwater flow calculation, a consolidation analysis or a fully coupled flow-deformation analysis can be defined at the extreme boundaries of the model. The options available are *Open* and *Closed*.
Precipitation	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 (on page 176)).

PseudoStatic	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 33: Resulting force direction r due to combinations of gravity and pseudo-static acceleration*
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 351).

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 26)). 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.

3.11.2 *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*.

3.11.3 *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 34 (on page 68).

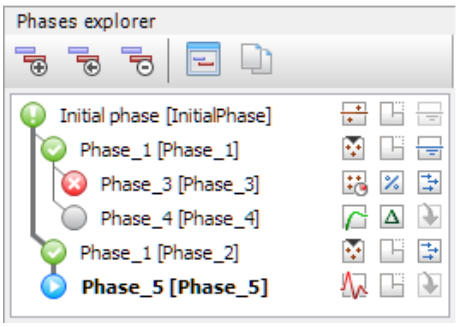


Figure 34: Phases explorer

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.

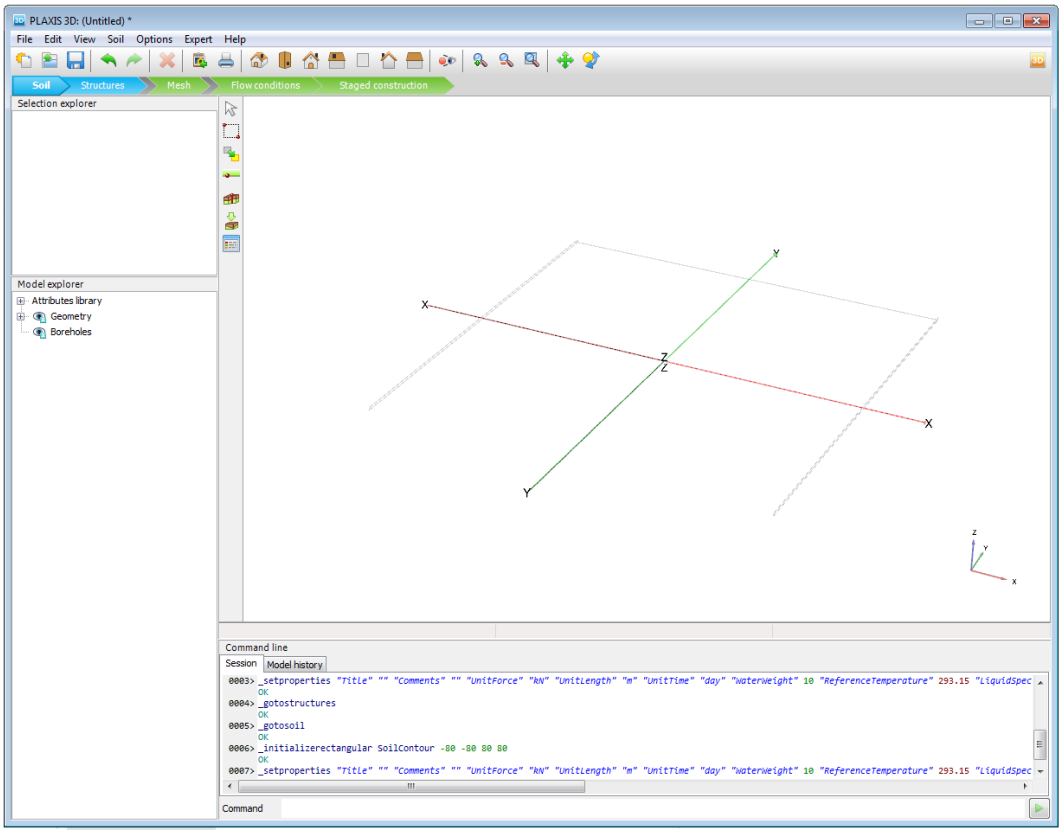


*Figure 35: View of a new project in the **Soil** mode*

4.1 Adjustment of the soil contour

- The model contour can be defined using the options available in the side toolbar. To modify the model contour click the *Adjust soil contour* button. The contour is defined in the drawing area. The buttons in the expanded menu can be used to modify the model contour.

- Click the *Move contour points/lines* button and drag any point or line in the contour to a new location.
- Click the *Insert contour points* button, click the contour where a new point is to be defined and drag it to the desired position of the new point. The soil contour will be modified accordingly.
- Click the *Delete contour points* button and click the contour point that is to be deleted.

Besides the expanded menu, the **Surface points** window pops up displaying the coordinates of the points defining the model contour. Points defining the contour can be relocated by entering the coordinates of the new location in the table. The options to move, insert and delete contour points are available in the menu appearing as the contour point number is right clicked in the table. A more detailed description on **Surface points** window is given in the Create surface (on page 105).

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 3D 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 36 (on page 71)). Boreholes and soil layers can be modified in this window.

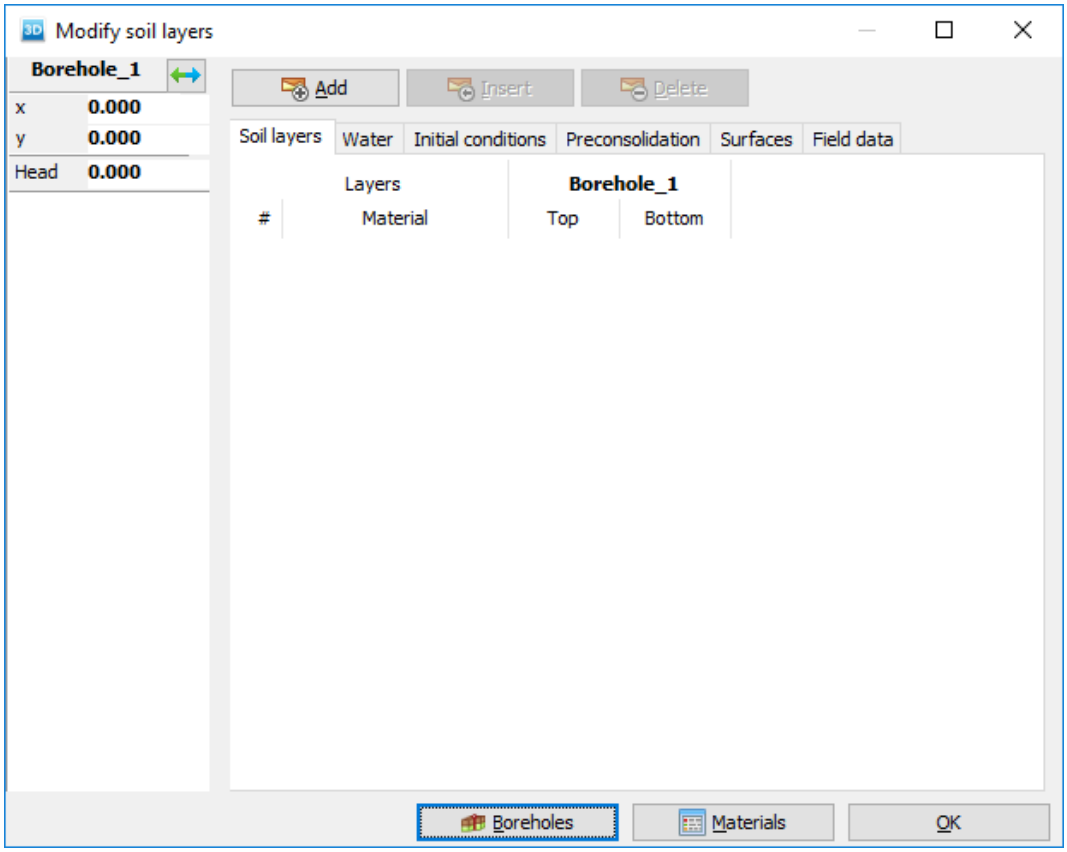


*Figure 36: **Modify soil layers** window*

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 37 (on page 71)).
- In the **Add** window define the position of the new borehole (*x* and *y* coordinates).
- Define the source borehole to copy initial layer boundaries from. By default, the source borehole is the nearest one.

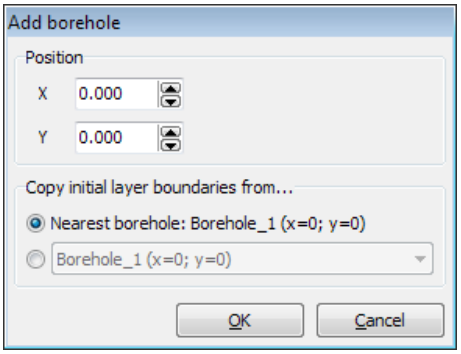


*Figure 37: **Add borehole** window*

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

4.2.2 *Creating multiple boreholes*

To create multiple copies of existing boreholes organised in a rectangular pattern:

- In the **Modify soil layers** window, click *Boreholes* and select the *Add array* option. The **Add borehole array** window pops up (Figure 38 (on page 72)).
- To select the source borehole click the corresponding check box. The thickness of the layers in the new borehole is equal to that in the source borehole.
- Select the appropriate option in the *Shape* drop-down menu.
- Define the number of columns (and/or rows) and the distance between them. A detailed description of the *Array* feature is given in Copy object - Array (on page 95).

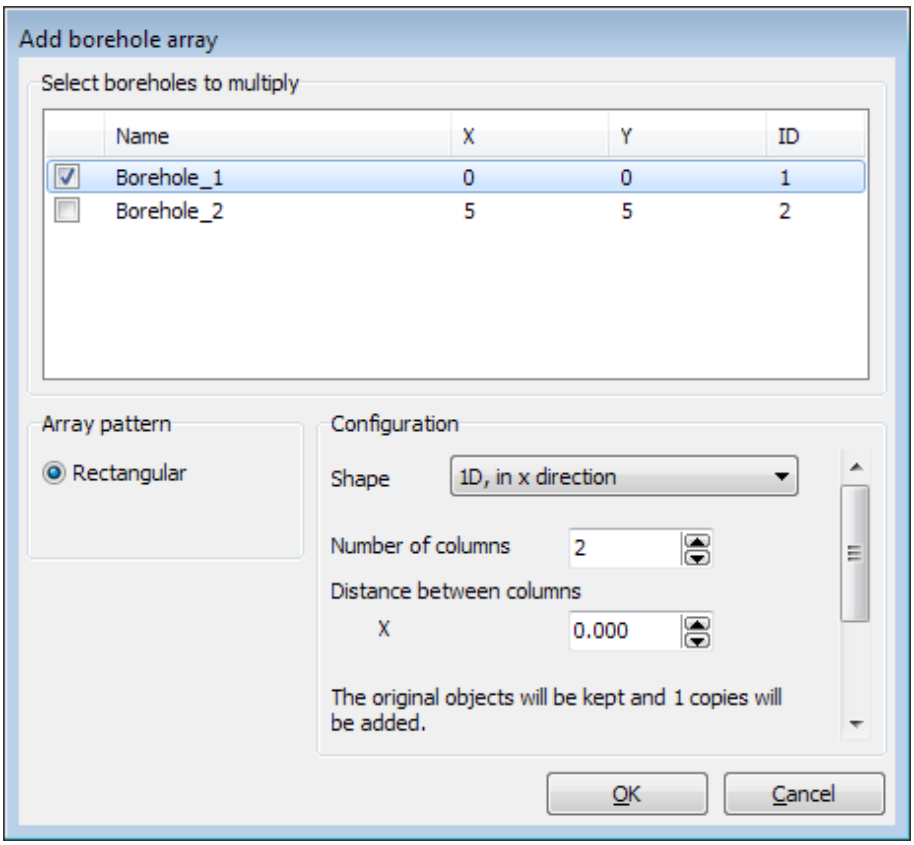


*Figure 38: **Add borehole array** window*

4.2.3 *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 36 (on page 71)):

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, $K0_x$ and $K0_x$, for all soil layers.
Preconsolidation tabsheet	Table where pre-overburden pressure (POP) can be defined.
Surfaces tabsheet	This tabsheet enables the import of predefined top and bottom soil surfaces.
Field data tabsheet	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 (on page 82).
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 39 (on page 75)).

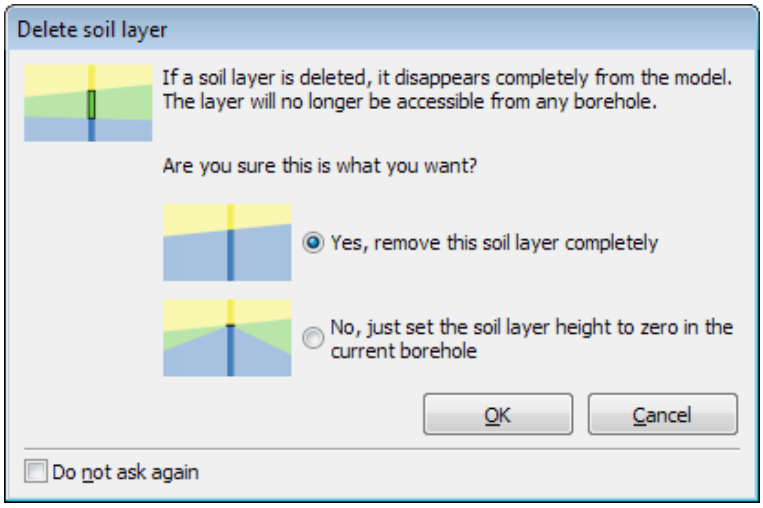


Figure 39: ***Delete soil layer*** *window*

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 3D 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 (on page 176). 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.

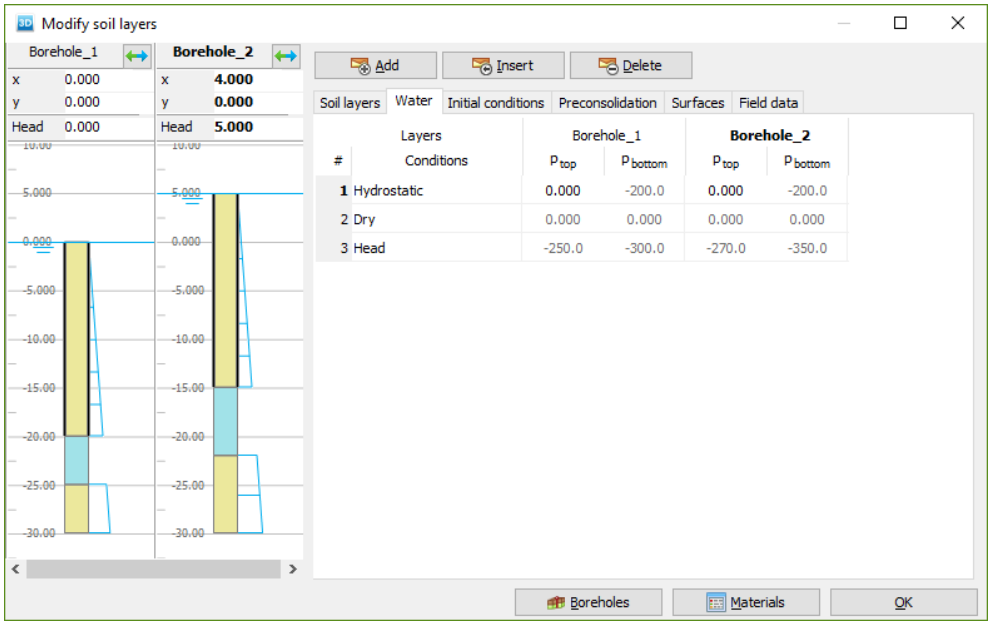


*Figure 40: **Water** tabsheet of the **Modify soil layers** window*

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 in the boundary with the layer above to the pressure in 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 3D 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 374).

By default, positive pore water stress (suction) in the unsaturated zone above the phreatic level is ignored. However, PLAXIS 3D 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 249)). It also requires that suction is allowed during the calculations (Flow control parameters (on page 372)).

4.3.3 *Initial Conditions of Soil*

The **Initial conditions** tabsheet in the **Modify soil layers** window offers (Figure 41 (on page 78)) 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.

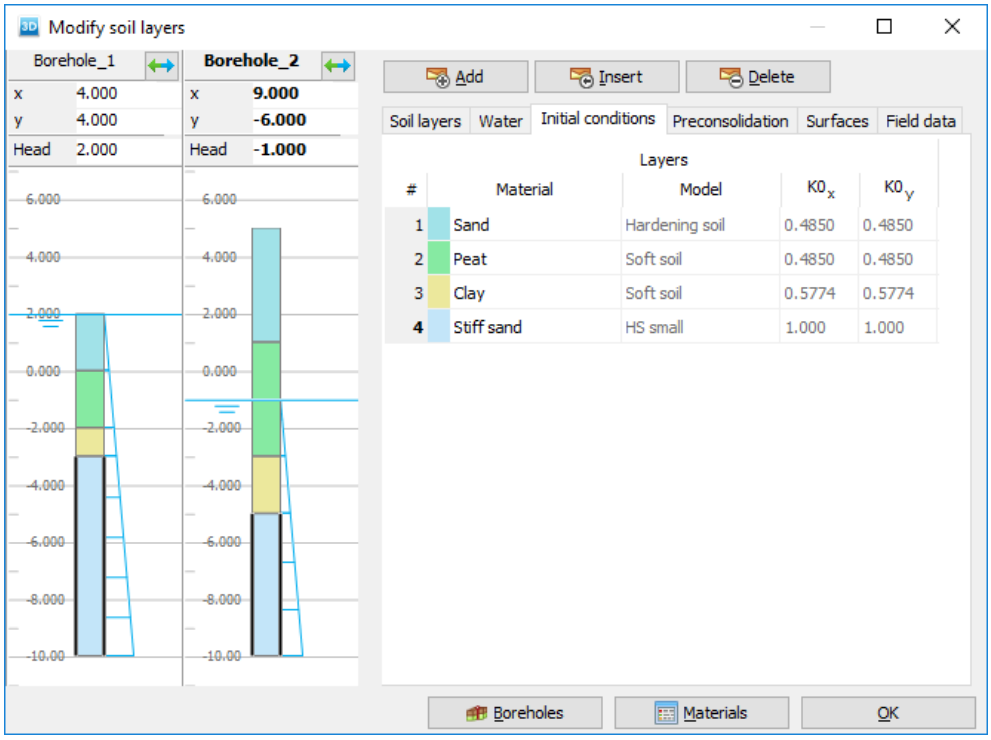


Figure 41: ***Initial conditions*** *tabsheet of the* ***Modify soil layers*** *window*

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 329).

If the *K0 procedure* is used, proper *K0* values need i.e. the initial ratio between horizontal effective stress and vertical effective stress needs to be specified for all layers. Two K_0 values can be specified, one for the x-direction and one for the y-direction.

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 42 (on page 79) and Initial tabsheet (on page 265)). 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.

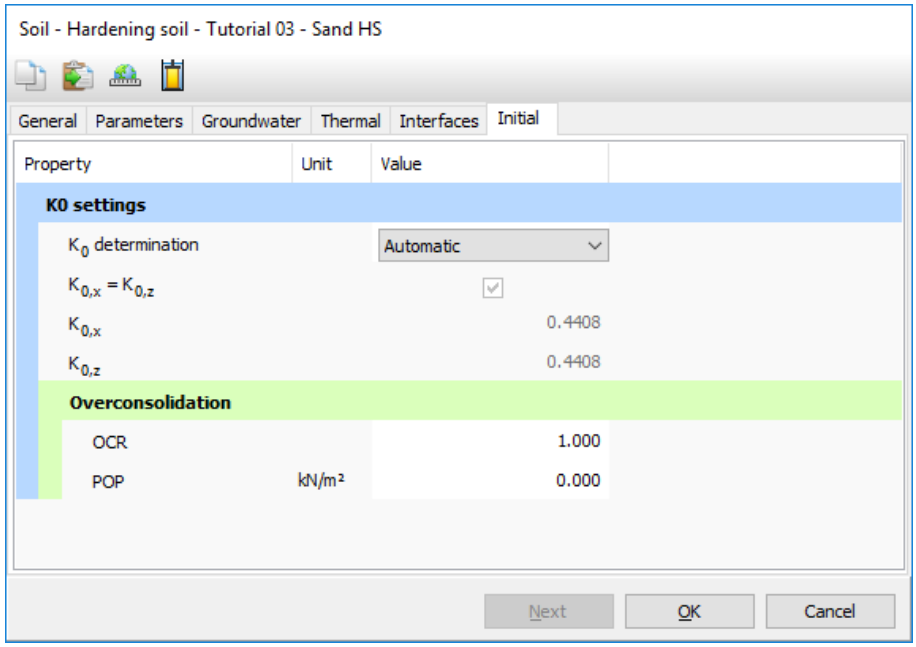


*Figure 42: **Initial** tabsheet in the **Soil** window*

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.

The two POP values are defined *From material* (Figure 43 (on page 80)), which means the preconsolidation stress is calculated based on the information (OCR or POP) as defined in the material data sets.

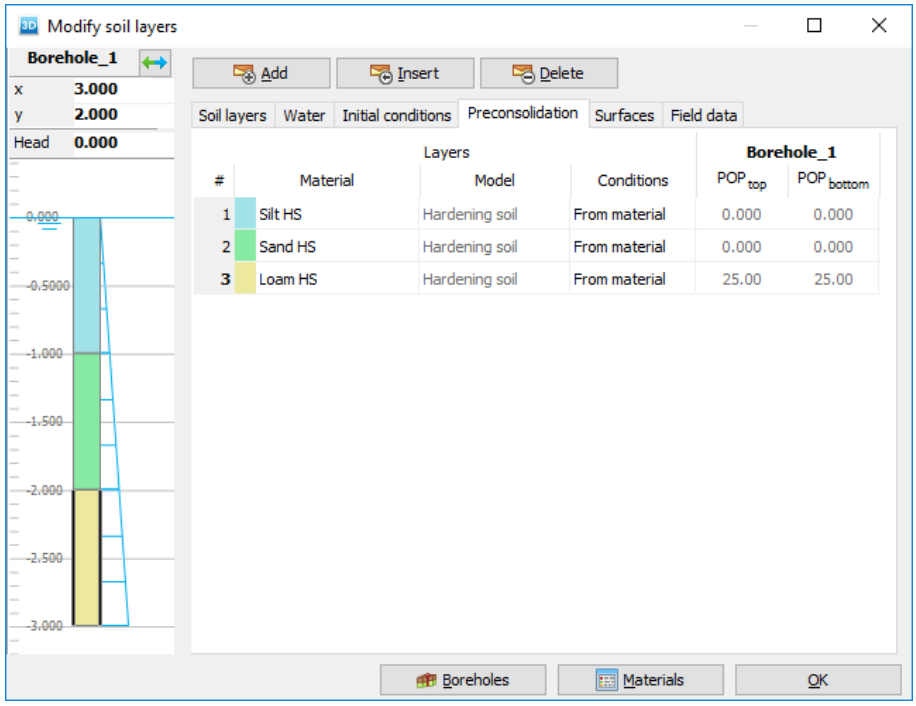


*Figure 43: The **Preconsolidation** tabsheet of the **Modify soil layers** window*

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 *Surfaces* GSE

By default, the ground level surface and the bottom layer of the soil are modelled using the information provided by boreholes, as indicated by the *From boreholes* option in the **Surfaces** tabsheet (Figure 44 (on page 81)). The surface level between two boreholes is linearly interpolated. Alternatively, predefined surfaces can be used to define the top and bottom surfaces of the soil volume using a single borehole.

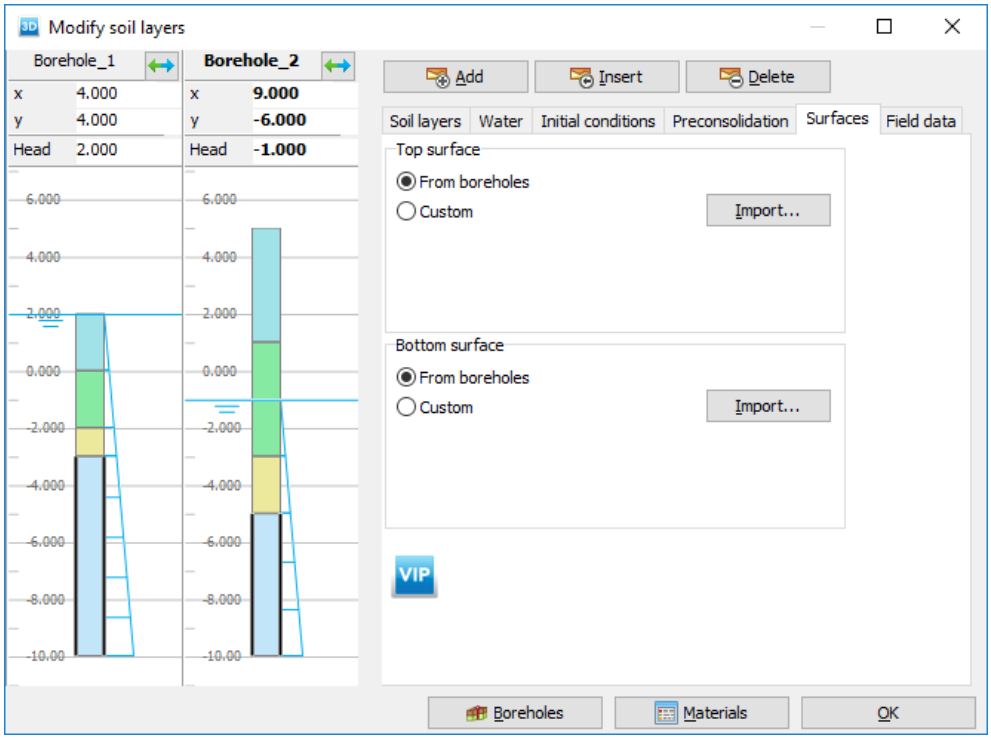


*Figure 44: **Surfaces** tabsheet of the **Modify soil layers** window*

In order to use predefined surfaces, the *Custom* option in the **Surfaces** tabsheet is selected. Next by clicking on the button *Import* an **Import (top/bottom) soil surface** window appears from which the desired surface can be selected.

Once the surface is imported, it can be further converted to a *NURBS* surface, by selecting the checkbox *Convert to NURBS*. The purpose of this option is to enable the conversion of a triangulated surface consisting of multiple triangles into an effective parametric surface that is described as a *NURBS surface* (Non-uniform Rational B-spline). The *NURBS surface* is an approximation and it is generated along a grid. The user provided surface is sampled along a grid and B-spline curves are generated in both directions along the grid. From the B-spline curves the *NURBS surface* is generated. The grid points may be changed in both direction if the resulting surface is not satisfactory.

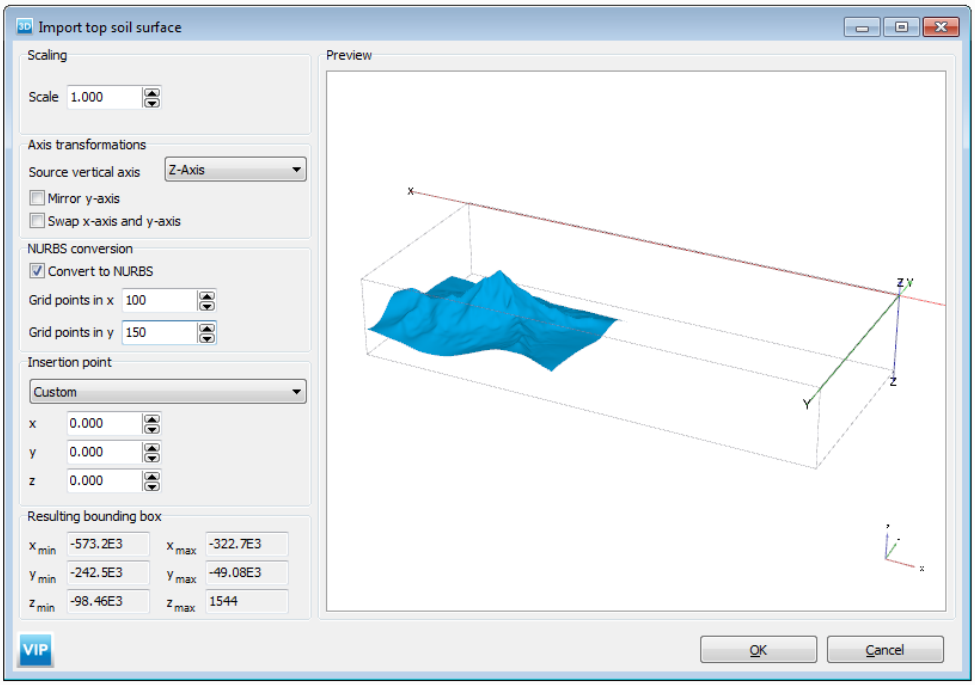


Figure 45: Importing of custom soil layers

4.3.6 *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 3D, 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 511)) 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.

The soil stratigraphy in a borehole can be specified using available CPT logs. More information on the required format is given in Appendix A.

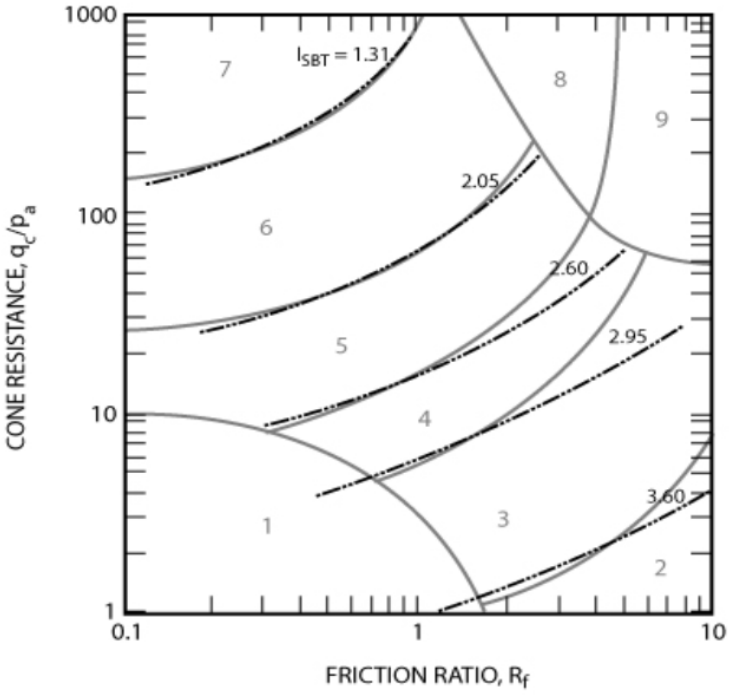
Robertson Method

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 46 (on page 83)).

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:

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



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 46: SBT chart based on cone resistance (qc/pa) and friction ratio Rf, showing contours of ISBT (after Robertson et al., 1986)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:

- Note that the value of p_a should be in the same units as the unit system selected in the *Project properties* window.

- Be careful when importing CPT data as simple text files, since the information (in terms of depth, q_c and f_s) should be expressed in the same unit system as the one chosen in PLAXIS.

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
$2.60 < I_{SBT} < 2.95$	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

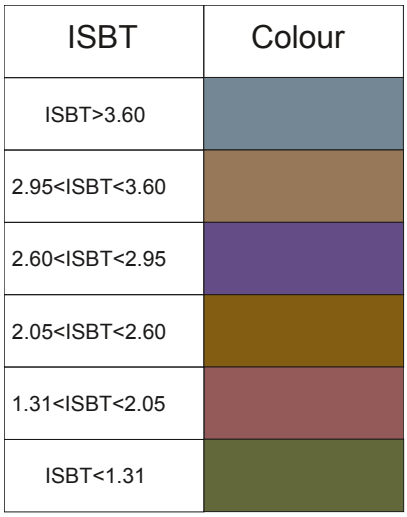


Figure 47: Colour scheme adopted in PLAXIS 2DPLAXIS 3D representing different layers for different ISBTPLAXIS 3D representing different layers for different I_{SBT}

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 47 (on page 84)
- 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 49 (on page 86).
- Click *Apply layers* to subdivide the soil cluster into the layers shown in the borehole.

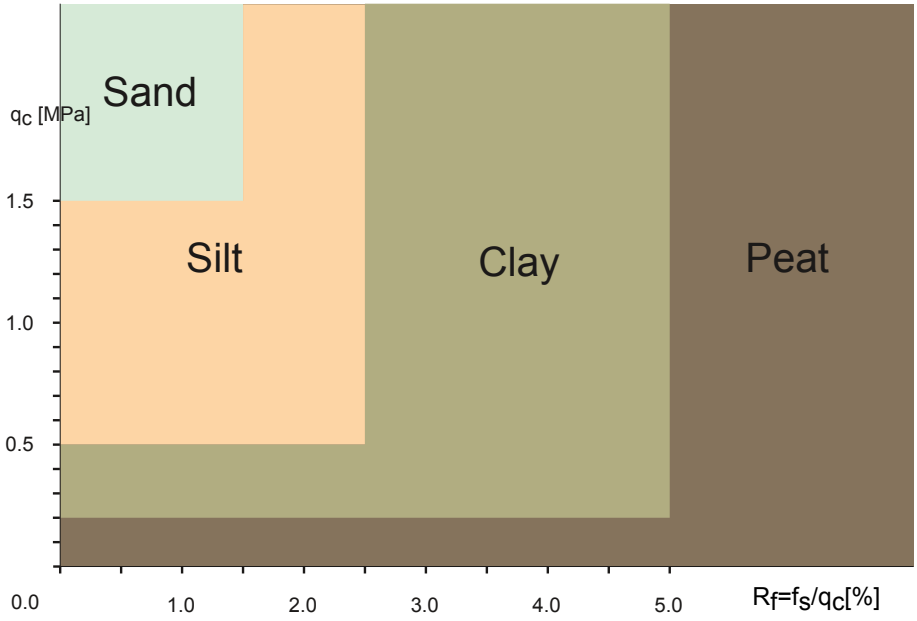


Figure 48: Layering criteria for CUR 3 layers methodCUR 3 layers method

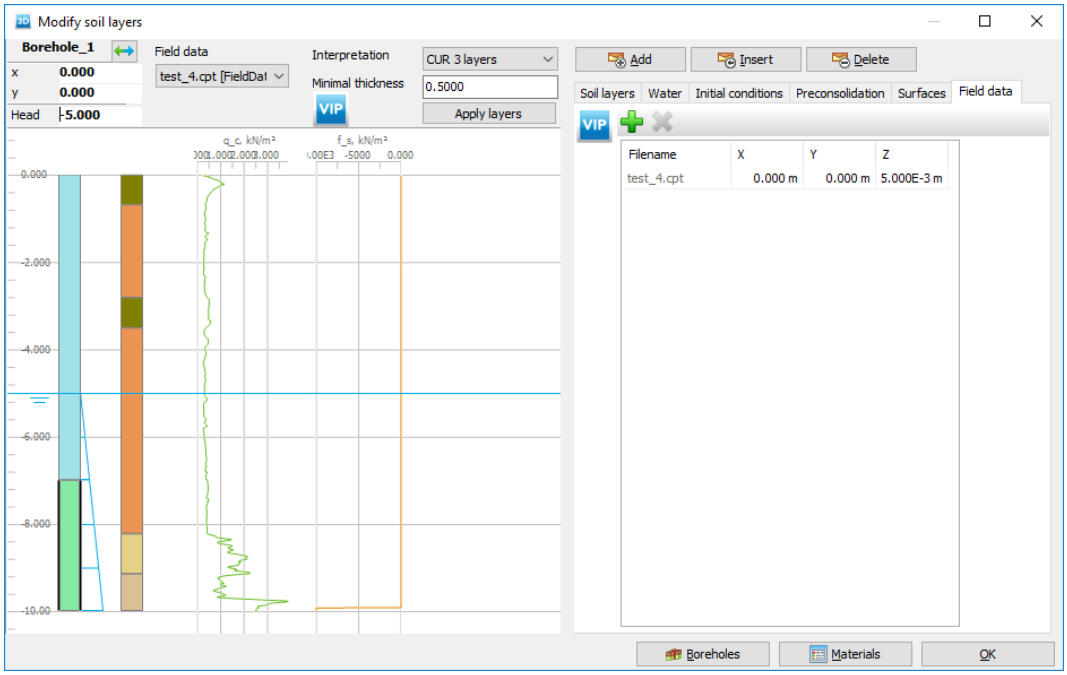


*Figure 49: CPT data in the Modify soil layers window**Modify soil layers** window*

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.

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. Note that only surfaces and volumes can be imported. 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.
*.brep	OpenCascade shape files.
*.step	3D STEP files.
*.stp	3D STP files.

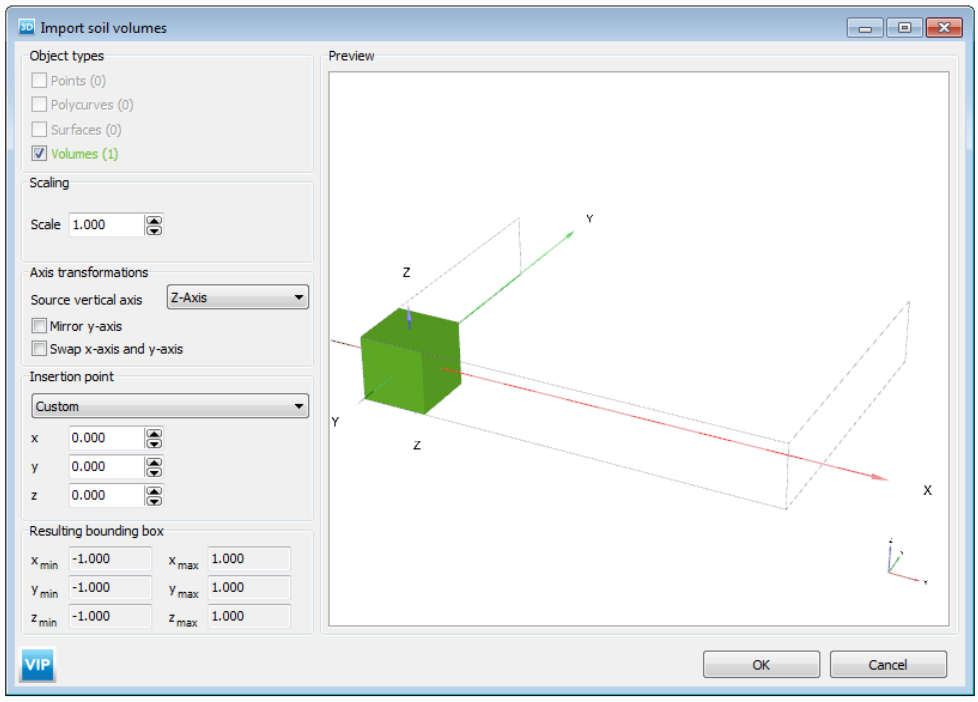


*Figure 50: The **Import soil volumes** window*

Before importing **.dxf* files, the *INSUNIT* global variable should be set to the desired unit Table 1 (on page 87).

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

Unit	*INSUNIT* value
Angstroms	11
Nanometers	12
Microns	13
Decimeters	14
Decameters	15
Hectometers	16
Gigameters	17

After selecting the file, the **Import soil volumes** window pops up (Figure 50 (on page 87)), where the geometry can be modified. It is possible to import only volumes by keeping the corresponding option checked in the **Import soil volumes** window.

4.4.1 *Scaling*

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

4.4.2 *Axis transformation*

The orientation of the imported geometry might need to be modified in order to be properly mounted in the project.

Source vertical axis	It enables reorienting the imported geometry such that the vertical axes of the imported geometry and the program coincide. The source vertical axis enables axis transformation according to the right hand rule of rotation. Note that the vertical direction in PLAXIS 3D is the z-axis.
Mirror y-axis	The geometry will be transformed such that the values of *y* coordinates will be changed to *-y*; coordinates (effectively in the xz-plane).
Swap x-axis and y-axis	The geometry will be transformed such that the values of *x* coordinates will be changed to *y* coordinates and vice versa.

4.4.3 *Insertion point*

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.

4.4.4 *Bounding box*

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

Note:

- Material properties of volumes or surfaces 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).

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. The soil layers created in the mode will appear in the drawing area of the **Structures** mode according to the default visibilities defined in the *Visualization settings* in the *Options* menu, i.e. somewhat transparent. Boreholes cannot be modified in the **Structures** mode.

5.1 Assisting tools

The assisting tools enable modifying the geometry by changing the location or the orientation 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 *Ctrl* 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, lines or surfaces) or multiple structural elements of one type (for example node-to-node anchors, beams 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 56)).

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 90))
- Click the *Move object* button and drag the selection to the new location.

5.1.3 *Rotate object*

A selection can be rotated over a global axis around a rotation point. The *Rotate object* button is active only after one or more objects have been selected.

To rotate a selection:

- Click the *Rotate object* button. The **Rotate selection** window pops up. Note that the name of the window varies according to the selection.
- Define the coordinates of the rotation point. The default rotation point is the object reference point. The rotation point can be relocated by dragging it in the drawing area.
- For cases where the rotation point is not visible, it is possible to drag the rotation point by clicking the *Drag point* button in the **Rotate object** window and dragging it in the drawing area.
- Define the rotation angles. The rotation angles are positive for counterclockwise rotation. Alternatively, the rotation may be defined by dragging one of the points on one of the circles.

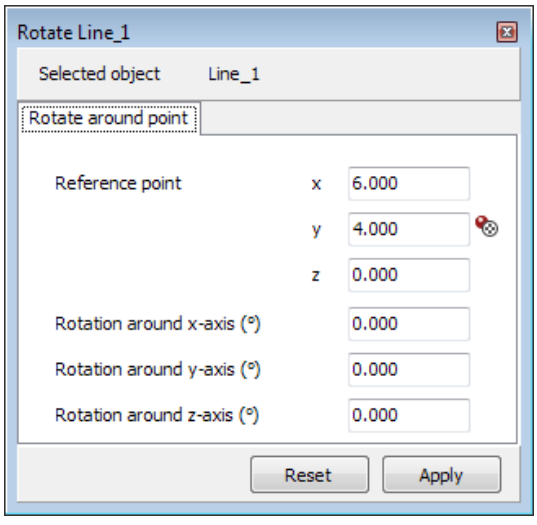


*Figure 51: **Rotate object** window for a line*

In the drawing area, the circle defined by rotation over the *x*-axis is marked in red colour. The circle defined by rotation over the *y*-axis is marked in green colour. The circle defined by rotation over the *z*-axis is marked in blue colour.

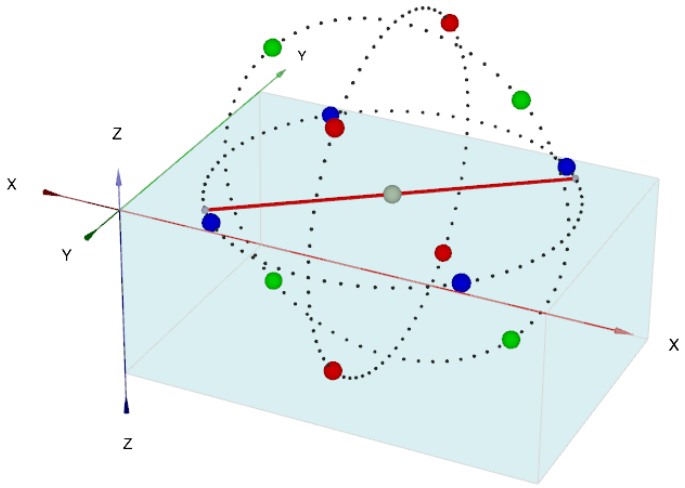


Figure 52: Rotation point and rotation axis for a selected line

Note: Note that the options to rotate or pan camera do not change the orientation of the objects in the model but only the view.

5.1.4 *Extrude object*

Points, lines and surfaces can be extruded to create lines, surfaces and volumes correspondingly. The *Extrude object* button is activated only after the selection of one or more such geometry entities.

The graphical definition of the extrusion is possible by dragging the entity. The coordinates defining the direction of extrusion and its magnitude will change accordingly as the entity is dragged in the drawing area. Note that the selected entity can only be dragged in the direction defined by its normal. Holding the *Shift* key pressed while moving the mouse enables lateral movement. For a group of selections the dragging direction is defined by the normal of the geometry entity clicked before dragging.

Extrusion

The extrusion of points, lines and surfaces can be defined in the **Extrusion** tabsheet in the **Extrude object** window. As an example, the extrusion of a line is described.

To extrude a line:

- Select a line in the drawing area.
- Click the *Extrude object* button. The **Extrude** window pops up as shown below.
- Define the components of the extrusion vector. The length of the extrusion vector is automatically calculated.
- If the length of the extrusion vector is redefined, the vector components are automatically recalculated according to their previous values and the new length.

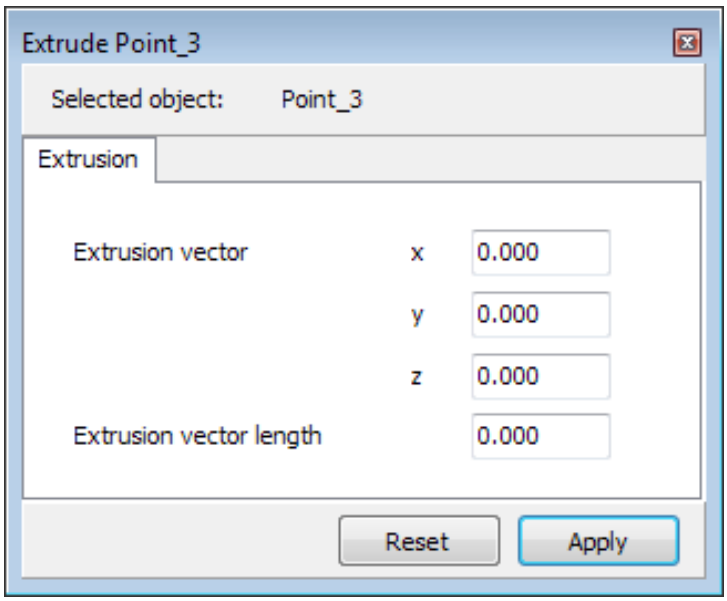
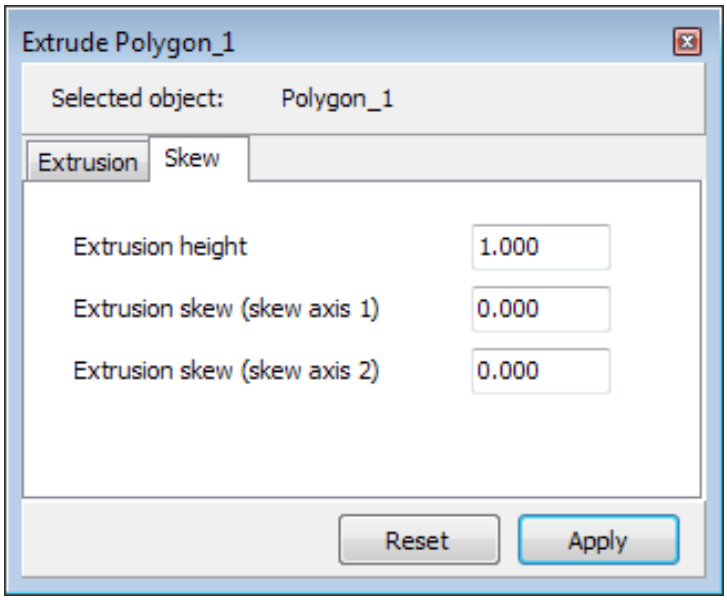


*Figure 53: **Extrude object** window for a point selection*

Extrude surface - Skew

A volume can be created by extruding a surface. In addition to a normal extrusion, the extrusion can be defined in the **Skew** tabsheet in the **Extrude object** window. Note that the *Skew* tabsheet is available only when a line or a surface is extruded (see figure below).



This option enables defining the extrusion by entering the skew components in the first and second local axis and the height of the extrusion in the direction of the third local axis

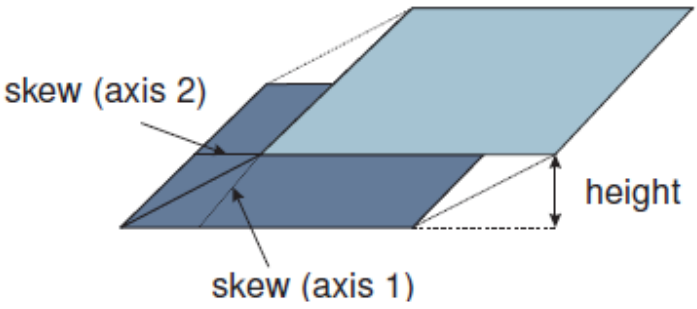


Figure 54: Skew components

Extrusion along a predefined path

PLAXIS 3D enables the extrusion of a polycurve (Create polycurve (on page 99)) or a polygon (Create surface (on page 105)) along a nonlinear path. A non-linear path can be defined using the *Polycurve* tool. A polygon must be non-self-intersecting and flat to be valid for extrusion along a nonlinear path.

To extrude a polycurve/polygon along a nonlinear path:

- Select the polycurve/polygon to be extruded.
- Keep the *Ctrl* pressed and select the polycurve representing the extrusion path.
- Click the right mouse button and select the *Extrude* option in the appearing menu.
- The extruded geometry is created in the model. The original geometric entities are still available.

Revolve around axis GSE

A volume can be created by revolution of a surface, and a surface can be created by revolution of a line. PLAXIS 3D enables the revolution of a surface or a line around an axis. A surface or line must produce non-self-intersecting results after revolution. A line or a surface must also be in-plane for revolution.

To revolve a line/surface around an axis:

- Select the line/surface required for revolution.
- Click the *Revolve* object button. The **Revolve** window pops up as shown below.
- Define the components of the revolution *Axis start*, *Axis end*, and *Rotation angle* as required.
- Click *Apply* to create the volume/surface.
- It is possible to reset all settings to inital conditions by clicking on *Reset*.

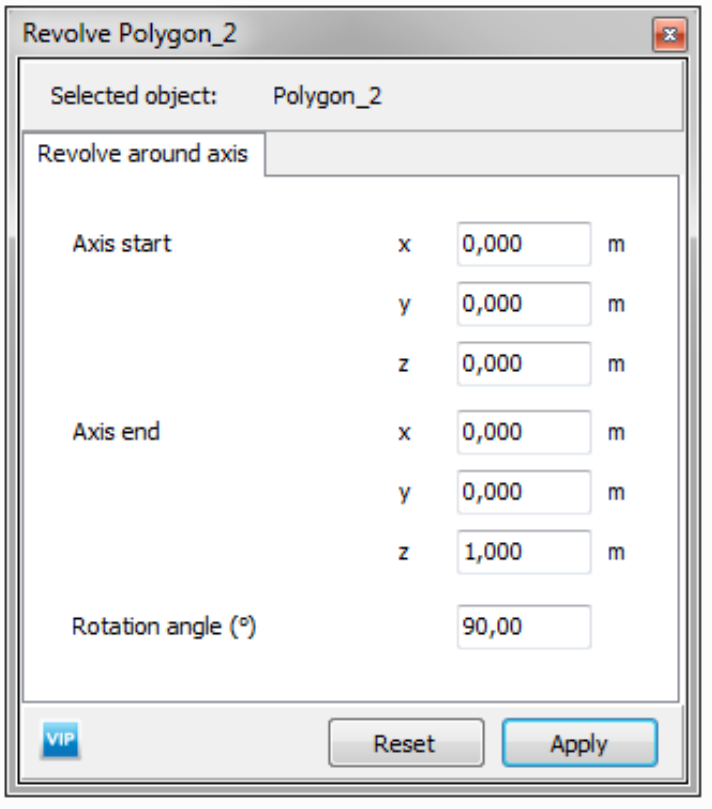


Figure 55: ***Revolve object*** *window for a surface selection*

Note: In case of revolution of a surface, then the axis of revolution should be in the same global plane as the surface itself.

5.1.5 *Copy object - Array* GSE

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.

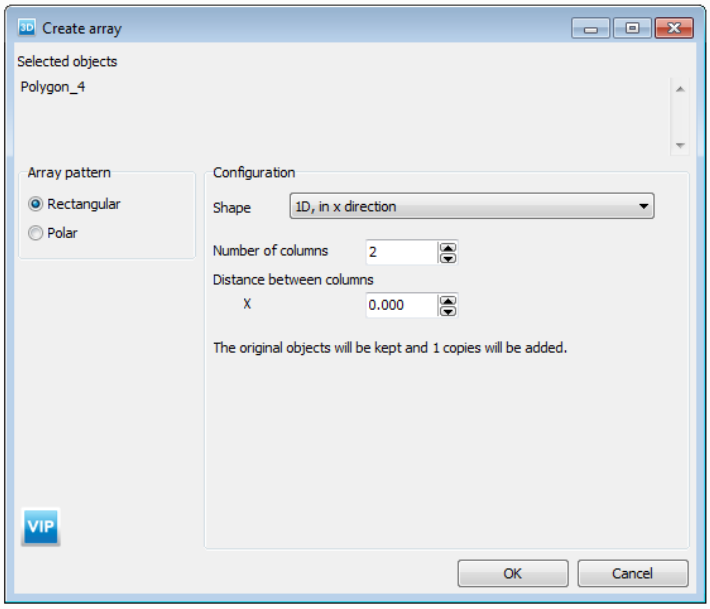


*Figure 56: **Create array** window, Rectangular array pattern*

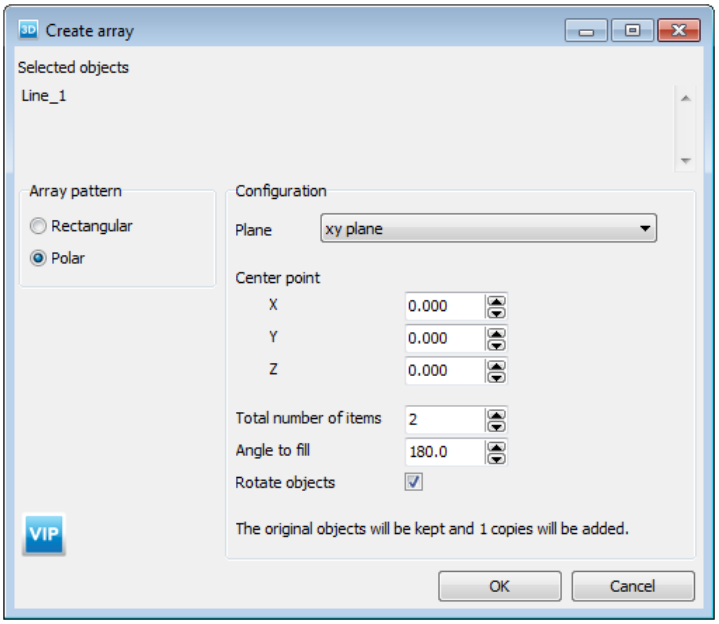


*Figure 57: **Create array** window, Polar array pattern*

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*.

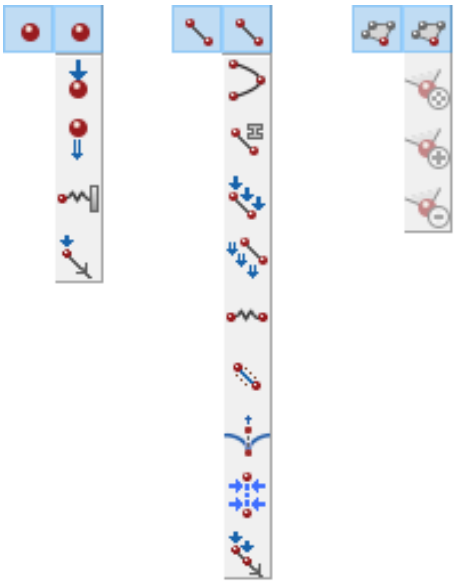


Figure 58: The expanded menus for Create geometric entities

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.

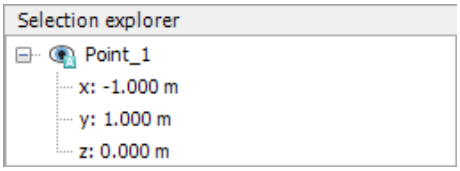


Figure 59: Point in the Selection explorer

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 *Beam*, *Embedded beam*, *Line load*, *Line prescribed displacement*, *Node-to-node anchor*, *Well* and *Line drain* can be assigned to a line.

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

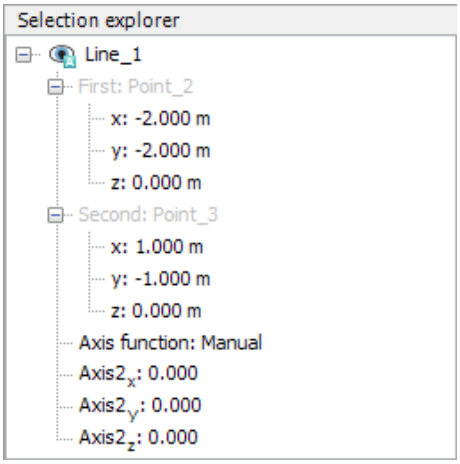


Figure 60: Line in the Selection explorer

Local coordinate system

A local coordinate system for lines is defined automatically by the program. Defining a local coordinate system for a line element is required in order to define the material properties of structural elements having orthotropic material behaviour and to show the forces in the Output program. The local coordinate system can be modified in the *Object explorers*.

For lines, the first local axis always corresponds to the axial line direction. The *Automatic* axis function defines automatically the second axis of the line according to the global coordinate system. When the *Manual* option in the *Axis function* drop-down menu in the *Object explorers* (Figure 60 (on page 98)) is selected, the user has to define the direction of the second axis. In both cases the third axis is determined automatically from the first and the second axis according to the right hand rotation rule.

5.2.3 *Create NURBS Curve*

NURBS (Non-uniform rational basis spline) curve is a collection of points that forms curves which are flexible and precise. Nurbs curves can be extruded using the *Extrude object* option to generate surfaces. This option is especially useful while modelling pipelines or tunnels.

To create a NURBS curve:

-  Click the *Create NURBS curve* button from the lines option in the side toolbar.
- Click the location where the NURBS curve is to be inserted. The **NURBS curve** table is displayed with the co-ordinates of the NURBS curve points. The table also displays options to import / open a text file with the NURBS curve points.

5.2.4 *Create polycurve*

A polycurve is a collection of lines and arcs that form a single continuous curve. Each line and arc composing the polycurve is defined as a segment. The starting point of each new segment is the end point of the previous segment.

To create a polycurve:

-  Click the **Create polycurve** button in the side toolbar.
- Click the location where the polycurve is to be inserted (this location is known as the *Insertion Point*). Subsequently, the **Polycurve designer** window is displayed.

Note: The general coordinates of the *Insertion Point* are displayed and can be modified in the **Selection Explorer** (outside the polycurve designer window).

The emerging **Polycurve designer** window consists of a toolbar with two tabsheets (**General** and **Segments**) where the properties of the polycurve are defined. The **Preview** window displays the created polycurve.

1. ***General tabsheet***

   The general settings of the polycurve are defined in the **General** tabsheet of the **polycurve designer** window(see Figure 61 (on page 100)) taking into account the following considerations:

   - The **Offset to begin point** can be used to specify the distance between the origin of the local polycurve axes "1" and "2" (the insertion point of the polycurve) and the *starting point* of the first segment of the polycurve. By default the offset values are zero, which means that the *starting point* of the first segment is equal to the *insertion point*. However, by setting the offset values it is possible, for instance, to make the center point of the polycurve the insertion point.
   - The **Orientation axis 1** and the **Orientation axis 2** define a plane in which the polycurve is drawn.

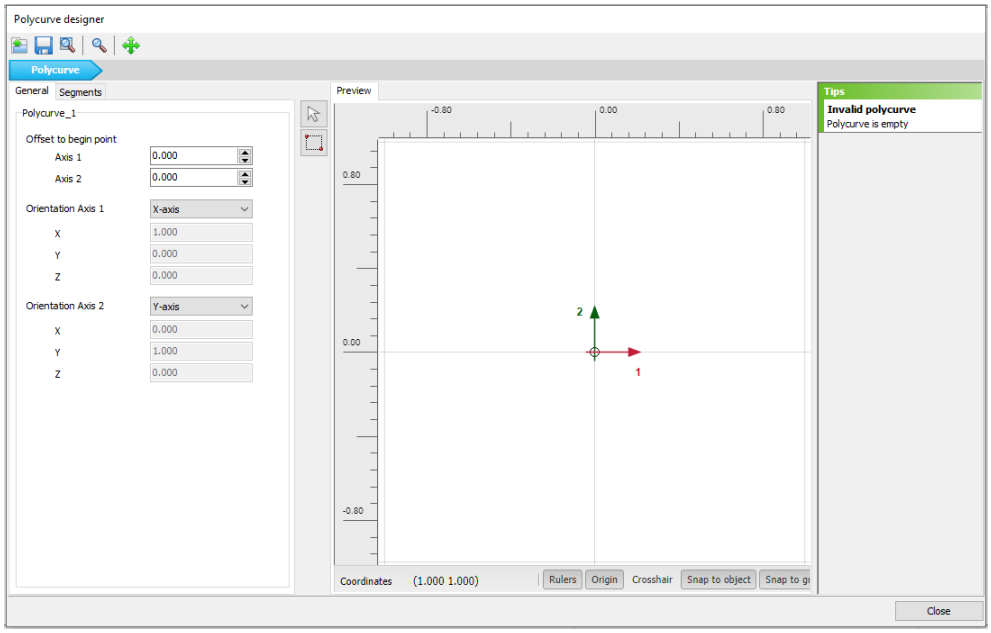


*Figure 61: **General** tabsheet of the Polycurve Designer window*

The general properties of the polycurve in the model are defined using the options available in the **General** tabsheet.

2. ***Shape tabsheet***

The segments of a polycurve can be defined in the **Segments** tabsheet of the **Polycurve designer** window.

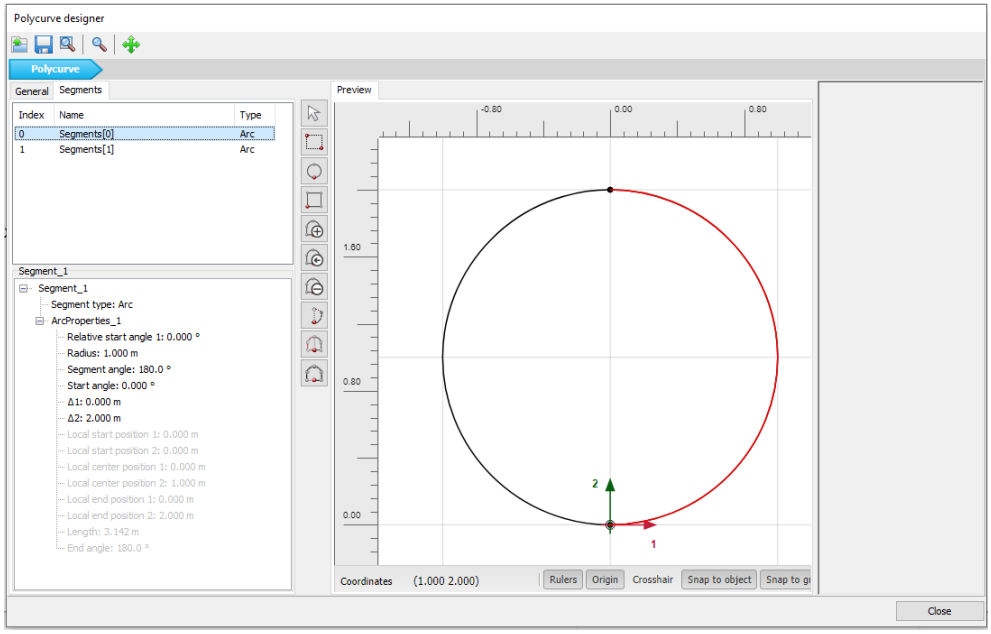


*Figure 62: **Segments** tabsheet of the **Polycurve designer** window*

- ***Definition of polycurve segments***

  To edit the polycurve/segments a toolbar is displayed with the following buttons:

  A new circular shape is created by clicking the **Make circle** button. Once this button is selected, insert the *radius* of the circle in the emerging window.

  A new square shape is created by clicking the **Make square** button. Once this button is selected, insert the *edge length* of the square in the emerging window.

  A new segment is added to the last segment in a polycurve by clicking the **Add segment** button available in the toolbar. The last point of the last segment in the polycurve is the first point of the newly added segment. The newly created segment is displayed in the segment list shown below.

  Note: A segment can be selected from the displayed segment list or by clicking the segment in the **Preview area**.

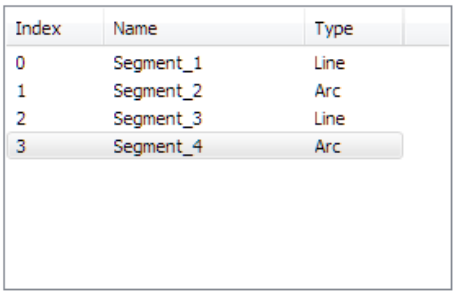


Figure 63: Segment list displaying the segments composing the polycurve

  A new segment is inserted before a selected segment in a polycurve 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 polycurve is deleted when the **Delete segment** button is clicked. The indices of the segments are updated according to their position in the list.

  To extend the polycurve to the axis of symmetry click the **Symmetry axis** button. A new segment is added connecting the last point of the polycurve to the axis of symmetry.

  To close the polycurve by mirroring the existing segments, click the **Symmetric close** button . This is only possible when one half is completed up to the symmetry axis.

  To close the polycurve click the **Close** button. A new segment is created connecting the last point of the polycurve to the first one.

- ***General properties of segments***

Type	To define the type of the segment. 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 as depicted below.

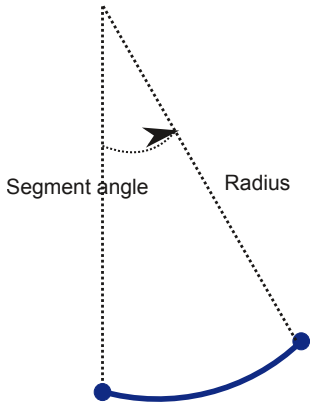


Figure 64: Properties of Arc segment

When specifying a negative radius for an *Arc* segment, the arc will be mirrored in the line described by the tangent vector.

Note: When the **Make a circle** option is selected, the created polycurve consists of two *Arc* segments with a *Segment angle* value of 180°.

Relative start angle	To define the value of the angle between the start tangent of the next segment and the end tangent of the previous segment as shown below.
Radius	Arc only. The value of the radius for an arc segment.
Length	Line only. The value of the line segment length.

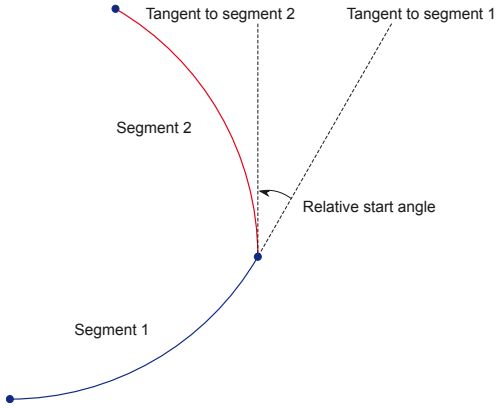


Figure 65: Description of Relative start angle

- ***Additional segment properties available in the Model explorer***

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 increment of the line in the local "1" direction formed by the start and end point of the segment. Δ1 is in the local polycurve space (defined by the orientation axes), not the global model space.
Δ2	The increment of the line in the local "2" direction formed by the start and end point of the segment. Δ2 is in the local polycurve space (defined by the orientation axes), not the global model space.

Note: Δ1 and Δ2 are perpendicular.

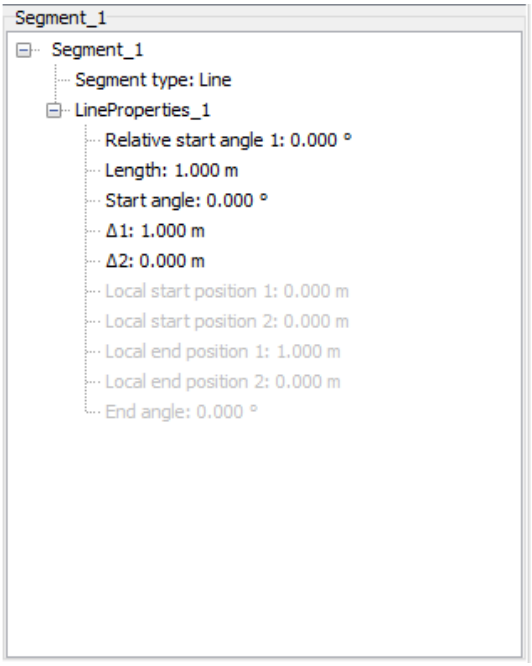


Figure 66: Properties of polycurve segment in Model explorer

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, program will always take the shortest segment. To design those ambiguous cases correctly, use the remaining parameters.

3. ***Modification of polycurves***

Modification of a polycurve is possible in the *Model explorer* after closing the **Polycurve designer** window.

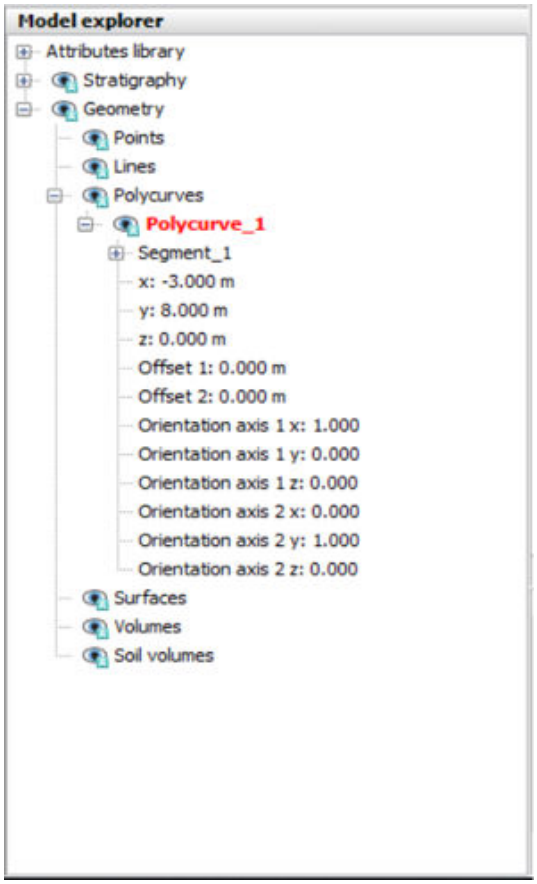


Figure 67: Polycurves in Object explorer

When a polycurve is right-clicked (RMB) either in the **Model/ Object explorer** or in the **drawing** area, a menu is displayed (See Figure 68 (on page 105)) with different options:

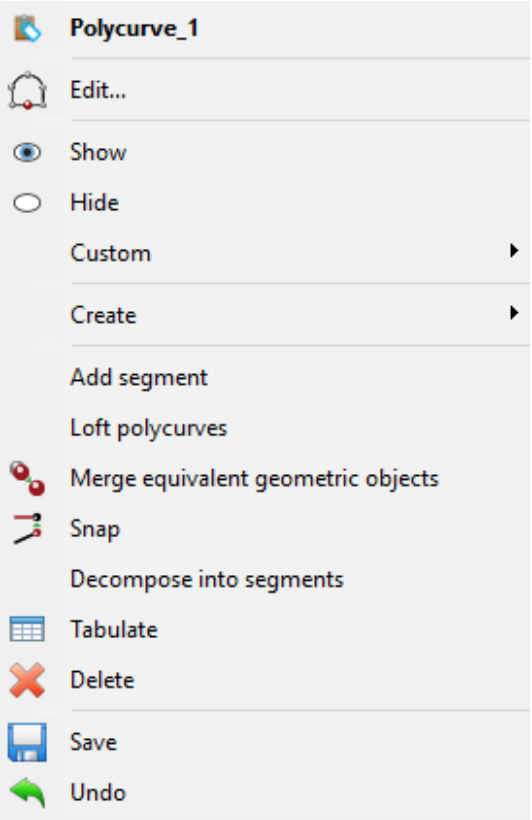


Figure 68: RBM menu of polycurves

Note: Complex geometries can be created by extruding a polycurve or a polygon along a predefined path (Extrude object (on page 92)).

5.2.5 *Create surface*

The third basic input item for the creation of a geometry is a surface. To create a surface click *Create surface* and then on the locations of the points that define the surface.

The first point defined in the surface is the reference point of the surface. After the first point is created, the user may draw a line by entering another point. After the first line is created, the user may draw a surface by entering another point. As the third surface point is defined, the **Surface points** window pops up.

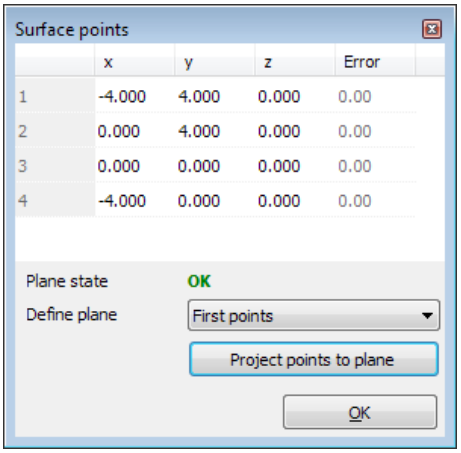


*Figure 69: **Surface points** window*

Note:

- The plane of the created polygon is displayed in the model as the third point is defined. The program allows creation of new points located only in this plane by clicking.
- To add new points out of the plane, right-click the points in the **Surface points** window and select either the *Insert point* or the *Add point* option.

The points defining the surface and their coordinates are listed in the **Surface points** window. Crossing lines are indicated in the *Plane state* option. Existence of crossing lines indicates that no valid surface can be created. When there are no crossing lines, the condition of the *Plane state* is *OK*. The options to define the plane of the surface are:

First points	The first three points in the **Surface points** window define the surface plane. Values in the error column indicate whether the surface is in or out of the defined plane.
Best fit	Best fit of surface points is considered to define a plane surface that fits the best to the defined points. The error column is not applicable.

The points defining the surface can be projected to the plane of the surface by clicking *Project points to plane* in the **Surface points** window. The creation of a surface is finished by clicking on the selection tool in the side tool bar.

Note:

- Note that the points and the lines defining a surface are not shown in the *Model explorer*.
- The points and lines defining the surface can be added to the model by selecting the *Decompose into outlines* option from the right mouse button menu.

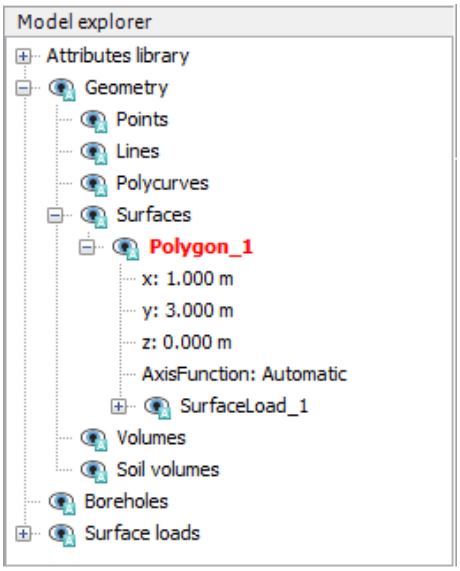


Figure 70: Surface in Model explorer

A *Plate*, *Geogrid*, *Interfaces*, *Surface load*, *Surface prescribed displacement*, *Contraction*, *Surface groundwater flow boundary condition* and *Surface drain* can be assigned to a surface (either created or imported).

Modification of surfaces

The points defining surfaces can be modified using the options available in the menu that appears when the point ID is right-clicked in the **Surface points** window. Using these options a point can be added, inserted or deleted.

The points defining the surface can be relocated by changing their coordinates in the corresponding cells in the **Surface points** window.

Note that the options to modify the points defining a surface are available in the *Create surface* submenu and they are selectable after the surface to be modified is double clicked first.



Figure 71: Create surface menu

Local coordinate system

A local coordinate system for surfaces is defined automatically by the program. Defining a local coordinate system for a surface element is required in order to define the material properties of structural elements having orthotropic material behaviour and to show the forces in the Output program.

For the *Automatic* axis function, the first and second local axes lie in the plane of the surface, whereas the third axis is perpendicular to the plane of the surface (Material data sets for plates (on page 293) of the Material Models Manual). The first axis is automatically assigned as follows:

- If the normal is parallel to the global *z* axis, the first axis points the global *x*-axis direction.
- If the normal is not parallel to the global *z* axis, the first axis is then $Axis_1 = n \times z$ (cross product of the unit vector of *z* pointing upwards and the normal).

When the *Manual* option in the *Axis function* drop-down menu in the *Object explorers* is selected, the user has to define the direction of the first axis. The third axis is determined by the normal and the second axis is determined as $Axis_2 = n \times z$. The coordinate system does not have to be orthogonal.

Note: In case of a straight tunnel it is convenient to use the direction of the tunnel as the direction of the first axis. The direction of the second axis is determined automatically from the first and third axis according to the right hand rule.

Imported surfaces

Predefined surfaces can be imported in the **Structures** mode (Importing geometry (on page 113)). The geometry of an imported surface can be modified after being decomposed into outlines.

Note: Volumes can be created by extruding a polygon along a predefined path created using the *Polycurve* or *NURBS curve* feature (Extrude object (on page 92)).

5.2.6 *Create volume*

In PLAXIS 3D volumes can be created by extruding surfaces (Extrude object (on page 92)). *Import volume* feature enables usage of predefined volumes (Importing geometry (on page 113)). Note that the soil volumes generated according to the soil layers defined in the **Modify soil layers** window and the volumes imported in the **Soil** mode are listed under *Soil volumes* in the *Geometry* subtree of the *Model explorer*. The volumes created by extruding surfaces or that are imported in the **Structures** mode are listed under *Volumes* in the *Geometry* subtree.

Note: The surfaces and the outlines defining a volume created or imported in the **Structures** mode can be added to the model by selecting the *Decompose into surfaces* or *Decompose into outlines* option respectively from the right mouse button menu.

Soil and interfaces material data sets, *Volumetric strain* and *Water conditions* can be assigned to volumes.

5.2.7 *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 surface) 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 below.

Table 2: Assignable features

Loads			
Point	Line	Surface	Volumes
Load Prescribed displacement	Load Prescribed displacement	Load Prescribed displacement	Volume strain
Assignable structures and conditions			
Fixed-end anchor	Beam Node to node anchor Embedded beam Well Line drain	Geogrid Plate Interface (Positive/ Negative) Contraction Surface drain Surface groundwater 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 or a node-to node anchor is assigned to a line in the model, no other features can be assigned to it.

Invert normal

The *Invert normal* option is available for surfaces in the right mouse button menu. Note that the location of an interface relative to the surface to which it is assigned depends on the type of the interface (positive or negative) and the normal of the surface. If the normal of a surface is inverted, the assigned interface will automatically be applied to the other side of the surface.

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

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 3D 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 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 options 1 and 3 are 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*, or *Lines* or *Surfaces* 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.
- Features that are sensitive to local axes need to be checked by the user after the merging.
- 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*).

Decompose

The *Decompose* option enables decomposing the selected geometric entity into lower entities. A surface is decomposed into outlines and a volume is decomposed into surfaces. An example of decomposition of a surface into outlines is given in the chapter *Loading of Suction Pile* of the PLAXIS 3D Tutorial Manual.

Intersect and recluster

A multiple selection of geometric entities can be intersected by selecting the corresponding option in the right mouse menu. The Figure 72 (on page 111) displays two surfaces before and after intersecting.

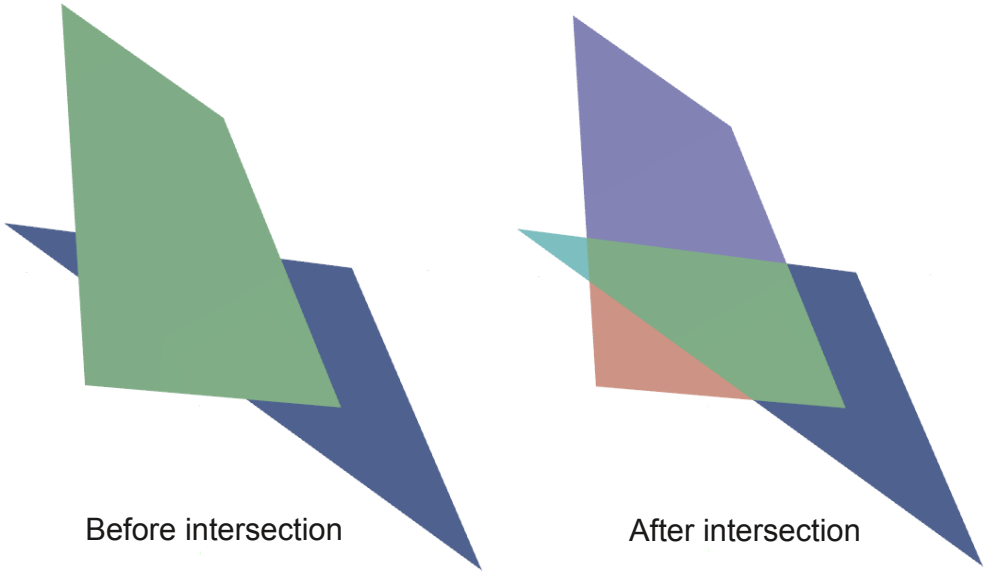


Figure 72: Example of intersection of surfaces

Combine

A multiple selection of geometric entities can be combined (merged/fused together) by selecting the corresponding option in the right-click mouse menu (RMB). The figure below displays two volumes before and after combining. After combining different entities into one new entity, the original entities do not exist anymore.

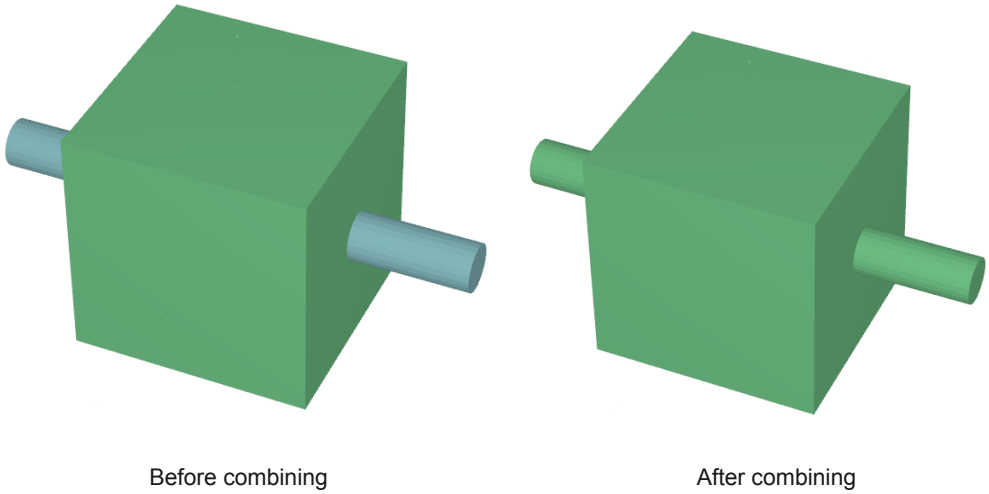


Figure 73: Example of combination of volumes

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.

Loft polycurves GSE

The *Loft polycurves* option can be used to create a surface between two or more different cross sections. To use the loft polycurves option: select the corresponding polycurves from the Model explorer or Structures window → Right click and select *Loft polycurves* option from the menu that appears. The lofted surfaces will be listed under the *Surfaces* subtree in the *Model explorer*. This functionality is especially useful when the user wants to model a tunnel with varying cross sections along its axis.

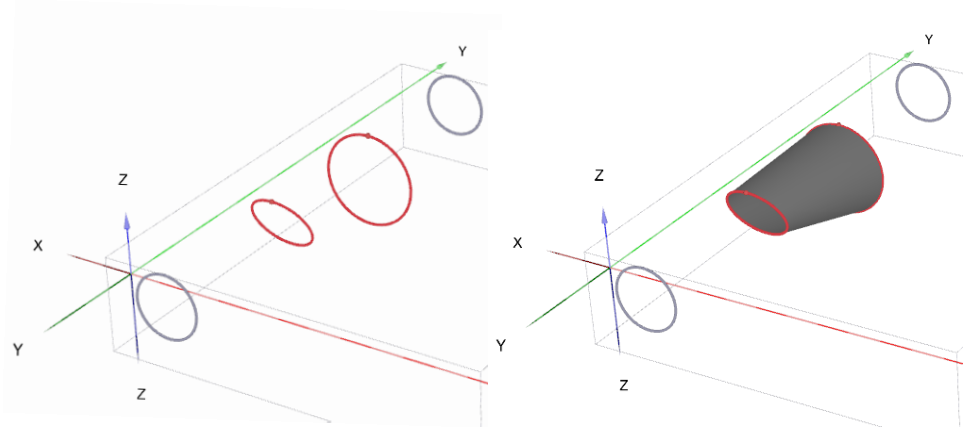


Figure 74: Creating a surface between two polycurves using Loft polycurves option

Blend surfaces

The blend surfaces option can be used to connect two or more surfaces. To use the blend surface option: select the corresponding surfaces from the *Model explorer* or *Structures* window. Right click and select *Blend surfaces* option from the menu that appears. If the surfaces that are blended have identical properties, the blended surface is assigned with the same properties, the non-identical properties are left unassigned. This functionality can be used to connect two or more tunnels that have different cross-sections and / or non-aligned axes.

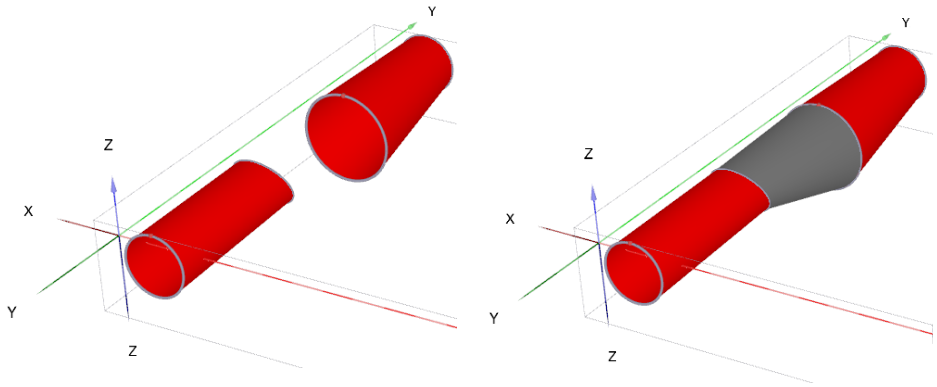


Figure 75: Connecting two surfaces using Blend surfaces option

5.3 Importing geometry



*Figure 76: **Import...** menu*


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 (on page 86). 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.

Extension	File Type
*.brep	OpenCascade shape files.
*.step	3D STEP files.
*.stp	3D STP files.

After selecting the file, the **Import structures** window pops up, where the geometry can be modified. It is possible to import only surfaces or volumes by keeping the corresponding option checked in the **Import structures** window.

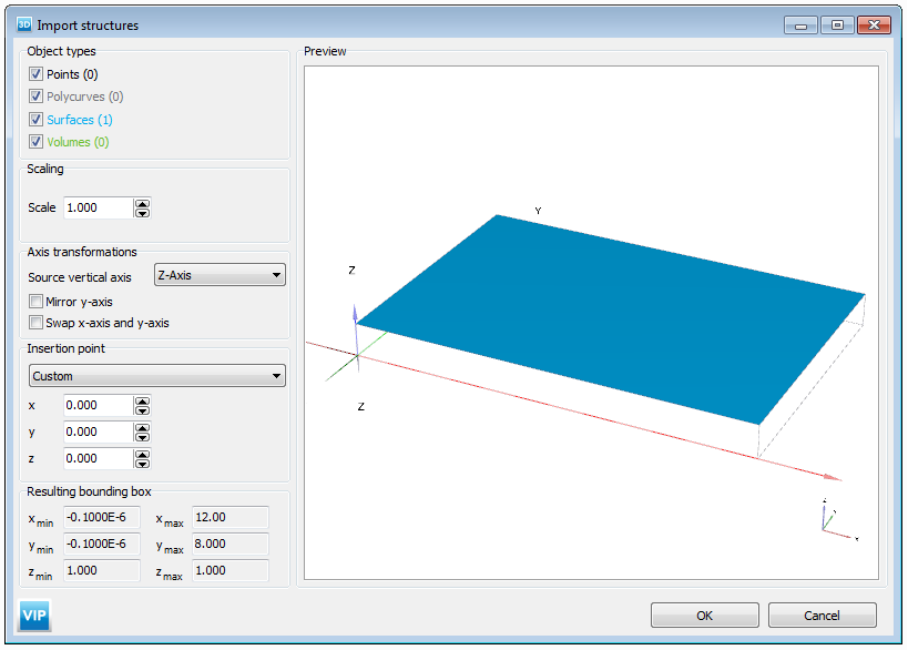


*Figure 77: The **Import structures** window*

5.3.1 *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.

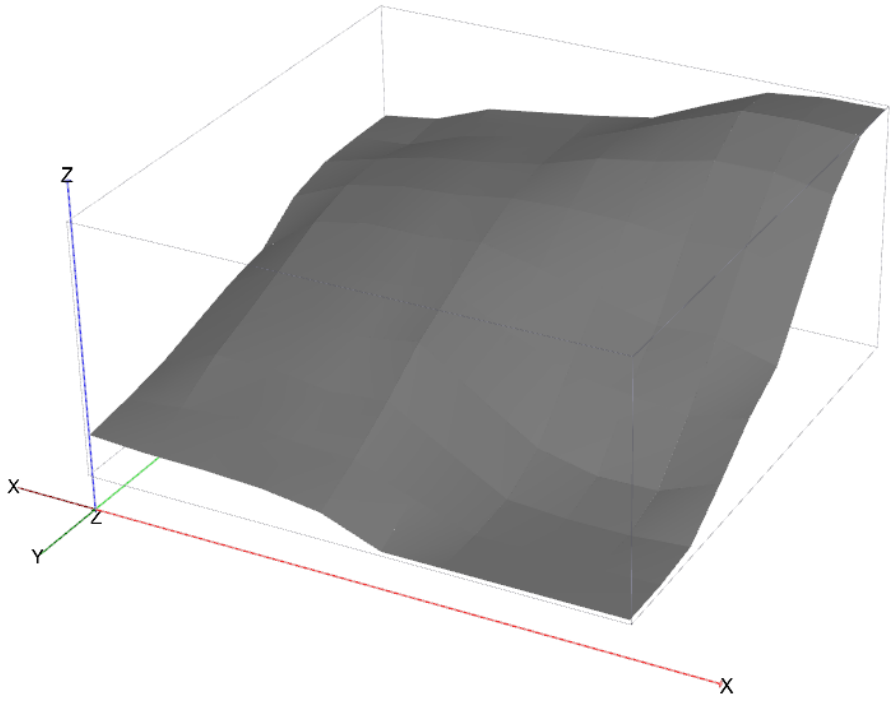


Figure 78: Imported point cloud file

5.3.2 *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.

To see more examples and parameters description, see `importifc` in Concrete model.

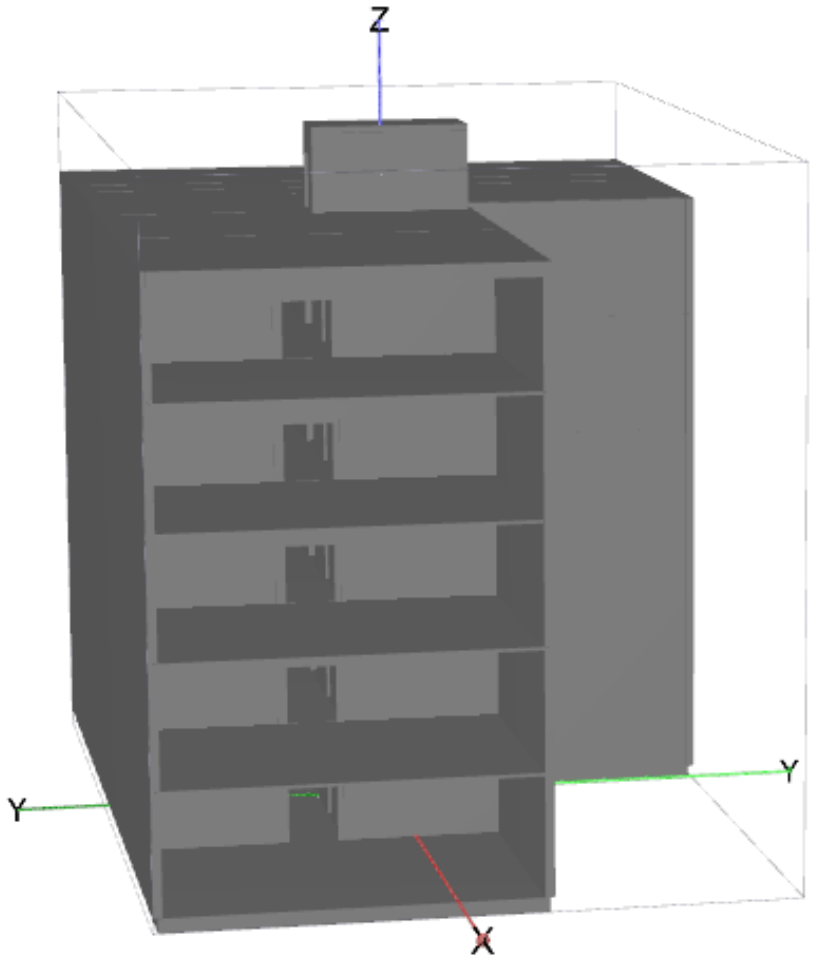


Figure 79: Imported IFC data

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. Alternatively, the options available in the *Create load* menu can be used. To assign loads to existing geometry entities:

- Select the button corresponding to the load from the menu appearing when the *Create load* is selected Figure 80 (on page 118).
- Drag the tool (keep the left mouse button down) and drop it on the geometric entity.

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.

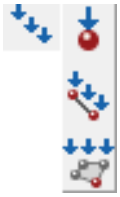


Figure 80: The Create load menu

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 384)).

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 directions.

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.

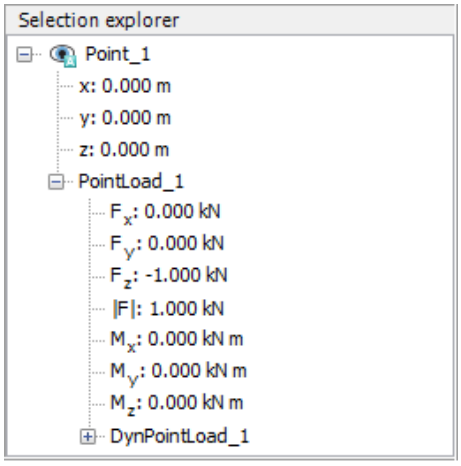


Figure 81: Static point load in the Object explorers

2. Dynamic load

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 126)) can be selected from the drop-down menu. The view of the dynamic load in the *Object explorers* is given in figure below.

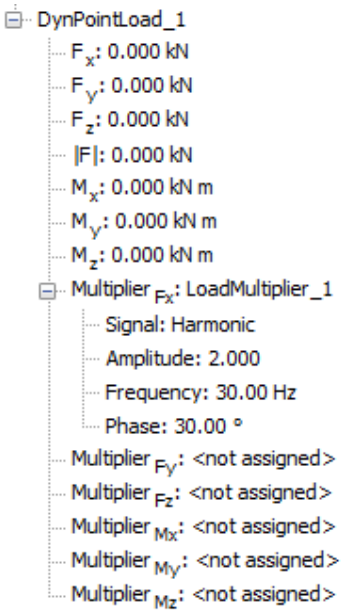


Figure 82: Dynamic point load in the Object explorers

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

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. The default value of point loads is one unit in negative z-direction. The x-, y- and z-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.

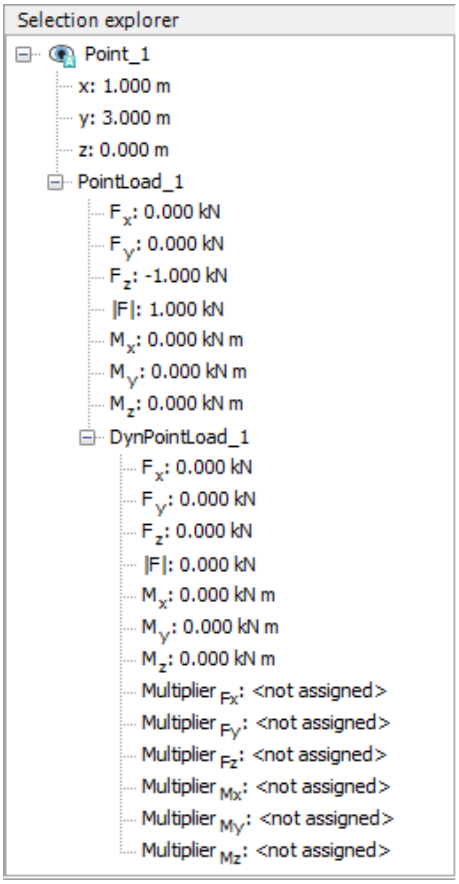


Figure 83: Point load in the Selection explorer

Note:

- Bending moments can only be applied to plates, beams or embedded beams.
- Bending moments applied to nodes that do not belong to the plates, beams or embedded beams are not included in the calculation.
- Plates and soil elements do NOT sustain torsion.
- If torsion is applied to beams/embedded beams, users should ensure that the attached elements have the ability to sustain torsion as well. As an example, if torsion is applied to the start point of a beam, the end point should be connected to a beam or an embedded beam and NOT a plate or the surrounding soil.

5.4.2 *Line load*

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

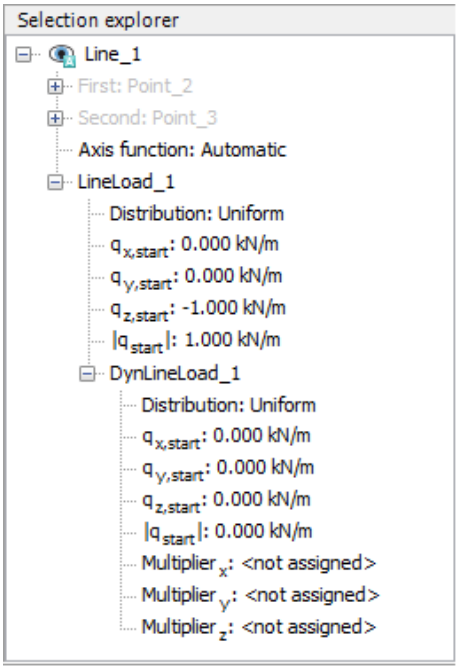


Figure 84: LineDistributed load in the Selection explorer

The distribution options for a line load are:

Uniform	To create a uniform line load.
Linear	A linearly varying line 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.

5.4.3 *Surface load*

A surface load can be created using the *Create surface load* button. The input values of a surface load are given in the unit of force per area (for example kN/m^2). Surface loads may consist of a *x*-, *y*- and/or *z*-component. The magnitude is automatically calculated by the program. If a different magnitude value is defined, the components are calculated accordingly.

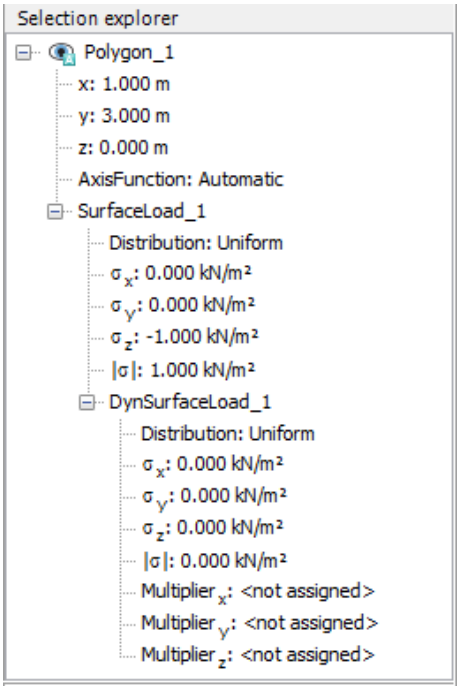


Figure 85: Surface load in Selection explorer

The options to define a surface load are:

Uniform	To create a uniformly distributed surface load.
x-aligned increment	To create a linearly distributed surface load varying in x-direction.
y-aligned increment	To create a linearly distributed surface load varying in y-direction.
z-aligned increment	To create a linearly distributed surface load varying in z-direction.
Vector aligned increment	To create a linearly distributed load varying in a specified direction.
Free increment	To create a general load varying in all directions.
Perpendicular	To create a uniformly distributed load perpendicular to the surface.
Perpendicular, vertical increment	To create a load perpendicular to the surface varying in the direction of gravity (-z) by defining the components and the magnitude of the load at the reference point and its increment in the negative z-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. Alternatively, the options available in the *Create prescribed displacement* menu can be used. 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.

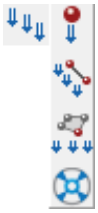


Figure 86: Create prescribed displacement menu

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 382)).

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*

Dynamic Multipliers can be defined for each of the static prescribed displacement components, to apply the prescribed displacement as a function of (dynamic) time in a dynamic calculation. One of the available dynamic multipliers defined in *Attributes library* (Definition of multipliers (on page 126)) can be selected from the drop-down menu. The view of the dynamic prescribed displacement in the *Object explorers* is given in figure below.

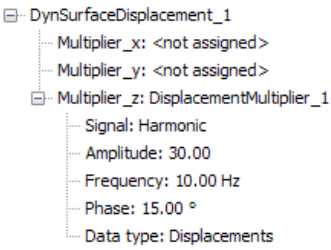


Figure 87: Dynamic prescribed displacement in the Object explorers

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 97)).

The default point prescribed displacement is free in x and y directions and it has a prescribed value of -1 unit in the *z*-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*..

5.5.4 *Line prescribed displacement*

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

The default line prescribed displacement is free in x and y directions and it has a prescribed value of -1 in *z*-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. 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 *Surface prescribed displacement*

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

The *Distribution* options to define a surface prescribed displacement are:

Uniform	A uniformly distributed surface prescribed displacement is defined.
x-aligned increment	A linearly distributed surface prescribed displacement varying in *x*-direction is defined.

y-aligned increment	A linearly distributed surface prescribed displacement varying in *y*-direction is defined.
z-aligned increment	A linearly distributed surface prescribed displacement varying in *z*-direction is defined.
Vector aligned increment	A linearly distributed surface prescribed displacement varying in a specified direction is defined.
Free increment	A general surface prescribed displacement varying in all the directions is defined.
Perpendicular	A uniformly distributed prescribed displacement perpendicular to the surface is defined.
Perpendicular, vertical increment	A prescribed displacement perpendicular to the surface varying in *z* direction is created by defining the components and the magnitude of the displacement at the reference point and its increment in *z* direction.

5.5.6 *Surface contraction*

Surface contraction can be used to simulate a volume loss around a tunnel lining or to apply a contraction to a surface 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 on surfaces, the *Create surface contraction*, the *Create line contraction* option should be selected from the menu appearing when the *Create prescribed displacement* icon is clicked in the side toolbar. Surface contraction may also be defined directly in the *Tunnel designer*.

Surface 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 volume loss around a tunnel, the value of surface contraction to be applied is equal to the volume loss as a percentage of the tunnel volume.
- In geogrid elements, a negative contraction can be used to simulate thermal expansion.

The options to define a surface contraction are:

Constant	A constant contraction is defined. The contraction is applied in the local Axis2-direction.

Axial increment	A linear contraction varying in the local Axis1-direction is defined (Figure 88 (on page 126)). The Axis1-direction is defined for each slice as shown in Figure 88 (on page 126) . The value of the contraction is then applied in the local Axis2-direction. When applied in the tunnel designer, the reference value is taken from the front of each slice. For all other cases, the reference value is defined according to the global axis.

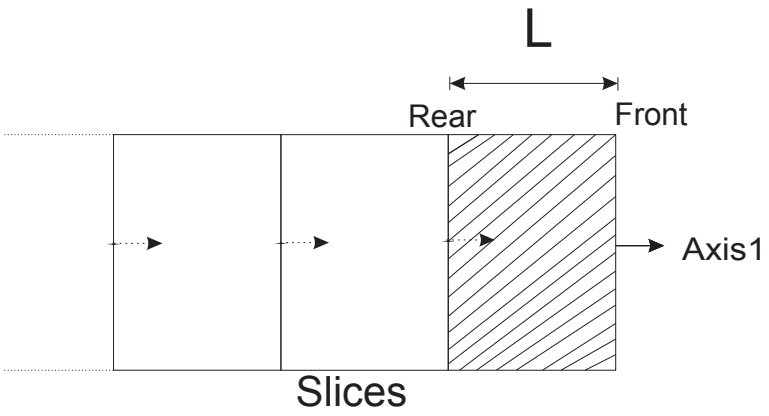


Figure 88: Definition of Axis1 while using Axial increment during surface contraction in tunnel slices, while using the tunnel designer

5.6 Dynamic loading

In PLAXIS 3D, a dynamic load is specified by means of an input value and a multiplier. The actual dynamic value at each time step equals to the input value times the multiplier. Dynamic multipliers can be assigned to the dynamic component of a load (Loads (on page 117)) or to a prescribed displacement (Prescribed Displacements (on page 122)). The dynamic multipliers to be applied in the model can be defined in the *Dynamic multipliers* subtree under the *Attributes library* in the *Model explorer*.

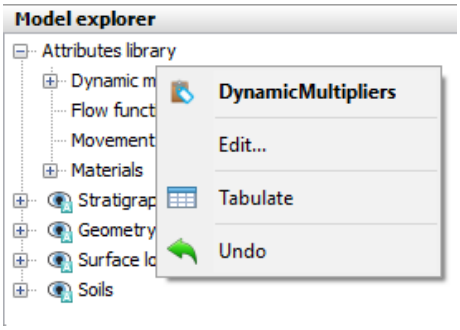


Figure 89: Dynamic multipliers branch in the Model explorer

5.6.1 *Definition of multipliers*

The multipliers can be defined in the **Multipliers** window that pops up when the *Dynamic 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 127) and Signal from table (on page 128) 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 3D 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.

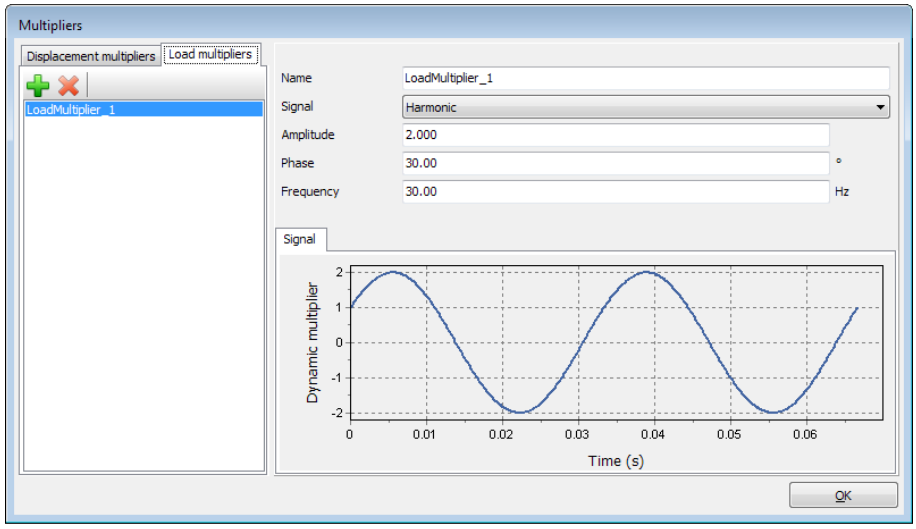


Figure 90: Definition and display of a Harmonic signal in the ***Dynamic multipliers*** *window*

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 131). 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.

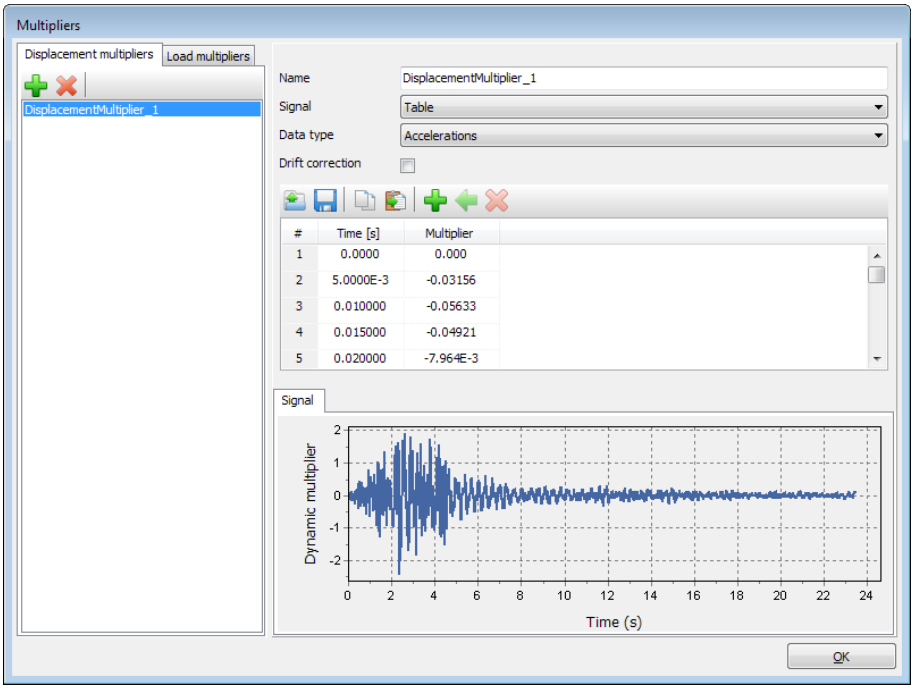


Figure 91: ***Dynamic multipliers*** *window*

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*.

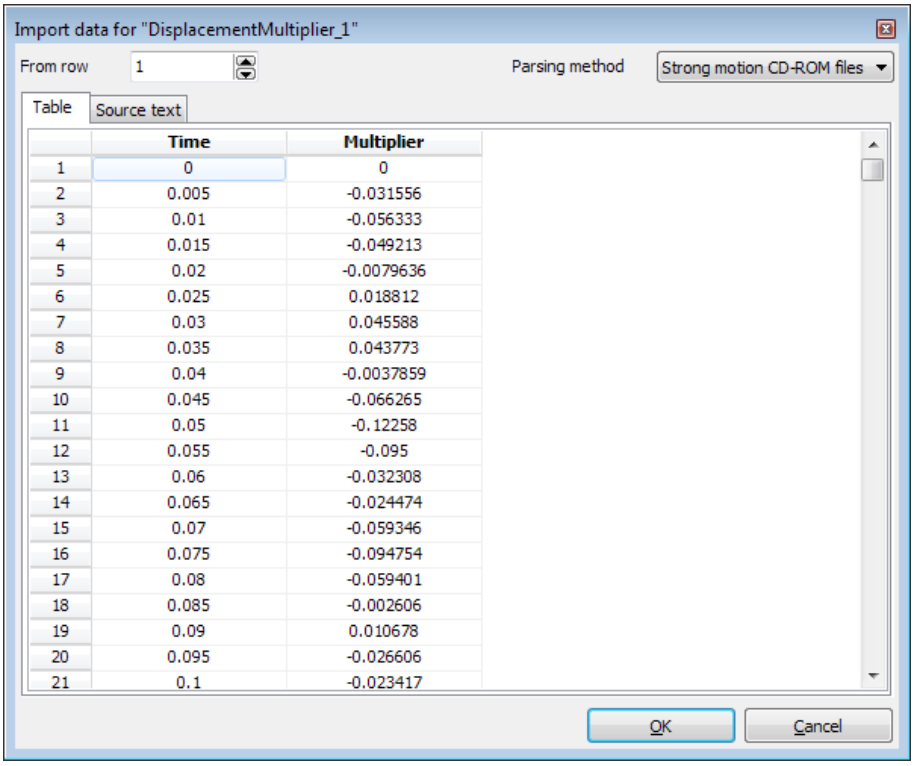


Figure 92: ***Import data*** *window for dynamic multipliers*

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 dynamic 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 dynamic 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:

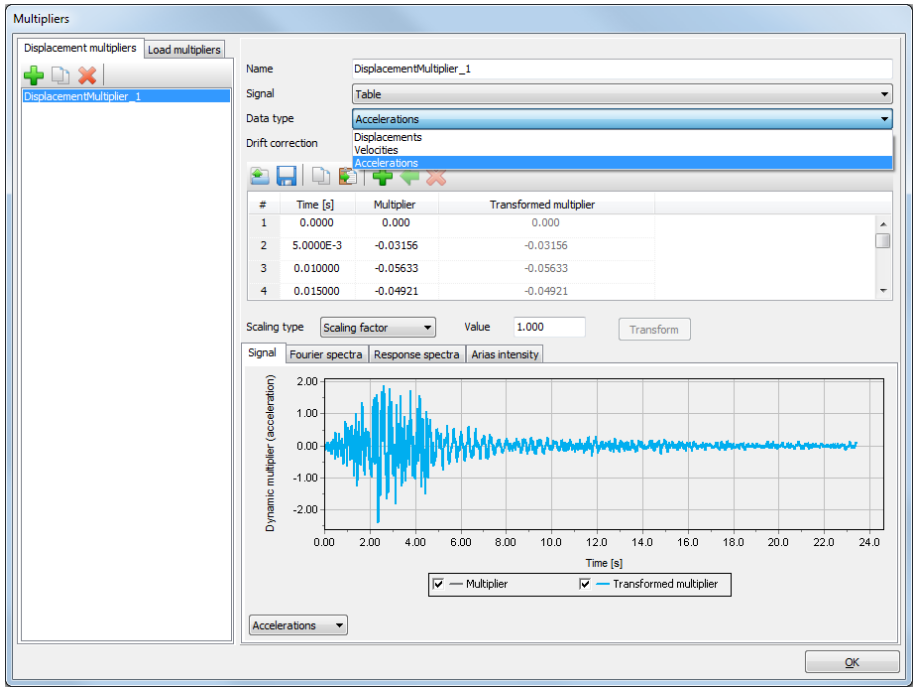


Figure 93: ***Import data*** *window for dynamic multipliers*

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*.

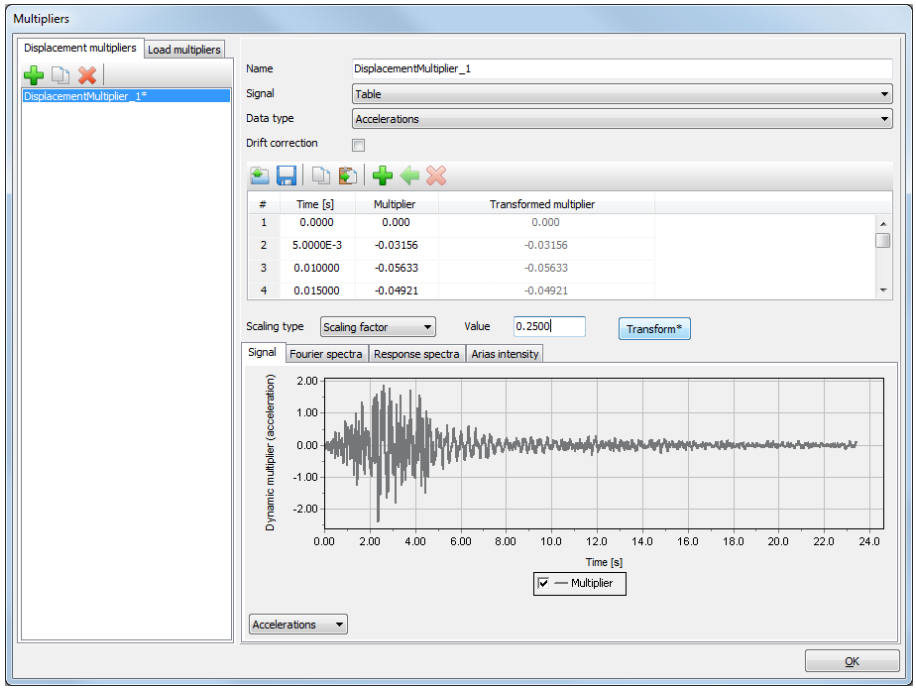


Figure 94: Input accelerograms tranformation

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.

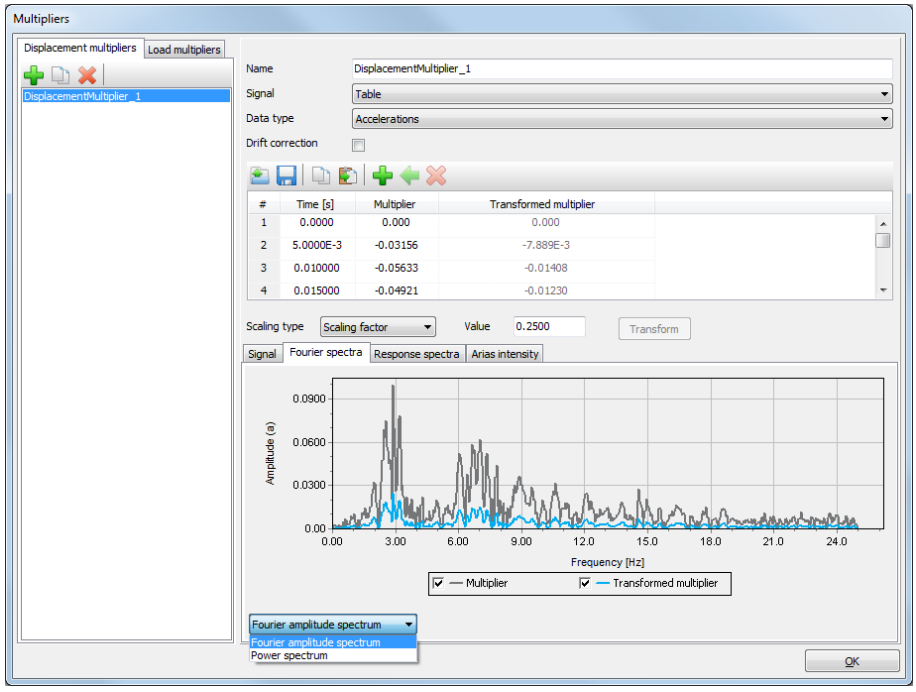


Figure 95: Fourier spectra for input accelerograms

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.

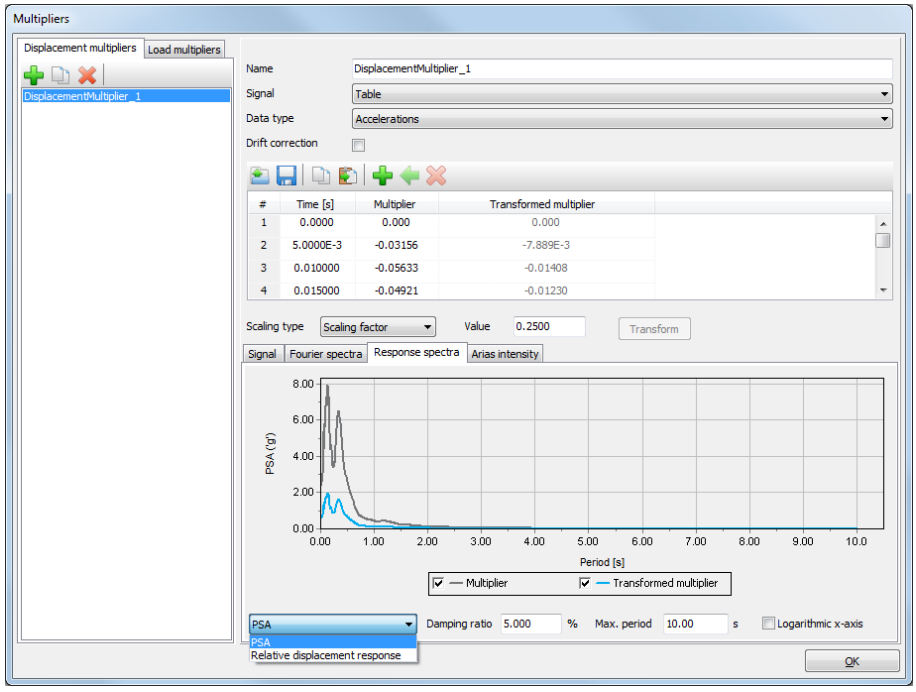


Figure 96: Response spectra for input accelerograms

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*.

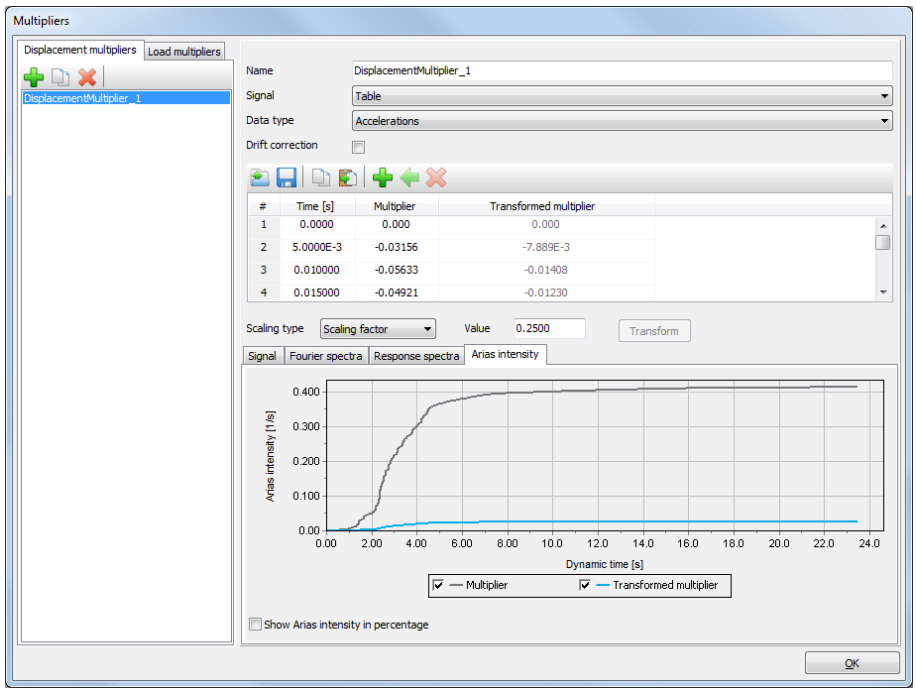


Figure 97: Arias intensity for input accelerograms

5.7 Moving loads GSE

In PLAXIS 3D, a moving load is specified by assigning a path and a movement function to a load. A *Point movement* and *Line movement* can be added by right clicking on a point load or line load and selecting *Add point movement* or *Add line movement* respectively. A *Movement* can be applied to both static and dynamic part of *Point* and *Line* loads. A moving load has two features namely : *Path* and *Movement function*. A *Movement function* is applied on a predefined line which acts as the *Path*.

5.7.1 *Path*

A *Path* is necessary for defining moving loads. A path is a predefined line, over which a line load or point load can have a movement function applied (see figure below).

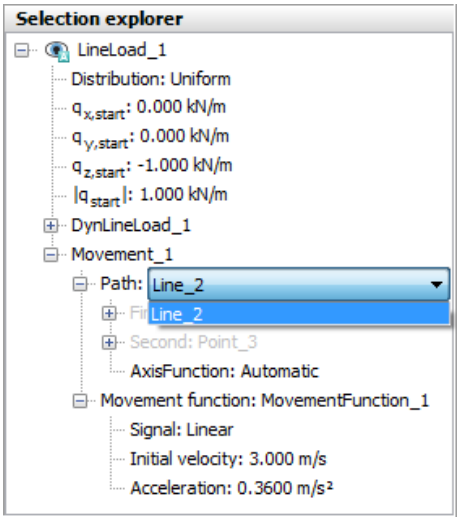


Figure 98: Assigning a Path to a moving load in the Selection explorer

5.7.2 *Definition of movement functions*

A **Movement function** can be defined when the *Movement function* subtree under the *Attributes library* in the *Model explorer* is right-clicked and the *Edit...* option is selected in the appearing menu.

All *Movement functions* can be shown and edited by using the menu *Structures > Show movement functions*.

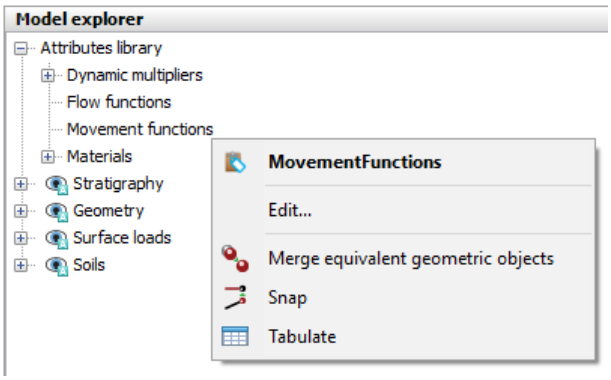


Figure 99: Movement function branch in the Model explorer

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

Name	The name of the movement function can be defined.
Signal	The type of the signal can be assigned. The options available are *Harmonic*, *Table* and *Linear*. More information on definition of a harmonic function, definition of a function in a table and definition of a movement function is given in Harmonic signal (on page 136), Signal from table (on page 137), Linear (on page 139).

In the lower panel, it is possible to view the *Signal* as both a *Velocity* and the relative *Position* of the load (Figure 100 (on page 137)).

5.7.3 *Harmonic signal*

In PLAXIS a harmonic movement is defined as:

$$V = V_0 + A\sin(\omega t) \qquad Eq.\ [3]$$

where:

V_0 Initial velocity

A Amplitude of the movement function

ω $2\pi f$ with f = Frecuency in *Hz*

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

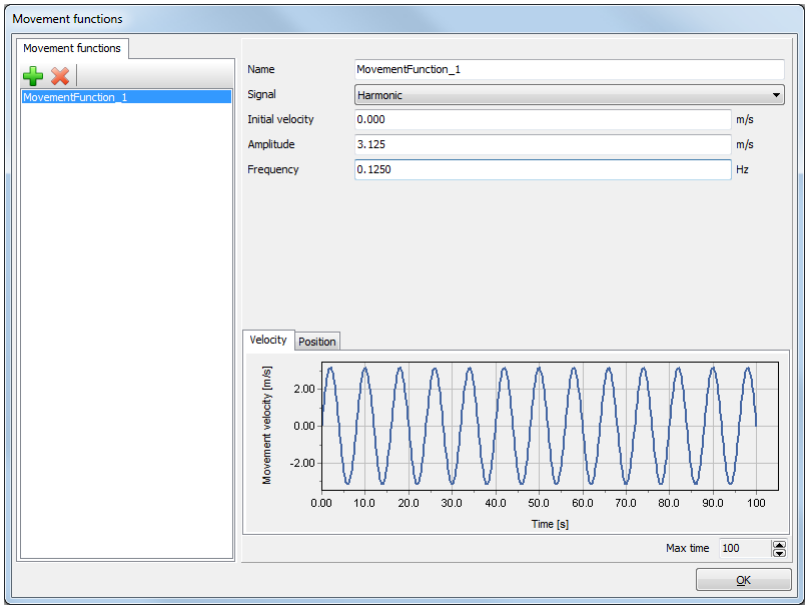


Figure 100: ***Movement function*** *harmonic signal definition window*

5.7.4 *Signal from table*

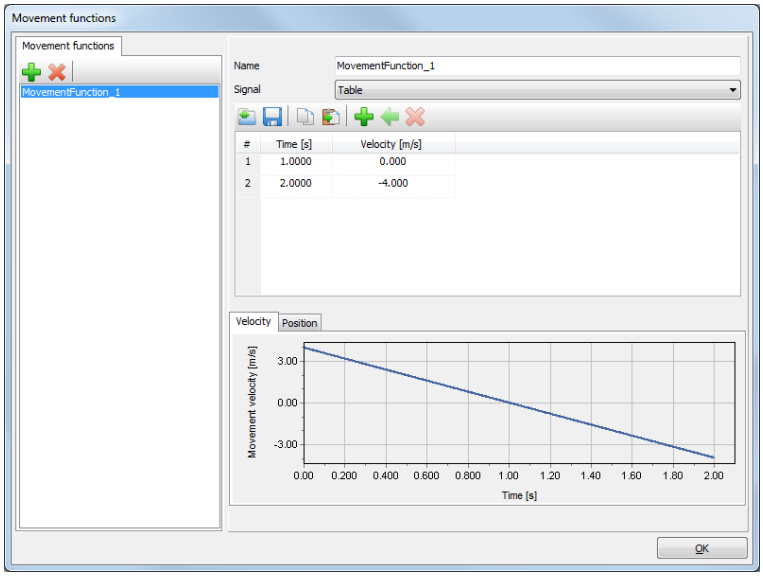


Figure 101: Movement functions with a signal defined by a Table.

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 *Velocity* 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.

Importing data for signals

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

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 138)). The data and the plot is displayed in the **Movement functions** window after pressing *OK*.

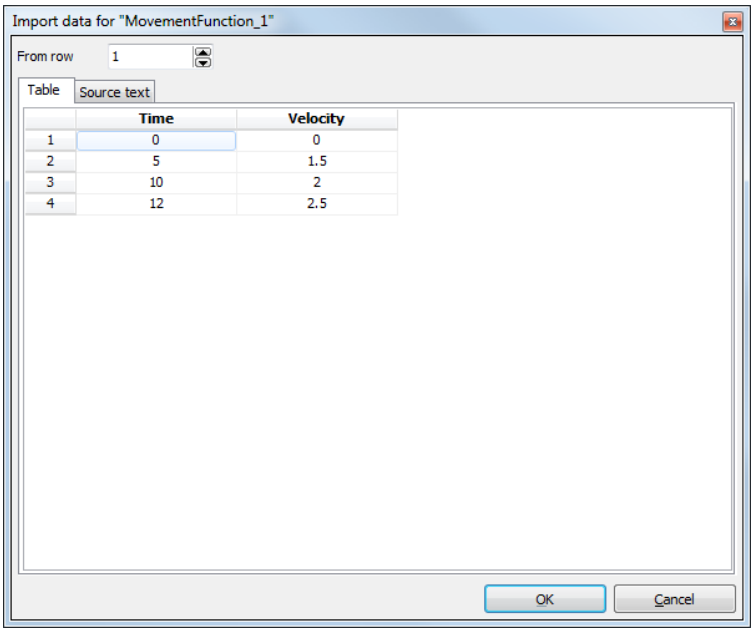


*Figure 102: **Import data** window for movement functions*

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 can be created with any text editor. For every line a pair of values (time and corresponding velocity) must be defined, leaving at least one space between them. Note that PLAXIS 3D only supports the English notation of decimal numbers using a dot. The resulting graph of the input data is shown in the **Velocity** tabsheet of the **Movement function** window.

5.7.5 *Linear*

This option can be used to describe a linear movement in time. For a linear variation of velocity, the input of the following parameters are required:

Initial velocity	This parameter is the initial velocity, specified in unit of length per time. Hence, together with the acceleration this parameter determines the rate of the velocity increase or decrease.
Acceleration	This parameter represents the acceleration for the movement function, expressed in unit of length per unit of time, squared.

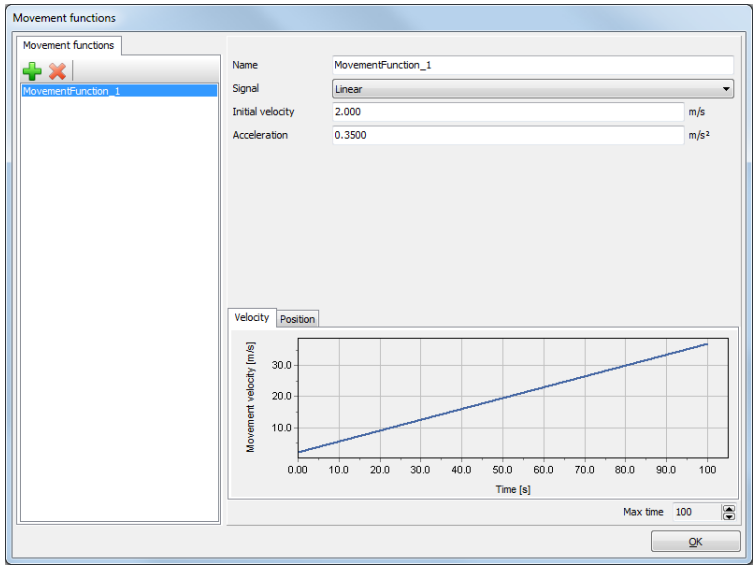


*Figure 103: **Linear** signal for velocity for the movement function*

5.8 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 104: 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* 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 318).

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

5.8.1 *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. 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 97)). The default direction is (1, 0, 0), which is in positive x-direction. The direction of the fixed-end anchor is defined starting from the fixed-end anchor connection point in the fixed-end anchor direction. The user may enter the fixed-end anchor direction by specifying the individual *x*-, *y*- and *z*-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 105 (on page 141)): 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.

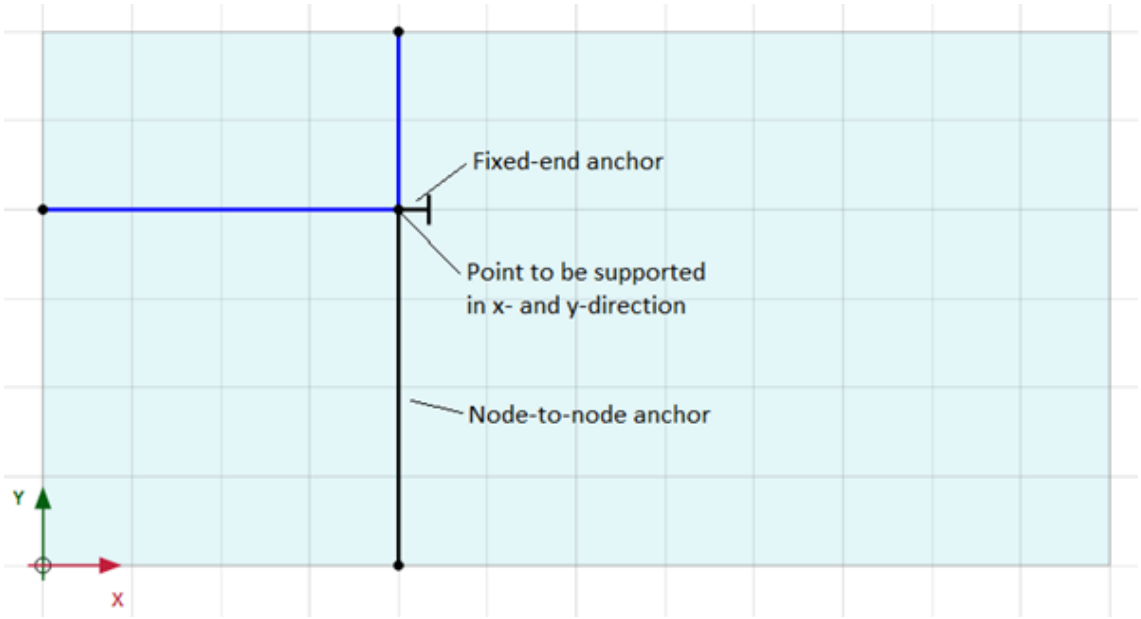


Figure 105: Representation of connection of fixed-end anchor and node-to-node anchor

Anchor material properties

The material properties of fixed-end anchors are contained in *Anchors* material data sets (Material data sets for anchors (on page 315)) 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 318)).

5.8.2 *Node-to-node anchors*

Node-to-node anchors can be used to model a spring connection between two points. Typical applications include the modelling of a cofferdam.

The creation of a node-to-node anchor is similar to the creation of a geometry line (Create line (on page 98)).

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 318)).

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

5.8.3 *Beams*



Beams are structural objects to model slender (one-dimensional) structures with a significant flexural rigidity (bending stiffness) and an axial stiffness. The creation of a beam is similar to the creation of a geometry line (Create line (on page 98)).

Beam elements

After meshing beams are composed of 3-node line elements (beam elements) with six degrees of freedom per node: three translational degrees of freedom (u_x, u_y and u_z) and three rotational degrees of freedom (φ_x, φ_y and φ_z). The beam elements are based on Mindlin's beam theory (Bathe, 1982 (on page 510)). 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. Element stiffness matrices are based on the properties as defined in the material data sets and numerically integrated from the four Gaussian integration points (stress points) along the length of the element and numerically integrated from the 4 (2x2) Gaussian integration points over the cross section of the element. These 4 stress points are located at a distance $\frac{\sqrt{3}d_{eq3}}{6}$ in the local *3*-direction of the beam and at a distance of $\frac{\sqrt{3}d_{eq2}}{6}$ in the local *2*-direction of the beam, where:

$$d_{eq3} = \sqrt{12\frac{I_3}{A}} \qquad Eq.\ [4]$$
$$d_{eq2} = \sqrt{12\frac{I_2}{A}}$$

Details about the element formulation are given in the Scientific Manual.

When a beam element is connected to another structural element, they share all degrees of freedom in the connected node, which implies that the connection is rigid (moment connection) unless a *(custom) Connection* is defined (Definition of connections (on page 385)).

When a beam extends to the model boundary, additional boundary conditions for the rotational degrees of freedom are automatically applied (Model conditions (on page 390)).

Beam material properties

The material properties of beams are contained in *Beams* material data sets (Material set (on page 303)) 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 318)).

5.8.4 *Embedded beams*


An embedded beam is a structural object (such as a pile, rock bolt or grout body) composed of beam elements that can be placed in arbitrary direction in the sub-soil and that interacts with the sub-soil by means of special interface elements. The interaction may involve a skin resistance as well as a foot resistance. The skin friction and the tip force are determined by the relative displacement between the soil and the pile. Although an embedded beam does not occupy volume, a particular volume around the pile (elastic zone) is assumed in which plastic soil behaviour is excluded. The size of this zone is based on the (equivalent) pile diameter according to the corresponding embedded beam material data set. This makes the pile behave almost like a volume pile. However, installation effects of piles are not taken into account and the pile-soil interaction is modelled at the centre rather than at the circumference.

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's necessary to properly evaluate these factors in order to simulate the pile installation effect.

The creation of a pile is similar to the creation of a geometry line. The point of connection of the pile to the surrounding geometry and the type of the connection can be specified in the Object explorers.

The behaviour options available for *Embedded beam* are:

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

Depending on the *Embedded beam* behaviour for *Pile* and *Rock bolts*, the connection point may be indicated as:

Top/Bottom	In the case the *Embedded beam* 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 that has the highest z-coordinate in the model. In the rare case of an exactly horizontal embedded beam defined as a *Pile*, *Top* refers to the point with the minimum x-coordinate. In case z-coordinate and x-coordinate are the same, the point that has the minimum y-coordinate is considered the *Top*.
First/Second	In the case the *Embedded beam* 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 is selected, whereas second refers to the second point of the corresponding geometry line.

In the case of the pile connection point coinciding with soil elements, the connection behaviour is as follows:

Free	The *Connection point* is free to move and rotate relative to the surrounding soil. The pile and the surrounding soil interact only through the special interface elements.
Hinged	The displacement at the connection point of the pile is directly coupled with the displacement of the soil element in which the pile connection point is located, which means that they undergo exactly the same displacement. However, their rotation may be different.
Rigid	The displacement and rotation at the pile connection point are both coupled with the displacement and rotation of the soil element in which the pile top is located, provided that this element has rotational degrees of freedom. This option only applies if the pile connection point coincides with structural elements like plates or beams.

In the case of the pile connection point coinciding with structural elements (without interface elements) the connection behaviour is as follows:

Free	The *Connection point* is free to move and rotate relative to the surrounding soil and the structural element. The pile interacts with the surrounding soil and the structural elements only through the special interface elements. Note that the structural elements are not free to move and rotate relative to the surrounding soil.
Hinged	The displacement at the connection point of the pile is directly coupled with the displacement of the structural element and/or the surrounding soil in which the pile connection point is located, which means that they undergo exactly the same displacement.
Rigid	The displacement and rotation at the pile connection point are both coupled with the displacement and rotation of the structural and/or soil element in which the pile top is located.

In the case of the pile connection point coinciding with structural elements which interaction with the surrounding soil is modelled using interfaces, the connection behaviour is as follows:

Free	The *Connection point* is free to move and rotate relative to the surrounding soil. The pile interacts with the surrounding soil and the structural elements only through the special interface elements. In the same way, the structural element interacts with the surrounding soil through the defined interface elements. This is the default option in the program.
Hinged	The displacement at the connection point of the pile is directly coupled with the displacement of the structural element in which the pile connection point is located, which means that they undergo exactly the same displacement. Note that the structural element interacts with the surrounding soil through the defined interface elements.

Rigid	The displacement and rotation at the pile connection point are both coupled with the displacement and rotation of the structural element in which the pile top is located. Note that the structural element interacts with the surrounding soil through the defined interface elements.

The options specified for the embedded beam in the *Object explorers* are valid for the connection of the anchor-pile connection to the surrounding soil.

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

Note:

- When embedded beams penetrate a volume 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 a 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.
- An alternative option for this is between an embedded beam and a volume 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 (linked to the elongation length), the user could insert an elastic plate at the volume cluster side where the connection with the embedded beam takes place. The beam can be now linked to the plate without elongating it in the volume cluster structure.

Embedded beam elements

An embedded beam consists of beam elements with special interface elements providing the interaction between the beam and the surrounding soil.

After meshing, the beam elements are 3-node line elements with six degrees of freedom per node: Three translational degrees of freedom (u_x, u_y and u_z) and three rotational degrees of freedom (φ_x, φ_y and φ_z). Element stiffness matrices are numerically integrated from the four Gaussian integration points (stress points). The element allows for beam deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied.

The special interface elements are different from the regular interface elements as used along walls or volume piles. Therefore, at the position of the beam element nodes, virtual nodes are created in the soil volume element from the element shape functions. The special interface forms a connection between the beam element nodes and these virtual nodes, and thus with all nodes of the soil volume element.

Pile forces (structural forces) are evaluated at the beam element integration points and extrapolated to the beam 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 Scientific Manual.

Embedded beam material properties

The material properties of embedded beam are contained in *Embedded beamsEmbedded beam rows* material data sets (Material set (on page 308)) 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 318)).

More information regarding the material properties of the embedded beam is given in Interaction properties: bearing capacity (on page 312).

5.8.5 *Plates*

Plates which are actually shell elements are structural objects used to model thin two-dimensional structures in the ground with a significant flexural rigidity (bending stiffness). The creation of a plate is similar to the creation of a geometry surface (Create surface (on page 105)).

Plate elements

After meshing plates are composed of 6-node triangular plate elements with six degrees of freedom per node: three translational degrees of freedom (u_x, u_y and u_z) and three rotational degrees of freedom (φ_x, φ_y and φ_z). The plate elements are based on Mindlin's plate theory (Bathe, 1982 (on page 510)). This theory allows for plate deflections due to shearing as well as bending. In addition, the element can change length when an axial force is applied. Element stiffness matrices are based on the properties as defined in the material data sets and numerically integrated from the three pairs of Gaussian integration points (stress points). Within each pair, stress points are located at a distance $\frac{\sqrt{3}}{6}d$ above and below the plate centre-line.

It is important to note that a change in d will change 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, d must remain unchanged. Details about the element formulation are given in the Scientific Manual.

Plate properties

The material properties of plates are contained in *Plates* material data sets (Material set (on page 293)) 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 318)).

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 - *Chapter 18 - 3D Embedded beam*.

5.8.6 *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.

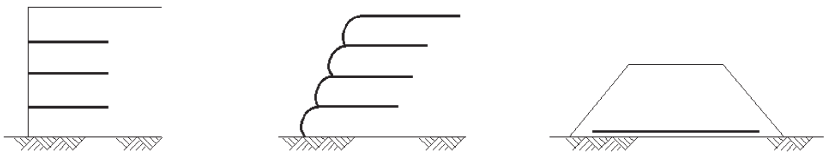


Figure 106: Applications in which geogrids are used

Geogrid elements

After meshing, geogrids are composed of 6-node triangular surface elements with three translational degrees of freedom per node (u_x, u_y and u_z). Element stiffness matrices are based on the properties as defined in the material data sets and numerically integrated from the three Gaussian integration points (stress points). The element can change length when a tension force is applied. Details about the element formulation are given in the Scientific Manual. When a geogrid element is connected to another structural element, they share the translational degrees of freedom in the connecting node(s).

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 Material set (on page 298) and Reference Manual.

When using geogrid elements in PLAXIS 3D 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.

5.8.7 *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 volumes.


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 surface (Create surface (on page 105)). In this case a surface with an interface assigned to it will be created. When the geometric entity (surface) 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 108).

Distinction is made between a positive interface (the side of the surface at the positive local *z*-direction) and a negative interface (the side of the surface at the negative local *z*-direction).

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

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

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 107 (on page 148)).

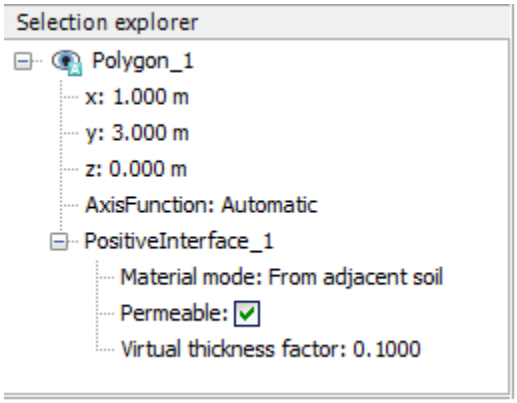


Figure 107: Interfaces in the Selection Exploter

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 258).
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 permeable. Impermeability can be introduced by assigning an interface to the geometric entity, besides the structural element. In principle, interface permeabilities (cross permeability and parallel permeability) are defined in the corresponding material data set (Interfaces tabsheet (on page 258)). However, note that for an interface a checkbox is available in the *Objects explorers*, to make the interface either *Active in flow* or *Inactive in flow*. This has consequences for the actual interface permeability being used in a calculation phase (Table 4 (on page 148)). In *Flow only* calculations no structural element needs to be assigned to the geometric entity as deformations will not be taken into account. Hence, it is sufficient to use only interfaces to block the flow. Note that the end points of an interface are always permeable.

Table 4: Resulting settings for permeabilities in *Interfaces*

Setting in material database	Active in flow	Inactive in flow	Remarks

Cross permeability	Impermeable	Infinite cross resistance (or zero permeability)	Infite cross permeability (or zero resistance)	No output of flow q_n available.
	Semi-permeable	Input value of hydraulic resistance used (Interfaces tabsheet (on page 258))	Infite cross permeability (or zero resistance)	Output of flow q_n available when active.
	Fully permeable	Infite cross permeability (or zero resistance)	Infite cross permeability (or zero resistance)	No output of flow q_n available.
Parallel permeability	Drainage conductivity	Input value of drainage conductivity used (Interfaces tabsheet (on page 258))	Zero parallel flow	Output of discharge Q_1 available when active.

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 322)). 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 258).

Interface elements

After meshing, interfaces are composed of 12-node interface elements. Interface elements consist of pairs of nodes, compatible with the 6-noded triangular side of a soil element or plate element. In some output plots, 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.

The stiffness matrix for triangular interface elements is based on the properties as defined in the material data sets and obtained by means of Gaussian integration using 6 integration points. The position of these integration points (or stress points) is chosen such that the numerical integration is exact for linear stress distributions. For more details about the element formulation reference, see Scientific Manual.

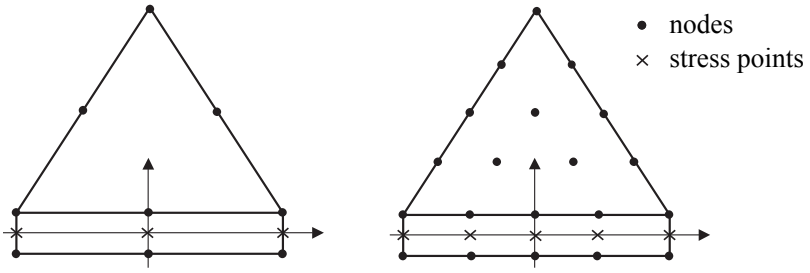


Figure 108: Distribution of nodes and stress points in interface elements and their connection to soil elements

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 a beam), interface element node pairs are locally 'degenerated' to single nodes to avoid a disconnection between both structural elements.

Interfaces around corner points

Figure 109 (on page 150) and Figure 110 (on page 150) 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 110 (on page 150).

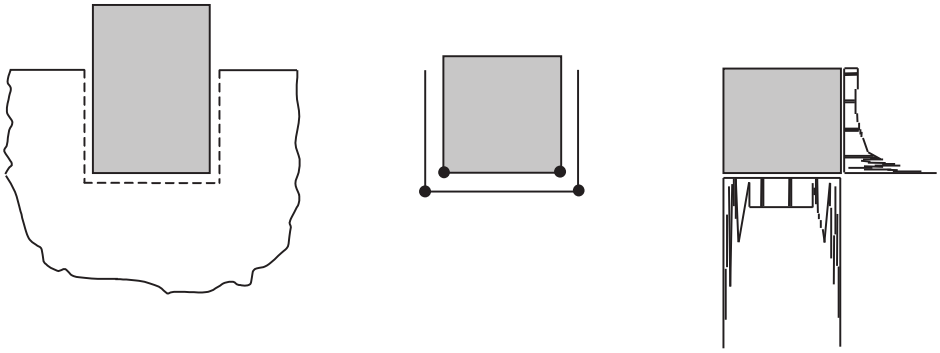


Figure 109: Inflexible corner point, causing poor quality stress results

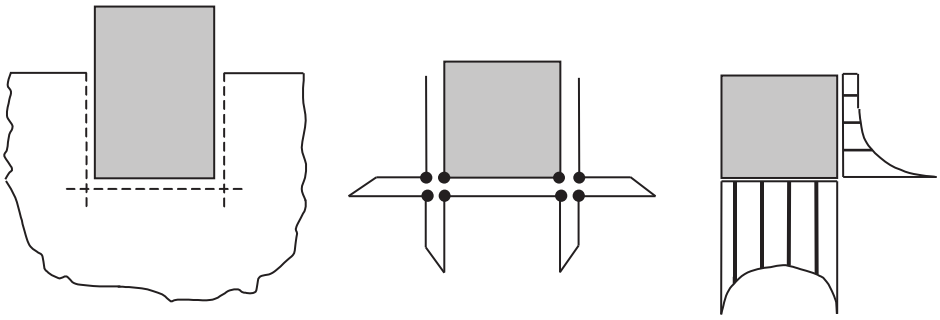


Figure 110: Flexible corner point with improved stress results

Figure 110 (on page 150) 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 258)).

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. Also when structural elements are connected perpendicular to each other (for example, a plate is connected to a beam), interface element node pairs are locally 'degenerated' to single nodes to avoid a disconnection between both structural elements, connecting them rigidly. A description on how to define a non-rigid connection is given in Definition of connections (on page 385). Additional theoretical details on this special use of interface elements is provided by Goodman, Taylor & Brekke (1968) (on page 511) and Langen & Vermeer (1991) (on page 512) .

5.8.8 *Rigid bodies*

When modelling soil-structure interaction problems, the structures are usually much stiffer and stronger than the soil. In many situations it is still relevant to model structures with their actual stiffness and strength properties in order to calculate realistic deformations and structural forces in those structures. Moreover, contact stresses between soil and structures are influenced by their relative stiffnesses; for example contact stresses behind a soil retaining wall can vary from 'passive' (K_p) to 'active' (K_A), depending on the stiffness of the wall and the stiffness of wall supports (struts, anchors).

In situations where deformations of structures and structural (internal) forces are not supposed to be relevant in output data, these structures may simply be modelled as 'stiff' structures, with an arbitrary stiffness that is sufficiently large compared to the stiffness of the soil. However, considering an arbitrarily large stiffness may deteriorate the condition of the global stiffness matrix, and should therefore be used with care. As an alternative to this, PLAXIS 3D provides the option to create so-called 'rigid body' objects based on predefined geometry components (surfaces and volumes).

Examples where rigid bodies may be used are the following:

- Rigid foundation plate (footing, mud mat)
- Rigid wall
- Bucket foundation
- Spud can (foundation for jack-up platform legs)

To create a rigid body object in the **Structures** mode, create the required geometry components first; then (multi-) select the corresponding surfaces and volumes that are to be included in the rigid object, and select the *Create rigid body* option from the context menu. Multiple rigid bodies may be created in this way in a PLAXIS 3D model. The rigid bodies will appear in the *Model explorer*. Each rigid body has a *Reference point* associated to it and it should be given a combination of forces / moments and/or displacements / rotations in x, y and z-direction (Figure 111 (on page 152)), in order to apply external forces and to set its boundary conditions.

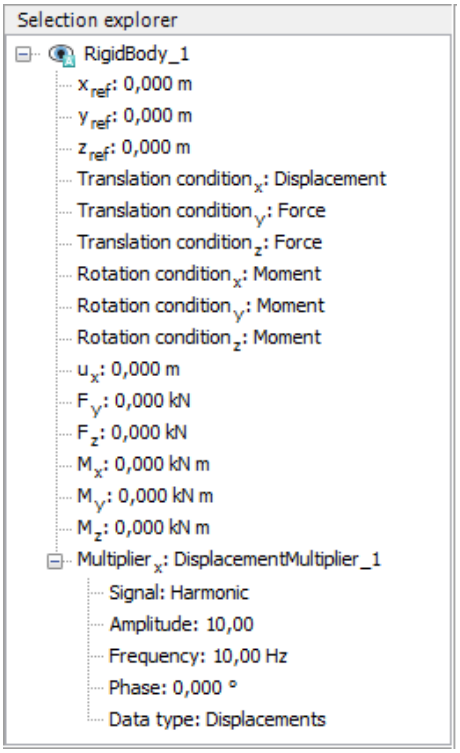


Figure 111: Rigid body in the Selection explorer

The reference point of a rigid body can be an arbitrary point (independent from existing geometry points), defined by its three coordinates (x y z). Upon mesh generation, this point will NOT appear as mesh node. However, during calculations, the reference point is associated with six independent degrees-of-freedom (u_x, u_y, u_z, φ_x, φ_y, φ_z) and six corresponding force components (F_x, F_y, F_z, M_x, M_y, M_z) in the global system of equations. During calculations, all stiffness components related to any of the nodes involved in the rigid body's surface(s) and volume(s) will be transformed into stiffness components related to the movement of the rigid body reference point, while the original nodal degrees-of-freedom are linked to the movement of the rigid body reference point. In a similar way, during dynamic calculations, mass components related to any node of the rigid body will be transformed into mass components related to the rigid body reference point. In this way, the rigid body itself will not deform (it will move, but not deform) and therefore it is not possible to calculate internal stresses and structural forces. On the other hand, contact stresses and forces between the rigid body and the soil are properly calculated, as long as there are interface elements between the rigid body and the soil.

Conditions

Since the reference point involves separate degrees-of-freedom that are independent from the rest of the finite element model, boundary conditions need to be specified for all six degrees-of-freedom of a rigid body. These boundary conditions may involve a *Force condition* (*Force* or *Moment*) or a *Displacement condition* (*Displacement* or *Rotation*). It is possible to add dynamic boundary conditions to rigid bodies by adding multipliers when the *Translation Condition* is selected as *Displacement*. This is only possible in *Dynamic* calculation phases. Note that the *Moment* and *Rotation* conditions refer to a moment or rotation around the corresponding axis, where a positive value corresponds to a clock-wise rotation when looking into the axis positive direction. The type of condition for each of the degree-of-freedom must be selected by the user. The user is responsible for selecting rigid body conditions that are compatible with the boundary conditions of the full finite element model.

Instead of or in addition to conditions in the reference point, external loads (point loads, line loads or distributed loads) may be defined at a rigid body in the geometry. Considering half a symmetric model of a rigid plate, while the y=0 plane is the symmetry plane (Figure 112 (on page 153)), the plate can only move in x- and z-direction and rotate around the y-axis. Hence, in this case the user has to make sure that u_y=0, φ_x=0 and φ_z=0 are specified as conditions for the rigid body, while the remaining three components can be set as desired.

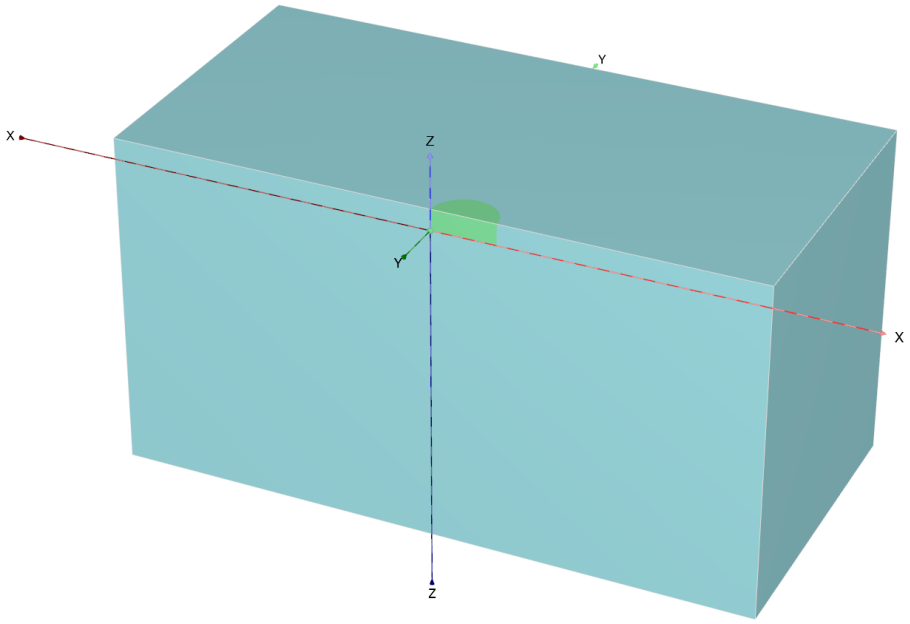


Figure 112: Rigid body

Limitations

The rigid body element in PLAXIS 3D is an advanced feature. The user has to be aware of the limitations of this feature and is responsible for setting the right conditions to ensure the consistency of the model. Incompatibilities in the conditions of the rigid body are not checked by the PLAXIS 3D user-interface. In addition to the aforementioned example where appropriate symmetry conditions have to be applied at the rigid body reference point, there are some other limitations related with the use of rigid bodies during calculations.

- Note that prescribed displacements that are applied on nodes as a part of an active rigid body, are not taken into account during the calculation. Instead, prescribed displacements that need to be applied to a rigid body have to be specified as *Displacement* conditions at the rigid body reference point.
- In case a rigid body is used in dynamic calculations, the user has to ensure that the location of its reference point coincides with the centre-of-gravity of the rigid body, considering all mass components of the nodes involved. If the reference point does not coincide with the centre of gravity, out-of-balance forces might develop which may not be solved during the calculation. Note that if a rigid body is directly connected to the surrounding soil (i.e. it shares the same nodes in the boundary between the rigid body and the soil), the boundary nodes of the rigid body contain weight components of the surrounding soil elements, and it might be difficult to identify the exact location of the centre-of-gravity. In that case it is recommended to add interface elements around the rigid body to ensure that only weight components of the rigid body itself are considered.

5.9 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.

5.9.1 *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.

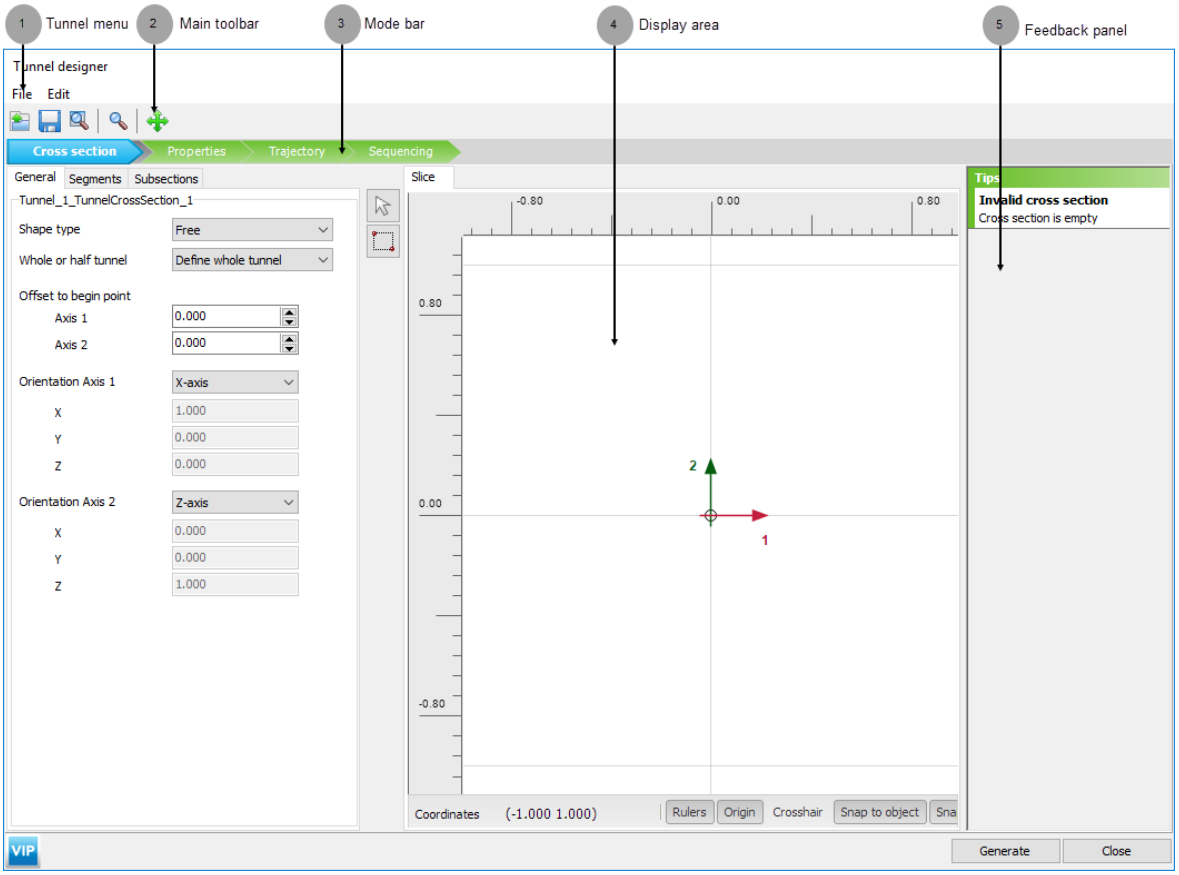


*Figure 113: The **General** tabsheet of the **Cross-section** mode*

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.

5.9.2 *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 (Tunnel splitting tool (on page 167)).
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 114 (on page 157)).

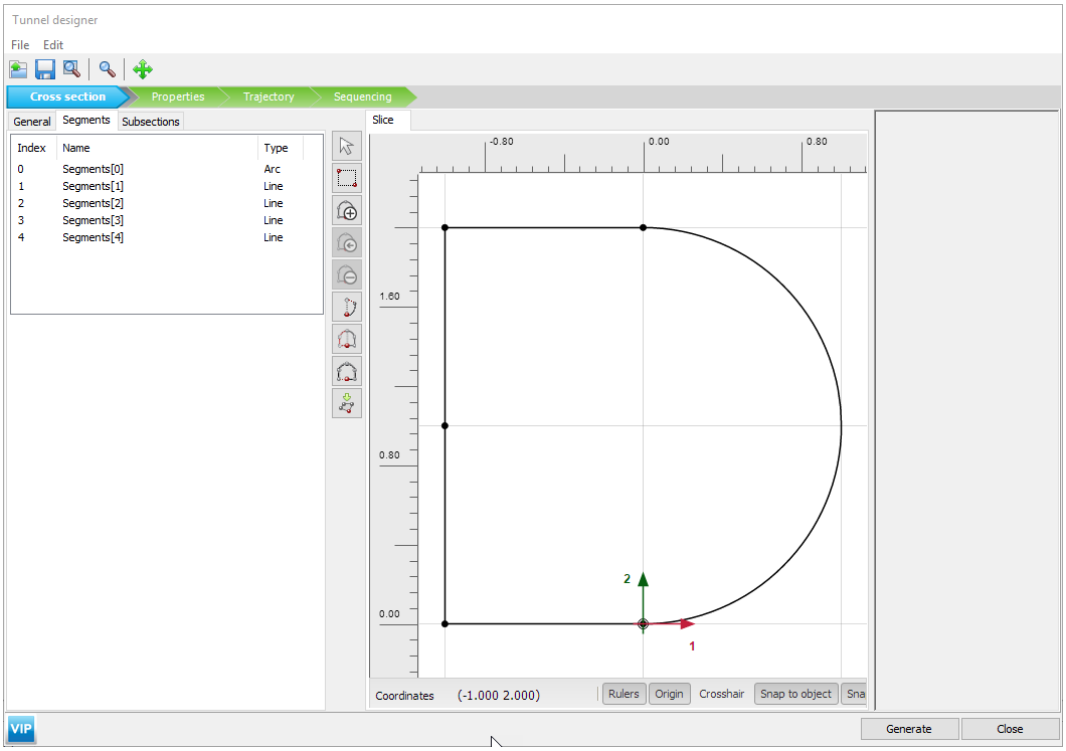


*Figure 114: The **Segments** tabsheet of the **Cross-section** mode*

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 114 (on page 157).

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 115 (on page 158)).
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 116 (on page 159)).
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.

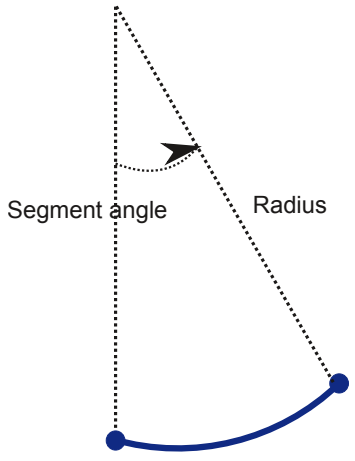


Figure 115: Properties of Arc segment

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.

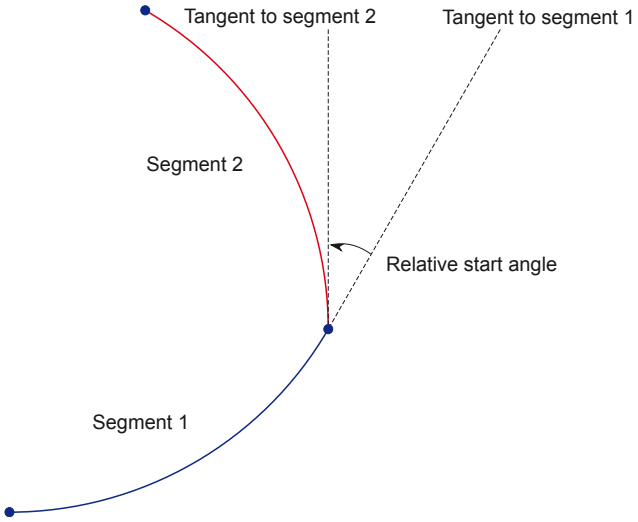


Figure 116: Description of Relative start angle

- 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.

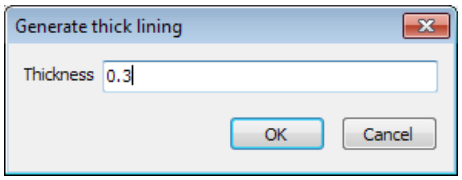


*Figure 117: The **Generate thick lining** window*

5.9.3 *Import tunnel geometry* GSE

Existing tunnel geometry can be imported in the **Segments** tabsheet in the **Cross section** mode of the **Tunnel designer** window. The program supports *. pntl, *.dxf, *.step, *.stp and *.brep files. *.pntl files can also be exported and saved (Importing geometry (on page 113)). For an imported geometry to be a valid cross section the following requirements must be met:

- It must contain a polyline, or circle, defining the outer boundary of the tunnel. This may consist of line segments of both line and arc types. The component of the outer boundary will be listed as segments.
- If the imported geometry contains more than one polycurve, the polycurve with the largest bounding box is imported as the cross-section and the others as subsections.
- Entities of incompatible types are omitted from the import.

5.9.4 *Definition of tunnel trajectory*

The trajectory (path) of the tunnel in the model can be created in the corresponding tabsheet in the *Tunnel designer*.

General

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

Layout of the trajectory in the model: The general properties of the trajectory 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 trajectory in the model.

Segments

The number of segments the *Trajectory* must contain is defined in the **Segments** tabsheet.

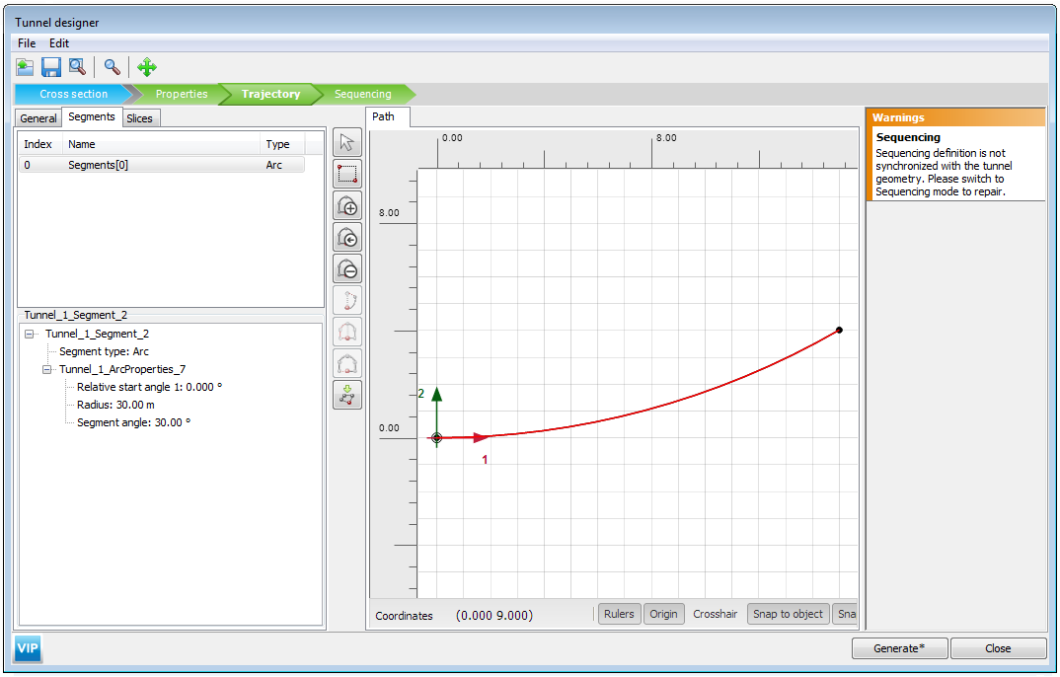


Figure 118: Definition of Segments in a trajectory

Definition of segments: The segments of a cross section can be defined in the *Segments* tab of the *Trajectory* tabsheet (Figure 119 (on page 162)).

- A new segment is added to the trajectory of a tunnel 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.
- A new segment is inserted before a selected segment to the trajectory of a tunnel as 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 trajectory is deleted as the *Delete segment* button is clicked. The indices of the segments are updated according to their position in the list.
- Import a polycurve to define the trajectory of the tunnel. The polycurve can only contain lines and arcs positioned on the XY-plane. The program supports *.dxf, *.step, *.stp and *.brep files (Importing geometry (on page 113)).

Slices

The number of slices for the tunnel trajectory may be defined in the *Slices* tab of the *Trajectory* tabsheet.

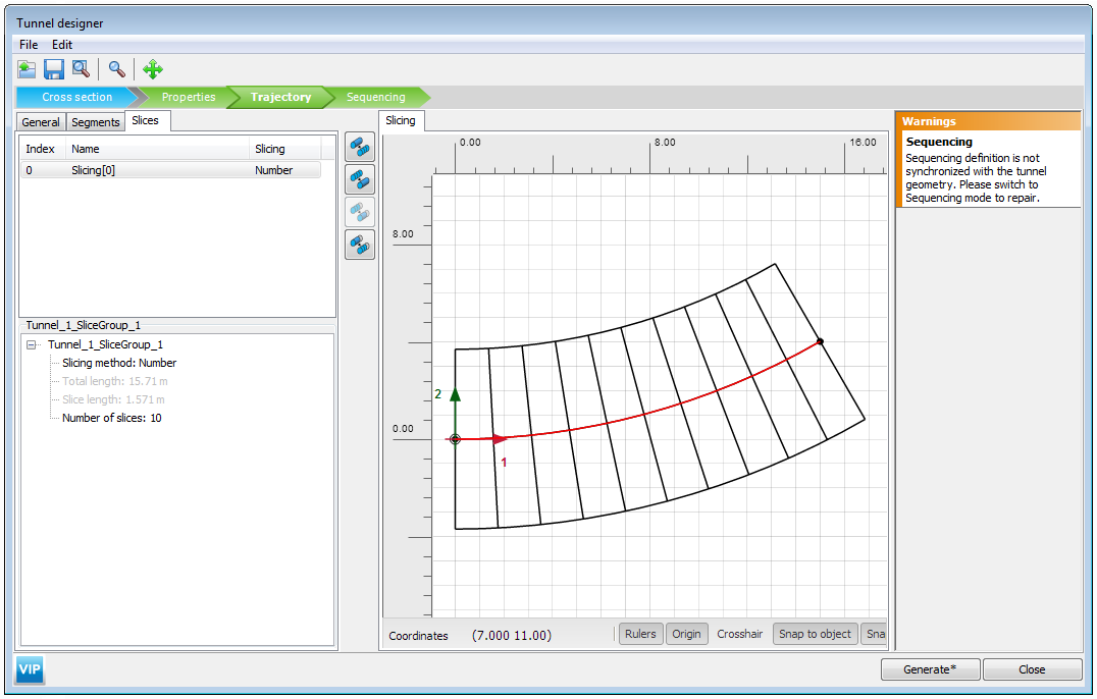


Figure 119: Definition of Segments in a trajectory

Definition of slices: A new slice is automatically created when a segment is added to the trajectory.

Slicing method: The available types for slicing methods are:

Length	Allows to create slices based on *Slice length*. This can be done by setting a particular value to the length. The slicing can start from the beginning or the end of the segment, which can be selected from the dropdown menu of *Starting from*.
Number	Allows to create slices based on *Number of slices* in a selected segment.

- To merge specific slice groups, multi-select consecutive slice groups of interest and then click on the button *Merge slice groups* on the left side of the drawing panel. The merged slicing group now has properties which have been copied from the first slice group.
- To merge all slice groups click on the button *Merge all slice groups*. The resulting slice group has properties which have been copied from the first slice group. This action is possible only if all the slicing group have similar properties with respect to slicing method and slice length.
- To reset all slicing for every slice group of the segment click on the button *Reset slicing* on the left side of the drawing panel. It removes all properties and resets them to the original state.
- To unmerge slice groups, click on the button *Unmerge slice groups* on the left side of the drawing panel.

5.9.5 *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 surface load	To assign a surface load to the selected segment (or subsection).
Create surface prescribed displacement	To assign a surface prescribed displacement to the selected segment (or subsection).
Create surface contraction	To assign a surface contraction to the selected segment (or subsection).
Create surface drain	To assign a surface drain 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	To assign boundary conditions for flow to the selected segment (or subsection).
Create rigid body	To assign a rigid body to the selected segment.

The features available in the Plane panel are:

Create beam	To assign a beam to the line or polycurve between two adjacent segments. This feature can be used to define girders (ribs, wailings etc.) along the perimeter or around the cross section of a tunnel at slice boundaries.

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 surface 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 (z-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 beams. 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 120 (on page 164)). 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.
- The configuration of a reinforcement may be set to *Rock bolt* or *Umbrella arch*.
- For more information about material properties of rock bolts refer Material set (on page 308).
- It is possible to delete a reinforcement, by clicking on the *Delete* button in the drawing area.

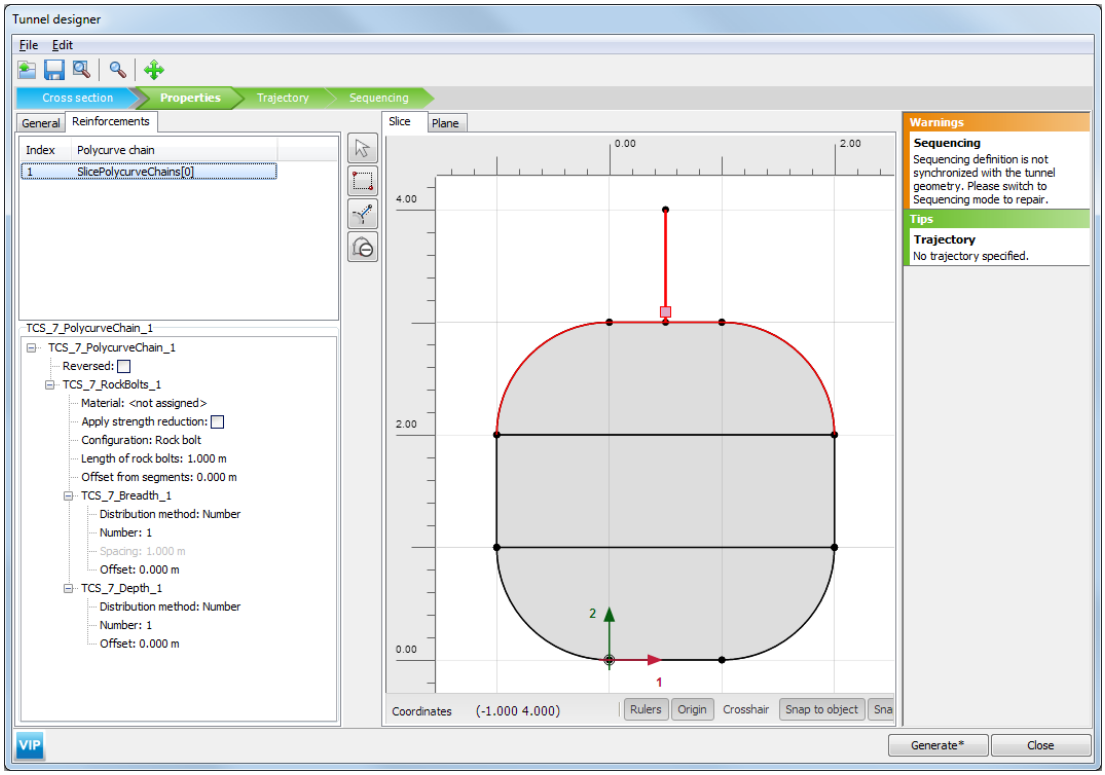


*Figure 120: Creation of **Reinforcements** in the Properties tabsheet*

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.

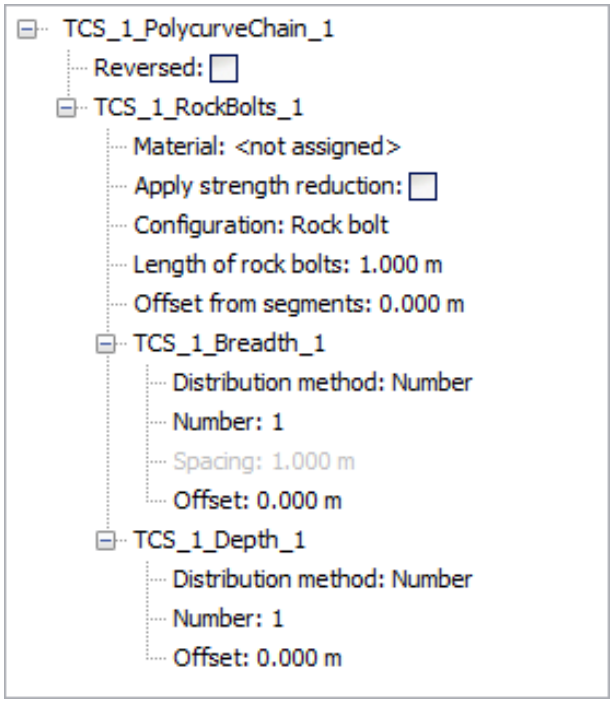


Figure 121: Rock bolt parameters

Input parameters required when the configuration is set to *Umbrella arch* (Figure 121 (on page 165)) are *Material*, *Apply strength reduction* and the following as described below :

Length of rock bolt	Length of rock bolts. Rock bolts can be created perpendicular to the tunnel contour or with an angle.

Offset from segments	Umbrella arch rock bolts can have an offset from the tunnel contour.
Angle	Angle of rock bolts in an umbrella arch with respect to the out of plane direction. The angle of rock bolts for an umbrella arch can have values greater than 0° and lesser than 180°.
Distribution	Rock bolts can be added in both breadth and depth directions which correspond to latitudinal and longitudinal directions respectively. 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 (Figure 123 (on page 167)).
Offset	Distance from the first tunnel point to the first rock bolt along the tunnel contour.

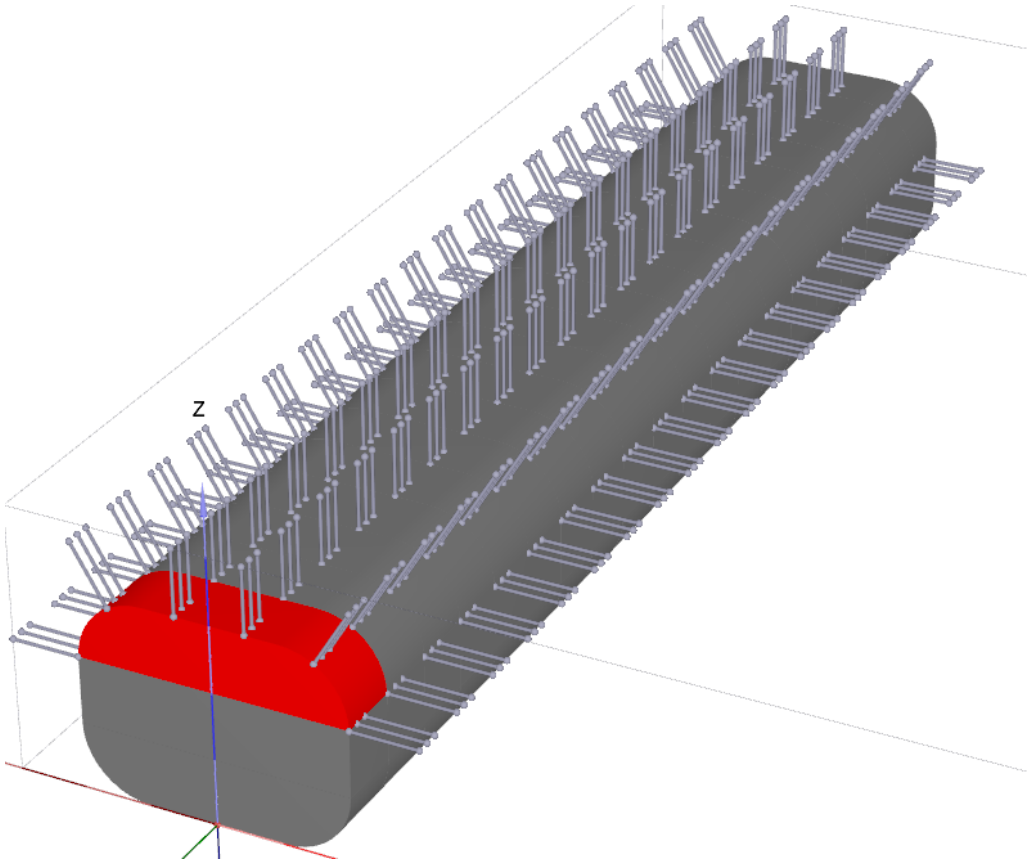


Figure 122: Rock bolts in breadth and depth of a tunnel

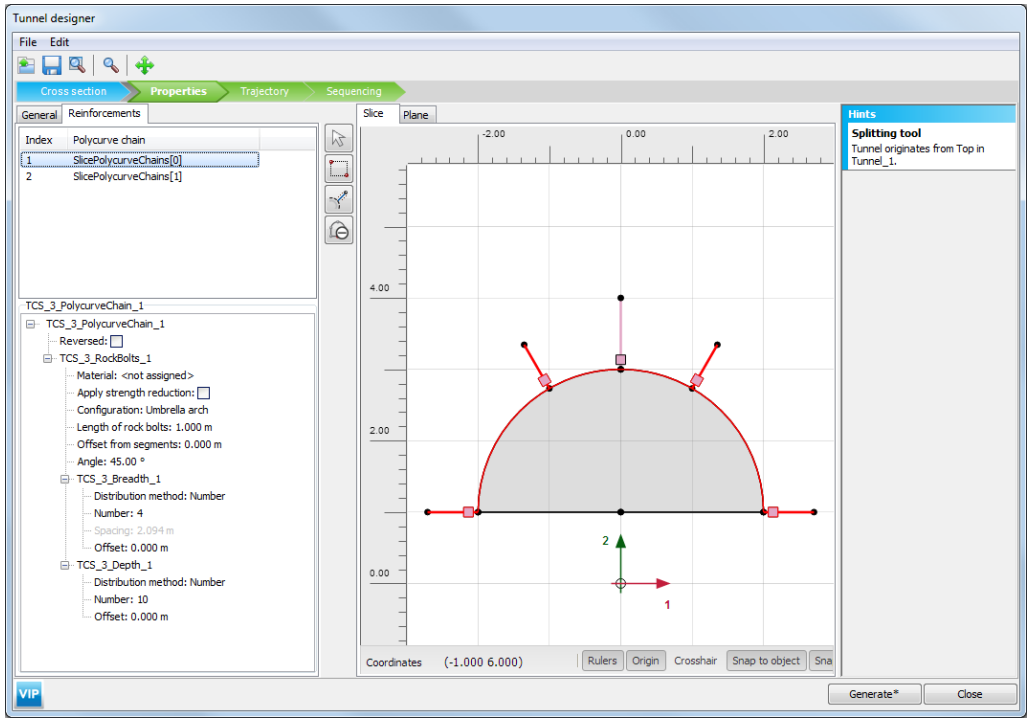


*Figure 123: **Umbrella arch** rock bolt properties*

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

5.9.6 *Tunnel splitting tool*

A tunnel can be split into multiple separate tunnels using the *Splitting tool*. For example in NATM tunnels, they can be split into *crown*, *base* and *invert* (base). Once they are split into separate tunnels, they can have their own sequencing definition to create the separate (sub)parts of the tunnel (Sequencing (on page 172)). This can be done by first creating a complete tunnel and then using the splitting action as described below.

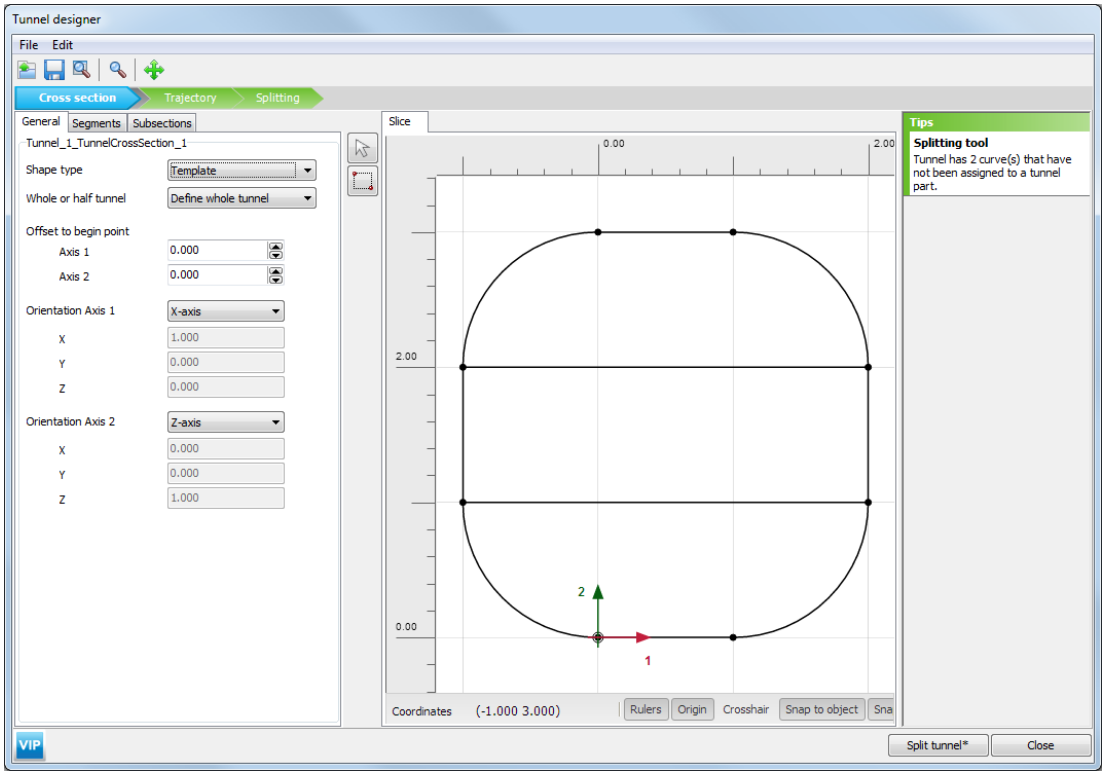


Figure 124: Tunnel splitting tool

The splitting tool can be accessed by first choosing the option *Template* for *Shape type* in the *General* tabsheet of the tunnel designer. When the *Template* shape type is selected, the *Properties* and *Sequencing* mode tabs are removed, leaving the *Geometry* modes of *Cross-section* and *Trajectory*. If a cross-section has already been defined, it will remain but any specified properties and sequencing definition will be reset.

A new mode is now introduced as *Splitting* mode.

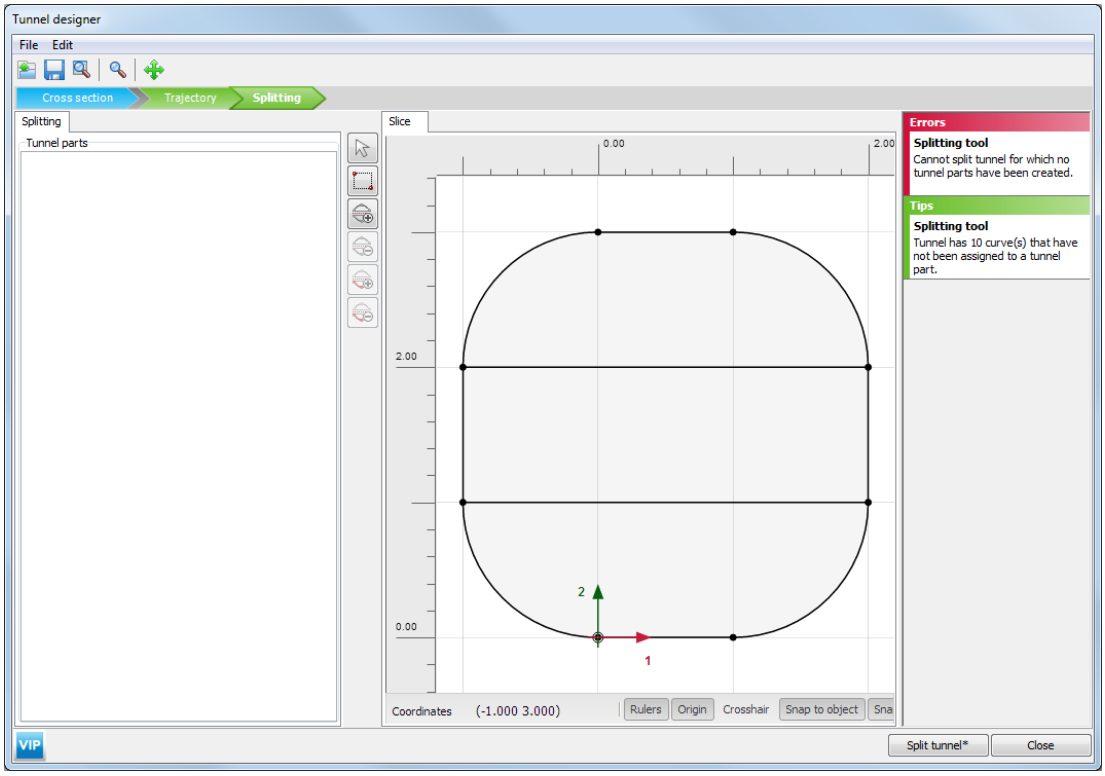


Figure 125: Tunnel splitting mode

In the *Splitting mode* different tunnel parts can be created which will represent separate parts of a split tunnel. Each of these tunnel parts will be shown in the corresponding tree in the left panel. Each tunnel part will contain one or more references to curves in the cross-section. Curves can be added or removed from tunnel parts. New tunnel parts can be created using the buttons on the right hand drawing panel.

- To add a new tunnel part, click on the *Add tunnel part (including selected parts)* button.
- To delete a tunnel part, click on the tunnel part in the left panel and then click on the *Delete tunnel part* button.
- Curves can be added to an already created tunnel part(s) by first selecting curve(s) in the left panel and then clicking on the *Add curve(s) to tunnel part* button.
- Curves can be deleted from an already created tunnel part(s) by first selecting curve(s) in the right panel and then clicking on the *Delete curve(s) from tunnel part* button. Some tunnel part curves can be part of multiple tunnel parts. Then, a user will have to select a tunnel part, then select a number of tunnel part curves, and then press delete curve from tunnel part button.

If a *Tunnel part* is selected in the left hand input panel, then all the cross-section curves in the tunnel part will be highlighted in the visualisation panel.

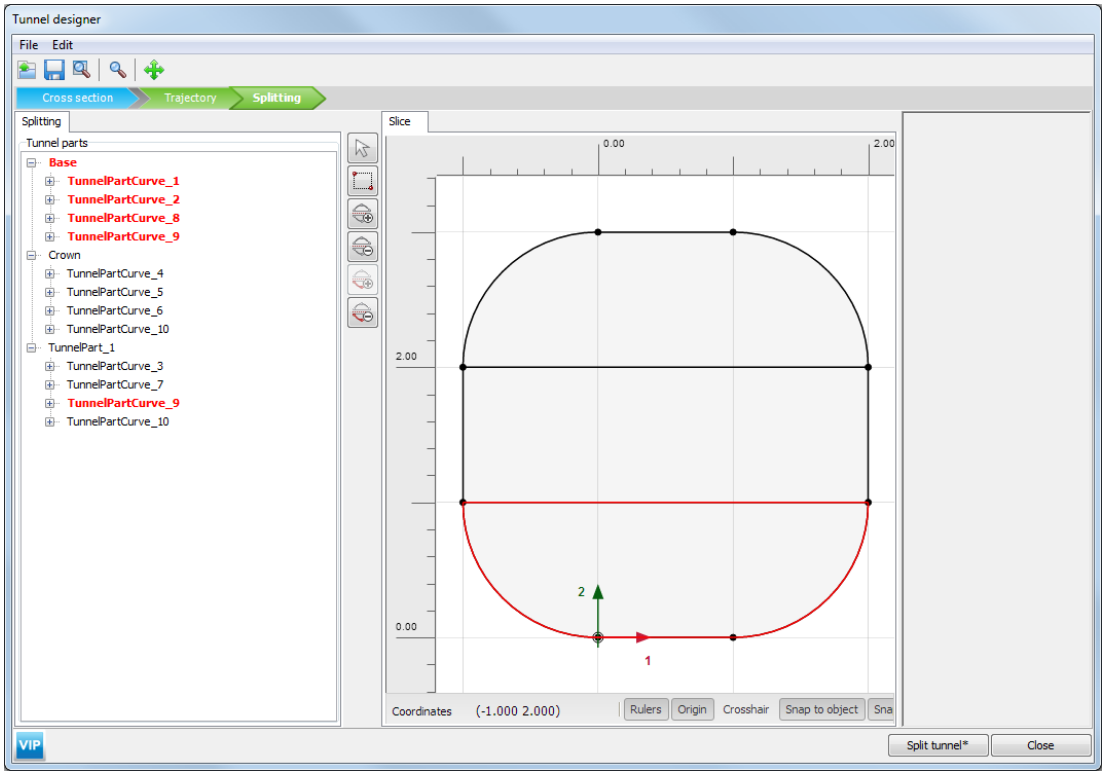


Figure 126: Selection of Tunnel part and cross-section curves

If cross-section curve(s) are selected in the right hand visualisation panel, then the *Add tunnel part (including selected curves)* is activated and can be then used to make a new tunnel part.

Note: When a cross-section curve which is not a part of a *Tunnel part* is selected, it is then possible to select an existing *Tunnel part* without losing the selection. Now, the *Add curve (s) to tunnel part* button is activated and can be used to add the curve to the selected tunnel part.

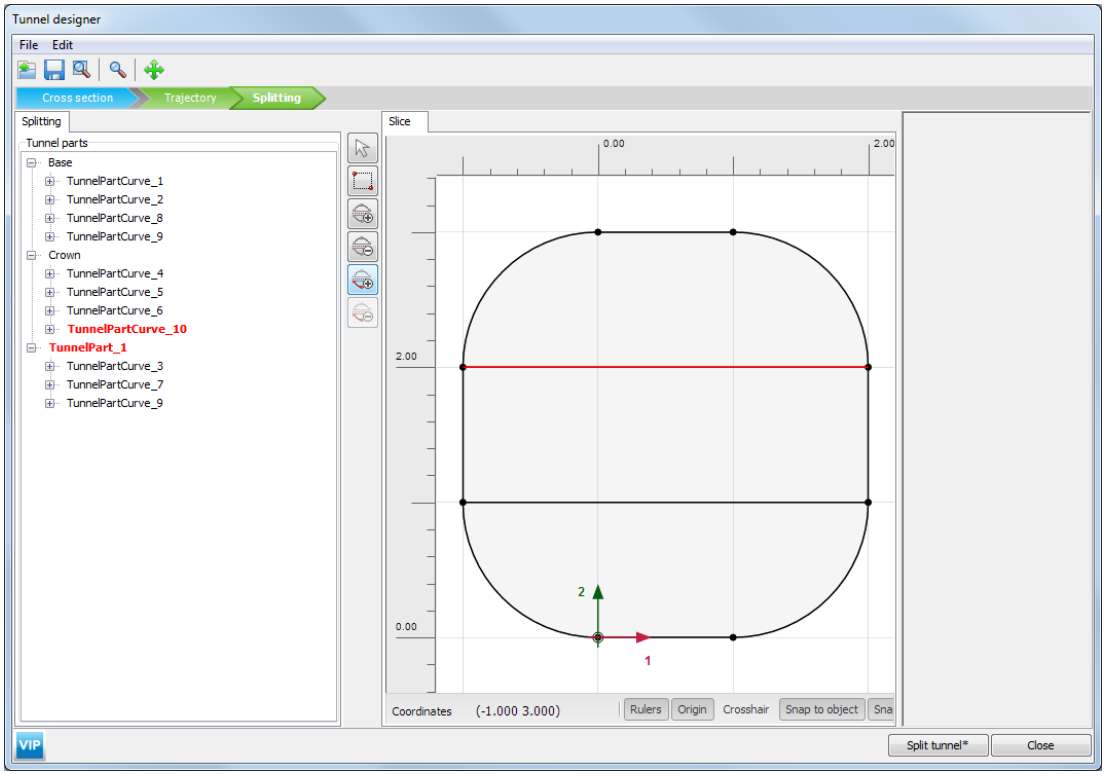


Figure 127: Adding a cross-section curve to an existing Tunnel part

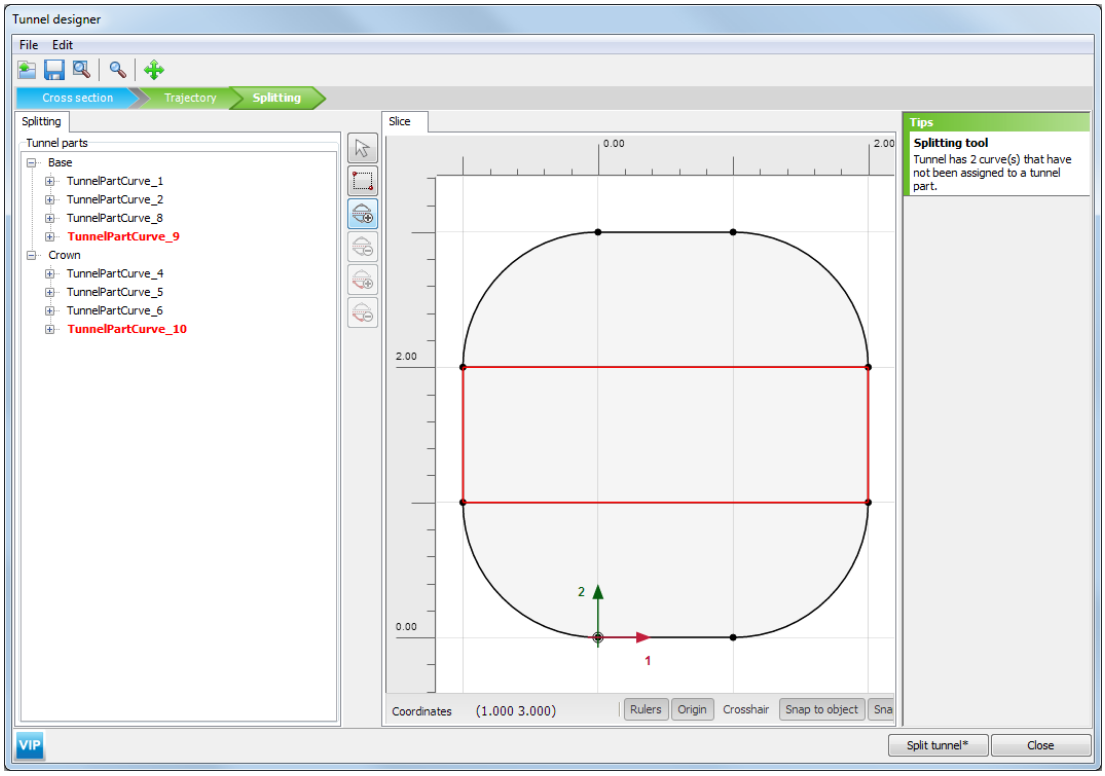


Figure 128: Adding a new Tunnel part from cross-section curves

Note:

- A tip is displayed in the right hand panel of the splitting tool when some of the cross-section curves have not been assigned to a tunnel part. However it is still possible to split tunnel.
- Note that only curves which are not yet in the *Tunnel part* will be added.
- It is recommended to define the geometry of trajectory before the splitting action is performed to have an efficient workflow.

The *Feedback panel* displays all the errors, hints and warnings. All errors must be solved before proceeding to split a tunnel. Hints and warnings do not block the process of splitting a tunnel.

A tunnel can be split by clicking the *Split tunnel* button on the right bottom of the tool. If changes are made to the template (parent) tunnel, these changes will be regenerated towards the split-off (child) tunnels, once the split tunnel command is executed again (e.g. by pressing the split tunnel button).

Each individual split tunnels still needs to get a proper definition for the properties (e.g. interfaces, plate elements, loads, etc), trajectory with slicing and tunnel sequencing, and then needs to be generated before it actually appears in the geometry of the model. Using the tunnel splitting tool, it is possible to create more complex tunnel excavations than the default tunnel designer, since the user can control the sequencing and progression in staged construction separately for each individual part.

5.9.7 *Sequencing*

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. These steps can be defined in the *Sequencing* tabsheet of the *Tunnel Designer*.

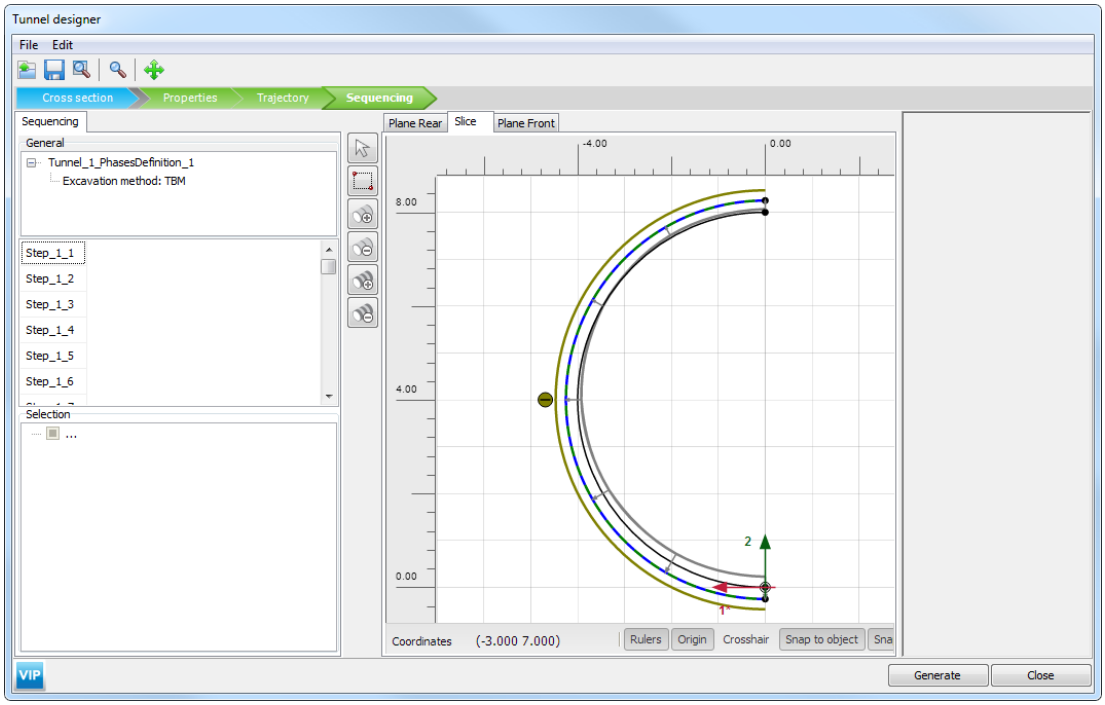


*Figure 129: **Sequencing** tabsheet*

The *Sequencing* tabsheet consists of a left panel, in which various steps can be defined, and a right panel showing a graphical section of the model with different tabs for the 'Rear' (*Plane Rear*) cross section, the 'Slice' (= volume section), and the 'Front' (*Plane Front*) cross section. The 'Slice' tab can be used to define operations on volumes and surfaces in the tunnel advancement direction, whereas the 'Rear' and 'Front' tabs can be used to define operations on surfaces and lines in cross sections perpendicular to the tunnel advancement direction. The buttons in the middle are used to select one or more objects in the right panel, or to create and delete steps in the left panel.

Excavation method

The *General* settings of the *Sequencing* tabsheet allows to choose the *Excavation method*. There are two available options: *TBM* and *NATM*.

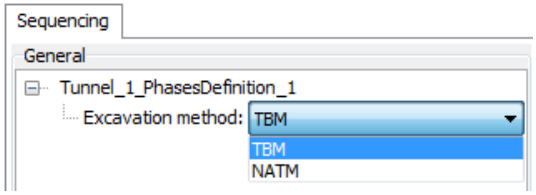


Figure 130: Excavation methods

TBM: The front of the tunnel moves forward during each advancement step.

NATM: The front of the tunnel can be fixed or delayed for a specified number of steps:

Slices in active scope	*Slices in active scope* works similar to a 'caterpillar '. It allows to fix the front for the defined number of steps while the back moves forward.
Delayed steps	Allows to delay the advancement from moving into the next slice for the defined number of steps.

Definition of steps

In principle, 'steps' are associated with the subsequent sections of the tunnel train, numbered from the first section in the front towards the back of the tunnel train.

 Click the *Add tunnel phase step* button to create a first step (Step_1_1). While this step is selected in the left panel, make the necessary settings in the right panel to model the changes that are required when the first section of the tunnel train is entering into the soil. This can involve, for example, a combination of

- some excavation (de-activation of soil volume in the slice)
- a TBM or tunnel lining to be installed (activation of plate in the slice)
- and optionally a face pressure applied at the tunnel heading (surface load in the front plane).

 If the tunnel train consists of more construction steps, click the *Add tunnel phase step* button again to introduce the next step (Step_1_2). Initially, the settings of this step are copied from the previous step. This is OK if the situation as defined in the first step persists for another section length (or more). However, if the next step (section) involves changes with respect to the previous step (section), the required changes shall be made in the right panel.

The process of defining steps shall be continued until the full tunnel train has been defined.

 A step can be deleted by clicking on the *Remove tunnel phase step* button on the left panel of the drawing canvas.

 Click the *Add tunnel phase routine* button to add a new routine (Step_2_1).

Initially, the settings of this routine are copied from the previous routine. Routines can deal with differences in the configuration of a same step.

 A routine can be deleted by clicking on the *Remove tunnel phase routine* button on the left panel of the drawing canvas.

NATM examples

In order to model a sequential excavation tunnel in which the 'tunnel train' consists of six sections of equal length:

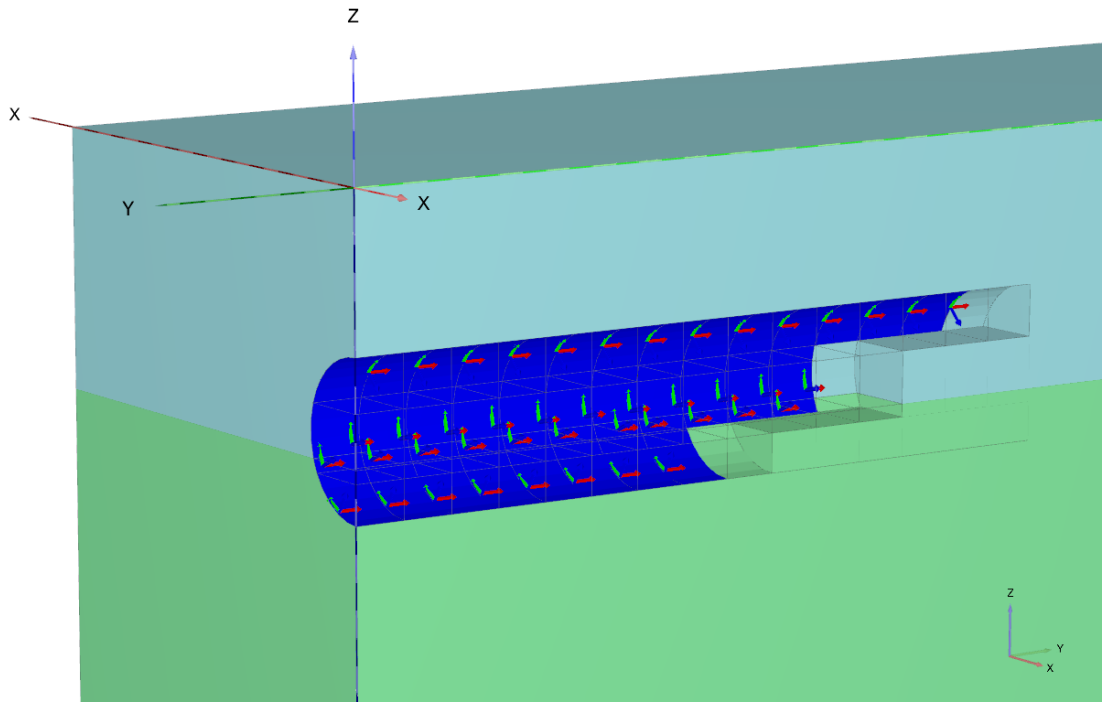


Figure 131: Sequential excavation tunnel

- Top heading excavation (unsupported excavation) (1 section)
- Top heading (secured with shotcrete lining) (2 sections)
- Bench excavation (unsupported) (1 section)
- Bench (secured with shotcrete lining) (2 sections)
- Invert excavation (unsupported) (1 section)
- Invert (secured with shotcrete lining) (1 section, but in fact all sections behind this one will be the same)

Hence, the 'tunnel train' consists of 8 sections. In order to advance the tunnel train over one section in a new calculation phase, define a total of 8 steps:

- Step_1_1: Excavate top heading
- Step_1_2: Install top lining
- Step_1_3: (no change)
- Step_1_4: Excavate bench
- Step_1_5: Install middle lining
- Step_1_6: (no change)
- Step_1_7: Excavate invert
- Step_1_8: Install bottom lining

In this case *Slices in active scope* can be set to 8, in this way the front of the tunnel is fixed for 8 steps and then it moves to the next slice. If different steps are required for the top heading, the bench and the invert, three different tunnels can be defined assigning different steps and excavation properties.

With these settings, it is easy to define the calculation phases for the construction of the tunnel in multiple stages (Advancing a tunnel (on page 389)).

Note: It is highly recommended to start tunnel advancements in a phase subsequent to the *Initial Phase*, as otherwise the staged construction process may be disrupted. Particularly, during tunnel sequencing, the *Initial Phase* is referred to as the initial state, and it may not be possible to revert modifications after the initial state has been modified.

5.10 Hydraulic conditions

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 132 (on page 176)), provide a faster definition of conditions.



Figure 132: 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 382)).

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 390)). Note that specific hydraulic conditions have priority over global model conditions.

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 (on page 185)).

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 98)). 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 133 (on page 177)).

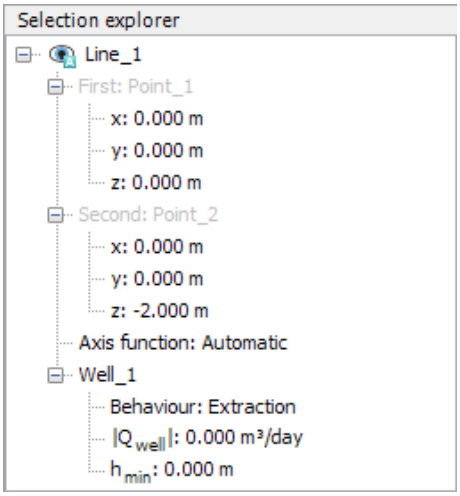


Figure 133: Well definition in the Selection explorer

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.
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 **Flow parameters** tabsheet of the material dataset.

5.10.2 *Line drain*

 In order to define a line drain, the *Create line 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 98)). Line drains are used to prescribe lines inside the geometry model where (excess) pore pressures are reduced.

Together with the creation of a line 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 line drain can be modified in the *Object explorers* (Figure 134 (on page 178)).

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.

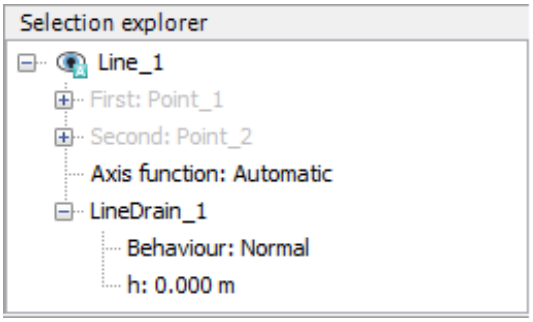


Figure 134: Normal line drain definition in the Selection explorer

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

5.10.3 *Surface drain*



In order to define a surface drain, the *Create surface 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 surface (Create surface (on page 105)). Surface drains are used to prescribe surfaces inside the geometry model where (excess) pore pressures are reduced.

Together with 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. Pore pressures lower than the equivalent to the given head are not affected by the drain. Drains can be activated or de-activated in calculation phases. The properties of a line drain can be modified in the *Object explorers* (Figure 135 (on page 179)).

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. 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.

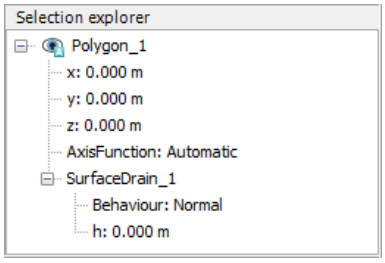


Figure 135: Normal surface drain definition in the Selection explorer

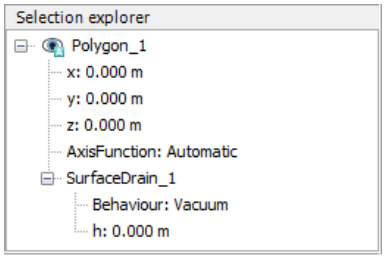


Figure 136: Vacuum surface drain definition in the Selection explorer

5.10.4 *Surface flow boundary conditions*


In order to define a surface flow boundary condition, the *Create surface groundwater flow boundary condition* option should be selected from the menu appearing as the *Create hydraulic conditions* button is clicked. The creation of a surface flow boundary condition in the geometry model is similar to the creation of a surface (Create surface (on page 105)).

The behaviour of the surface flow boundary conditions can be specified by selecting the corresponding option in the *Selection explorer* (Figure 137 (on page 180)).

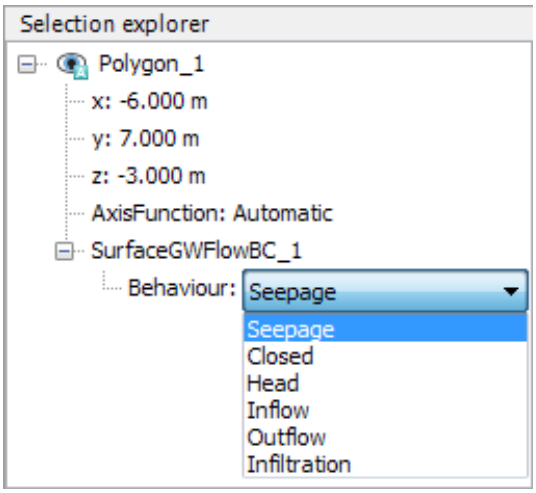


Figure 137: Options for the behaviour of Surface groundwater flow boundary conditions

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 *Surface flow boundary conditions* feature always prevail over the *Model conditions*. As an example, if a surface flow boundary conditions is placed on the top surface and precipitation is specified in *Model conditions* only the behaviour specified for the surface 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 3D 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 surface, such as a dam or a dike).

If a surface 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 surface. The part of the geometry surface above the transition line is treated as a boundary above the water level, whereas the part of the geometry surface below the transition line is treated as a boundary below the water level. Hence, different conditions can apply to such a geometry boundary surface. This is possible because, in general, a geometrysurface 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 surfaces.

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 surface 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 (on page 380)), 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 138 (on page 182) 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.
Linear	A linear variation of the head along the boundary is specified. The parameters required to specify a linear variation are: • h_{ref}: The reference head value at the specified reference level. • $h_{inc,z}$: Increment value specifying the change of the head with depth. • z_{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 $h_{inc,z}$ value (h = h_{ref} + z_{ref} - $zh_{inc,z}$).

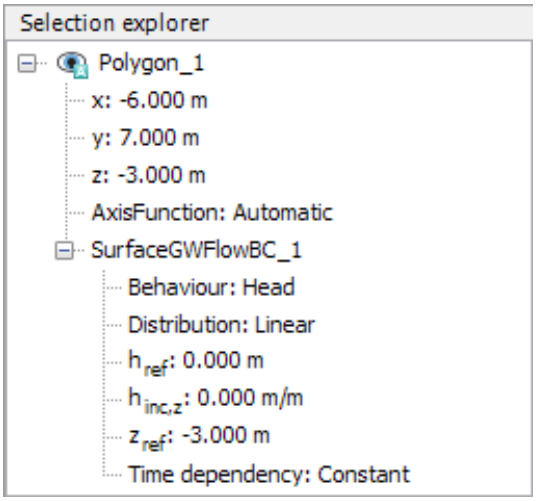


Figure 138: Head definition in the Selection explorer

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

Constant	The head in the boundary does not change with time.
Time dependent	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 (on page 185).

Inflow

Inflow to the model through a boundary can be specified by selecting the corresponding *Behaviour* option in the *Selection explorer*. Figure 139 (on page 183) 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.

Linear	A linear variation of the head along the boundary is specified. The parameters required to specify a linear variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level. • $q_{inc,z}$: Increment value specifying the rate of variation of discharge along the boundary. • z_{ref}: The reference value for which the reference head 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 $\lvert q_{inc,z} \rvert$ value.

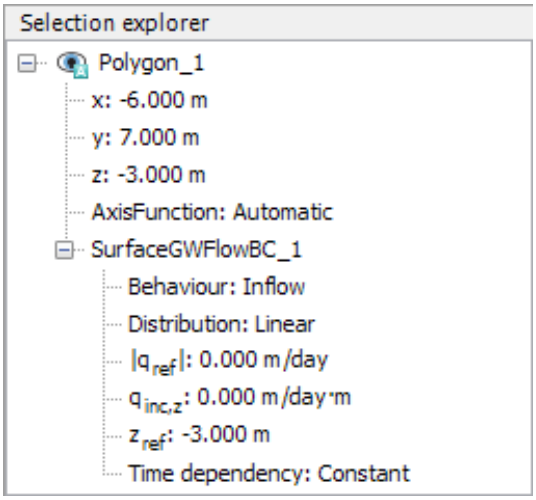


Figure 139: Inflow definition in the Selection explorer

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

Constant	The discharge does not change with time.
Time dependent	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 (on page 185).

Outflow

Outflow from the model through a boundary can be specified by selecting the corresponding option for *Behaviour* in the *Selection explorer* Figure 140 (on page 184).

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.

Linear	A linear variation of the discharge along the boundary is specified. The parameters required to specify a linear variation are: • $\lvert q_{ref} \rvert$: The reference discharge value at the specified reference level. • $q_{inc,z}$: Increment value specifying the rate of variation of discharge along the boundary. • z_{ref}: The reference value for which the reference head 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 $\lvert q_{inc,z} \rvert$ value.

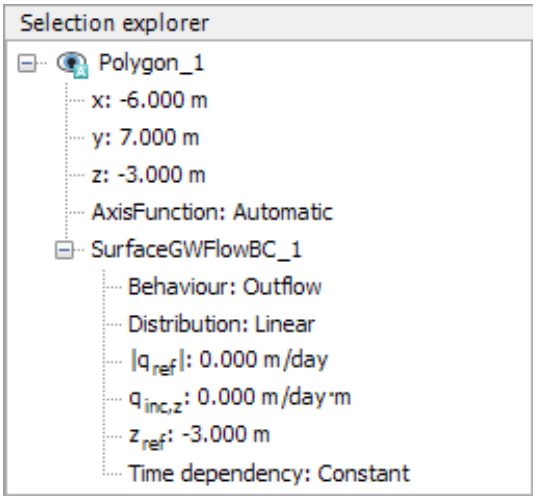


Figure 140: Outflow definition in the Selection explorer

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

Constant	The discharge does not change with time.
Time dependent	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 (on page 185).

Infiltration

Apart from the automatic generation of infiltration boundary conditions from precipitation (Flow boundary conditions (on page 380)), 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	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 (on page 185).

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

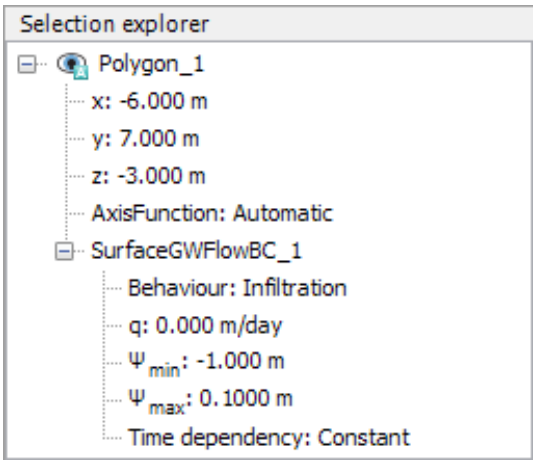


Figure 141: Infiltration definition in the Selection explorer

5.11 Flow functions

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 142 (on page 185)).

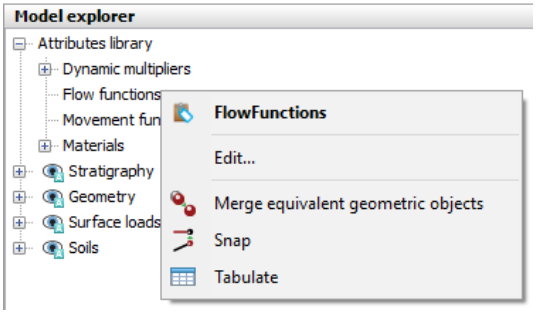


Figure 142: Flow functions subtree in Model explorer

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 143 (on page 186).

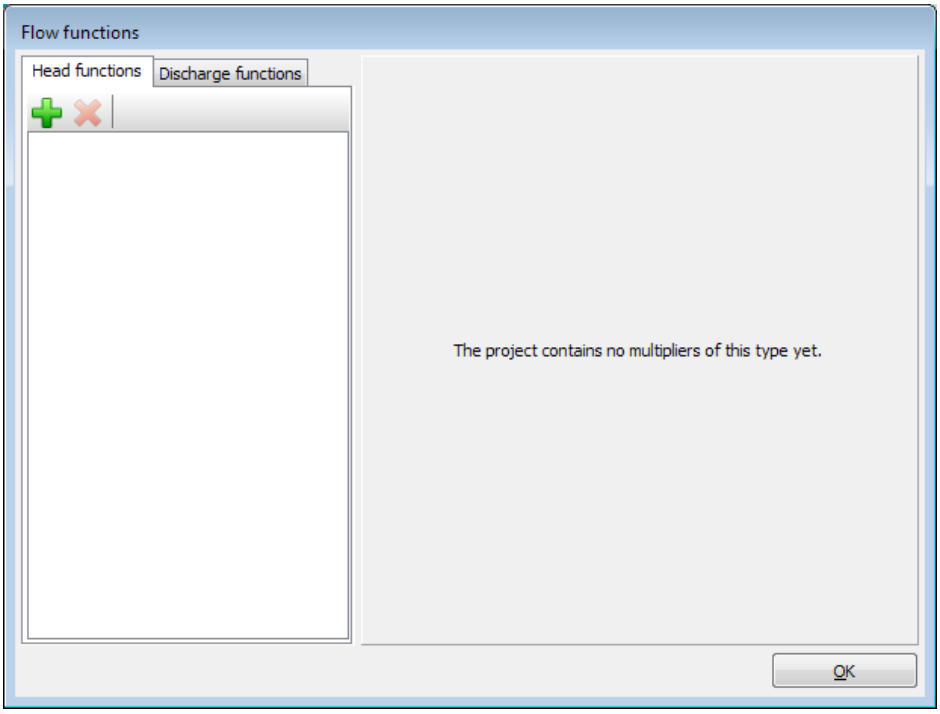


Figure 143: ***Flow functions*** *window*

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:

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

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:

$$y'_0 = y_0 + A\sin(\omega_0) \qquad Eq.\ [6]$$

Figure 144 (on page 187) shows the **Flow functions** window where a Harmonic signal is defined and plotted.

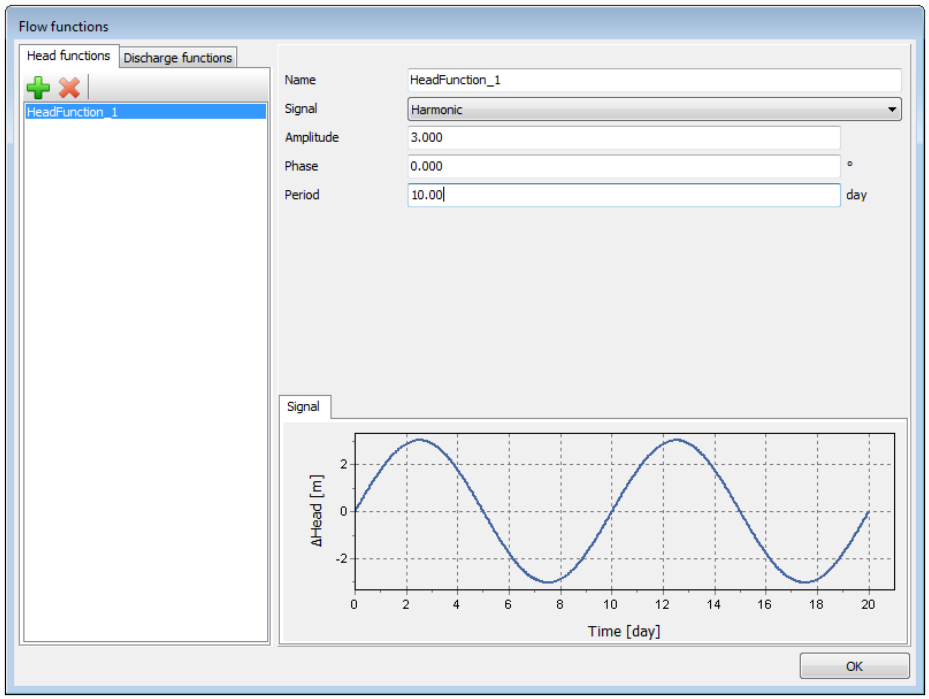


*Figure 144: Definition and display of a Harmonic signal in the **Flow functions** window*

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 3D 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 145 (on page 188)). 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*.

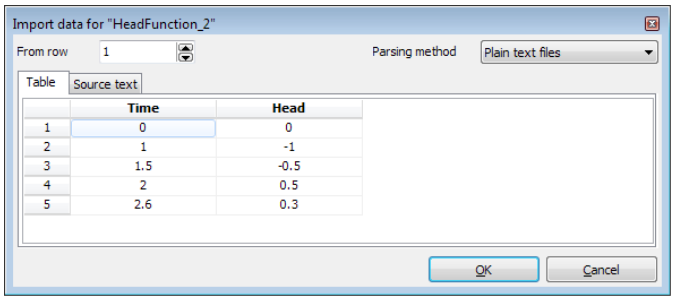


Figure 145: ***Import data*** *window for flow functions*

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 3D 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:

ΔDischarge	This parameter, specified in unit of volume per unit of time, 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 3D 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 Surface flow boundary conditions (on page 180), two new options are added for boundary conditions in the **Phase definition** mode (Figure 146 (on page 189)). 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 3D 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.

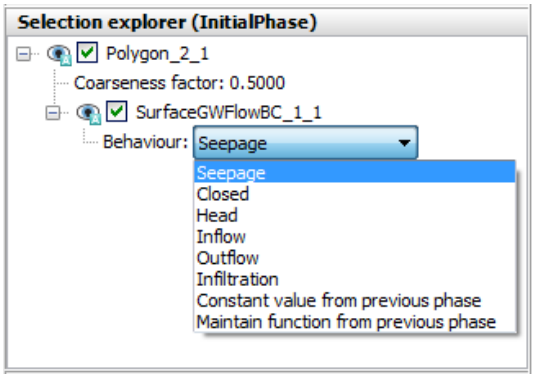


*Figure 146: Options for boundary condition behaviour in **Phase definition** modes*

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 3D 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.

6 Material properties and material database

In PLAXIS 3D, 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, beams, embedded beams, plates, geogrids 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 activated 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 147 (on page 191).

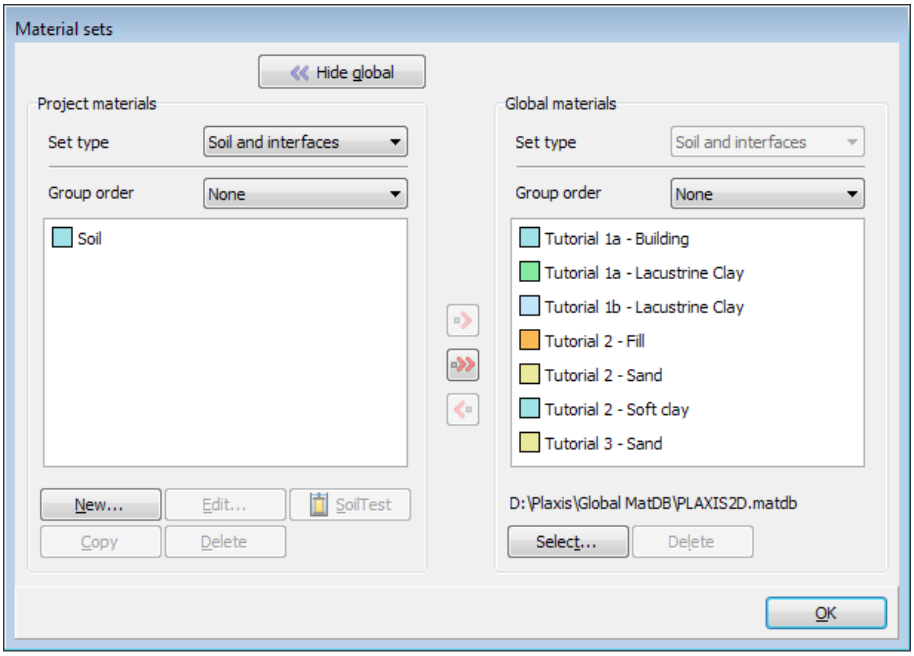


*Figure 147: **Material sets** window showing the project and the global database*

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*, *Plates*, *Geogrids*, *Beams*, *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 'SoilMat.matdb'. This file is compatible with other PLAXIS 3D database files for soil and interfaces and is stored in the installation folder of PLAXIS 3D. Material data sets for structural elements will be contained in separate files. Similarly, the global data bases for are contained in the files 'PlateMat3D.matdb', 'GeogridMat.matdb', 'BeamMat.matdb', 'EmbeddedPileMat.matdb' and 'AnchorMat3D.matdb' respectively.

Note that besides the global material files (*.matdb), it is possible to select project material files (*.plxmat) and legacy project material files (*.mat) as global database.

In addition, databases with data sets of standard sheet-pile wall profiles are available from the PLAXIS Knowledge Base (publication *Material parameter datasets for sheetpiles and beams*).

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 (on page 272)).
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 191)) 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 148 (on page 193). The properties in the data sets are divided into five tabsheets: **General**, **Parameters**, **Groundwater**, **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 3D 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.

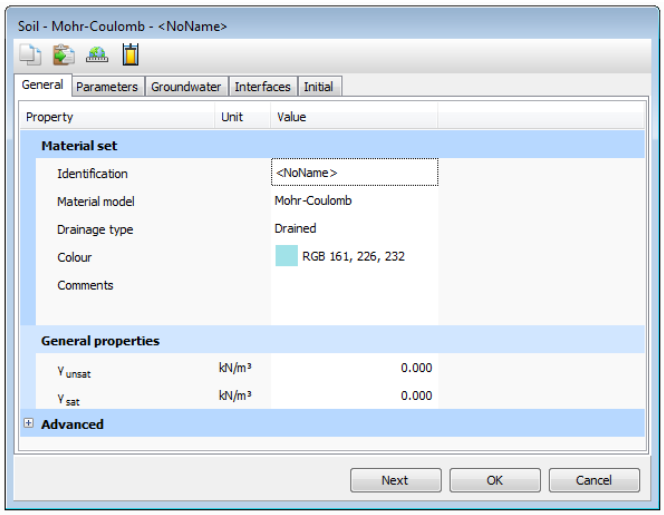


*Figure 148: **General** tabsheet of the **Soil** window*

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 3D 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}).

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.

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.

Joined 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.

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.

GSE **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.

GSE **UDCAM-S (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 (on page 234)) to obtain the degraded parameter set based on the type of analysis.

GSE **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.

UBC3D-PLM (UBC3D-PLM): This is an advanced model for the simulation of liquefaction behaviour in dynamic 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.

Concrete model (Concrete): 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.

User-defined Soil Models (UDSM): With this option it is possible to use other constitutive models than the standard PLAXIS 3D 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 3D 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 3D 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 *Dynamic* analysis, is defined by the *Drainage type* parameter. PLAXIS 3D 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 269) 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 *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 *Permeability* of the material.

Saturated and unsaturated weight (γ_{sat} and γ_{unsat})

The saturated and the unsaturated 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 3D 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 329)).

Advanced general properties

Additional properties for advanced modelling features can be defined in the *Advanced* subtree in the **General** tabsheet (Figure 148 (on page 193)).

Void ratio (e_{init}, e_{min}, e_{max}): 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). In addition to e_{init}, a minimum value e_{min} and a maximum value e_{max} can be entered. These values are related to the maximum and minimum density that can be reached in the soil. When the Hardening Soil model or Hardening Soil model with small-strain stiffnessis used with a certain (positive) value of dilatancy, the mobilised dilatancy is set to zero as soon as the maximum void ratio is reached (this is termed dilatancy cut-off). For other models this option is not available. To avoid the dilatancy cut-off in the Hardening Soil modelor Hardening Soil model with small-strain stiffness, the option may be deselected in the *Advanced general properties* subtree.

DYN **Rayleigh α and β:** Material damping in dynamic calculations is caused by the viscous properties of soil, friction and the development of irreversible strains. All plasticity models in PLAXIS 3D 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. There is one model in PLAXIS 3D that includes viscous behaviour, which is the Soft Soil Creep model. Using the model in dynamic calculations may lead to viscous damping, but also the Soft Soil Creep model hardly shows any creep strain in load / reload cycles. There is also one model in PLAXIS 3Dthat includes hysteretic behaviour in loading / reload cycles, which is the Hardening Soil model with small-strain stiffness (Material Models Manual - *Chapter 7*). 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 dynamic 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 **Parameters** tabsheet of the **Soil** window shown in Figure 149 (on page 198).

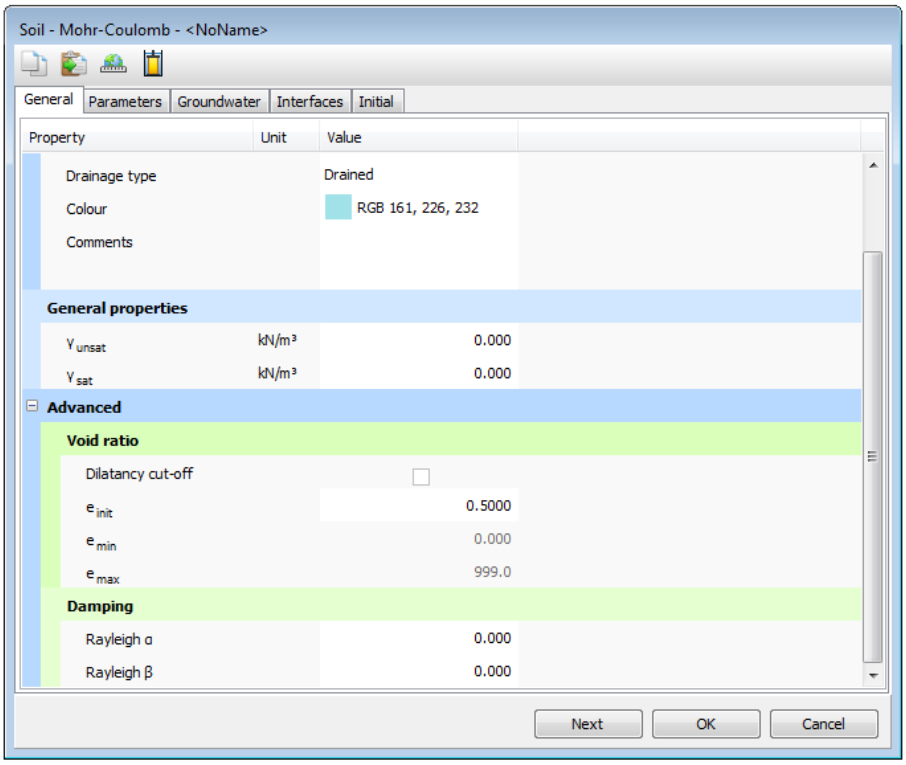


*Figure 149: Damping parameters in the **General** tabsheet*

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 3D, 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).

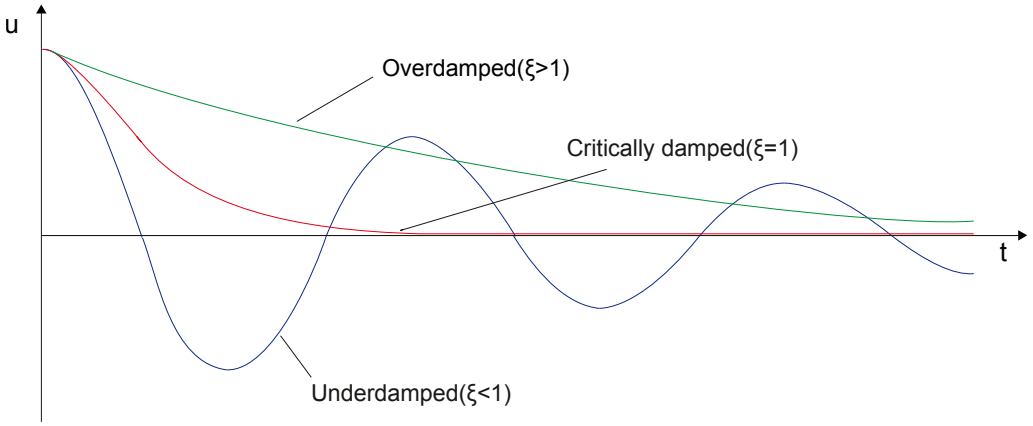


Figure 150: Role of damping ratio ξ in free vibration of a single degree-of-freedom system

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 *α = 1.2698* and *β = 0.002681*. From Figure 151 (on page 199) 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.

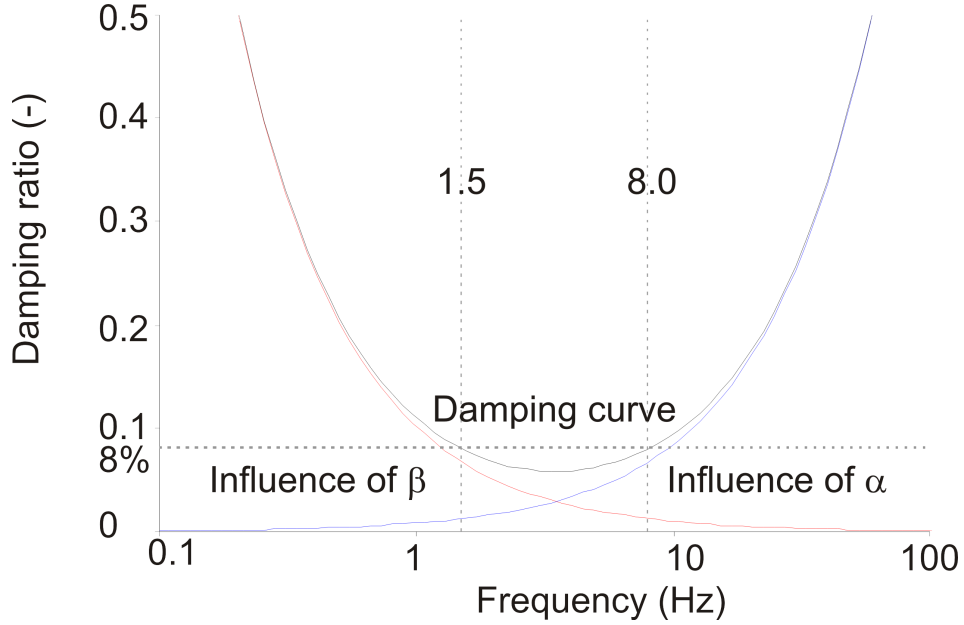


Figure 151: Rayleigh damping parameter influence

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 511) and Hashash & Park (2002) (on page 511) 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 510) 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.

The damping parameters (α and β) can be automatically calculated by the program when the target damping ratio (ξ) and the target frequencies (f) are specified in the panel displayed in the **General** tabsheet when one of the cells corresponding to the damping parameters is clicked. Figure 152 (on page 201) shows the damping ratio as a function of the frequency.

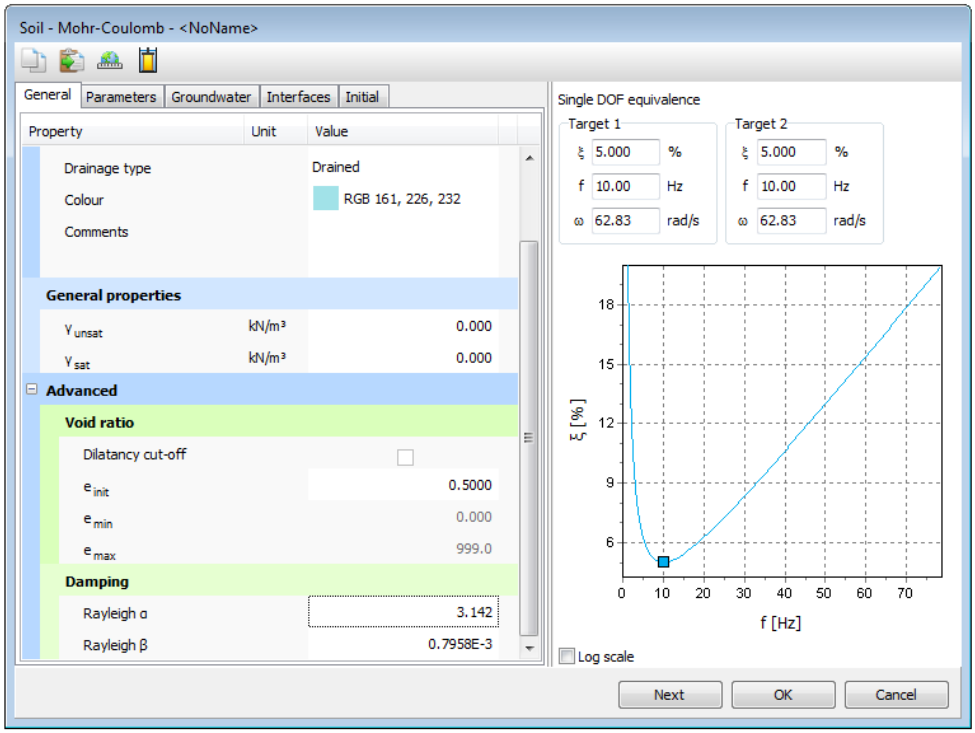


Figure 152: Input of ξ and f

6.1.2 *Parameters tabsheet*

The **Parameters** 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 **Parameters** tabsheet for the Linear Elastic model(drained behaviour) is shown in Figure 153 (on page 202).

The model involves two elastic stiffness parameters, namely the effective Young's modulus E' and the effective Poisson's ratio ν'.

E'	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	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 (γ_u) are used.

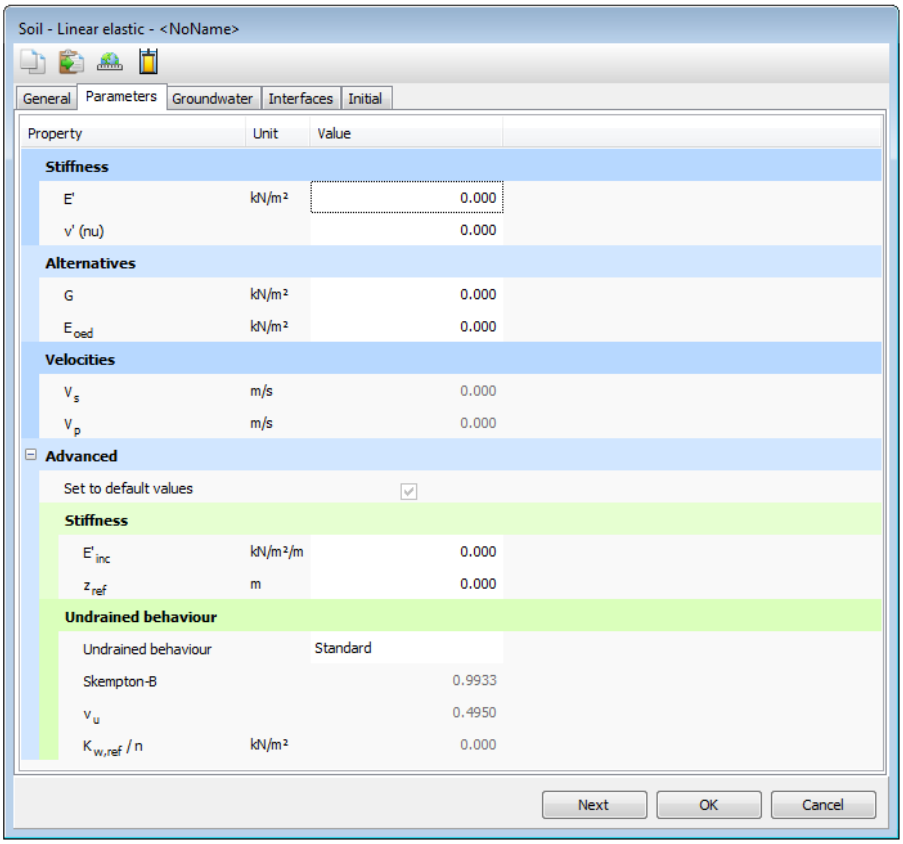


Figure 153: ***Parameters*** *tabsheet for the Linear Elastic model(drained behaviour)*

Note that the alternatives are influenced by the input values of E' and ν'. Entering a particular value for one of the alternatives G or E_{oed} results in a change of the Young's modulus E'.

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 z_{ref} becomes relevant. For any z-coordinate above z_{ref}, the stiffness is equal to E'_{ref}. For any z-coordinate below z_{ref} the stiffness is given by:

$$E'(z) = E' + (z_{ref} - z)E'_{inc}; \quad z < z_{ref} \qquad Eq.\ [12]$$

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 volume 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.

DYN Beside the parameters related to strength and stiffness of the soil, the velocities of wave propagation in soil can be defined in the **Parameters** 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 $\rho = \gamma / g$.

Note:

- Note that 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.
- 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.

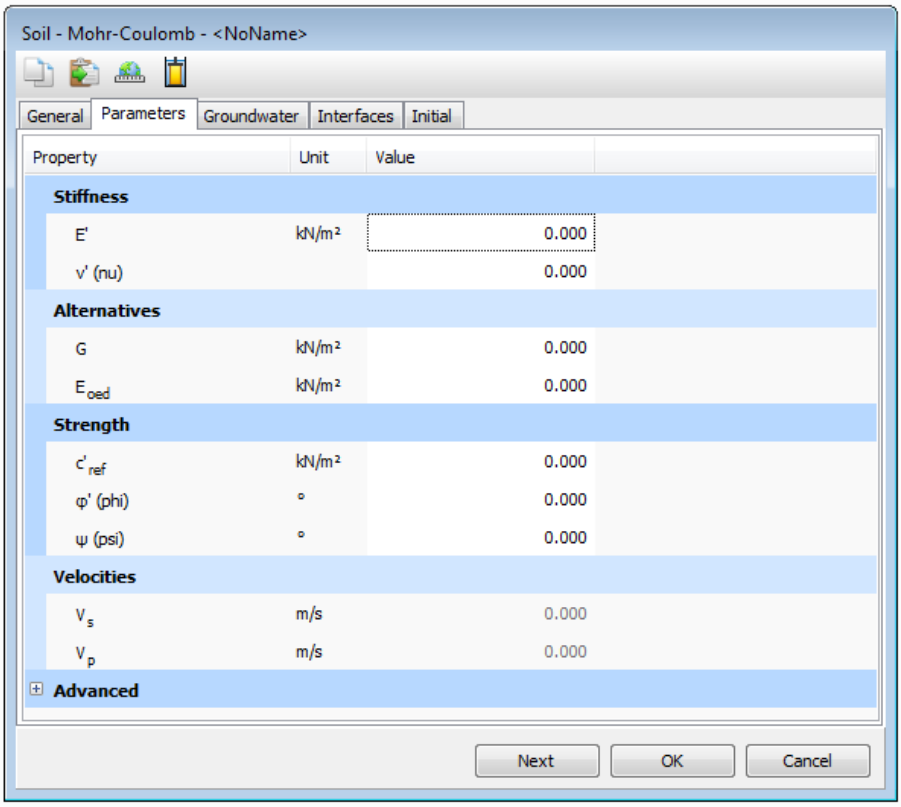


Figure 154: ***Parameters*** *tabsheet for the Mohr-Coulomb model (drained behaviour)*

The stiffness parameters of the Mohr-Coulomb model (drained behaviour) are:

E'	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	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 that the alternatives are influenced by the input values of E' and ν'. Entering a particular value for one of the alternatives G or E_{oed} results in a change of the Young's modulus E'.

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 z_{ref} becomes relevant. For any z-coordinate above z_{ref} the stiffness is equal to E'_{ref}. For any z-coordinate below z_{ref} the stiffness is given by:

$$E'(z) = E' + (z_{ref} - z)E'_{inc}; \quad z < z_{ref} \qquad \textit{Eq. [13]}$$

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 z_{ref} becomes relevant. For any z-coordinate above z_{ref} the cohesion is equal to c'_{ref}. For any z-coordinate below z_{ref} the cohesion is given by:

$$c'(z) = c'_{ref} + (z_{ref} - z)c'_{inc}; \quad z < z_{ref} \qquad \textit{Eq. [14]}$$

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 3D 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.

Note:

- Note that 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.
- Velocities of wave propagation in soil can be defined only for models with stress independent stiffness.

Hardening Soil model (HS)

The **Parameters** tabsheet for the Hardening Soil modelis shown in Figure 155 (on page 206).

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').

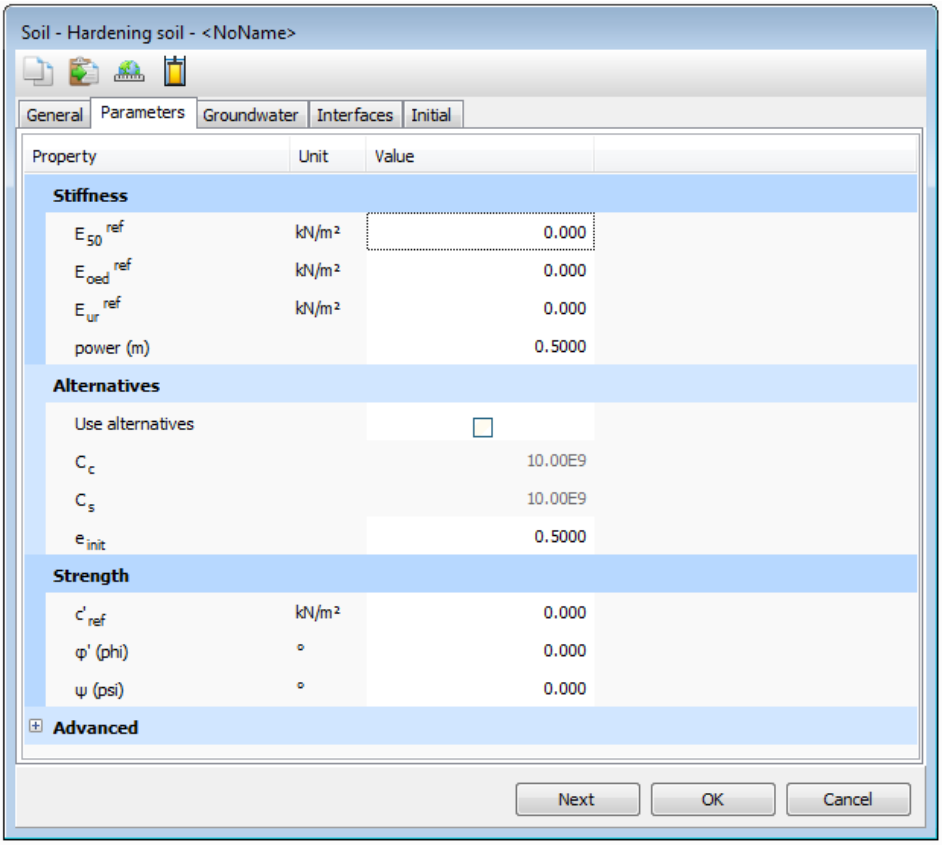


*Figure 155: **Parameters** tabsheet for the Hardening Soil model(drained behaviour)*

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]
m	Power for stress-level dependency of stiffness	[-]

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, advanced parameters can be defined for stiffness (it is advised to use the default setting):

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]
K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - sin(\varphi)$)	[-]

The strength parameters of the present hardening model coincide with those of the non-hardening Mohr-Coulomb model:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]

In addition, advanced parameters can be defined for strength:

c'_{inc}	As in Mohr-Coulomb model (default c_{inc} = 0)	[kN/m^3]
z_{ref}	Reference level	[m]
R_f	Failure ratio q_f / q_a (default R_f = 0.9)	[-]
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 3D 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 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 **Parameters** tabsheet for the Hardening Soil model with small-strain stiffness is shown in Figure 156 (on page 208).

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').

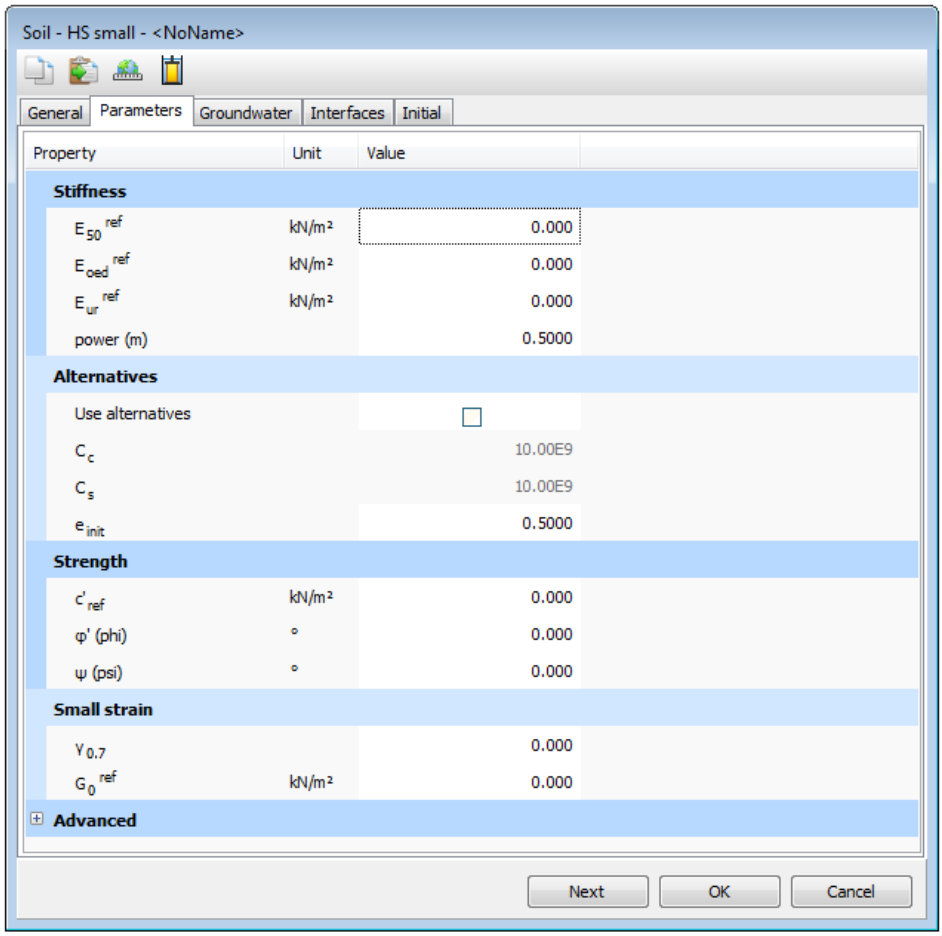


Figure 156: ***Parameters*** *tabsheet for the Hardening Soil model small(drained behaviour)*

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	[-]

Advanced parameters for stiffness:

ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.2)	[-]
p^{ref}	Reference stress for stiffnesses (default p^{ref} = 100 kN/m^2)	[kN/m^2]
K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - sin(\varphi)$)	[-]

Parameters for strength:

c'_{ref}	Effective cohesion	[kN/m^2]
φ'	Effective friction angle	[°]
ψ	Dilatancy angle	[°]

Advanced parameters for strength:

c'_{inc}	As in Mohr-Coulomb model (default c_{inc} = 0)	[kN/m^3]
z_{ref}	Reference level	[m]
R_f	Failure ratio q_f / q_a (default R_f = 0.9)	[-]
Tension cut-off	To be selected when tension cut-off is considered	[-]
Tensile strength	The allowable tensile strength	[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]

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 3D 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 dynamic 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 510).

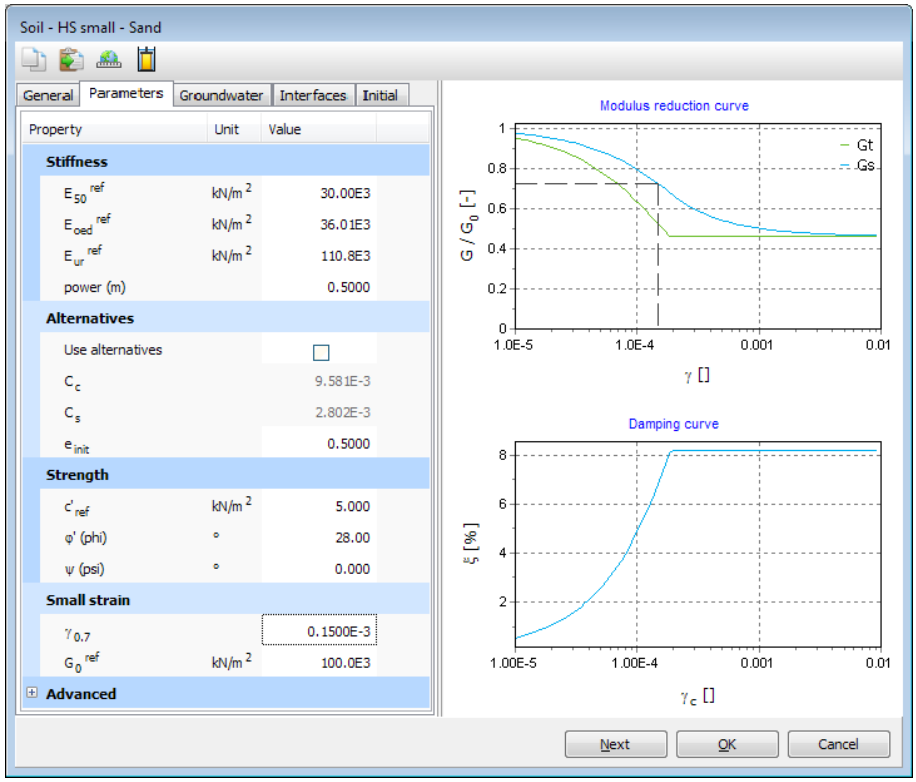


Figure 157: Effect of small strain stiffness parameters on damping

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.

Soft Soil model (SS)

The **Parameters** tabsheet for the Soft Soil model is shown in Figure 158 (on page 211).

Note:

- Optional drainage types when Soft Soil modelis selected are: *Drained* and *Undrained (A)*.
- In the case of *Undrained (A)* drainage type, the same parameters are used as for drained behaviour.

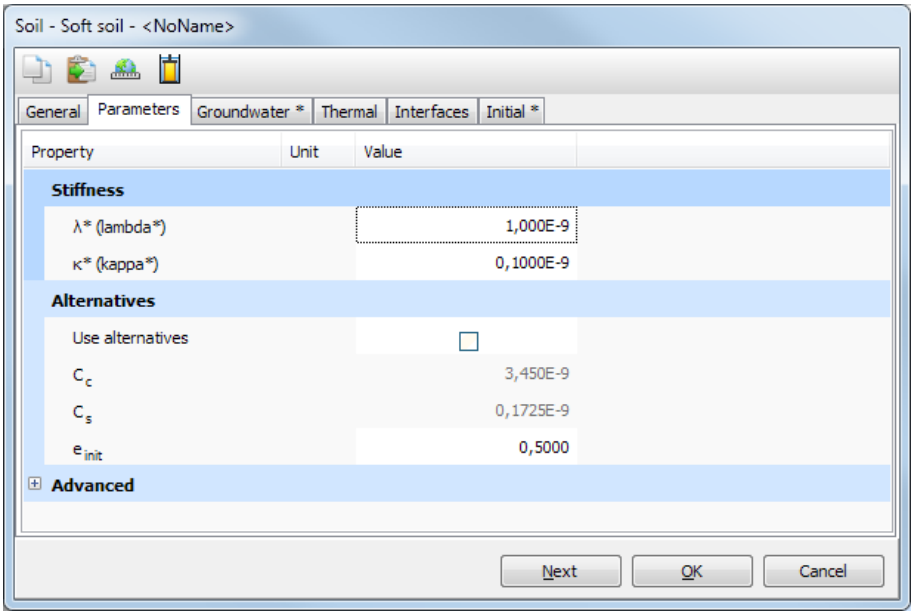


*Figure 158: **Parameters** tabsheet for the Soft Soil model (drained behaviour)*

The parameters for stiffness are:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]

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	[°]

Advanced parameters (use default settings):

ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.15)	[-]
K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - \sin(\varphi)$)	[-]
M	K_0^{nc}- related parameter	[-]

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 3Danalysis 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.

Soft Soil Creep model (SSC)

The **Parameters** tabsheet for the Soft Soil Creep model is shown in Figure 159 (on page 212).

Note:

- Optional drainage types when Soft Soil Creep 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.

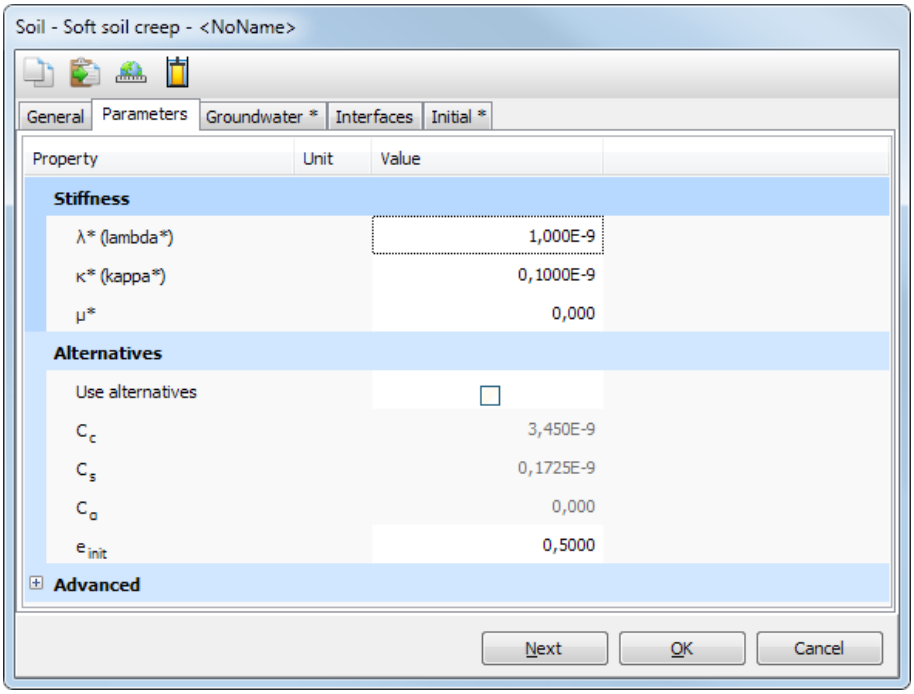


*Figure 159: **Parameters** tabsheet for the Soft Soil Creep model (drained behaviour)*

The parameters for stiffness are:

λ^*	Modified compression index	[-]
κ^*	Modified swelling index	[-]

The parameter taking time effect into account is:

μ^*	Modified creep index	[-]

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	[°]

Advanced parameters (use default settings):

ν_{ur}	Poisson's ratio for unloading-reloading (default ν = 0.15)	[-]
K_0^{nc}	K_0-value for normal consolidation (default $K_0^{nc} = 1 - sin(\varphi)$)	[-]
M	K_0^{nc}- related parameter	[-]
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 3D 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.

Joined Rock model (JR)

The **Parameters** tabsheet for the Joined Rock model is shown in Figure 160 (on page 214).

Note:

- Optional drainage types when Joined Rock model is selected are: *Drained* and *Non-porous*.
- In the case of *Non-porous* drainage type, the same parameters are used as for drained behaviour.

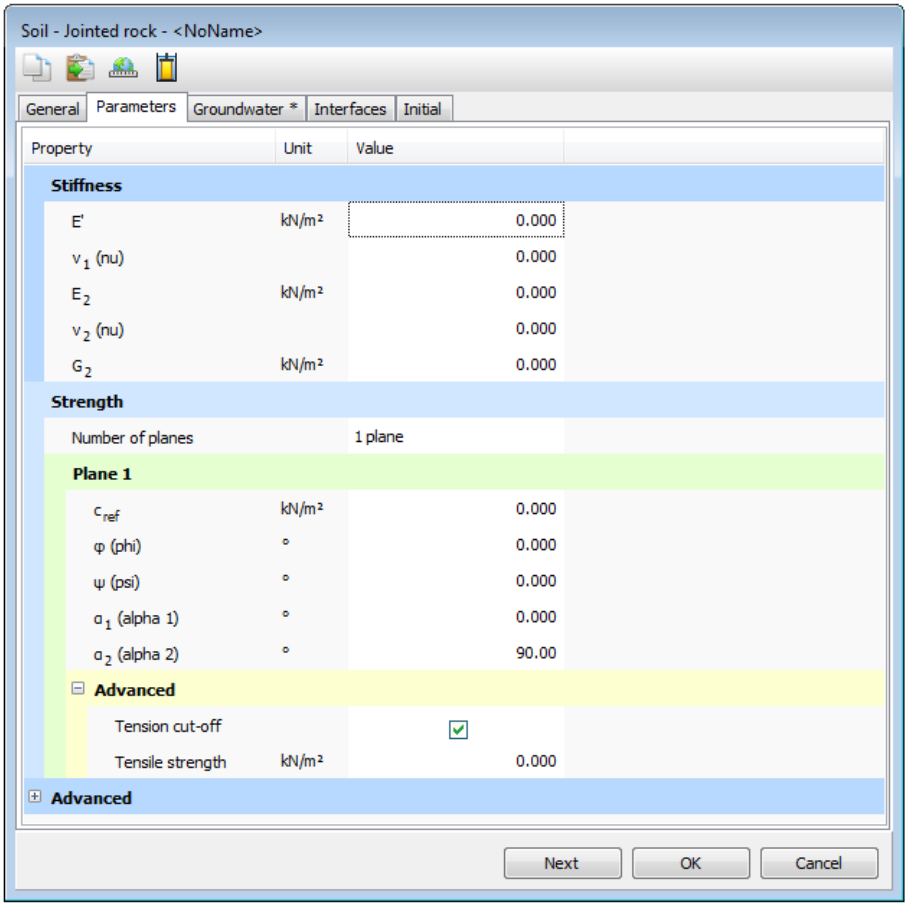


Figure 160: ***Parameters*** *tabsheet for the Joined Rock model*

Parameters for stiffness:

E_1	Young's modulus for rock as a continuum	[kN/m^2]
ν_1	Poisson's ratio for rock as a continuum	[-]

Anisotropic elastic parameters 'Plane 1' direction (e.g. stratification direction):

E_2	Young's modulus perpendicular to 'Plane 1' direction	[kN/m^2]
G_2	Shear modulus perpendicular to 'Plane 1' direction	[kN/m^2]
ν_2	Poisson's ratio perpendicular to 'Plane 1' direction	[-]

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):

n	Number of joint directions ($1 \le n \le 3$)	[-]
$\alpha_{1,i}$	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 3D 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.

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 **Parameters** tabsheet for the Modified Cam-Clay model is shown in Figure 161 (on page 215).

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.

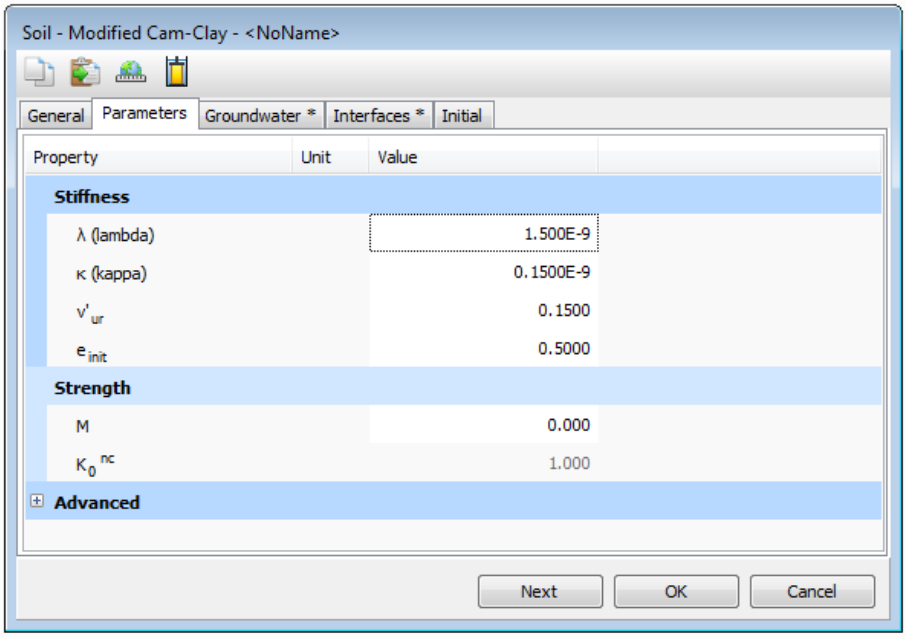


*Figure 161: **Parameters** tabsheet for the Modified Cam-Clay model (drained behaviour)*

Parameters for stiffness:

λ	Cam-Clay compression index	[-]
κ	Cam-Clay swelling index	[-]
ν_{ur}	Poisson's ratio	[-]
e_{init}	Initial void ratio for loading/unloading	[-]

Parameters for strength:

M	Tangent of the critical state line	[-]
K_0^{nc}	Coefficient of lateral stress in normal consolidation derived from M. The relationship between M and K_0^{nc} is given in the Material Models Manual *Chapter 10- Parameters of the soft soil creep model.*	[-]

NGI-ADP model (NGI-ADP) GSE

The NGI-ADP model may be used for capacity, deformation and soil-structure interaction analysis involving undrained loading of clay-type materials. The **Parameters** tabsheet for the NGI-ADP model is shown in Figure 162 (on page 217).

Note:

Optional drainage types when NGI-ADP model is selected are: *Drained*, and *Undrained (C)*.

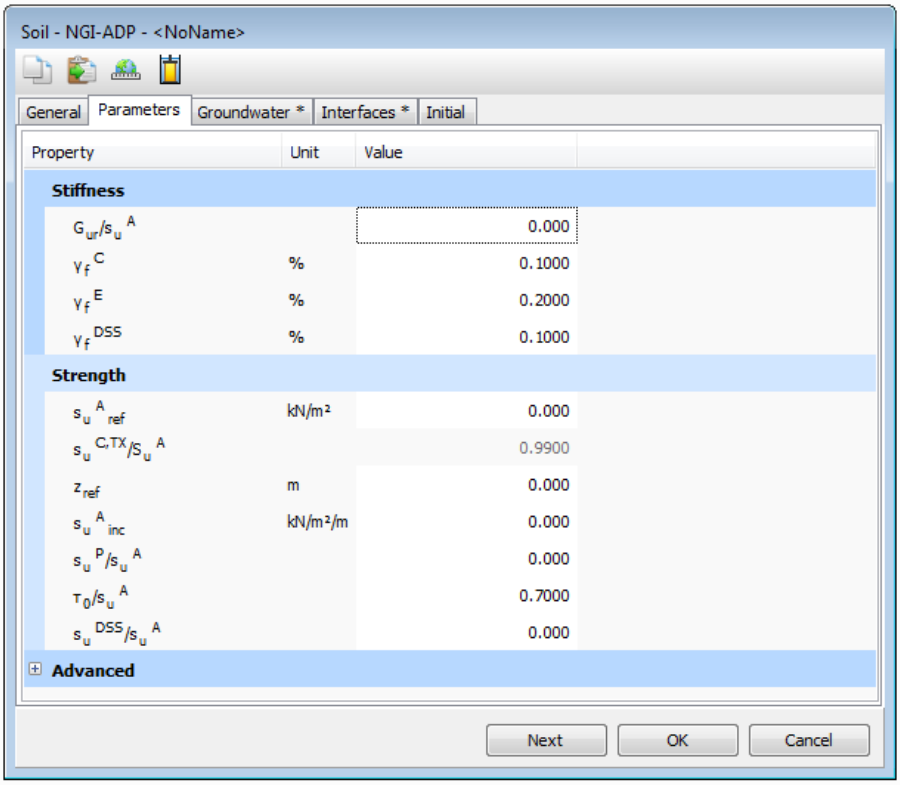


Figure 162: ***Parameters*** *tabsheet for the NGI-ADP model*

- *Parameters for stiffness:*

G_{ur} / s_u^A	Ratio unloading/reloading shear modulus over (plane strain) active shear strength	[-]
γ_f^C	Shear strain in triaxial compression ($\lvert\gamma_f^C = 3/2\ \varepsilon_1^C\rvert$)	[%]
γ_f^E	Shear strain in triaxial extension	[%]
γ_f^{DSS}	Shear strain in direct simple shear	[%]

- *Parameters for strength:*

$s_u^{A,\,ref}$	Reference (plane strain) active shear strength	[kN/m^2]
$s_u^{C,TX}/s_u^A$	Ratio triaxial compressive shear strength over (plane strain) active shear strength (default = 0.99)	[-]
z_{ref}	Reference level	[m]
$s_{u,inc}^A$	Increase of shear strength with depth	[kN/m^2/m]
s_u^P/s_u^A	Ratio of (plane strain) passive shear strength over (plane strain) active shear strength	[-]

τ_0/s_u^A	Initial mobilization (default = 0.7)	[-]
s_u^{DSS}/s_u^A	Ratio of direct simple shear strength over (plane strain) active shear strength	[-]

- *Advanced parameters:*

ν'	Effective Poisson's ratio	[-]
ν_u	Undrained Poisson's ratio	[-]

UDCAM-S model GSE

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 **Parameters** tabsheet for the UDCAM-S model is shown in Figure 163 (on page 219). 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:

Optional drainage types when UDCAM-S model is selected are: *Undrained (B)* and *Undrained (C)*.

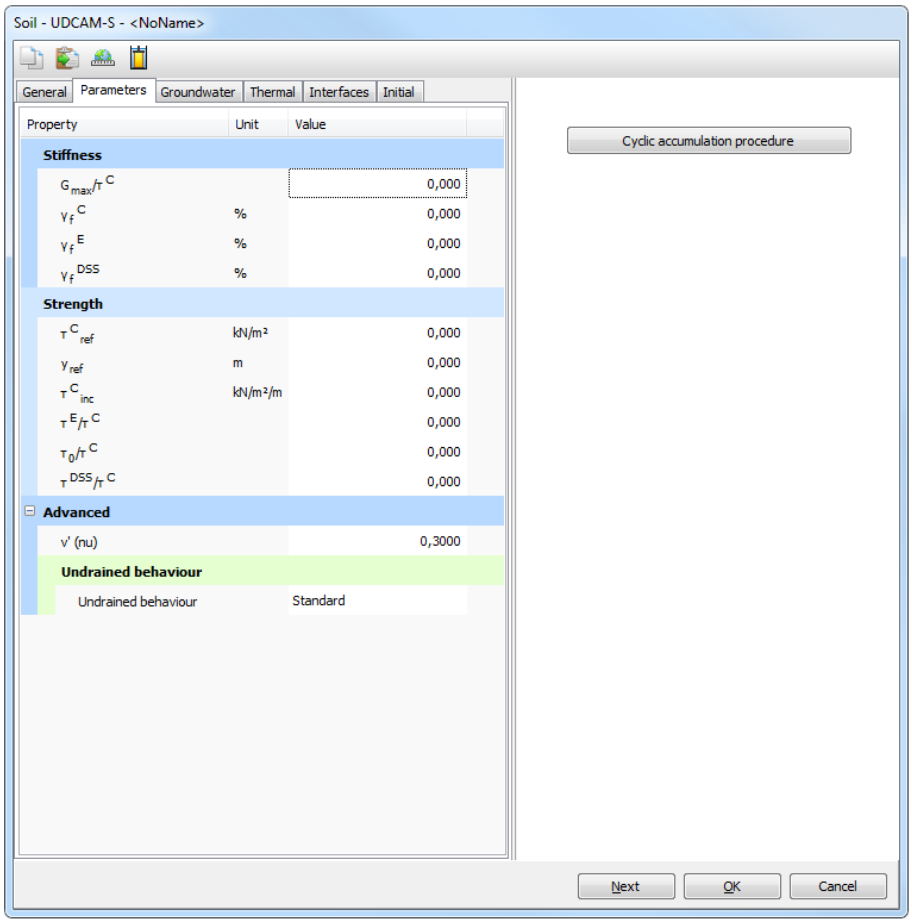


Figure 163: ***Parameters*** *tabsheet for the UDCAM-S model*

- *Parameters for stiffness:*

G_{max}/τ^C	Ratio unloading/reloading shear modulus over (plane strain) active shear strength	[-]
γ_f^C	Shear strain in triaxial compression	[%]
γ_f^E	Shear strain at failure in triaxial extension	[%]
γ_f^{DSS}	Shear strain at failure in direct simple shear	[%]

- *Parameters for strength:*

τ_{ref}^C	Degraded reference shear strength in triaxial compression	[kN/m^2]
z_{ref}	Reference level	[m]
τ_{inc}^C	Increase of degraded triaxial compression shear strength with depth	[kN/m^2/m]
τ^E/τ^C	Ratio of degraded triaxial compression shear strength triaxial extension over degraded triaxial compression shear strength	[-]

τ_0/τ^C	Initial mobilisation	[-]
τ^{DSS}/τ^C	Ratio of degraded direct simple shear strength over degraded triaxial compression shear strength	[-]

- *Advanced parameters:*

ν'	Effective Poisson's ratio	[-]
ν_u	Undrained Poisson's ratio	[-]

Note:

The model should be only used for undrained conditions.

Hoek-Brown model (HB) GSE

The **Parameters** tabsheet for the Hoek-Brown model is shown in Figure 164 (on page 221). 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*.
- In the case of *Non-porous* drainage type, the same parameters are used as for drained behaviour.

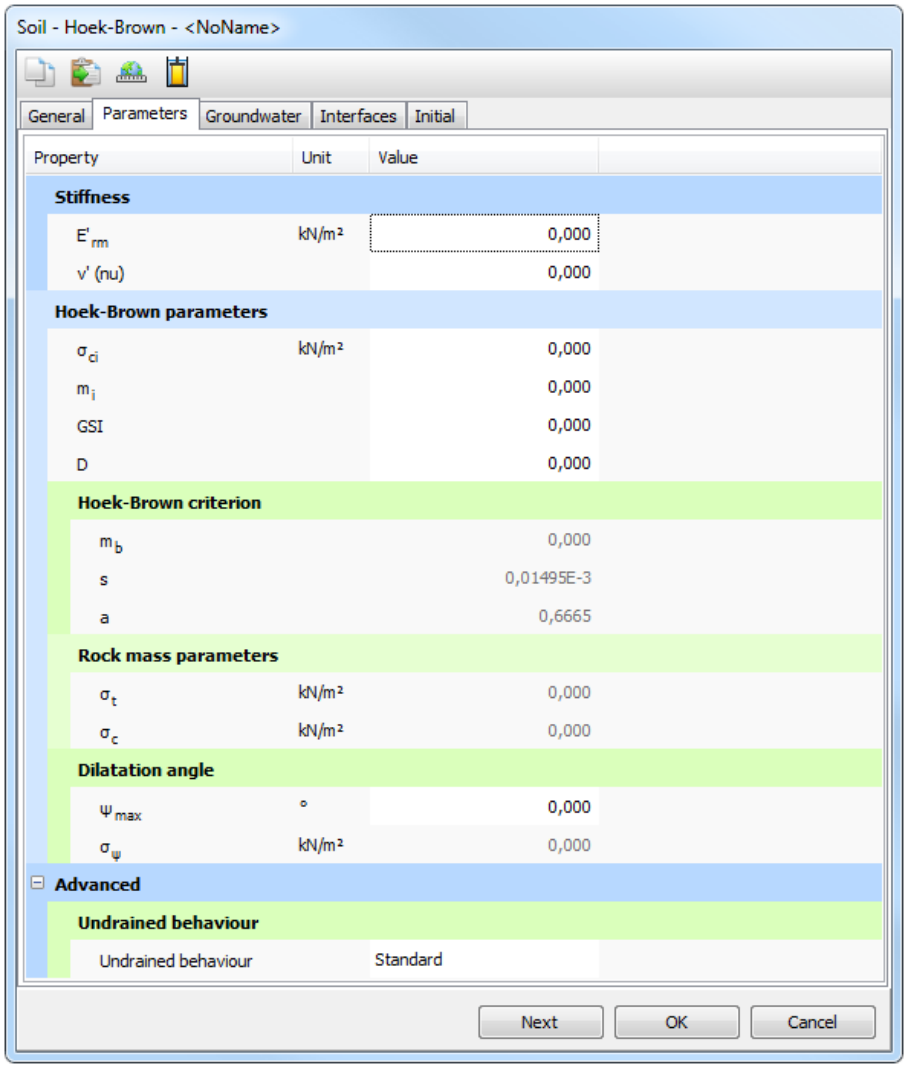


Figure 164: ***Parameters*** *tabsheet for the Hoek-Brown model (drained behaviour)*

The stiffness parameters of the Hoek-Brown model are:

E_{rm}	Rock mass Young's modulus	[kN/m^2]
ν	Poisson's ratio	[-]

The side panel supports the selection of the Hoek-Brown parameters. The Hoek-Brown parameters are:

$\lvert\sigma_{ci}\rvert$	Uni-axial compressive strength of the intact rock (>0)	[kN/m^2]
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	[-]
ψ_{max}	Dilatancy at zero stress level	[°]
σ_{ψ}	Poisson's ratio	[kN/m^2]

UBC3D-PLM model GSE

The **Parameters** tabsheet for the UBC3D-PLM model is shown in Figure 165 (on page 223).

Note:

Drainage types when UBC3D-PLM model is selected are: *Drained* and *Undrained A*. The other undrained calculations are not available due to the effective stress nature of the model.

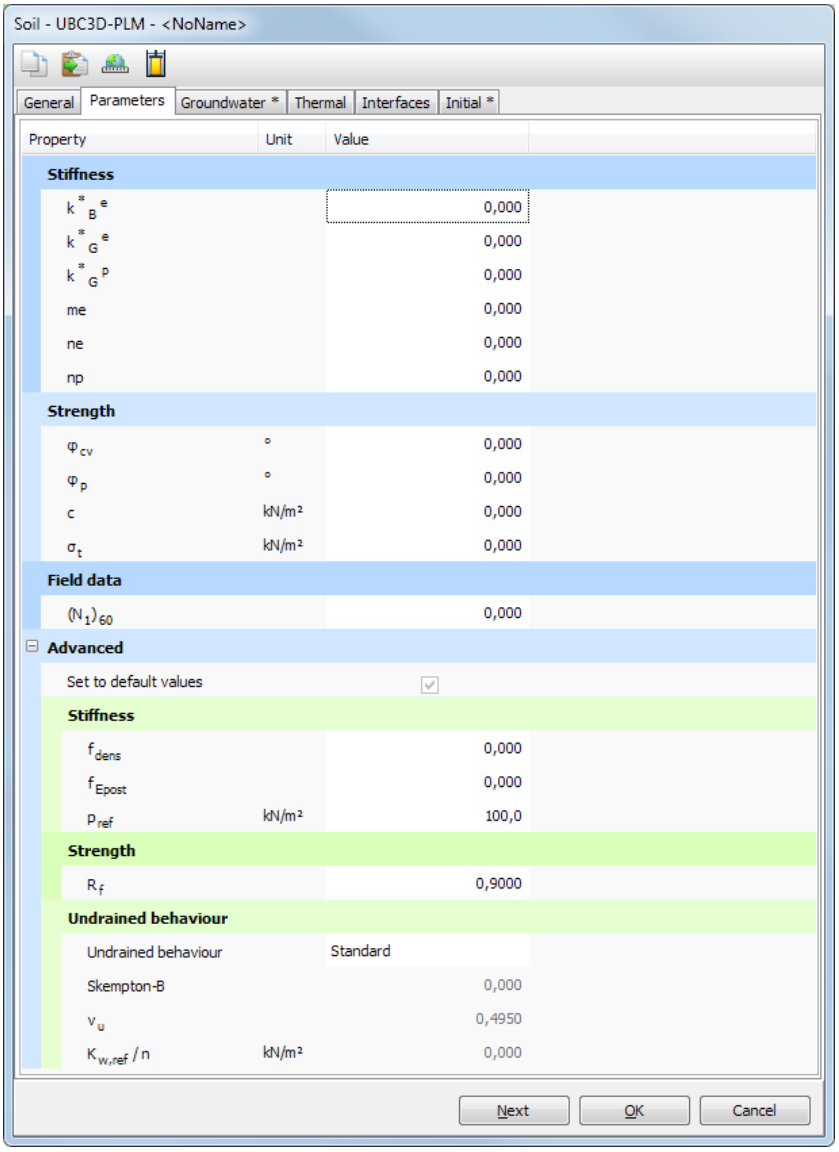


Figure 165: Parameters tabsheet for the UBC3D-PLM model

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	Elastic bulk modulus index	[-]
ne	Elastic shear modulus index	[-]

np	Plastic shear modulus index	[-]
p_{ref}	Reference pressure	[kN/m^2]

Note:

The implicit Poisson's ratio that is defined based on $k^e{}_B$ and $k^e{}_G$ is suitable for dynamic 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:

φ_{cv}	Constant volume friction angle	[°]
φ_p	Peak friction angle	[°]
c	Cohesion	[kN/m^2]
σ_t	Tension cut-off and tensile strength	[kN/m^2]

Advanced parameters for strength and stiffness:

R_f	Failure ratio	[-]
$(N_1)_{60}$	Corrected SPT value	[-]
f_{dens}	Densification factor	[-]
f_{Epost}	Post-liquefaction stiffness factor	[-]

Concrete model GSE

The **Parameters** tabsheet for the Concrete model are shown in Figure 166 (on page 225).

Note:

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.

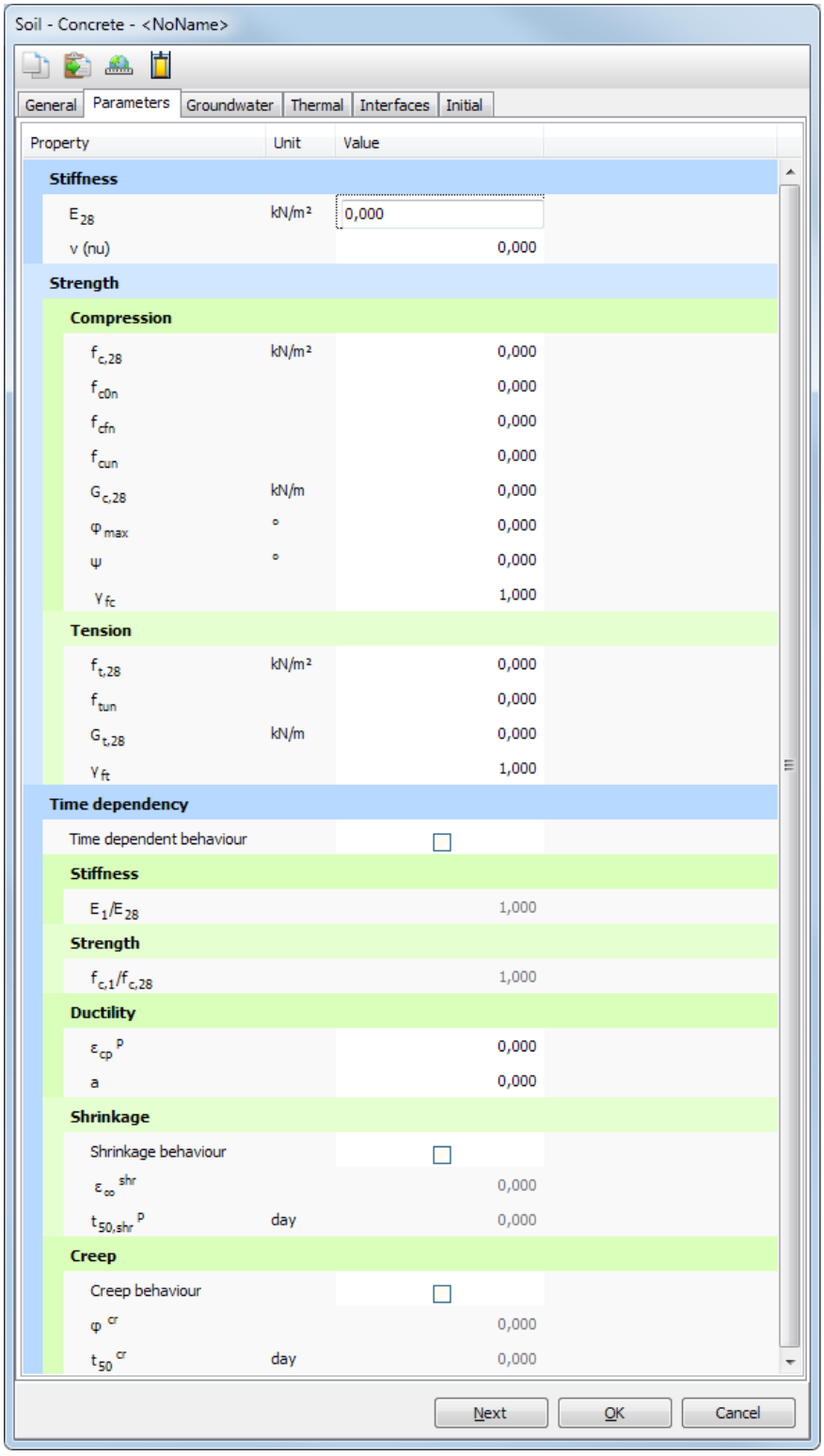


Figure 166: ***Parameters*** *tabsheet for the Concrete model*

- *Parameters for stiffness:*

E_{28}	Young's modulus for rock as a continuum	[kN/m^2]
E_1 / E_{28}	Time-dependency ratio of elastic stiffness	[-]
ν	Poisson's ratio	[-]

- *Parameters for strength in compression:*

$f_{c,28}$	Young's modulus for rock as a continuum	[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	[°]

- *Time-dependency of compressive strength:*

$f_{c,1} / f_{c,28}$	Time-dependency of compressive strength	[-]
t_{hydr}	Time for full hydration	[day]

- *Parameters for tensile strength:*

$f_{t,28}$	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]

- *Parameters for 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)	[-]
a	Normalised residual strength	[-]

- *Parameters for creep:*

φ^{cr}	Ratio between creep and elastic strains	[-]

t_{50}^{cr}	Time for 50% of creep strains	[day]

- *Parameters for shrinkage:*

$\varepsilon_{\infty}^{shr}$	Safety factor for compressive strength	[-]
t_{50}^{cr}	Time for 50% of shrinkage strains	[day]

- *Safety factor:*

γ_{fc}	Safety factor for compressive strength	[-]
γ_{ft}	Safety factor for tensile strength	[-]

Note:

- When simulating shotcrete the tensile strength is essential for tunnel stability. Neglecting or considering low values of it 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) GSE

The **Parameters** 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 3D 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*.

Advanced parameters for Undrained behaviour and Coupled Analysis

The advanced parameters available in the **Parameters** tabsheet can be used to model the *Undrained behaviour* of soils and the properties of the pore water in coupled analysis. The advanced parameters for the *Undrained behaviour* shows three drop-down menus (Figure 167 (on page 228)) namely *Standard*, *Manual - Stiffness dependent* K_w and *Manual - Constant* K_w. By default, the solid material of the soil (i.c the grains) is supposed to be incompressible (α_{Biot} = 1) and the undrained Poisson's ratio is taken as *0.495*. This is the *Standard* behaviour. Alternatively, one can select *Manual - Stiffness dependent* K_w; here the Skempton's B-parameter can be specified, K_w and $K_{w,ref}/n$ are calculated accordingly. In the most advanced third option *Manual - Constant* K_w, the compressibility of the solid material is considered and one can select α_{Biot} together with K_w and ν_u; Skempton B and $K_{w,ref}/n$ are calculated accordingly.

Skempton-B	A parameter that determines which portion of a change in mean stress is carried by the pore water	[-]
ν_u	Undrained Poisson's ratio	[-]
α_{Biot}	Biot alpha pore pressure coefficient	[-]
K_w	Bulk modulus of water	[-]
$K_{w,ref} / n$	The corresponding reference bulk stiffness of the pore fluid	[kN/m^2]
$C_{v,ref}$	Consolidation coefficient (Mohr-Coulomb modelonly)	[m^2/day]

More detailed information is available in the Material Models Manual.

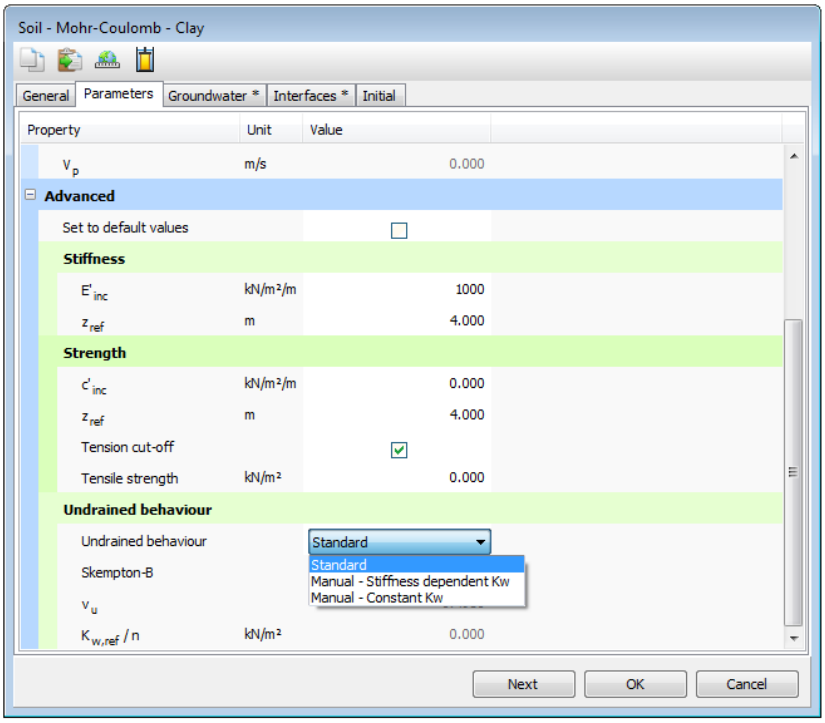


Figure 167: Advanced parameters for Undrained Behaviour

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 3D implements in the *Parameters* 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 3D 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 3Dautomatically 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 229) 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	$m_i \pm m$
Agglomerate	Igneous	Coarse	19	3
Amphibolites	Metamorhpic	Medium	26	6

Name	Rock type	Texture	m_i	$m_i \pm m$
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
Migmatite	Metamorhpic	Coarse	29	3

Name	Rock type	Texture	m_i	m_i \pm
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 168 (on page 232) and *Flysch* Figure 169 (on page 233). 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.

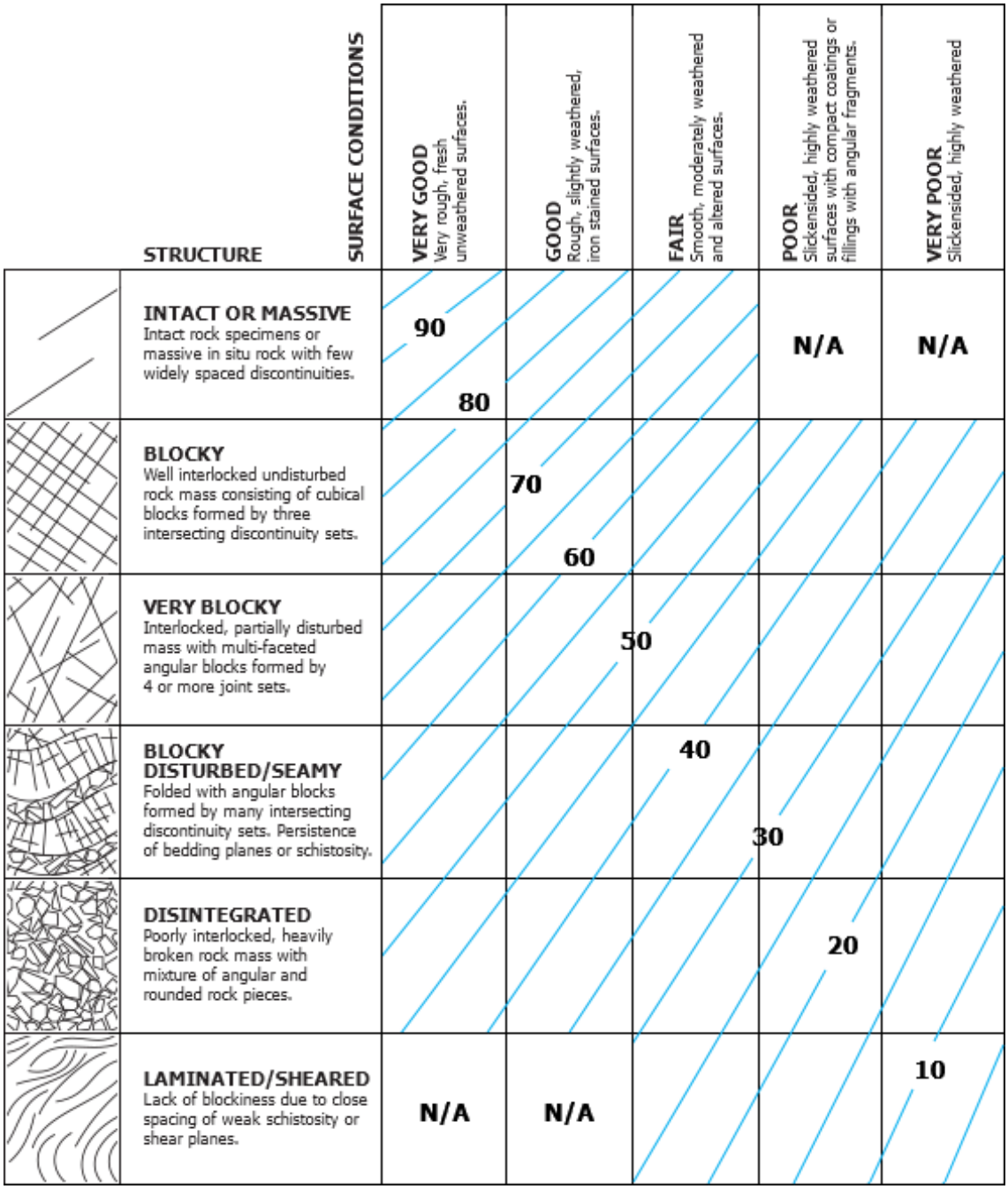


Figure 168: GSI chart for General rock type

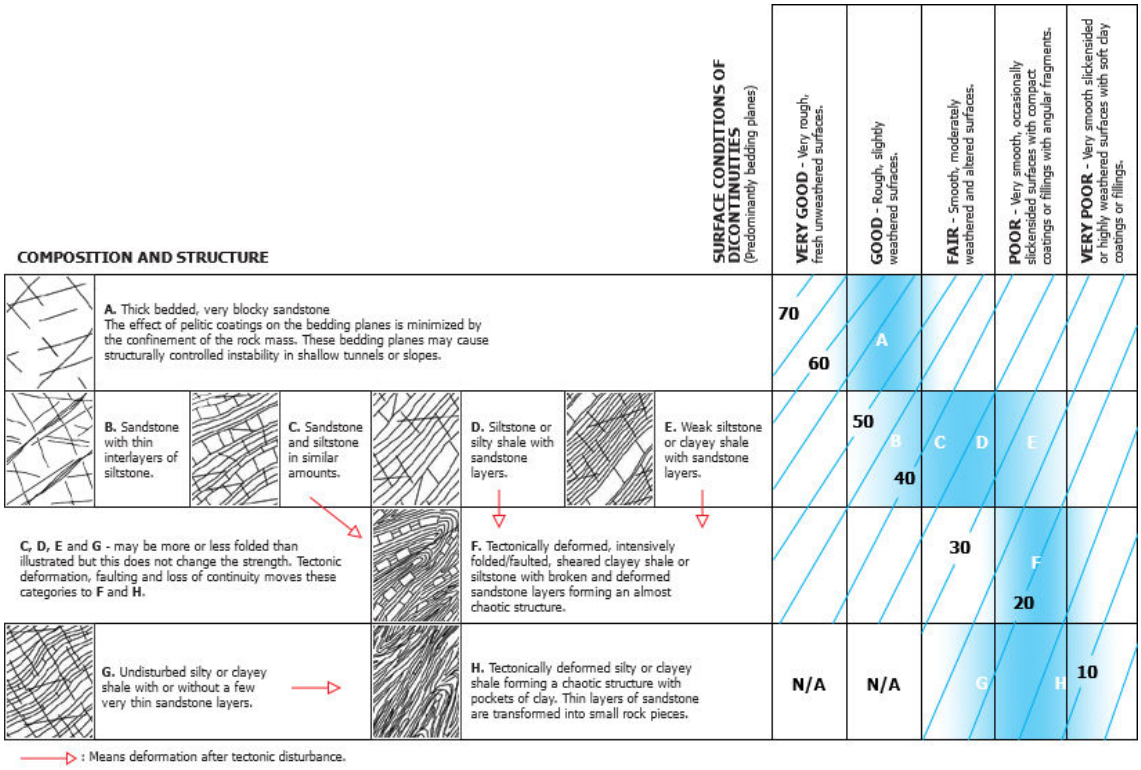


Figure 169: GSI chart for Flysch rock type

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 234) and slopes Table 8 (on page 234).

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 234)Table 8 (on page 234).
- 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 234) 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*

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 510)). 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 510)).

Starting the tool

To start the tool, the *Cyclic accumulation and optimisation tool* is clicked from the side panel in the **Parameters** tabsheet of the UDCAM-S model . The window consists of two blue tabs and one orange tab, see Figure 170 (on page 236) for an overview.

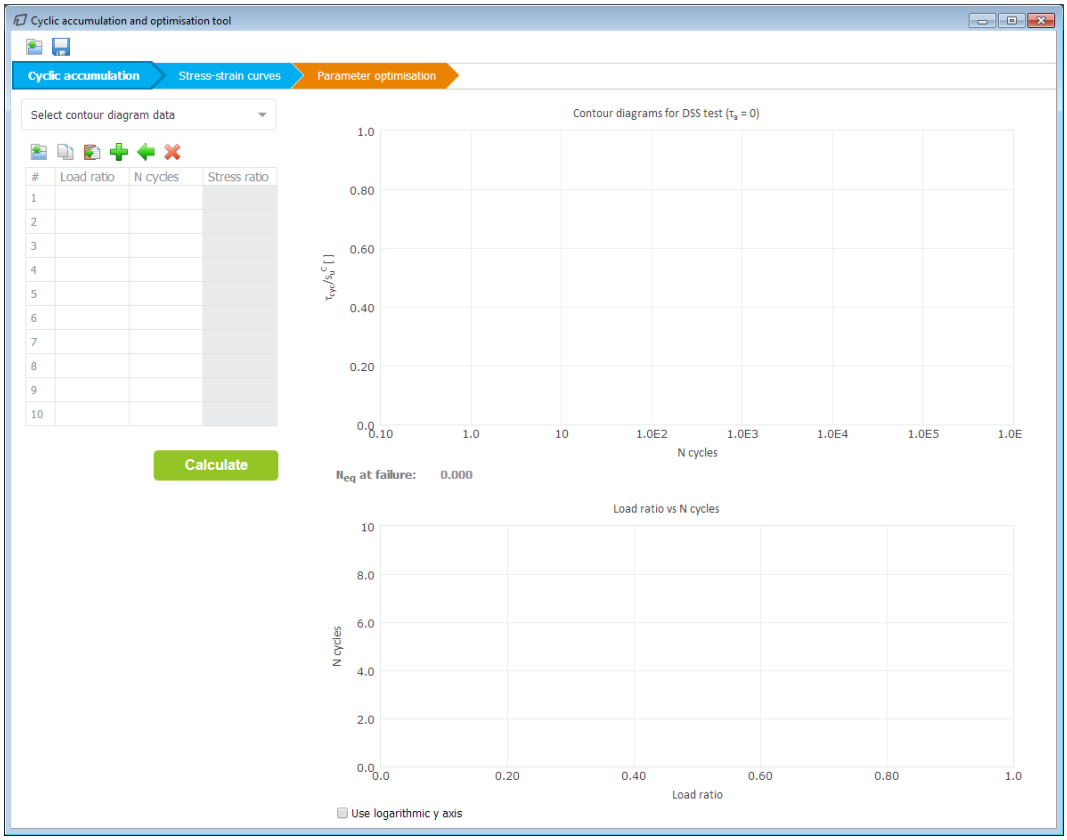


Figure 170: Cyclic accumulation and optimisation tool

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 171 (on page 236)) 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.

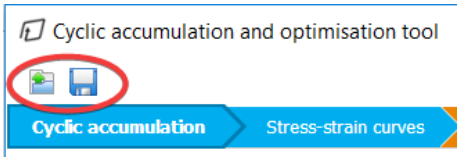


Figure 171: Load and save app state

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 511))) or the recently developed method, for more information see Noren - Cosgriff, Josatd & Madshus (2015) (on page 511)).

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 volume. 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 510)) 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 172 (on page 237)) 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 172: 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 510)) 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 (on page 519).

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 510)).
- 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 510)).

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 173 (on page 239) .

The buttons in the toolbar can be used to modify the Figure 173 (on page 239).



#	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 173: 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 174 (on page 240).

- 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 173 (on page 239). 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.

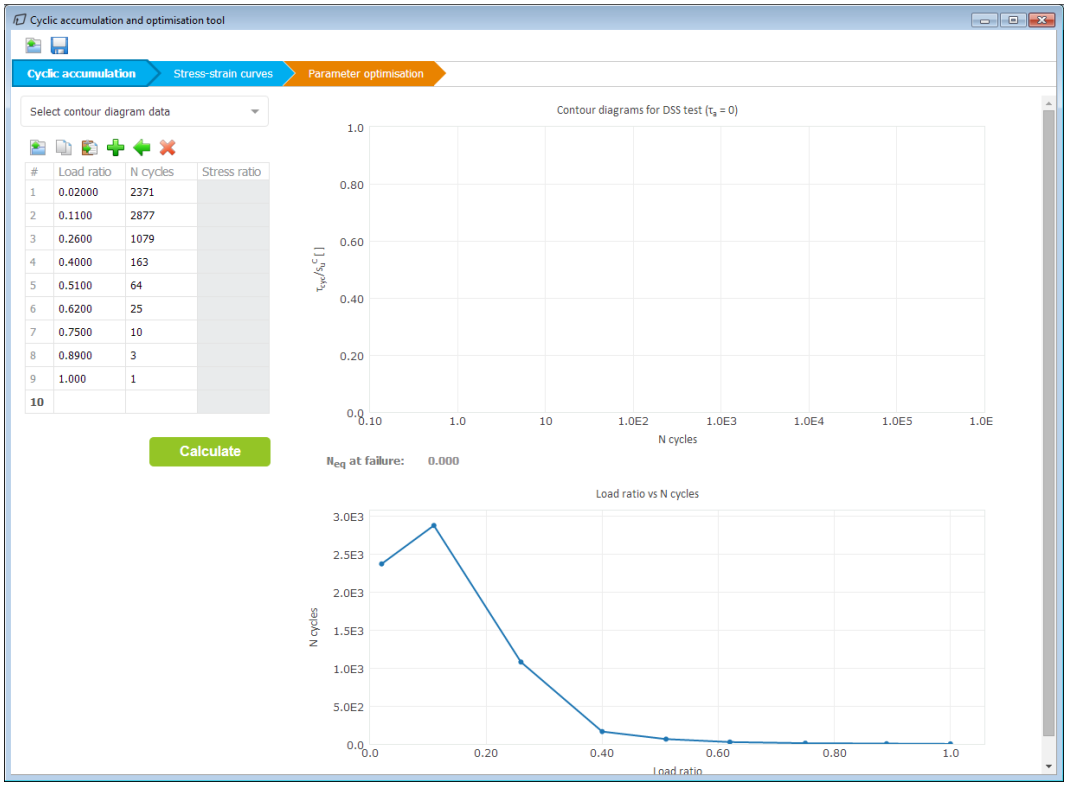


Figure 174: Automatic update of Load ratio graph when specifying parcels

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}.

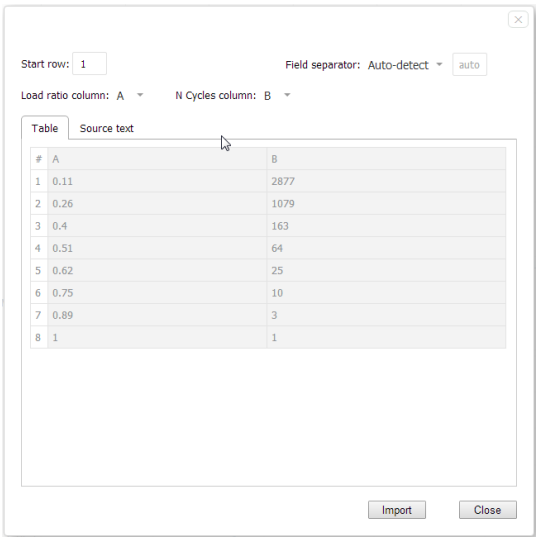


Figure 175: Import load parcels settings dialog

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 586).

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 519) 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 510)). 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 176: 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.

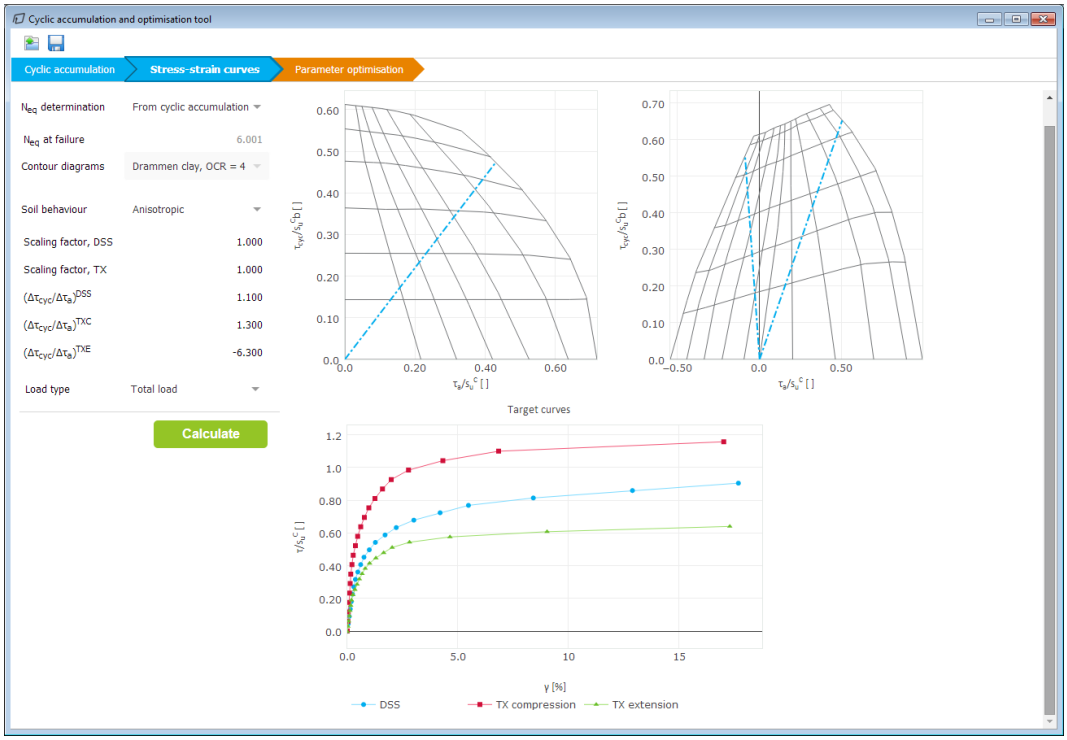


Figure 177: Stress-strain curves tabsheet displaying results after calculation

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 178 (on page 245)) 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.

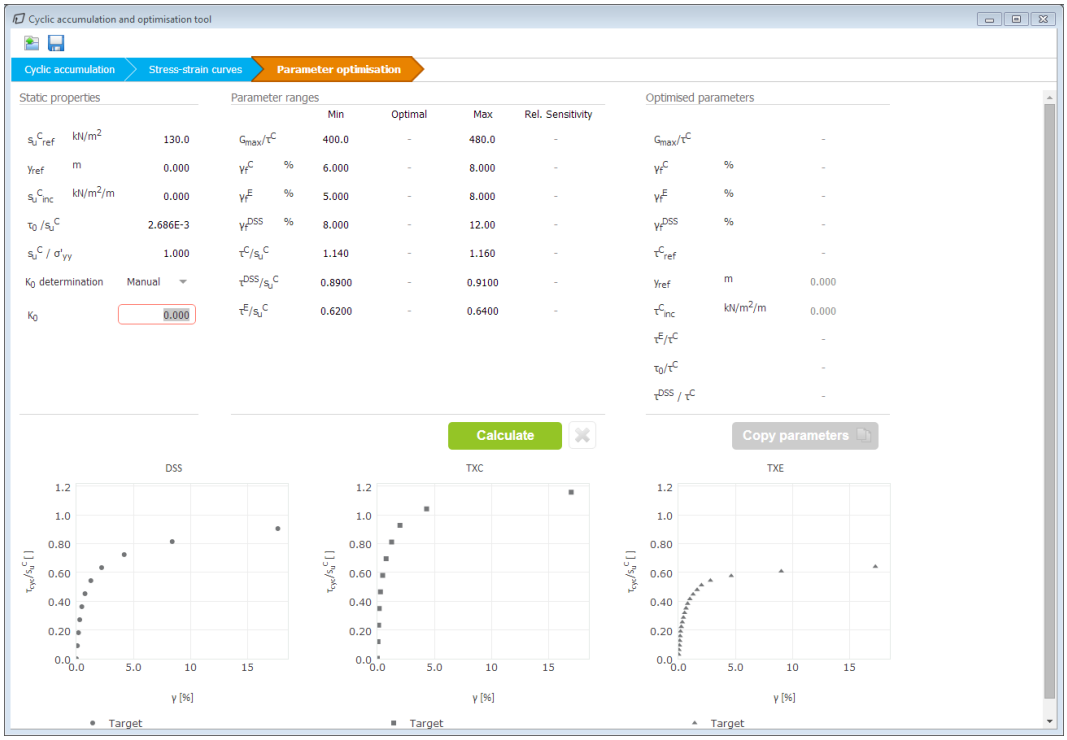


Figure 178: Parameter optimisation tabsheet

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 179 (on page 246)). 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.

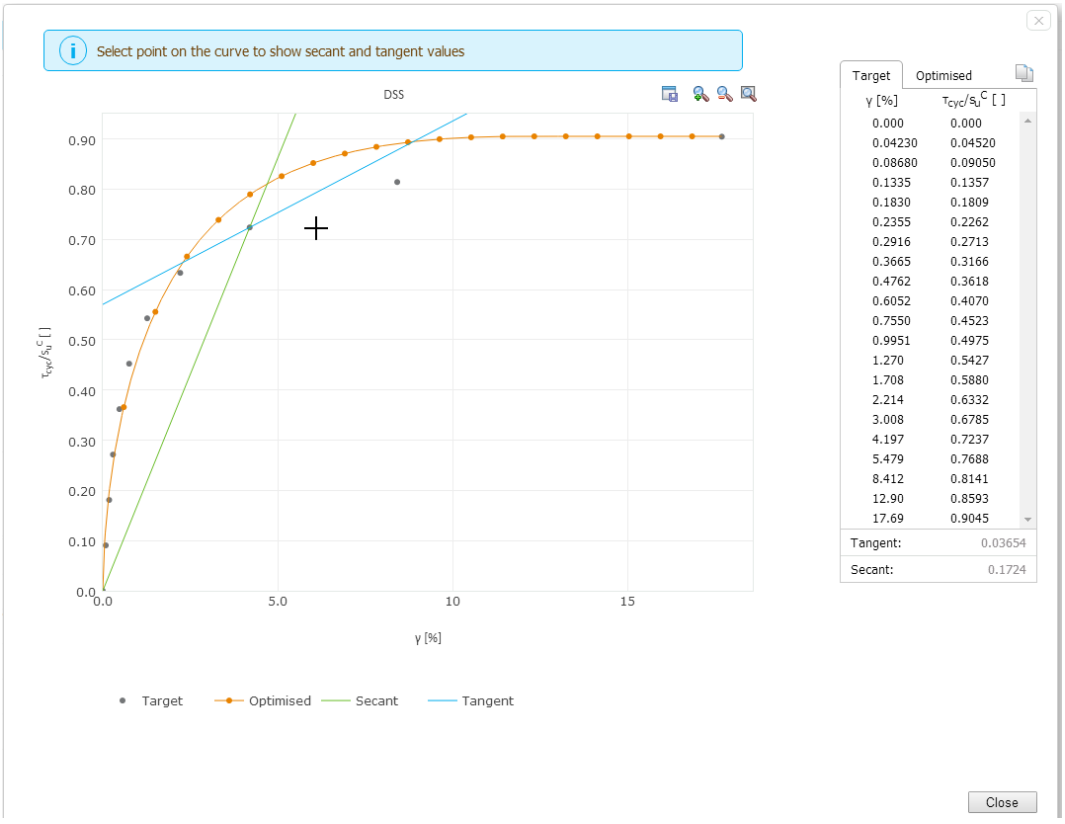


Figure 179: Chart displaying target, optimised, secant and tangent curves, and corresponding table of values.

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 510)). 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 178 (on page 245)).



Figure 180: 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]
τ_{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 - *Chapter 14*.

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 594).

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 3D 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:

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 512)).

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 181 (on page 250)). The particle fractions can also be defined by clicking on the corresponding location in the soil texture triangle or by directly typing the values.

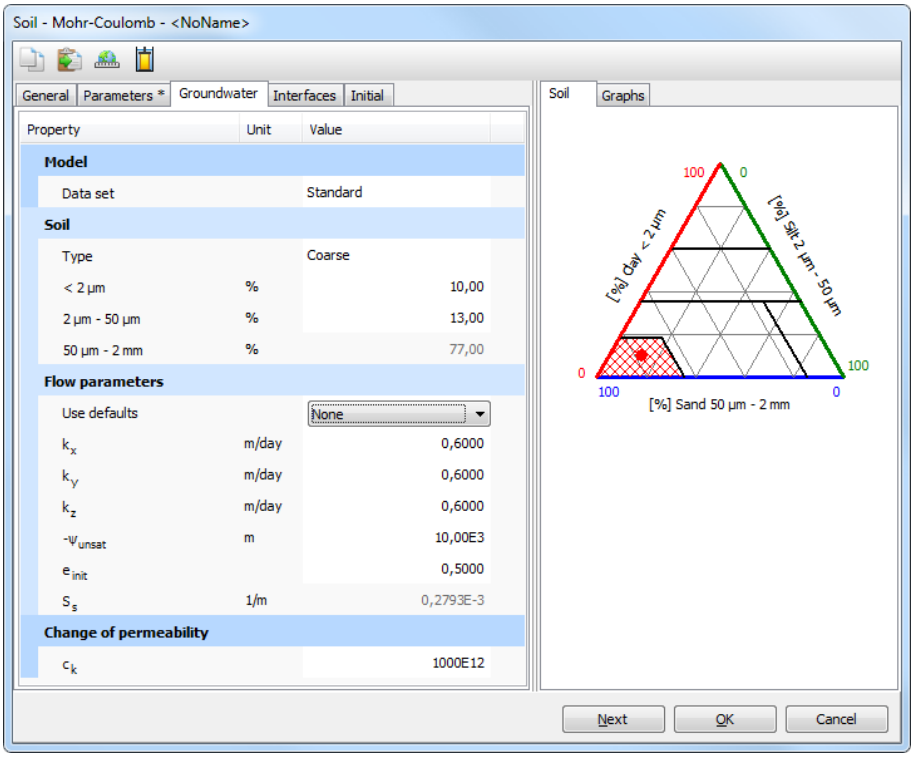


Figure 181: Groundwater parameters for Standard data set (Wösten, Lilly, Nemmes & Bas, 1999)

Hypres: The *Hypres* series is an international soil classification system (Wosten, Lilly, Nemmes & Bas, 1999 (on page 512)). The hydraulic models available for *Hypres* data set are the *Van Genuchten* model and the *Approximate Van Genuchten* (see Material Models Manual - *Chapter 19*).

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 182 (on page 251)).

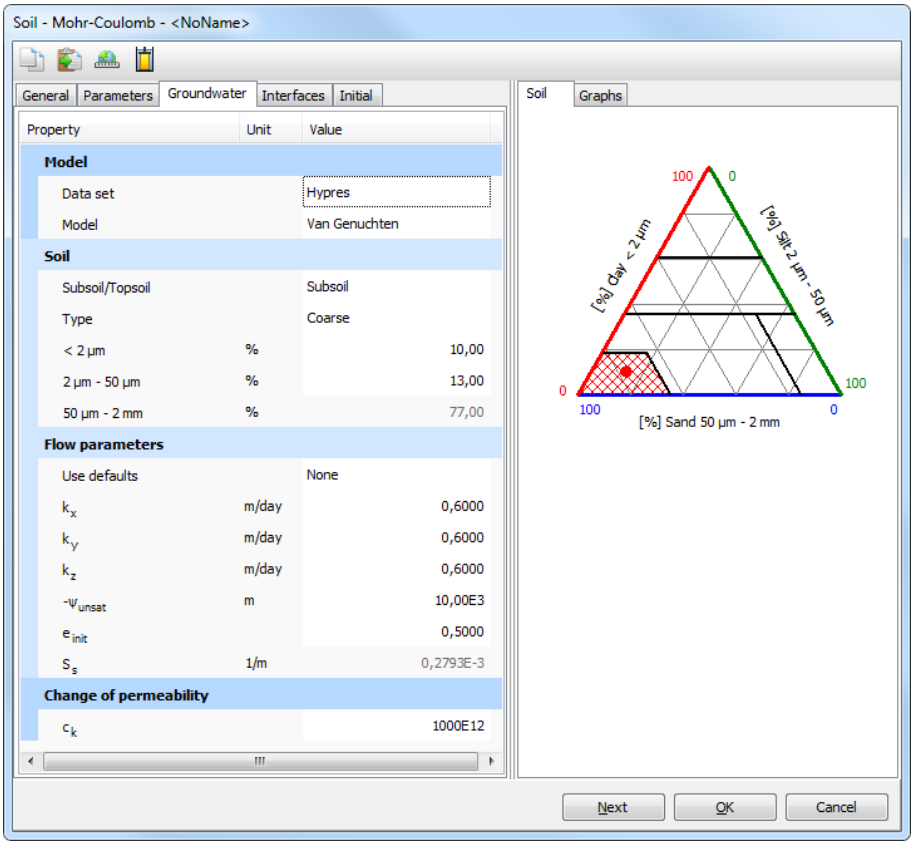


Figure 182: Groundwater parameters for Hypres data set (Wösten, Lilly, Nemmes & Bas, 1999)

The predefined parameters for both the *Van Genuchten* model as well as the *Approximate Van Genuchten* model are shown in Table 10 (on page 251) and Table 11 (on page 252).

Table 10: *Hypres* series with Van Genuchten parameters (Wösten, Lilly, Nemmes & Bas, 1999)

Topsoil:	θ_r	θ_s	K_{sat} (m/day)	g_a (1/m)	g_l (-)	g_n (-)
coarse	0.025	0.403	0.6	3.83	1.25	1.3774
medium	0.01	0.439	0.121	3.14	-2.3421	1.1804
medium fine	0.01	0.43	0.0227	0.83	-0.5884	1.2539
fine	0.01	0.52	0.248	3.67	-1.9772	1.1012
very fine	0.01	0.614	0.15	2.65	2.5	1.1033
Subsoil:						
coarse	0.025	0.366	0.7	4.3	1.25	1.5206
medium	0.01	0.392	0.108	2.49	-0.7437	1.1689

Subsoil:						
medium fine	0.01	0.412	0.04	0.82	0.5	1.2179
fine	0.01	0.481	0.085	1.98	-3.7124	1.0861
very fine	0.01	0.538	0.0823	1.68	0.0001	1.073
organic	0.01	0.766	0.08	1.3	0.4	1.2039

Table 11: *Hypres* series with Approximate Van Genuchten parameters

Topsoil	ψ_s(m)	ψ_k (m)
coarse	-2.37	-1.06
medium	-4.66	-0.5
medium fine	-8.98	-1.2
fine	-7.12	-0.5
very fine	-8.31	-0.73
Subsoil:		
coarse	-1.82	-1
medium	-5.6	-0.5
medium fine	-10.15	-1.73
fine	-11.66	-0.5
very fine	-15.06	-0.5
organic	-7.35	-0.97

USDA: The *USDA* series is another international soil classification system (Carsel & Parrish, 1988 (on page 511)). 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 183 (on page 253)).

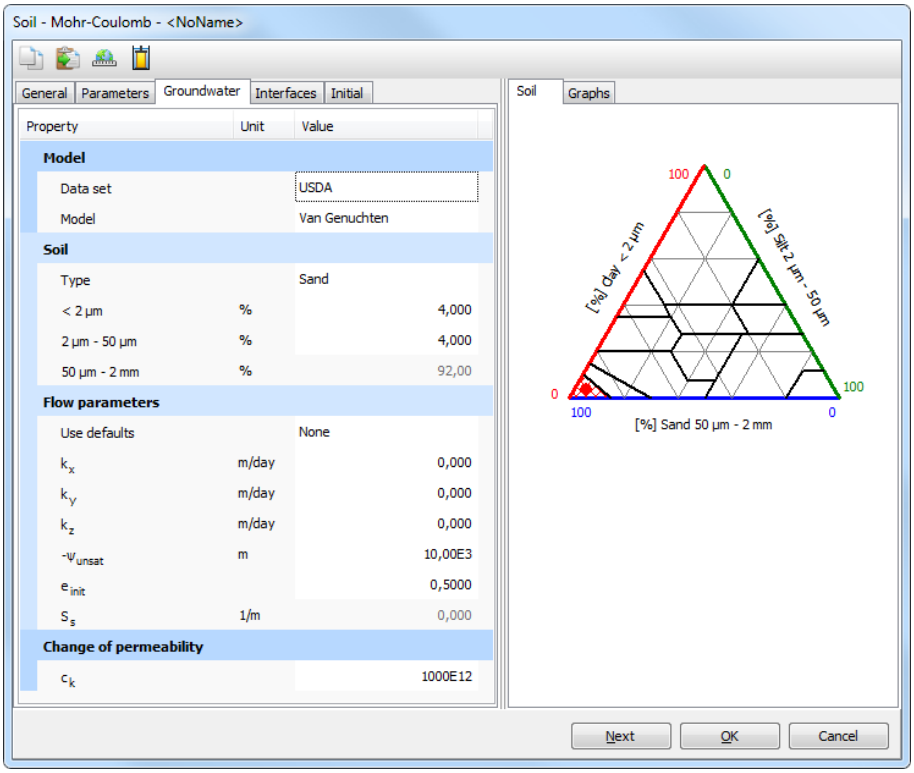


Figure 183: Groundwater parameters for USDA data set (Carsel & Parrish, 1988)

The parameters for the *Van Genuchten* and the *Approximate Van Genuchten* models are shown in Table 12 (on page 253) and Table 13 (on page 254) .

Table 12: USDA series with Van Genuchten parameters, $g_l = 0.5$ (Carsel & Parrish, 1988)

Soil type	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	3.5	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	0.6	1.6	1.37
silty loam	0.067	0.45	0.108	2	1.41
sandy clay loam	0.1	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	1.23

Soil type	θ_r (-)	θ_s (-)	K_{sat} (m/day)	g_a (1/m)	g_n (-)
sandy clay	0.1	0.38	0.288	2.7	1.23
silty clay	0.07	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 type	ψ_s (m)	ψ_k (m)
sand	-1.01	-0.5
loamy sand	-1.04	-0.5
sandy loam	-1.2	-0.5
loam	-1.87	-0.6
silt	-4	-1.22
silty loam	-3.18	-1.02
sandy clay loam	-1.72	-0.5
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	-0.6

Note:

In Table 10 (on page 251) to Table 12 (on page 253)the symbols θ_r and θ_s stand for the residual and the saturated water content correspondingly. Based on van Genuchten (1980) (on page 512), 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 3D, 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 526)) , 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 251) to Table 12 (on page 253) as $S_r = \theta_r / \theta_s$.

Staring: The *Staring* series is a soil classification system which is mainly used in The Netherlands (Wosten, Veerman, DeGroot & Stolte, 2001 (on page 512)). The hydraulic models available for *Staring* data set are the *Van Genuchten* model and the *Approximate Van Genuchten* (see the Material Models Manual - *Chapter 19*).

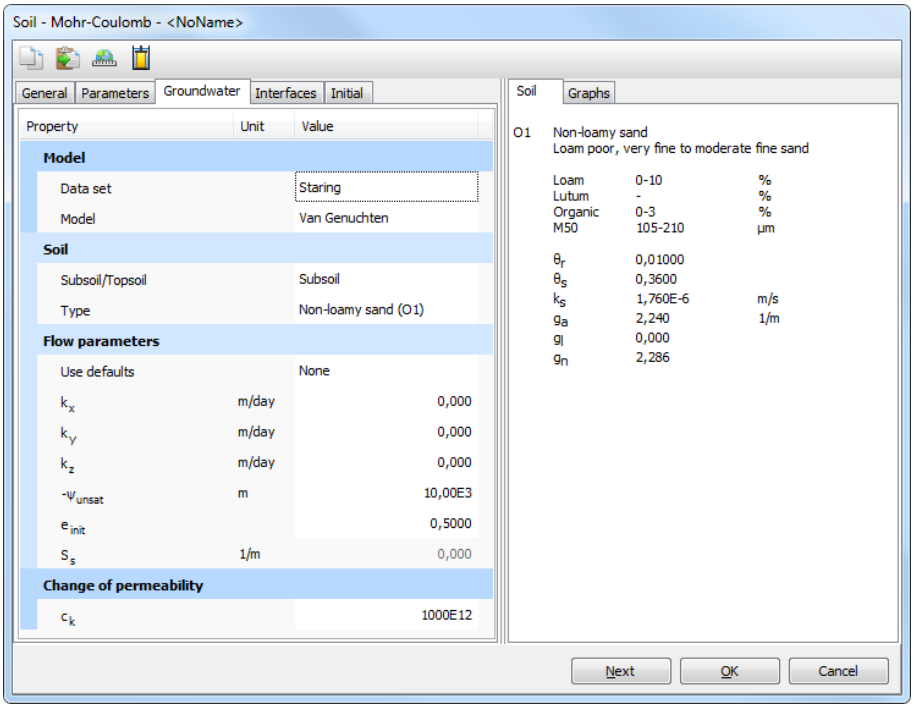


Figure 184: Groundwater parameters for Staring data set (Wösten, Veerman, DeGroot & Stolte, 2001)

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 184 (on page 255)) contains the following subsoils (Wosten, Veerman, DeGroot & Stolte, 2001 (on page 512)):

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 **Flow parameters** tabsheet.

Note:

Only soil layers that are located not more than *1* m below the ground surface are considered to be *Top* soils.

GSE **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- *Chapter 19*).

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 3D will continuously use the saturated permeabilities for soil layers where a *Saturated* data set was assigned.

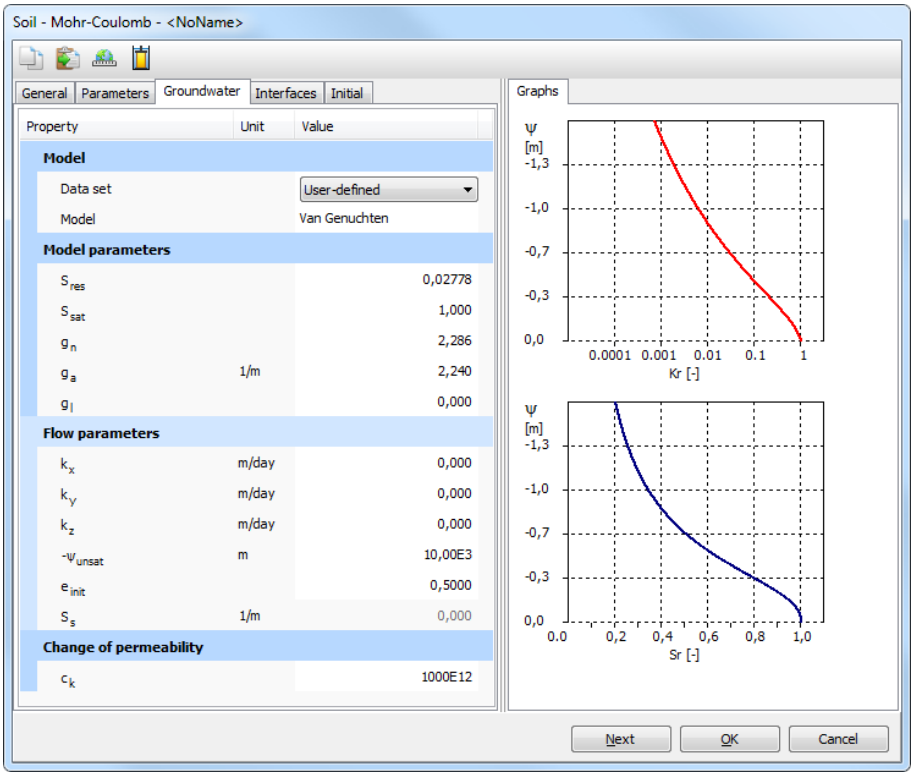


Figure 185: Groundwater parameters for user-defined data set

Saturated permeabilities (k_x, k_y and k_z)

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 consolidation calculations. In such calculations, it is necessary to specify the coefficient of permeability for all drained or undrained clusters, including almost impermeable layers, except for fully impervious layers with a non-porous drainage type. PLAXIS 3D allows for the anisotropic permeability of soils where the anisotropy directions coincide with the principal axes *x*, *y* and *z*.

Note that the input field for permeabilities are greyed out when the *Non-porous* option is selected.

One of the advanced features is to account for the change of permeability during a consolidation analysis. This can be applied by entering a proper value for the change of permeability parameter c_k and the void ratio's e_{init}, e_{min} and e_{max} in the **General** tabsheet of the **Soil** window.

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:

$$d_g = \exp\left(m_{cl}\ln\left(d_{cl}\right) + m_{si}\ln\left(d_{si}\right) + m_{sa}\ln\left(d_{sa}\right)\right) \qquad \textit{Eq. [17]}$$

$$\sigma_g = \exp\left[\sum_{i=1}^{3} m_i \ln\left(d_i\right)^2 - \left(\sum_{i=1}^{3} m_i \ln\left(d_i\right)\right)^2\right]^2 \qquad \textit{Eq. [18]}$$

where m_{cl}, m_{si} and m_{sa} are particle fractions for clay, silt and sand ; d_{cl}, d_{si} and d_{sa} are particle size limits separating clay, silt and sand respectively (i.e. d_{cl} = 2 μm , d_{si} = 50 μm, d_{sa} = 2 mm)

According to Aukenthaler, Brinkgreve & Haxaire (2016) (on page 510) 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:

$$k_x = k_y = k_z = K_{sat}$$

$$K_{sat} = 4 \times 10^{-5}\left(\frac{0.5}{1-\theta_{sat}}\right)^{1.3b} \times \exp\left(-6.88 m_{cl} - 3.63 m_{si} - 0.025\right) \qquad \textit{Eq. [19]}$$

$$b = d_g^{-0.5} + 0.2g$$

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.

The available options from the drop-down menu for *Flow parameters* are:

None	This is the default option, and the user must provide values for permeabilities.
From data set	The values of k_x, $k_y k_z$ are obtained from the selected data set.
From grain size distribution	The values for k_x, $k_y k_z$ are obtained from particle size distribution as explained above. The value for θ_{sat} is calculated internally using the equation, $\theta_{sat} = e / (1+e)$.

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 186 (on page 258)). It is used to limit the relative permeability K_r and degree of saturation for high unsaturated zones.

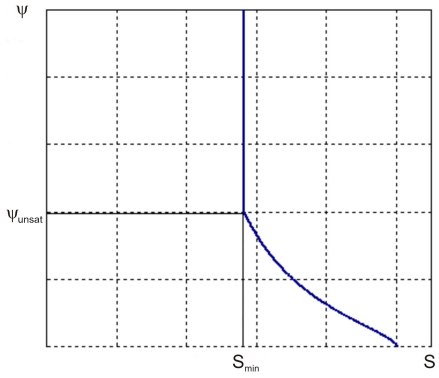


Figure 186: Relative permeability vs. Degree of saturation

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.

Volumetric specific storage (S_s)

The specific storage S_s of a saturated aquifer is defined as the volume of water that a unit volume of aquifer releases from storage under decline in hydraulic head.

For groundwater flow calculations in which there is no deformation, we can assume that:

$$S_s = \gamma_{water} \times \left(\frac{n}{K_w}\right) \qquad \textit{Eq. [20]}$$

Change of permeability (c_k)

This advanced feature is to account for the change of permeability during a consolidation analysis. This can be applied by entering a proper value for the c_k parameter and the void ratio's. On entering a real value, the permeability will change according to the formula:

$$\log\left(\frac{k}{k_0}\right) = \frac{\Delta e}{c_k} \qquad \textit{Eq. [21]}$$

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_z). 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}.

6.1.6 *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 Joined Rock model or 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 187 (on page 259)). 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 theMaterial Models Manual - *Chapter 17 - Input of UD model parameters via user-defined.*

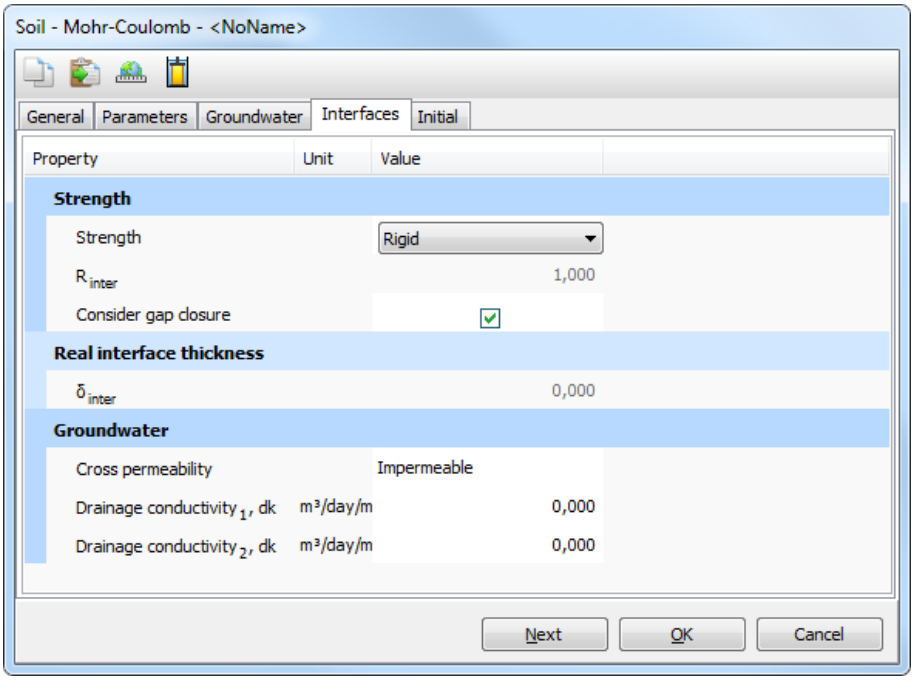


Figure 187: ***Interfaces*** *tabsheet of the* ***Soil*** *window*

Stiffness

The user may assign two distinct material sets to each interface set and use different interface stiffness values to define material properties properly.

Kn/Ks: *Kn/Ks* stiffness option allows inserting K_n and K_s values.

K_n	Elastic interface normal stiffness	kN/m^3
K_s	Elastic interface shear stiffness	kN/m^3

Standard: By default, stiffness is set to *standard*. The shear and compression moduli are related by the expressions:

$$E_{oed,i} = 2G_i \frac{1 - \nu_i}{1 - 2\nu_i}$$

$$G_i = R_{\text{int } er}^2 G_{soil} \leq G_{soil} \qquad \textit{Eq. [22]}$$

$$\nu_i = 0.45$$

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*.

Interface 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 Joined Rock model or the Hoek-Brown 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 109 (on page 150)) 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:

$$Elastic\ gap\ displacement = \frac{\sigma_n}{K_N} = \frac{\sigma_n t_i}{E_{oed,i}} \qquad Eq.\ [23]$$

$$Elastic\ slip\ displacement = \frac{\tau}{K_S} = \frac{\tau t_i}{G_i} \qquad Eq.\ [24]$$

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 147)), 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.

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 strength (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:

$$| \tau | < -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [25]}$$

where σ_n is the effective normal stress.

For plastic behaviour τ is given by:

$$| \tau | = -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [26]}$$

where φ_i and c_i are the friction angle and cohesion of the interface. The strength properties of interfaces are linked to the strength properties of a soil layer. Each data set has an associated strength 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:

$$c_i = R_{\text{inter}} c_{soil}$$
$$\tan(\varphi_i) = R_{\text{inter}} \tan(\varphi_{soil}) \leq \tan(\varphi_{soil}) \qquad \textit{Eq. [27]}$$

$$\psi_i = 0 \text{ for } R_{\text{inter}} < 1, \text{ otherwise } \psi_i = \psi_{soil} \qquad \textit{Eq. [28]}$$

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 Parameters tabsheet (on page 201)), also applies to interfaces (if not deactivated):

$$\sigma_n < \sigma_{t,i} = R_{\text{inter}} \sigma_{t,soil} \qquad \textit{Eq. [29]}$$

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:

$$| \tau | < s_u \qquad \textit{Eq. [30]}$$

where s_u is the undrained shear strength.

For plastic behaviour τ is given by:

$$| \tau | = s_u \qquad \textit{Eq. [31]}$$

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:

$$s_{u,i} = R_{\text{inter}} s_{u,soil} \qquad \textit{Eq. [32]}$$

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 (and UDCAM-S model), $s_{u,soil}$ is equal to s_u^{DSS} (and τ^{DSS}), taking the increase with depth into account. For more information about the NGI-ADP model, reference is made to the *Chapter 13* of 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.

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.

GSE **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:

$$| \tau | \leq -\sigma_n \tan(\varphi_i) + c_i \qquad \textit{Eq. [33]}$$

$$\sigma_n \leq \sigma_{t,i} \qquad \textit{Eq. [34]}$$

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:

$$\sin(\varphi) = \frac{\overline{f}'}{2 + \overline{f}'}$$
$$c = \frac{1 - \sin(\varphi)}{2\cos(\varphi)}\left(\overline{f} + \frac{2\sigma'_3 \sin(\varphi)}{1 - \sin(\varphi)}\right) \qquad \textit{Eq. [35]}$$

where

$$\overline{f} = \sigma_{ci}\left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^a$$
$$\overline{f}' = a m_b \left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^{a-1} \qquad \textit{Eq. [36]}$$

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}:

$$\tan(\varphi_i) = R_{\text{inter}} \tan(\varphi)$$
$$c_i = = R_{\text{inter}} c \qquad \textit{Eq. [37]}$$
$$\sigma_{t,i} = R_{\text{inter}} \sigma_t = R_{\text{inter}} \frac{s\sigma_{ci}}{m_b}$$

For more information about the Hoek-Brown model and an explanation of its parameters, reference is made to *Chapter 4* of the Material Models Manual.

Interfaces using the Modified Cam-Clay model If the Modified Cam-Clay model is selected in the **Parameters** 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:

$$Elastic\ gap\ displacement = \frac{\sigma_n}{K_N} = \frac{\sigma_n t_i}{E_{oed,i}} \qquad Eq.\ [38]$$

$$Elastic\ slip\ displacement = \frac{\tau}{K_S} = \frac{\tau t_i}{G_i} \qquad Eq.\ [39]$$

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 147)). 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:

$$E_{oed,i} = \frac{3}{\lambda}\frac{(1-\nu_i)}{(1+\nu_i)}\frac{\sigma_n}{(1+e_0)}$$

$$G_i = \frac{3(1-2\nu_i)}{2(1+\nu_i)}\frac{\sigma_n}{\lambda(1+e_0)} \qquad Eq.\ [40]$$

$$\nu_i = 0.45$$

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. 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

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 across the element as well as flow in the interface longitudinal direction. Therefore, distinction is made in *Cross permeability*, defined by means of the *Hydraulic resistance*, and *Parallel permeability*, defined by means of the *Drainage conductivity*.

The cross permeability can be set using the following options:

Impermeable: 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 model explorer. 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 model explorer. The output program will show the amount of groundwater flow q_n crossing the interface.

Fully permeable: 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 model explorer. Note that there is no output of groundwater flow available when choosing this option.

Hydraulic resistance and Drainage conductivity: The hydraulic conductivity or resistance of interfaces is not only defined by permeability, but also by the 'thickness' of the structure or the soil-structure interaction zone.

However, this 'thickness' is not always a well-defined quantity. Therefore, the hydraulic properties of interfaces are defined by two quantities that have a well-defined meaning:

- Hydraulic resistance: to define the hydraulic conductivity across the interface or structure
- Drainage conductivity: to define the hydraulic conductivity in the interface longitudinal direction

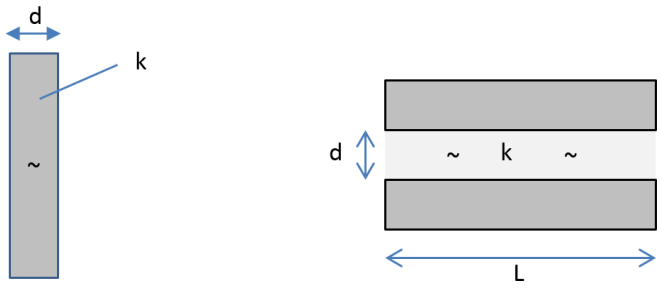


Figure 188: Hydraulic and drainage conductivity

Hydraulic resistance: The hydraulic resistance defines the permeability across the interface (perpendicular to the interface longitudinal direction).

Considering a semi-permeable wall with a thickness d and permeability k, the hydraulic resistance is defined by d/k, expressed in the unit of time. Considering Darcy's law, $q = k\,\Delta\varphi/dl$, where k is the cross permeability and $\Delta\varphi/\Delta\, l$ is the gradient of the groundwater head across the wall, which is the difference between the groundwater head left and right of the wall over the wall thickness ($\Delta\varphi / d$). Hence, for a given hydraulic resistance d/k, the specific discharge $q = k\,\Delta\varphi /d = \Delta\varphi /(d/k)$. To determine d/k, one needs to measure the average discharge q through a wall (per unit of area) for a given head difference $\Delta\varphi$, so $d/k = \Delta\varphi/q$.

Note that, for the hydraulic resistance, the actual thickness d, and the permeability, k, do not really matter. The hydraulic resistance represents, practically, the ratio of the thickness of the wall for the cross permeability.

Drainage conductivity: 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 model explorer. 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. Considering Darcy's law, as listed above, the gradient is now defined by the difference in groundwater head over the length of the gap (in longitudinal direction) divided by the gap length, such that $q = k\,\Delta\varphi / L$. The total amount of water Q is q times the thickness d times the unit width b, where $b = 1$ length unit for a plane strain application, so $Q/b = dk\,\Delta\varphi / L$. 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 $\Delta\varphi$ and a given length of the gap L, such that $dk = (Q/b)\,(L/\Delta\varphi)$.

Note that, for the drainage conductivity, the actual thickness, d, and permeability, k, do not really matter. The drainage conductivity represents, practically, the product of the thickness of the wall and the cross permeability.

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.

Interfaces below or around corners of structures

When interfaces are extended below or around corners of structures to avoid stress oscillations (Interfaces (on page 147)), 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 done by right-clicking the interface in the drawing area and selecting the corresponding soil data set from the *Set material* options in the right mouse button menu.

6.1.7 *Initial tabsheet*

The **Initial** tabsheet contains parameters to generate the initial stresses (*K0 procedure*) (Figure 189 (on page 265)).

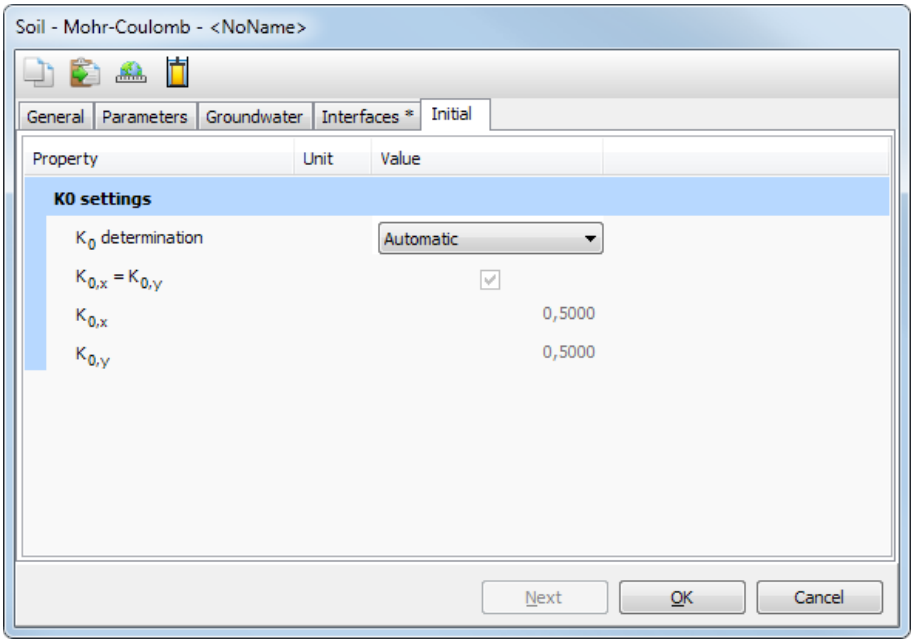


*Figure 189: **Soil** window (**Initial** tabsheet of the Mohr-Coulomb model)*

The K_0-values can be defined automatically by selecting the option *Automatic* in the *K_0 determination* drop-down menu or manually by selecting the option *Manual*.

K_0-values

In general, two K_0-values can be specified, one for the *x*-direction and one for the *y*-direction:

$$K_{0,x} = \sigma'_{xx} / \sigma'_{zz} \quad K_{0,y} = \sigma'_{yy} / \sigma'_{zz} \qquad Eq.\ [41]$$

The checkbox can be used to set the $K_{0,y}$ value equal to the $K_{0,x}$ value.

The default K_0-values are then in principle based on Jaky's formula:

$$K_0 = 1 - \sin(\varphi) \qquad Eq.\ [42]$$

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:

$$K_{0,x} = K_{0,y} = K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1) + \frac{K_0^{nc} POP - \frac{\nu_{ur}}{1-\nu_{ur}} POP}{\sigma'^{0}_{zz}} \qquad Eq.\ [43]$$

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 190 (on page 266)).

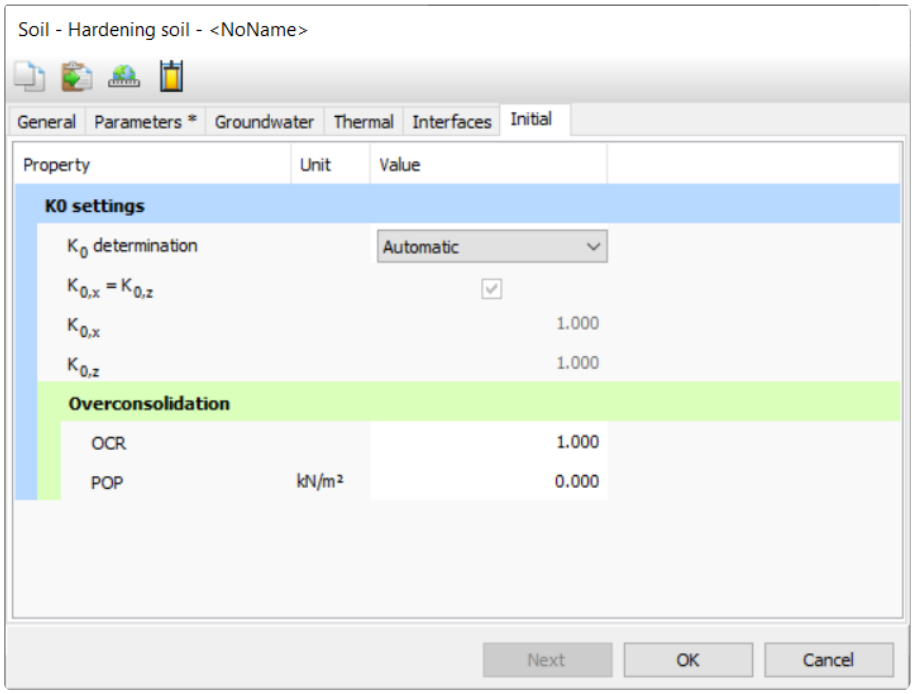


*Figure 190: **Soil** window (**Initial** tabsheet of the Hardening Soil model*

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:

$$\frac{1-\sin(\varphi)}{1+\sin(\varphi)} < K_0 < \frac{1+\sin(\varphi)}{1-\sin(\varphi)} \qquad Eq.\ [44]$$

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) 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 3D 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 191 (on page 267)), and the in-situ effective vertical stress, σ'^{0}_{zz}.

$$OCR = \frac{\sigma_p}{\sigma'^{0}_{zz}} \qquad Eq.\ [45]$$

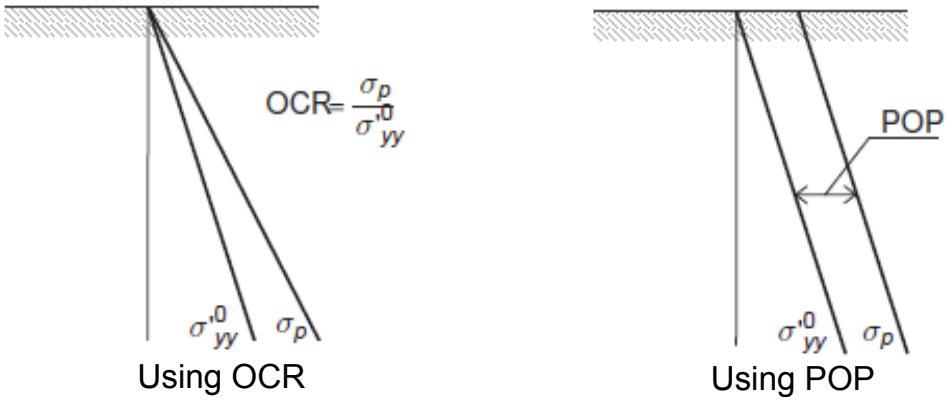


Figure 191: Illustration of vertical preconsolidation stress in relation to the in-situ vertical effective stress

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:

$$POP = \left| \sigma_p - \sigma'^{0}_{zz} \right| \qquad Eq.\ [46]$$

These two ways of specifying the vertical preconsolidation stress are illustrated in Figure 191 (on page 267).

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:

$$\begin{aligned} \sigma'_{xx,c} &= K_0^{nc} OCR \sigma'_{yy} \\ \sigma'_{yy,c} &= OCR \sigma'_{yy} \\ \sigma'_{zz,c} &= K_0^{nc} OCR \sigma'_{yy} \\ \sigma'_{xy,c} &= \sigma'_{xy} \end{aligned} \qquad Eq.\ [47]$$

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-sinφ*). 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 14: 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	$K_0^{nc} OCR - \frac{\nu_{ur}}{1-\nu_{ur}}(OCR - 1)$
Joined 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 3D 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 193)). More details about these methods are in the Material Models Manual - *Chapter 2.*

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 **Flow parameters** 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 3D 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 3D. 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 3D 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 249)).

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 526)). 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 527) 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 534)). 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 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 192 (on page 273)). Alternatively, the *SoilTest* option can be reached from the **Soil** dialog.

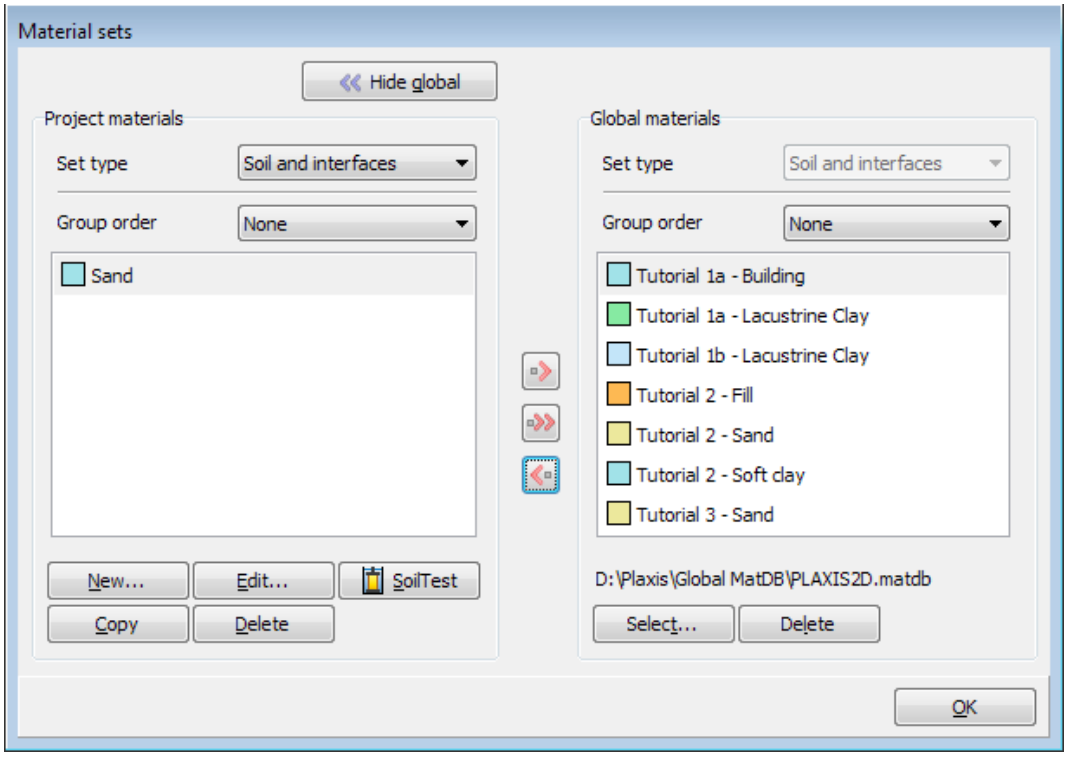


*Figure 192: **Material sets** window showing the project and the global database*

Once the *SoilTest* option has been selected, a separate window will open (Figure 193 (on page 274)). This window contains a menu, a toolbar and several smaller sections. The various items are described in more detail below.

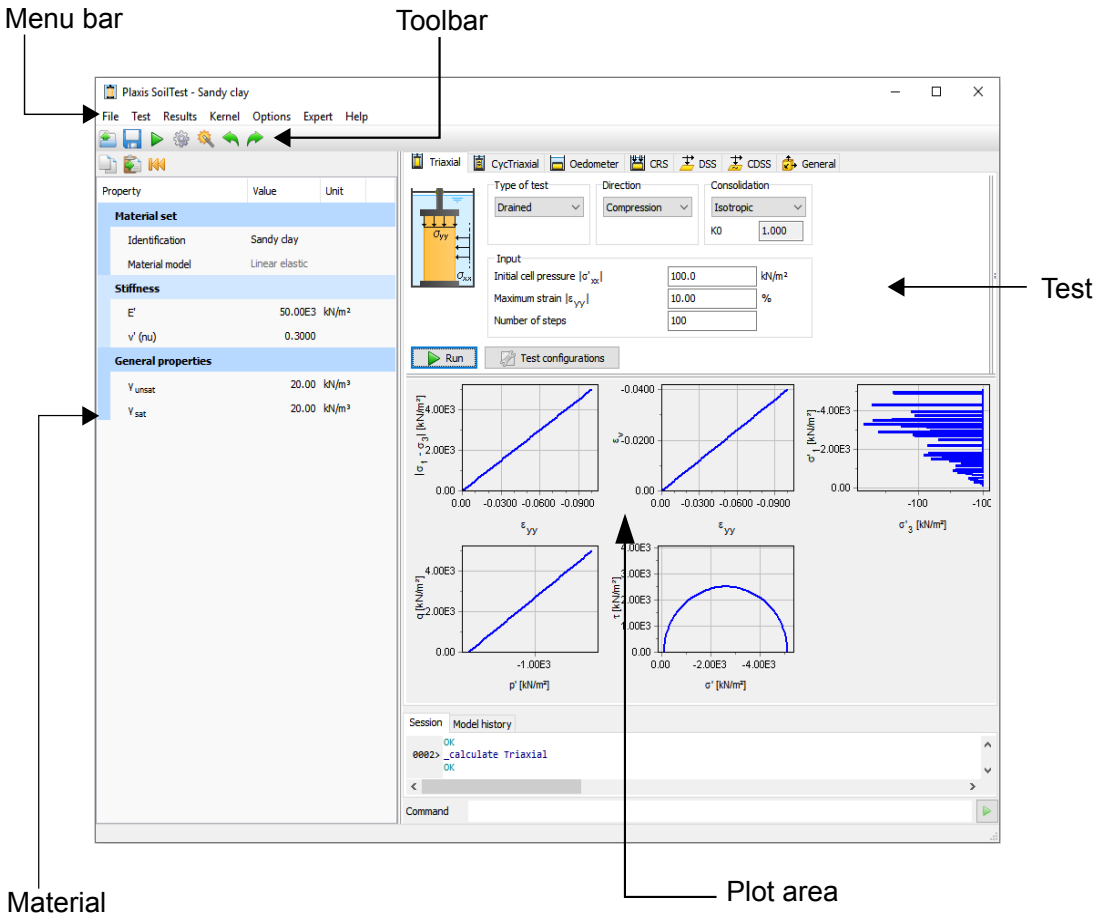


*Figure 193: **SoilTest** window showing drained triaxial test input*

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 285)). 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 194 (on page 275)).

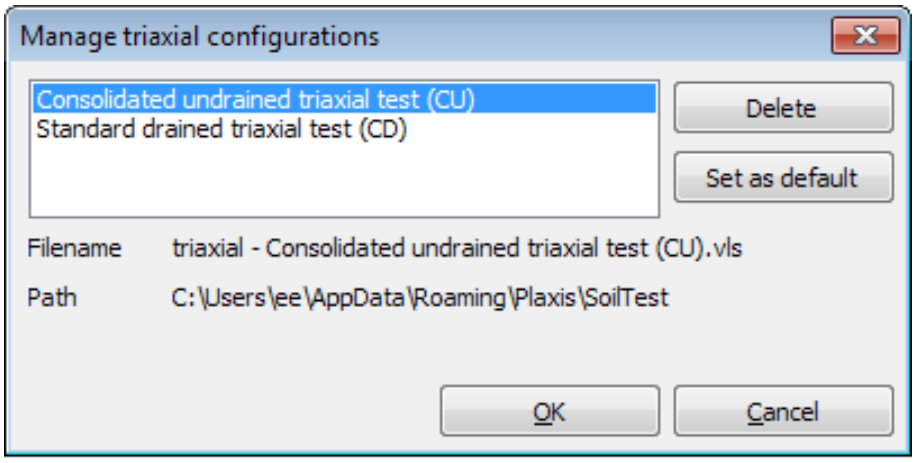


Figure 194: ***Manage configurations*** *window for triaxial tests*

   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.

6.3.1 *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 269)), 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 - *Chapter 2- 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.

6.3.2 *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 269)), 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 - *Chapter 2- 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 $\lvert\sigma'_{xx}\rvert$	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
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 195 (on page 279)). 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 195 (on page 279)), 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.

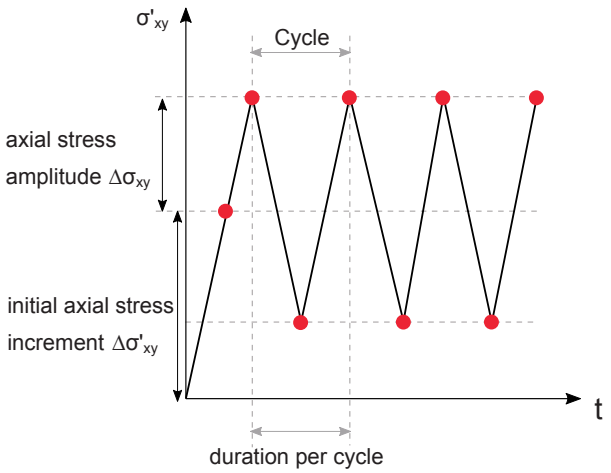


Figure 195: Cyclic triaxial test conditions

6.3.3 *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 - *Chapter 2- 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.

6.3.4 *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 - *Chapter 2- 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.

6.3.5 *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 269)), 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 - *Chapter 2- 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.

6.3.6 *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 269)), 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 - *Chapter 2- 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 196 (on page 283) (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 196 (on page 283)).
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 $\Delta\sigma_{xy} = CSR\ \sigma_{yy}$.

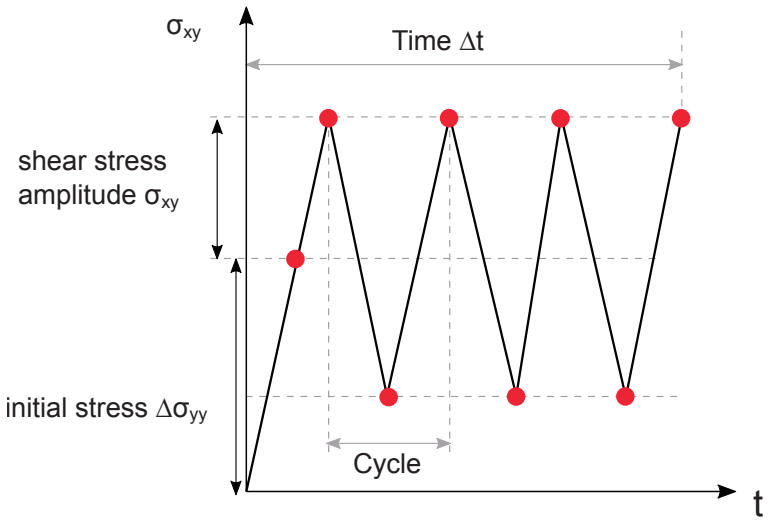


Figure 196: Cyclic DSS test conditions

Note:

Note that cycle boundaries and phase boundaries do not match except when $t = 0$ and $t = \Delta t$

6.3.7 *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 - *Chapter 2 - 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.

6.3.8 *Results*

The **Results** window shows several predefined typical diagrams to display the results of the current test in terms of stress and strain quantities. Double-clicking one of the graphs opens the selected diagram in a larger window (Figure 197 (on page 285)). 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.

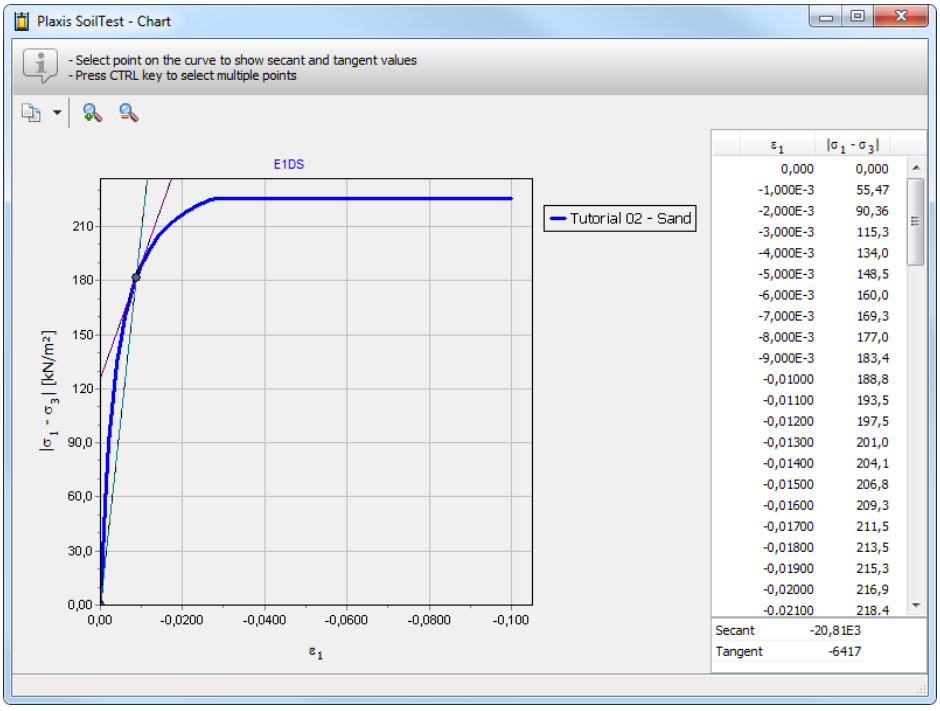


Figure 197: Results diagram

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).

6.3.9 *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 198 (on page 286)). The first tab (*Select parameters*) is active.

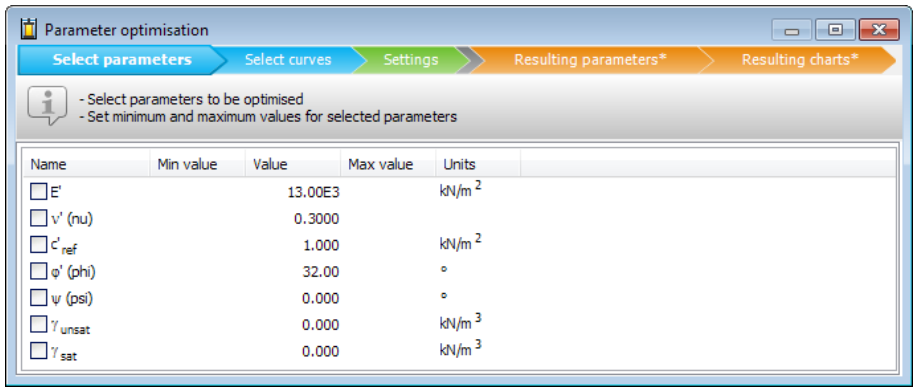


Figure 198: ***Parameter optimisation*** *window*

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 199 (on page 287)). 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.

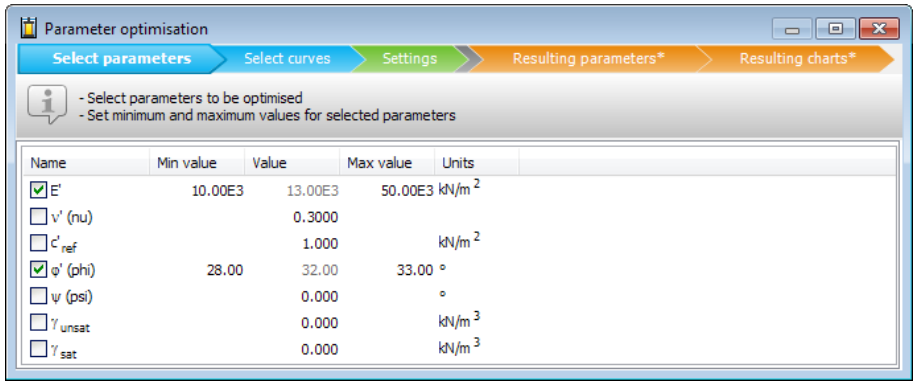


*Figure 199: Selection of the parameters in the **Select parameters** tabsheet*

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 3D 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 200 (on page 287)).

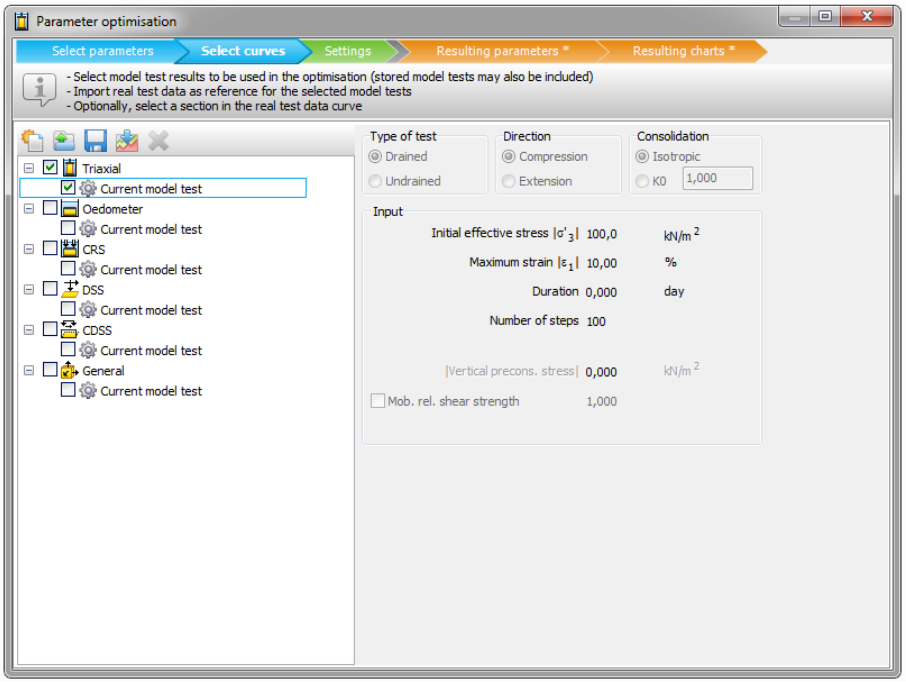


*Figure 200: Selection of the test curves in the **Select curves** tabsheet*

- 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 201 (on page 288)).
- 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 3Dsoil test project (<test>.vlt).

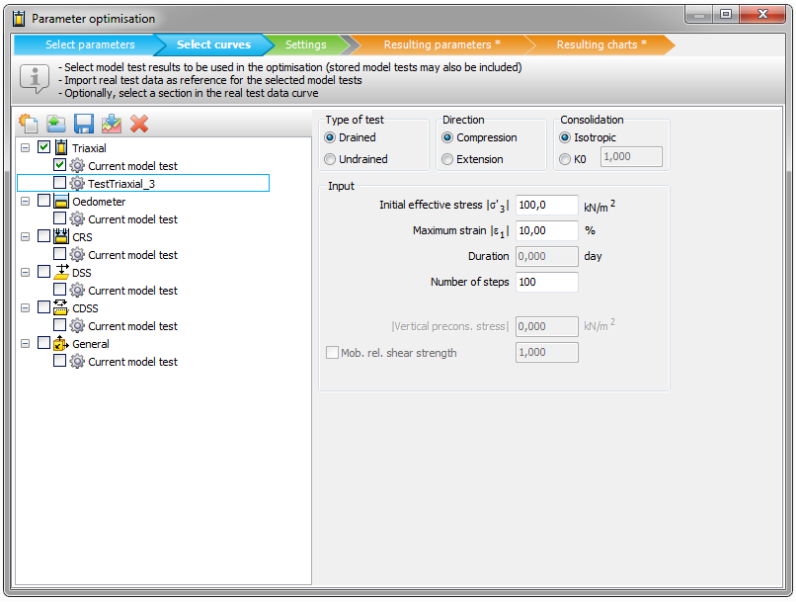


*Figure 201: Custom test definition in the **Select curves** tabsheet*

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 202 (on page 289)). 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.

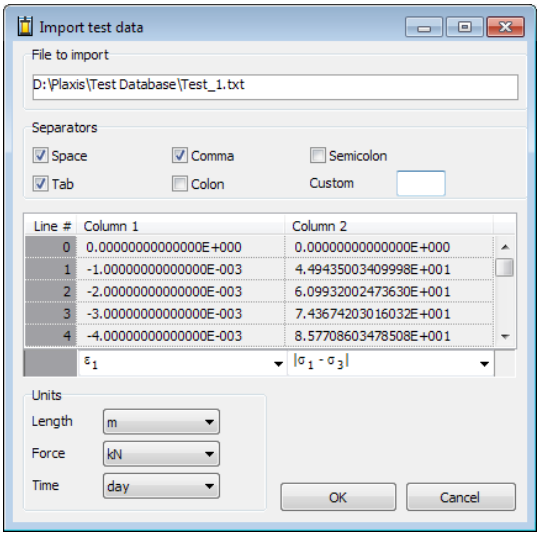


*Figure 202: **Import test data** window*

- 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 203 (on page 289)).

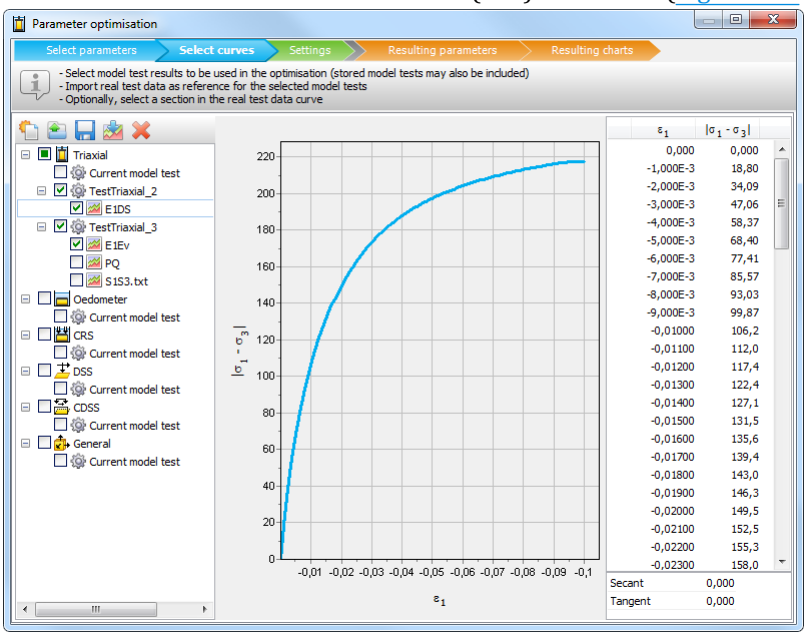


Figure 203: Display of the imported test curves

- 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 204 (on page 290)). 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.

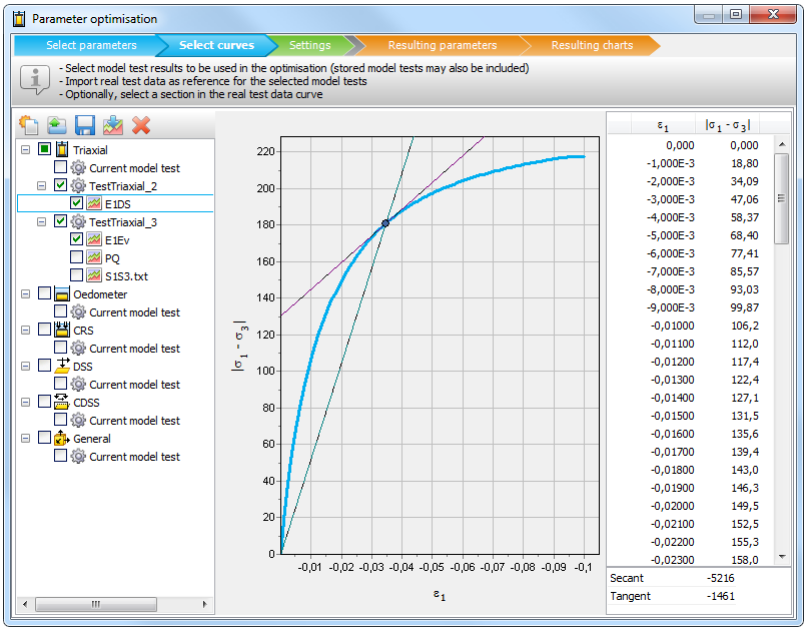


Figure 204: Importing data from SoilTest

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 205 (on page 291)). 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.

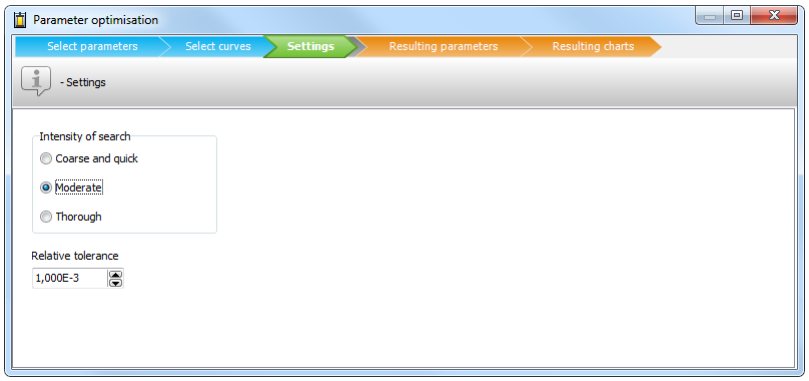


Figure 205: Settings window

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 206 (on page 291)). 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.

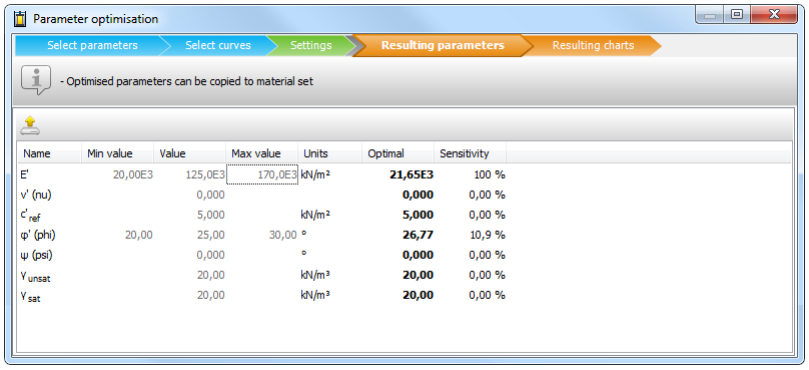


Figure 206: Resulting parameters window

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 207 (on page 292)).

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.

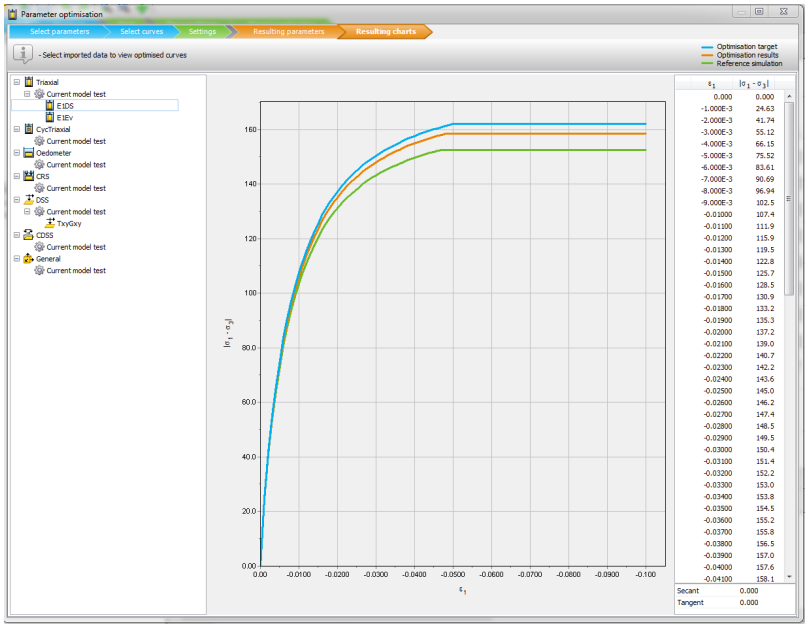


Figure 207: Resulting charts window

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.

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. 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.

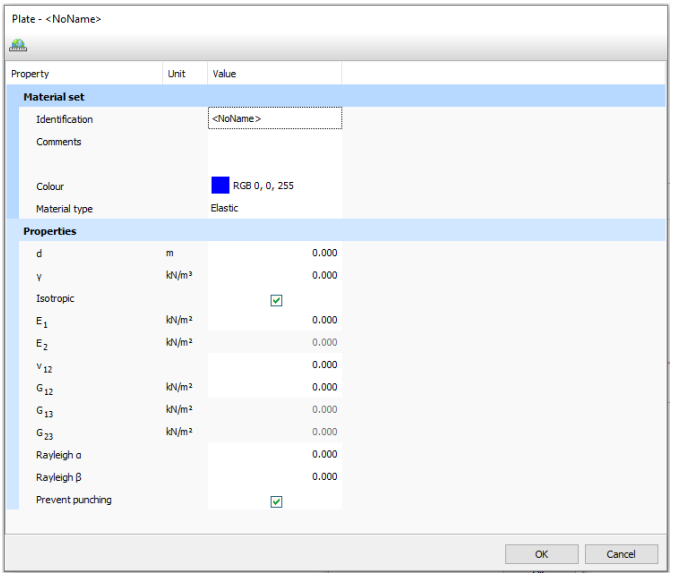


*Figure 208: **Plate** window*

6.4.1 *Material set*

Several data sets may be created to distinguish between different types of plates. Figure 208 (on page 293) shows the **Plate** 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.

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 three available options, describing the material type of a plate. These options are *Elastic* and *Elastoplastic*. The availability of the parameters defined in the *Properties* box depends on the selected material type.

6.4.2 *Mechanical properties*

The properties required for plates can be grouped into general properties and stiffness properties for *Elastic* plates. If the *Elastoplastic* type is chosen, the plastic properties are also required.

General properties

A plate has two general properties:

d	The (equivalent) thickness (in the unit of length) is the material cross section area of the plate across its major axial direction per 1 m width. For massive plates without a particular profile this is just the plate thickness, but for plates that have a certain profile (such as sheet-pile walls or sandwich plates), the thickness is relatively small and should be properly calculated from the above definition.
γ	The unit weight is the unit weight of the material from which the plate is composed. The product $\gamma \cdot d$ determines the distributed weight of the plate.

Note:

In a finite element model, plates are superimposed on a continuum and therefore 'overlap' the soil. Especially for massive 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 plate material.

Stiffness properties

Plate stiffnesses will be linear. PLAXIS 3D allows for orthotropic as well as anisotropic material behaviour in plates, which is defined by the following parameters:

E_1	Young's modulus in first axial direction.
E_2	Young's modulus in second axial direction.
G_{12}	In-plane shear modulus.
G_{13}	Out-of-plane shear modulus related to shear deformation over first direction.
G_{23}	ut-of-plane shear modulus related to shear deformation over second direction.
ν_{12}	Poisson's ratio.

These parameters appear in the following (approximate) relationships for structural forces:

$$\begin{bmatrix} N_1 \\ N_2 \end{bmatrix} = \begin{bmatrix} E_1 d & \nu_{12} E_2 d \\ \nu_{12} E_2 d & E_2 d \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \end{bmatrix} \qquad \textit{Eq. [48]}$$

$$\begin{bmatrix} Q_{12} \\ Q_{13} \\ Q_{23} \end{bmatrix} = \begin{bmatrix} G_{12} d & 0 & 0 \\ 0 & k G_{13} d & 0 \\ 0 & 0 & k G_{23} d \end{bmatrix} \begin{bmatrix} \gamma_{12} \\ \gamma_{13}^* \\ \gamma_{23}^* \end{bmatrix} \qquad \textit{Eq. [49]}$$

$$\begin{bmatrix} M_{11} \\ M_{22} \\ M_{12} \end{bmatrix} = \begin{bmatrix} \frac{E_1 d^3}{12} & \frac{\nu_{12} E_2 d^3}{12} & 0 \\ \frac{\nu_{12} E_2 d^3}{12} & \frac{E_2 d^3}{12} & 0 \\ 0 & 0 & \frac{G_{12} d^3}{12} \end{bmatrix} \begin{bmatrix} \kappa_{11} \\ \kappa_{22} \\ \kappa_{12} \end{bmatrix} \qquad \textit{Eq. [50]}$$

Where *k* represents the modification factor to the shear modulus and is taken equal to 5/6.

Figure 209 (on page 296) visualises the plate's local system of axes and the major quantities. The local system of axes in a plate element is such that the first and the second local axis lie in the plane of the plate whereas the third axis is perpendicular to the plane of the plate.

If the *Isotropic* option is checked the input is limited to E_1 and ν_{12}, where as $E_2 = E_1$ and $G_{12} = G_{13} = G_{23} = E / 2(1 + \nu_{12})$.

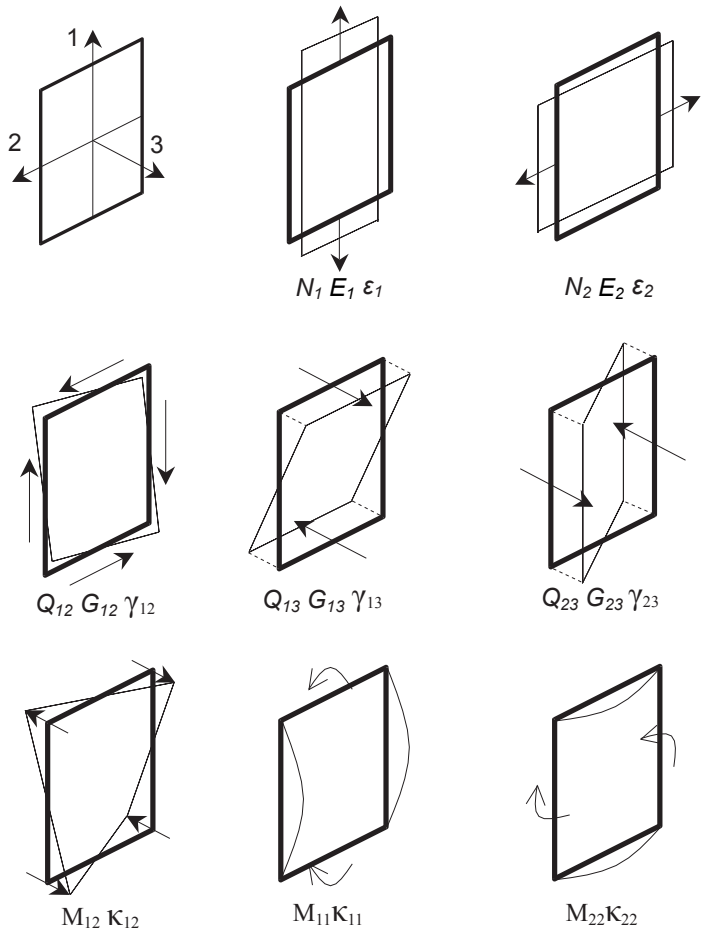


Figure 209: Definition of positive normal forces (N), shear forces (Q) and bending moments (M) for a plate based on local system of axes

Strength properties: Elastoplastic

Elastoplastic plates are defined by the following parameters:

Yield stress $\sigma_{y,11}$	Yield stress parameter in first axial direction.
Yield stress $\sigma_{y,22}$	Yield stress parameter in second axial direction.
W_{11}	Plastic section modulus in first axial direction.
W_{22}	Plastic section modulus in second axial direction.

If the *Isotropic* box is checked, then $\sigma_{y,11} = \sigma_{y,22}$ and $W_{11} = W_{22}$.

In order to obtain the plastic axial forces N_p, the equivalent yield stresses are calculated:

$$\sigma_{y,11,eq} = \sigma_{y,11}/\sqrt{3} \qquad \textit{Eq. [51]}$$

$$\sigma_{y,22,eq} = \sigma_{y,22}/\sqrt{3} \qquad \textit{Eq. [52]}$$

The plastic properties are calculated as follows:

$$N_{p,1,eq} = d \quad \sigma_{y,11,eq} \qquad Eq$$

$$N_{p,2,eq} = d \quad \sigma_{y,22,eq} \qquad Eq$$

$$M_{p,11} = W_{11} \quad \sigma_{y,11} \qquad Eq$$

$$M_{p,22} = W_{22} \quad \sigma_{y,22} \qquad Eq$$

The resulting yiel surfaces are a diamond shape in the N_1-M_{11} plane and in the N_2 - M_{22} plane. The two directions are treated separately.

Note:

1. If the plate is primarily subjected to bending moments (or it is expected to fail in bending), the plastic bending moment M_p is the most important parameter and it should be properly evaluated using the real σ_y and W values as input parameters. In this case, the plastic axial force N_p is calculated based on the equivalent yield stress $\sigma_{y,eq}$ and will not match the real N_p of the plate: $N \,/\, N_{p,eq} = (N \,/\, Np) \,/\, (\sqrt{3}d^{\,2}) \,/\, (6W)$ (Figure 210 (on page 297)).
2. If the plate is primarily subjected to axial forces (or it is expected to fail in tension/compression), the plastic axial force N_p should be properly evaluated by scaling the yield stresses such that $N_{p,eq}$ will be equal to the real N_p of the plate cross section. In this case, the plastic bending moment resulting from this operation $M_{p,eq}$ will not match the real plastic bending moment M_p of the plate: $M \,/\, M_{p,eq} = (M \,/\, Mp) \,/\, (\sqrt{3}d^{\,2}) \,/\, (6W)$ (Figure 6.59).

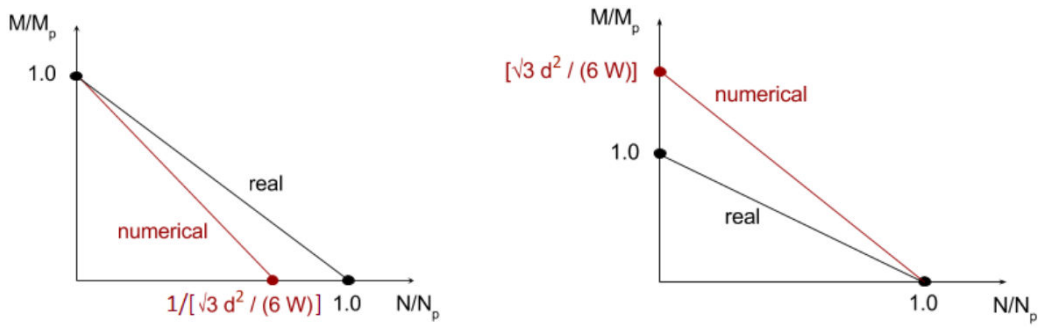


Figure 210: Plate primarily subjected to: bending moments (on the left) and axial forces (on the right)

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 corresponding 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 $D_{eq} = \sqrt{12EI \,/\, EA}$.

It is not the idea that this end bearing represents the true end bearing capacity of the wall. It just presents 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 theoritically determined by PLAXIS 3D.

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 end bearing option shall not be used for sheet pile walls. It is always advised to check the introduced tip-resistance does not exceed the expected end bearing, especially for cases where the equivalent plate thickness does not represent the true physical thickness.

More information about the behaviour and structural forces in plates can be found in the Material Models Manual - *Chapter 18 - 3D plates.*

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 elements that represent a grid or sheet of fabric. Geogrids cannot sustain 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 *Material set*

Several data sets may be created to distinguish between different types of geogrids or geosynthetics. Figure 211 (on page 299) shows the **Geogrid** 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.
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 in the *Properties* box depends on the selected material type.

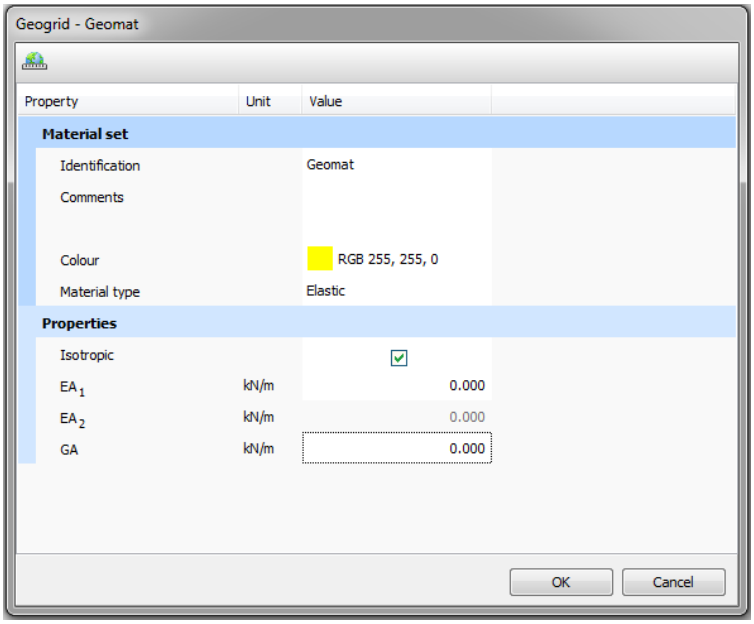


*Figure 211: **Geogrid** window*

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 stiffnesses in-plane and out-of-plane may be considered. The latter is most relevant for 3D or axisymmetric models when modelling geogrids with an anisotropic pattern. If this is not the case, the *Isotropic* option may be selected to ensure that both stiffness are equal.

Stiffness properties

For elastic behaviour, the axial stiffness *EA* should be specified. PLAXIS 3D allows for orthotropic as well as anisotropic material behaviour in geogrids, which is defined by the following parameters:

EA_1	The normal elastic stiffness in 1-direction (in-plane).
EA_2	The normal elastic stiffness in 2-direction.
GA	In-plane shear stiffness (anisotropic behaviour).

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):

$$EA = \frac{F}{\Delta l / l} \qquad Eq.\ [53]$$

If the *Isotropic* option is checked the input is limited to EA_1 where as $EA_2 = EA_1$ and $GA = EA_1 / 2$.

Strength properties: Elastoplastic

Strength parameters are required in case of elastoplasticity:

$N_{p,1}$	The maximum force in 1-direction.
$N_{p,2}$	The maximum force in 2-direction.

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 $N_{p,2} = N_{p,1}$.

Strength properties: Elastoplastic (*N*-ε)

A non-linear *N*-ε diagram may be specified in case of elastoplasticity (*N*-ε):

N_1-eps_1	The *N*-ε diagram in 1-direction.
N_2-eps_2	The *N*-ε diagram in 2-direction.

The user can add both *N*-ε diagrams, by specifying values as a table or importing a text file containing values (Figure 212 (on page 301)). 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.

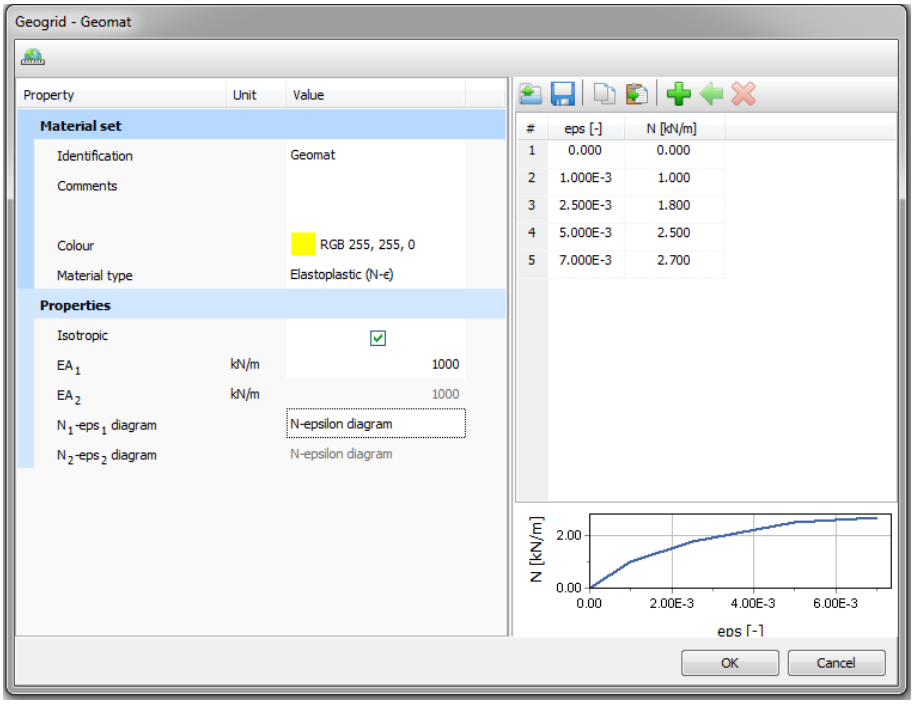


Figure 212: Geogrid window

Time-dependent properties: 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.

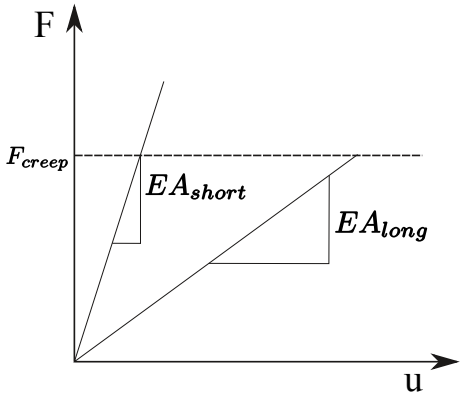


Figure 213: Force versus displacement showing EA_{short} and EA_{long}

Considering one *Kelvin-Voigt* element (see Figure 214 (on page 302)),

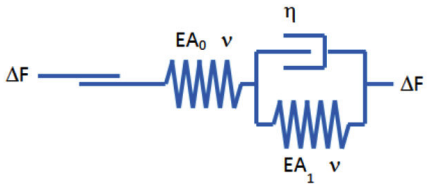


Figure 214: Kelvin-Voigt single element representation

$$EA_0 = EA_{short}$$

$$\frac{1}{EA_0} + \frac{1}{EA_1} = \frac{1}{EA_{long}}$$

$$EA_1 = \frac{1}{\left(\frac{1}{EA_{long}} - \frac{1}{EA_0}\right)} \qquad \textit{Eq. [54]}$$

$$\eta = EA_1 \times \text{Retardation time}$$

where η is viscous damping, EA_0 and EA_1 are internal stiffness used in the geogrid material model.

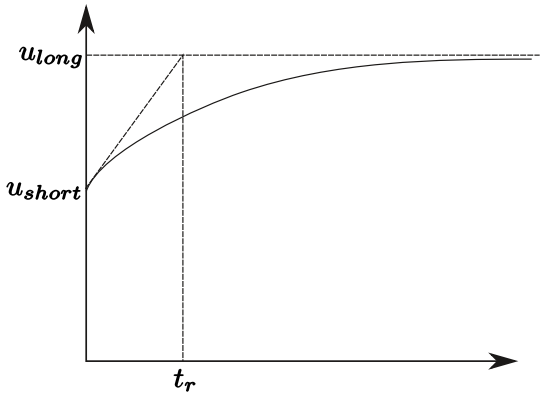


Figure 215: Displacement versus time in a Creep test

Retardation time can be obtained from a creep test in which a force F is applied, using the following equations (see Figure 215 (on page 302)):

$$t_{\text{retardation}} = \frac{\eta_1}{E_1}$$

$$EA_{short} = \frac{F}{u_{short}}$$

$$EA_{long} = \frac{1}{EA_{short}} \qquad \textit{Eq. [55]}$$

$$u_{short} = \frac{F}{EA_{short}}$$

$$u_{long} = \frac{F}{EA_{long}}$$

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 216 (on page 303)):

$EA_{1,short}$	Elastic stiffness during initial (instantaneous) strain increment in 1-direction.
$EA_{2,short}$	Elastic stiffness during initial (instantaneous) strain increment in 2-direction.
$EA_{1,long}$	Elastic stiffness during (infinitely) long strain increment in 1-direction.
$EA_{2,long}$	Elastic stiffness during (infinitely) long strain increment in 1-direction.
$N_{p,1}$	The maximum force in 1-direction.
$N_{p,2}$	The maximum force in 1-direction.
Retardation time	The time where a linear extrapolation of the initial creep rate intersects the long-term displacement line.

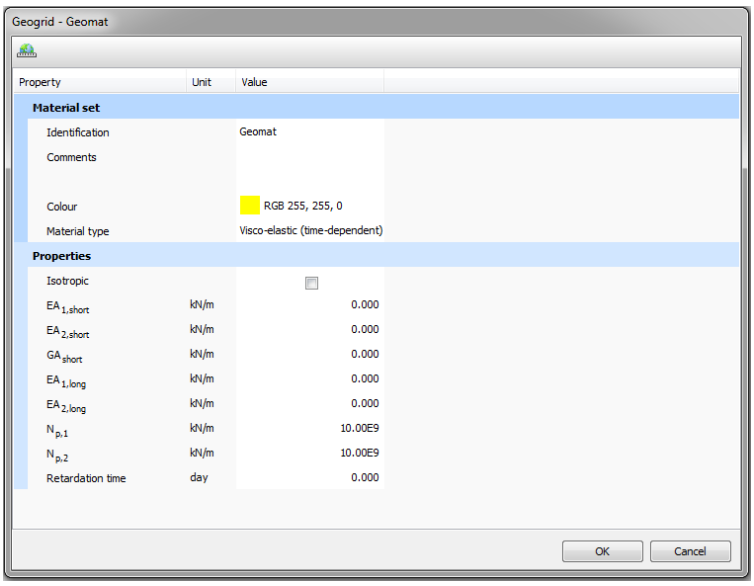


Figure 216: Geogrid window

6.6 Material data sets for beams

In addition to material data sets for soil and interfaces, the material properties and model parameters for beams are also entered in separate material data sets. A data set for beams generally represents a certain type of material or beam profile, and can be assigned to the corresponding (group of) beam elements in the geometry model.

6.6.1 *Material set*

Several data sets may be created to distinguish between different types of beams. Figure 217 (on page 304) shows the **Beam** 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.
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 two available options, describing the material type for beams. These options are *Elastic* and *Elastoplastic*. The availability of the parameters defined in the *Properties* box depends on the selected material type.

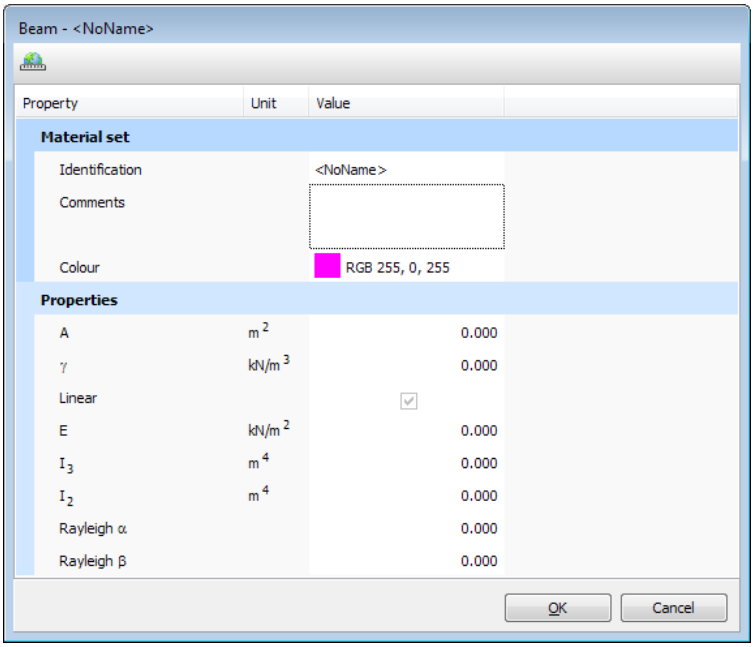


Figure 217: Beam window

6.6.2 *Beam properties*

The properties required for beams are:

E	Young's modulus.
γ	Unit weight of the beam material.

Note:

- In a finite element model, beams 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 beam material. For partially overlapping beams the reduction of the unit weight should be proportional.
- Please note that when reducing the unit weight, the axial forces in the beam may not be realistic.

Geometric properties

A beam requires several geometric parameters used to calculate additional properties:

Beam type	Either a *Predefined* or a *user-defined* type can be selected.
Predefined beam type	A list of predefined types is available (*Solid circular beam*, *Circular tube*, *Solid rectangular beam*). After entering the required data, the parameters A, I_3, I_2 and W_3 (or W_2) are filled automatically (Table 15 (on page 306)) accordingly to the material type.
Diameter	The beam diameter is to be defined for *Solidolid circular beam* and *Circular tube* predefined beam types. The beam diameter determines the size of the elastic zone in the soil around the beam in which plastic soil behaviour is excluded.
Height	The beam height is to be defined for a *Solid rectangular beam* predefined beam type. The beam height is recalculated into an equivalent diameter.
Width	The beam width is to be defined for a *Solid rectangular beam* predefined beam type. The beam width is recalculated into an equivalent diameter.
Thickness	The wall thickness needs to be defined for a *Circular tube* predefined beam type.

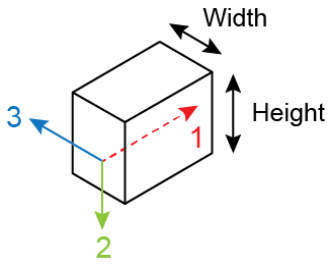


Figure 218: Height is measured in local direction 2 and Width in local direction 3, for a Solid rectangular beam

Alternatively, a user-defined type may be defined by means of the beam cross section area, A, and its respective moments of inertia I_3 and I_2:

A	The cross section area is the actual area (in the unit of length squared) perpendicular to the beam axis. For beams that have a certain profile (such as steel beams), the cross section area can be found in tables that are provided by steel factories.
I_3	Moment of inertia against bending around the third axis.

I_2	Moment of inertia against bending around the second axis.
J	Torsion moment of inertia (bending around the first axis). The torsion moment of inertia, *J*, is calculated from the formulas below. For Solid rectangular cross sections where *a* is half the width of the cross section and *b* is half the height of the cross section: $J = \frac{1}{12}\frac{\left(\frac{a}{b}\right)^2 + 1ab^3}{1.2}$ *Eq. [56]* For Solid circular cross sections where *D* is the cross section diameter: $J = \frac{1}{32}\pi D^4$ *Eq. [57]* For user-defined cross sections where *A* is the cross section area: $J = \frac{1}{2}\pi^3 A^2$ *Eq. [58]* Torsion also involves the shear modulus G, which is assumed to be half the Young's modulus E, i.e. *G = E / 2*

Note: The torsion moment of inertia, *J*, is implicitly calculated for the PLAXIS kernel.

Strength properties: Elastoplastic

Yield stress σ_y	Yield stress parameter for a beam
Yield direction	Yield direction for a beam which can be either in *Local direction 2* or *Local direction 3*.
W_2	Plastic section modulus when the *Yield direction* is in local direction 2.
W_3	Plastic section modulus when the *Yield direction* is in local direction 3.

Note:

The yield stress parameter σ_y makes no distinction between compression and extension.

Table 15: Formulas used for predefined beam types

Solid circular beam	A	$\pi \times diameter^2/4$
	I_3	$\pi \times diameter^4/64$
	I_2	$\pi \times diameter^4/64$
	W_3	$\pi \times diameter^3/32$
	W_2	$\pi \times diameter^3/32$

Circular tube	A	$\pi \times ((diameter)^2 - (diameter-2\times thickness)^2)/4$
	I_3	$\pi \times ((diameter)^4 - (diameter-2\times thickness)^4)/64$
	I_2	$\pi \times ((diameter)^4 - (diameter-2\times thickness)^4)/64$
	W_3	$\pi \times ((diameter)^4 - (diameter-2\times thickness)^4)/(32\times diameter)$
	W_2	$\pi \times ((diameter)^4 - (diameter-2\times thickness)^4)/(32\times diameter)$
Solid rectangular beam	A	$width \times height$
	I_3	$width \times height^3/12$
	I_2	$height \times width^3/12$
	W_3	$width \times height^2/6$
	W_2	$height \times width^2/6$

The definition of various quantities according to the beam's local system of axes are visualised in Figure 219 (on page 307) .

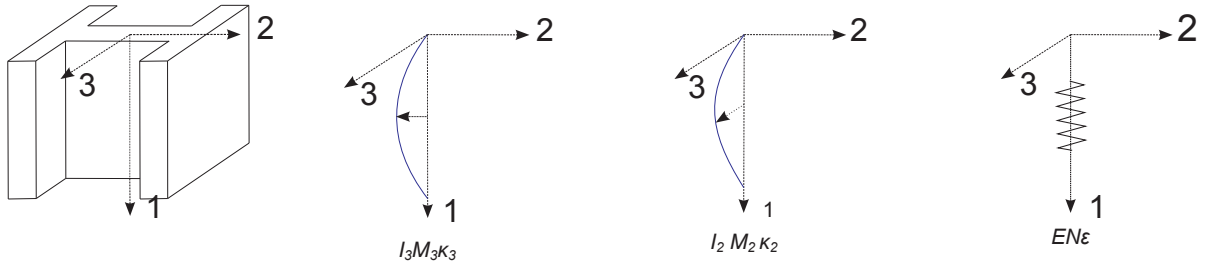


Figure 219: Definition of moment of inertia (I), positive bending moment (M), positive curvature (κ) and stiffness (E) for a horizontal beam based on local system of axes

Based on the input parameters A, I_2, I_3 and σ_y, the equivalent properties are calculated:

$$h_{2,eq} = \sqrt{12\frac{I_3}{A}}$$
$$h_{3,eq} = \sqrt{12\frac{I_2}{A}} \qquad \text{Eq. [59]}$$
$$A_{eq} = h_{2,eq} h_{3,eq}$$

$$W_{2,eq} = \frac{\sqrt{3}}{6} A_{eq} h_{2,eq}$$
$$W_{3,eq} = \frac{\sqrt{3}}{6} A_{eq} h_{3,eq} \qquad \text{Eq. [60]}$$

$$\sigma_{y,eq} = \sigma_y \frac{W_2}{W_{2,eq}} \quad \text{If local direction 2 is selected} \qquad \textit{Eq. [61]}$$

$$\sigma_{y,eq} = \sigma_y \frac{W_3}{W_{3,eq}} \quad \text{If local direction 3 is selected} \qquad \textit{Eq. [62]}$$

Based on the equivalent parameters, the plastic properties are calculated:

$$\begin{aligned} N_{p,eq} &= A_{eq}\sigma_{y,eq} \\ M_{p,3,eq} &= W_{3,eq}\sigma_{y,eq} \\ M_{p,2,eq} &= W_{2,eq}\sigma_{y,eq} \end{aligned} \qquad \textit{Eq. [63]}$$

More information about the behaviour and structural forces in beams can be found in the Material Models Manual - *Chapter 18 - 3D beam*.

6.7 Material data sets for embedded beams

Properties and model parameters for embedded beams 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, as well as the interaction properties with the surrounding soil or rock (bearing capacity).

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 and bearing capacity 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.

6.7.1 *Material set*

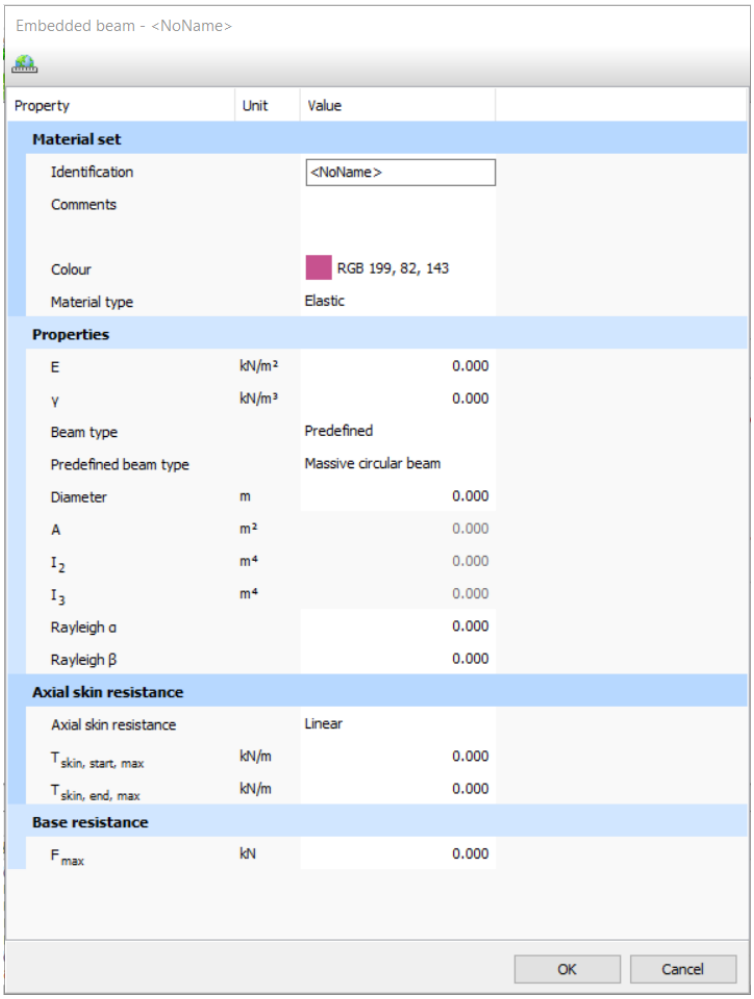


*Figure 220: **Embedded beam** window*

Several data sets may be created to distinguish between different types of embedded beams or spacings. Figure 220 (on page 309) shows the **Embedded beam row** 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.
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 two available options, describing the material type for embedded beams. These options are *Elastic* and *Elastoplastic*. The availability of the parameters defined in the *Properties* box depends on the selected material type.

6.7.2 *Beam properties*

The properties required for embedded beams are:

E	Young's modulus.
γ	Unit weight of the beam material.

Note:

- In a finite element model, embedded beams 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 beams material.
- Please note that when reducing the unit weight, the axial forces in the embedded beams may not be realistic.

Geometric properties

An embedded beam requires several geometric parameters used to calculate additional properties:

Beam type	Either a *Predefined* or a *user-defined* type can be selected.
Predefined beam 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 (Table 16 (on page 311) accordingly to the material type).
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 around the embedded beam in which plastic soil behaviour is excluded. This makes the embedded beam behave almost like a volume pile.
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.
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_3 and I_2:

A	The cross section area is the actual area (in the unit of length squared) perpendicular to the embedded beam axis (direction 1) 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 steel factories.

I_2	Moment of inertia against bending around the second axis.
I_3	Moment of inertia against bending around the third axis.

If the *Predefined* beam types are adopted, I_3 is equal to I_2.

From the embedded beam geometric properties, an equivalent radius for the elastic zone, R_{eq}, is determined:

$$R_{eq} = \max\left\{\sqrt{A/\pi}, \sqrt{2I_{avg}/A}\right\} \quad where \quad I_{avg} = (I_2 + I_3)/2 \qquad Eq.\ [64]$$

Strength properties: Elastoplastic

Yield stress σ_y	Yield stress parameter for a beam
Yield direction	In user-defined beam type, the yield direction can be either in *Local direction 2* or *Local direction* 3
W_2	Plastic section modulus when the *Yield direction* is in local direction 2
W_3	Plastic section modulus when the *Yield direction* is in local direction 3

If the *Predefined* beam types are adopted, W_3 is equal to W_2.

Note:

The yield stress parameter σ_y makes no distinction between compression and extension.

Table 16: Formulas used for predefined emdebbed beam types

Solid circular beam	A	$\varpi \times diameter^2/4$
	I_3	$\varpi \times diameter^4/64$
	I_2	$\varpi \times diameter^4/64$
	W_3	$\varpi \times diameter^3/32$
	W_2	$\varpi \times diameter^3/32$
Circular tube	A	$\varpi \times ((diameter)^2 - (diameter-2\times thickness)^2)/4$
	I_3	$\varpi \times ((diameter)^4 - (diameter-2\times thickness)^4)/64$
	I_2	$\varpi \times ((diameter)^4 - (diameter-2\times thickness)^4)/64$
	W_3	$\varpi \times ((diameter)^4 - (diameter-2\times thickness)^4)/(32\times diameter)$
	W_2	$\varpi \times ((diameter)^4 - (diameter-2\times thickness)^4)/(32\times diameter)$

Solid rectangular beam	A	*width × height*
	I_3	*width × height*^{3}*/12*
	I_2	*height × width*^{3}*/12*
	W_3	*width × height*^{2}*/6*
	W_2	*height × width*^{2}*/6*

Based on the input parameters A, I_2, I_3 and σ_y, the equivalent parameters for the plastic properties are calculated as reported in Material data sets for beams (on page 303).

The definition of various quantities according to the embedded beam's local system of axes are visualised in Figure 221 (on page 312). More information about the behaviour and forces in embedded beams can be found in the Material Models Manual -*Chapter 18-3D Beam*.

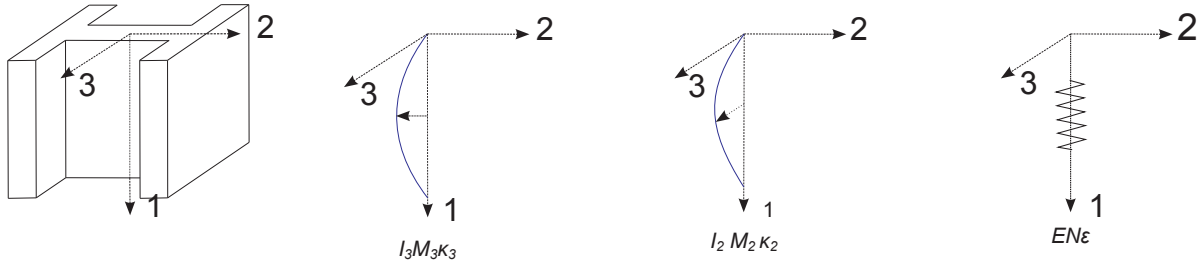


Figure 221: Definition of moment of inertia (I), positive bending moment (M), positive curvature (κ) and stiffness (E) for a vertical user-defined beam type (beam profile) based on the local system of axes

Dynamic properties

For dynamic behaviour, two additional parameters can be specified as material properties:

Rayleigh α	Rayleigh damping parameter determining the influence of mass in the damping of the system.
Rayleigh β	Rayleigh damping parameter determining the influence of the stiffness in the damping of the system.

For more information on Rayleigh damping, see Advanced general properties (on page 196).

6.7.3 *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. 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). 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:

$$| t_s | < T_{\max} \qquad Eq.\ [65]$$

where T_{max} is the equivalent local skin resistance at that point.

For plastic behaviour the shear force t_s is given by:

$$| t_s | = T_{\max} \qquad Eq.\ [66]$$

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:

$$N_{pile} = F_{\max} + F_{skin} \qquad Eq.\ [67]$$

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 ($F_{max} = 0$):

$$N_{\text{rock bolt}} = F_{skin} \qquad Eq.\ [68]$$

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.

In addition to displacement differences and shear forces in axial direction along the embedded beam 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 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 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 outside the elastic zone.

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.

Linear

The *Linear* option is the easiest way to enter the skin resistance profile. The input is defined by means of the skin resistance at the beginning of the embedded beam, $T_{skin,start,max}$ (in force per unit beam length), and the skin

resistance at the end of the embedded beam, $T_{skin,end,max}$ (in force per unit beam length). This way of defining the skin resistance is mostly applicable to piles in a homogeneous soil layer. Using this approach the total pile bearing capacity, N_{pile}, is given by:

$$N_{pile} = F_{\max} + \frac{1}{2} L_{pile} (T_{skin,start,\max} + T_{skin,end,\max}) \qquad \textit{Eq. [69]}$$

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:

$$N_{\text{rock bolt}} = \frac{1}{2} L_{\text{rock bolt}} (T_{skin,start,\max} + T_{skin,end,\max}) \qquad \textit{Eq. [70]}$$

where $L_{rock\ bolt}$ 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:

$$N_{pile} = F_{\max} + \sum_{i=1}^{n-1} \frac{1}{2} (L_{i+1} - L_i)(T_i + T_{i+1}) \qquad \textit{Eq. [71]}$$

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:

$$N_{\text{rock bolt}} = \sum_{i=1}^{n-1} \frac{1}{2} (L_{i+1} - L_i)(T_i + T_{i+1}) \qquad \textit{Eq. [72]}$$

where L is measured from the first point of the corresponding geometry line (L=0) to the second one (L= $L_{rock\ bolt}$).

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 258)) in which the pile/rock bolt is located:

$$\begin{aligned} \tau_i &= c_i + \sigma'_n \tan\varphi_i \\ c_i &= R_{\text{inter}} c_{soil} \\ \tan\varphi_i &= R_{\text{inter}} \tan\varphi_{soil} \end{aligned} \qquad \textit{Eq. [73]}$$

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 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 258). 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:

$$T_i = 2\pi R_{eq} \tau_i \qquad Eq.\ [74]$$

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 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.

6.8 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.

6.8.1 *Material set*

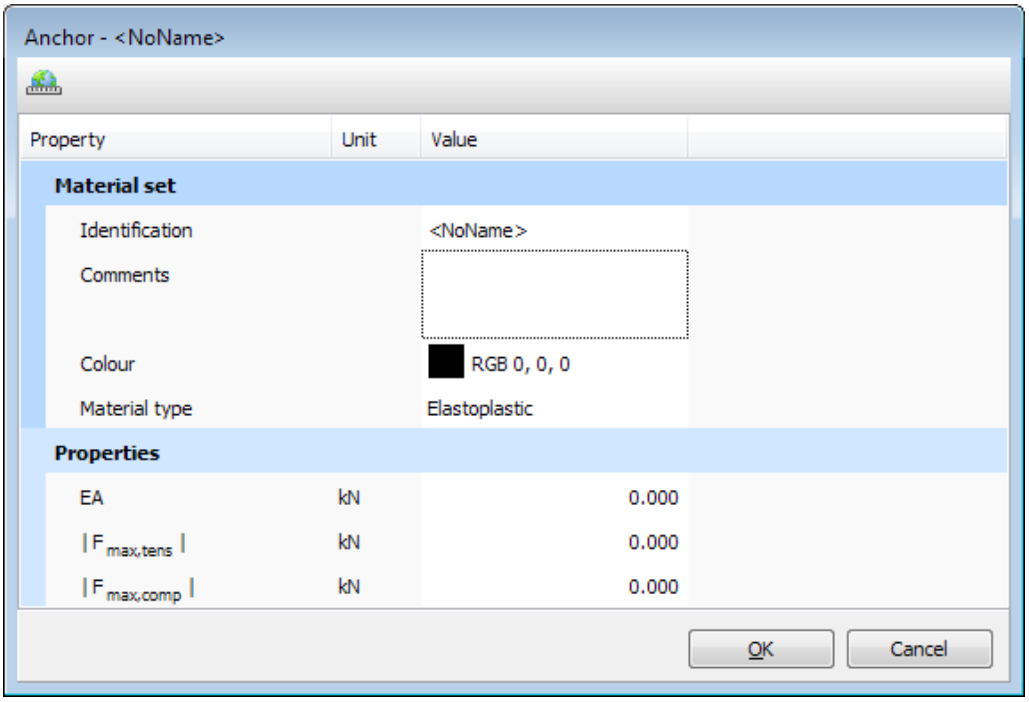


*Figure 222: **Anchor** window*

Several data sets may be created to distinguish between different types of anchors. Figure 222 (on page 316) shows 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.
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 three available options, describing the material type of an anchor. These options are *Elastic* and *Elastoplastic*. The availability of the parameters defined in the *Properties* box depends on the selected material type.

6.8.2 *Properties*

The properties required for anchors can be grouped into stiffness properties and strength properties in case of elastoplastic behaviour.

Stiffness properties

An anchor requires only one stiffness parameter:

EA	Axial stiffness, entered per anchor in the unit of force.

Strength parameters: Elastoplastic

If the material type is selected as *Elastoplastic*, two maximum anchor forces can be entered:

$F_{max,tens}$	Maximum tension force
$F_{max,comp}$	Maximum compression force

The Force-displacement diagram displaying the elastoplastic behaviour of the anchors is given in Figure 223 (on page 317).

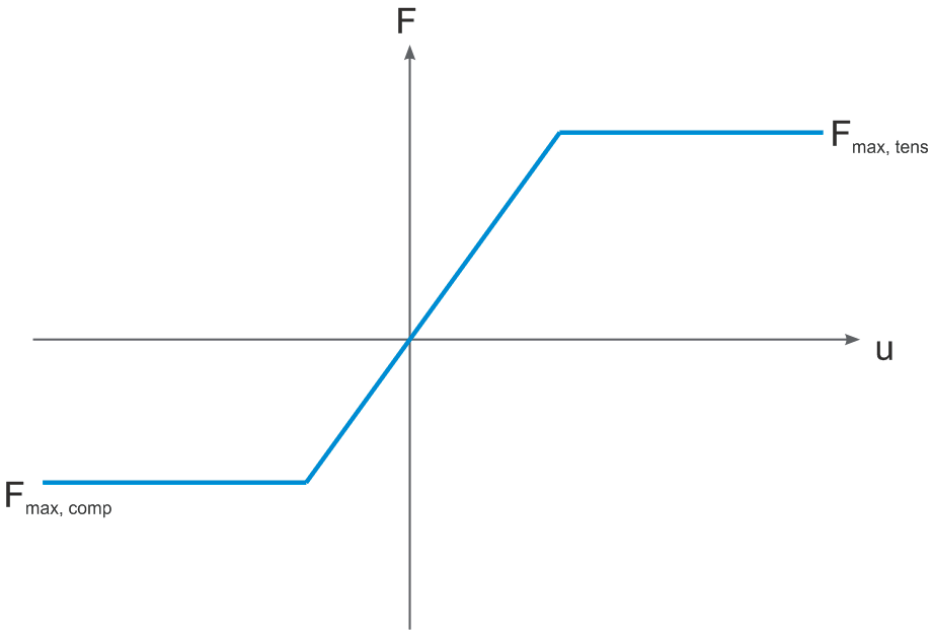


Figure 223: The force-displacement diagram displaying the elastoplastic behaviour of anchors

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.

Strength parameters: 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
$F_{max,comp}$	Maximum compression force
$F_{residual,tens}$	Residual tension force
$F_{residual,comp}$	Residual compression force

The Force-displacement diagram displaying the elastoplastic behaviour with residual strength of the anchors is given in Figure 223 (on page 317).

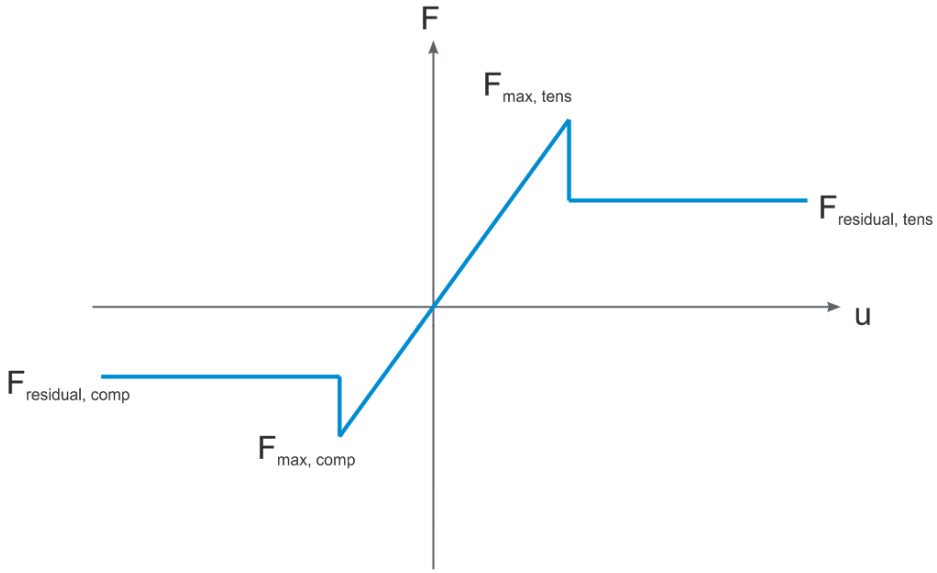


Figure 224: The force-displacement diagram displaying the elastoplastic behaviour with residual strength of the anchor

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 that 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 is a completely a new one.

Note: A reduced residual strength is not recommended to be used in *Safety* calculations.

6.9 *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 388)).

6.9.1 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.

6.9.2 Structures

Regarding structures (fixed-end anchors, beams, node-to-node anchors, embedded beams, 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 3D 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 3D 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 *Gravity loading* or *K0 procedure*. 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 328).

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 3D program uses fully

automatic generation of finite element meshes. 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 of the 3D finite element mesh are the 10-node tetrahedral elements (Figure 225 (on page 321)).

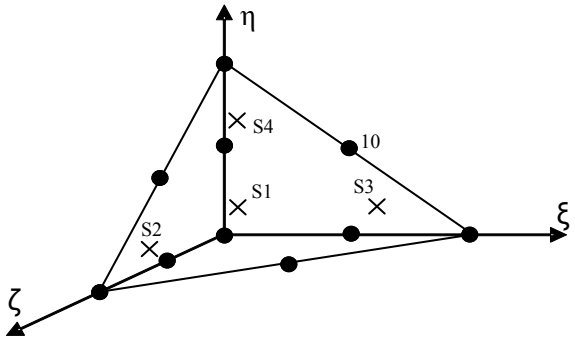


Figure 225: 3D soil elements (10-node tetrahedron)

In addition to the soil elements, special types of elements are used to model structural behaviour.

- For beams, 3-node line elements are used, which are compatible with the 3-node edges of a soil element.
- For plates and geogrids, 6-node plate and geogrid elements are used to simulate the behaviour.
- For soil-structure interaction, 12-node interface elements are used to simulate the behaviour.

The element formulations are given in the Scientific Manual.

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 226 (on page 321)). The options are explained in Global settings (on page 322), Local refinement (on page 323) and Enhanced mesh refinements (EMR) (on page 323).

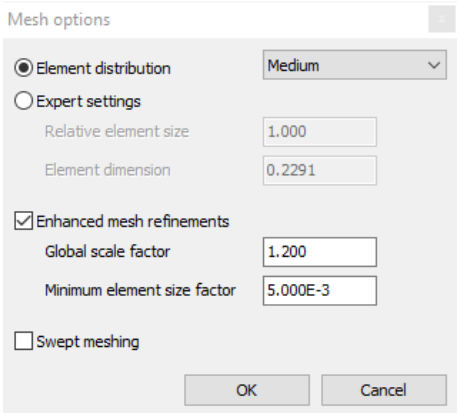


*Figure 226: **Mesh options** window*

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 3D this parameter is calculated from the outer geometry dimensions (x_{min}, x_{max}, y_{min}, y_{max}, z_{min}, z_{max}) and the *Element distribution* selected in the **Mesh options** window. The target element dimension is calculated using the Eq (on page 0).

$$l_e = r_e \times 0.05 \times \sqrt{(x_{\max} - x_{\min})^2 + (y_{\max} - y_{\min})^2 + (z_{\max} - z_{\min})^2}$$

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	2.0
Coarse	1.5
Medium	1.0
Fine	0.7
Very fine	0.5

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 exact number of elements depends on the shape of the geometry and optional local refinement settings.

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.

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 *0.5* 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 *0.0625* 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 (volume, surface, 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 (volume, surface, 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 3D will apply automatic mesh refinements. To affect the behaviour of these automatic mesh refinements, three parameters can be modified. The *Enhanced mesh refinement* parameters which can be applied are (Figure 226 (on page 321)):

Global scale factor	This parameter can be used to set the element size smaller. For example, if it is set to 0.25, this means that all EMR calculated element sizes will be 25% of what they would normally be, i.e. instead of generating one element along a line, four elements will be generated. This will scale to 16 times more elements in 2D and even 64 times more elements in 3D. Note that this scaling will only be notably visible in the mesh when EMR is determining the size. Also note that this won't be taken into account for surface curvature on a closed surface if this value is higher than 1, e.g. for a full cylinder you don't want to have less refinement as required here to have ~6 elements around the shaft of the cylinder) as otherwise it is not possible to create the mesh at all.
Minimum element size factor	This parameter sets the lower limit for any coarseness. No element size will be smaller than the minimum element size factor multiplied by the bounding box diagonal, except when the coarseness comes from a surface curvature on a closed surface to ensure a mesh.

Note:

The minimum allowed element size is *Minimum size factor* (see Enhanced mesh refinements (EMR) (on page 323)) multiplied by *Bounding box diagonal* of the geometry being meshed.

The third parameter is the *proximity* factor, which can be set only from the command line, by adding this parameter value to the arguments of the mesh command.

```
_mesh 0.02 256 True 1.2 0.005 2.0
```

In this example the last value (2.0) is the proximity factor. Please see *Help > Command reference* for further info on the arguments.

The *Proximity* factor affects the coarseness due to the length of edges, small distances in surfaces and small distances in volumes.

Note:

The automatic refinement considered can be turned off by unchecking the *Enhanced mesh refinements* option in the **Mesh options** window (Figure 226 (on page 321)).

7.2 Swept meshing

It is possible for some particular geometries to generate a swept mesh. A swept mesh starts at a source surface and sweeps along to a specified destination surface following a given trajectory (the swept direction). The option Swept meshing in the Mesh options window can be selected for that particular purpose. Swept mesh techniques usually result in faster mesh creation as well as it uncouples the mesh size in the source surface from the element size along the trajectory.

When using the swept meshing technique, PLAXIS will automatically detect volumes which are swept meshable along with their corresponding front surface and trajectory. The front surfaces of the swept meshable volumes will first be meshed. Then their volume meshes will be created by sweeping the initially meshed surfaces, using the coarseness of the volume to compute the number of mesh slices along the trajectory. Only once all swept meshable volumes have been meshed, PLAXIS will start considering the non-swept meshable volumes for

meshing. Note that neighboring swept meshable volumes should have the same coarseness to ensure consistent number of mesh slices along the trajectories. If not this will cause some volumes not being swept meshed despite being topologically perfectly suited for such a meshing technique.

For more information visit t Bentley Communities - Swept Meshing in PLAXIS 3D.

7.3 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*.

7.3.1 *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 227 (on page 325).

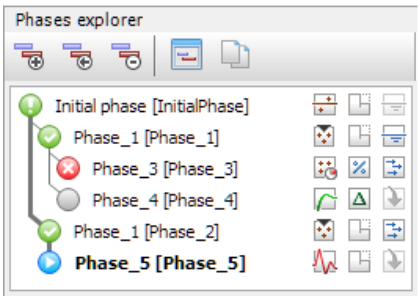


Figure 227: Phases explorer

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 329).

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 327)).

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 228 (on page 326)).

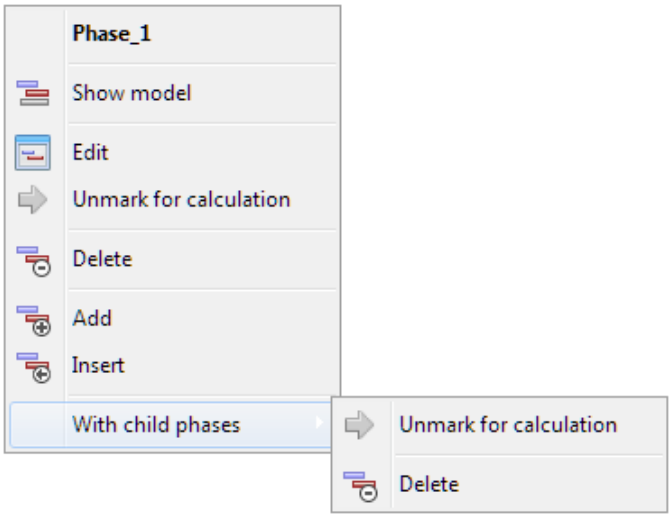


Figure 228: The expanded view of the right mouse menu of Phases explorer

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 356)).

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 328), Loading type (on page 345) and Water pressure calculation (on page 349) respectively.

7.3.2 *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.

7.3.3 *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 356)). 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 229 (on page 328)). 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.

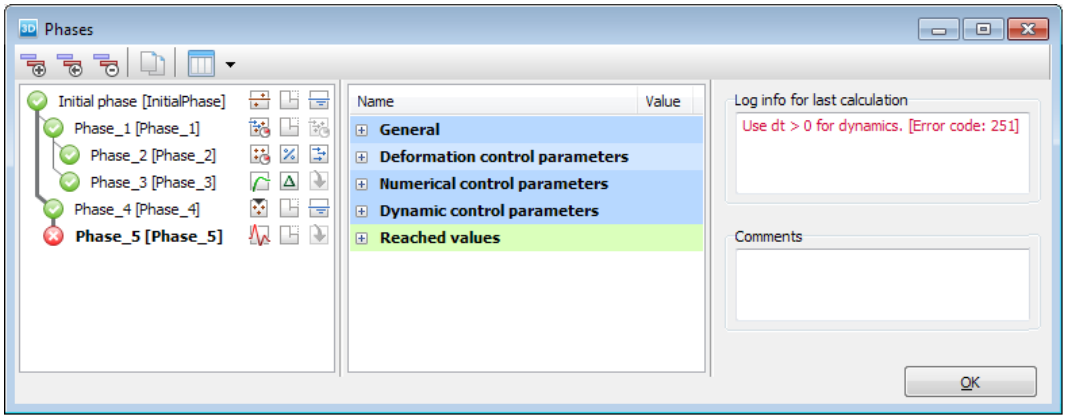


*Figure 229: Three panel view of the **Phases** window*

The *Spreadsheet* view displays all the calculation information in a spreadsheet form (Figure 230 (on page 328)) 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.

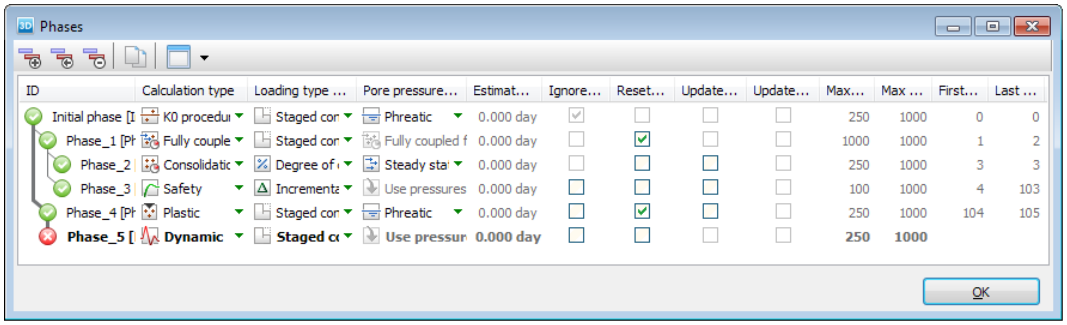


*Figure 230: Spreadsheet view of the **Phases** window*

In the **Phases** window, users need to select at least the *Calculation type* and the *Loading type* for each new phase. PLAXIS 3D 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.

7.4 Types of analysis

The first step in a PLAXIS 3D 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* and *Gravity loading* for the initial phase to generate the initial stress state of soil. The *Flow only* option can be used only if 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.

7.4.1 *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 3D, 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 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 265).

K0 procedure


K0 procedure is a special calculation method available in PLAXIS 3D 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 265)).

Two K_0 values can be specified, one for the *x*-direction and one for the *y*-direction.

$$K_{0,x} = \sigma'_{xx} / \sigma'_{zz} \qquad K_{0,y} = \sigma'_{yy} / \sigma'_{zz} \qquad \textit{Eq. [75]}$$

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:

$$K_0 = 1 - \sin\varphi \qquad \textit{Eq. [76]}$$

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), 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 265) and the Material Models Manual):

$$K_{0,x} = K_{0,y} = K_0^{nc}\text{OCR} - \frac{\nu_{ur}}{1-\nu_{ur}}(\text{OCR}-1) + \frac{K_0^{nc}\text{POP} - \frac{\nu_{ur}}{1-\nu_{ur}}\text{POP}}{\left|\sigma_{zz}^0\right|} \qquad \textit{Eq. [77]}$$

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 3D 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 459)).

For a cohesionless material it can easily be shown that to avoid Mohr-Coulomb plasticity, the value of K_0 is bound by:

$$\frac{1-\sin\varphi}{1+\sin\varphi} < K_0 < \frac{1+\sin\varphi}{1-\sin\varphi} \qquad \textit{Eq. [78]}$$

When the *K0 procedure* is adopted, PLAXIS 3D 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 334)). 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 GSE

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* and *Axis2*), see Figure 231 (on page 331). *Axis2* internally will be adjusted to create an orthogonal reference frame: (*Axis1*, *Axis2*, *Axis3*). *Axis3* is perpendicular to *Axis1* and (adjusted) *Axis2*. All axes are internally set to unit length.

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 and z) coefficients of the orientation axis do not need to form a vector of unit length.

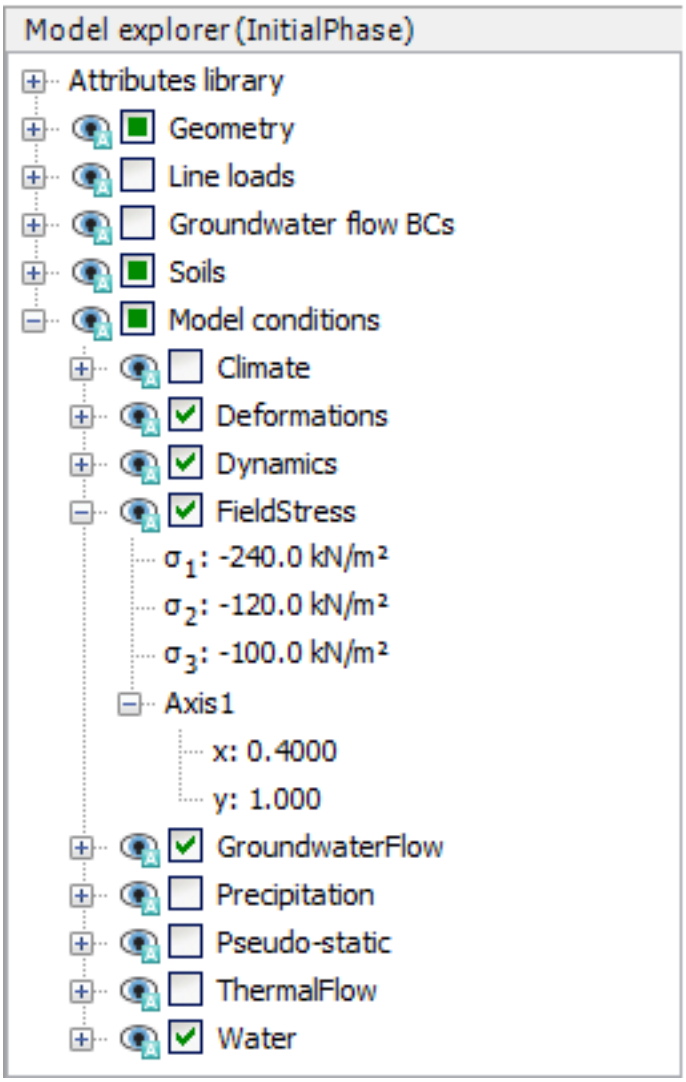


Figure 231: Field stress in Model explorer

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* and *Axis2*) 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 dynamic 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*, *BoundaryZMin* and *BoundaryZMax*) 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 principal stress directions according to the *Axis1* and *Axis2* parameters (Figure 232 (on page 332)). 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*.

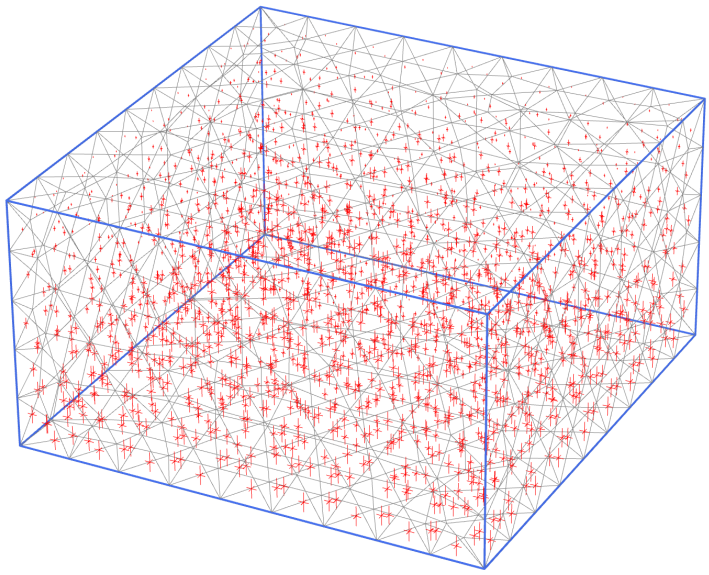


Figure 232: Initial stress generation using Field stress calculation

Gravity loading

 *Gravity loading* is a type of *Plastic* calculation (Plastic calculation (on page 334)), 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 388)). For one-dimensional compression an elastic computation will give:

$$K_0 = \frac{\nu}{1 - \nu} \qquad \textit{Eq. [79]}$$

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:

$$\frac{1 - \sin\varphi}{1 + \sin\varphi} < \frac{\nu}{1 - \nu} < 1 \qquad \textit{Eq. [80]}$$

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 421)). 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

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 343)). 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 358)).

7.4.2 *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 382)) it is also possible (for each calculation phase) to redefine the water boundary conditions and recalculate the pore pressures (Flow conditions (on page 374)). 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 382)).

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 349).

7.4.3 *Consolidation calculation*


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 3D 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 374)).

Note:

- In PLAXIS 3D, 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 3D 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 3D it is defined as the target minimum excess pore pressure over the maximum initial excess pore pressure $p_{max}/p_{max,initial}$.

7.4.4 *Fully coupled flow-deformation analysis*

 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.

Note:

It is not possible to use *Updated mesh* in a fully coupled flow-deformation analysis.

7.4.5 *Safety calculation (phi/c reduction)*

 The *Safety* calculation type is an option available in PLAXIS 3D 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 339)).

The total multiplier *ΣMsf* is used to define the value of the soil strength parameters at a given stage in the analysis:

$$\Sigma Msf = \frac{\tan\varphi_{input}}{\tan\varphi_{reduced}} = \frac{c_{input}}{c_{reduced}} = \frac{s_{u,input}}{s_{u,reduced}} = \frac{\text{Tensile strength}_{input}}{\text{Tensile strength}_{reduced}} \qquad \textit{Eq. [81]}$$

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 354)). 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:

$$SF = \frac{\text{available stregth}}{\text{stregth at failure}} = \text{value of } \Sigma\text{Msf at failure} \qquad \textit{Eq. [82]}$$

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 481)). 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 362)).

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 Joined 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 is not reduced in *Safety analysis*.
- When using the Modified Cam-Clay model the strength is not reduced at all since this model does 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 510).

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.

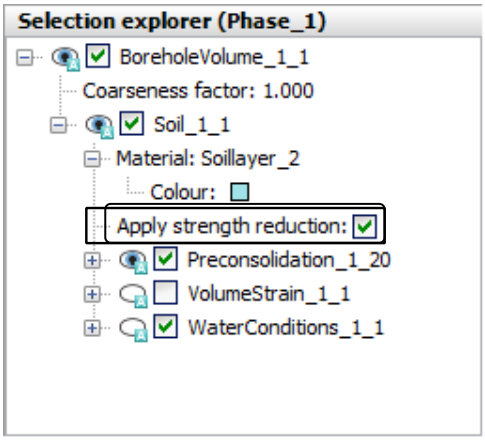


Figure 233: Enhanced safety for soil clusters

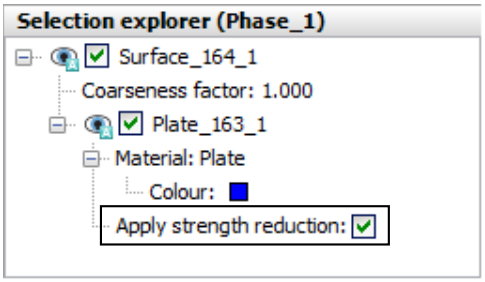


Figure 234: Enhanced safety for structures

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 445)). 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 GSE

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:

$$f_{HB} = \sigma'_1 - \sigma'_3 + \overline{f}_{red}(\sigma'_3) \qquad \textit{Eq. [83]}$$

with

$$\overline{f}_{red} = \frac{\overline{f}}{\eta} = \frac{\sigma_{ci}}{\eta}\left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^a \qquad \textit{Eq. [84]}$$

and

$$\eta = \frac{1}{2}\left(\sum Msf\,(2 - \overline{f}\,')\sqrt{1 + \frac{\left(\frac{1}{\sum Msf^2} - 1\right)\overline{f}\,'^2}{(2 - \overline{f}\,')^2}} + \overline{f}\,'\right) \qquad \textit{Eq. [85]}$$

where

$$\overline{f}\,' = \frac{\partial \overline{f}}{\partial \sigma'_3} = -am_b\left(m_b \frac{-\sigma'_3}{\sigma_{ci}} + s\right)^{a-1} \qquad \textit{Eq. [86]}$$

More details and a derivation of the above equations can be found in Benz, Schwab, Vermeer & Kauther (2007) (on page 510).

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 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 updated at the beginning of each load step even if the *Updated mesh* option is selected for the *Safety* phase.

7.4.6 *Dynamic calculation*

The *Dynamic* 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 3D it is possible to perform a dynamic 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 Dynamic calculation. For a more detailed description of the boundary conditions, see Model conditions (on page 390). A Dynamic time interval can be defined to specify the calculation duration and an automatic time stepping scheme (see Automatic time stepping (on page 355)) 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 Dynamic 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 dynamic 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 Dynamic 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 dynamic multiplier tab (Input accelerograms (on page 131)) 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 dynamic calculation. Special boundary conditions (Model conditions (on page 390)) 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 (Dynamic control parameters (on page 373)). Dynamic calculations can be conducted as drained, undrained or as Dynamic with consolidation. Automatic time stepping (Automatic time

stepping (on page 355)) 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 Dynamic calculations (on page 487) explains output curves generation options for dynamic 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 dynamic phases.

Note:

- Note that steady-state pore pressures in a *Dynamic* calculation are always taken from the steady-state pore pressures generated in the parent phase (see Calculation type (on page 349)).
- It is possible to calculate excess pore pressures in undrained soil layers in a dynamic analysis. However, the accuracy at which pore pressures are predicted depends on the capabilities of the soil models being used.
- A standard dynamic calculation may involve the generation of excess pore pressures, but not the dissipation of excess pore pressures. If the latter is required, a *Dynamic with consolidation* calculation should be performed.

Updated mesh

During a dynamic 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 343) for more information about *Updated mesh* calculations.


In a *Dynamic* calculation loading can be defined in the sense of applying a predefined combination of external loads as dynamic forces using dynamic multipliers activated in the **Staged construction** mode. Dynamic calculations can also deal with moving loads.

7.4.7 *Dynamic with consolidation calculation*


The *Dynamic 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 dynamic 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 *Dynamic 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 390).

In a *Dynamic with consolidation* calculation loading can be defined in the sense of applying a predefined combination of external loads as dynamic forces using dynamic multipliers activated in the **Staged construction** mode.

7.4.8 *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 358)).

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.

7.4.9 *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 3D 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 3D when the *Updated mesh* option is selected.

It should be clear from the descriptions given above that the updated mesh procedures used in PLAXIS 3D 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 510)). Implementation of this formulation within PLAXIS 3D 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 512)).

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. 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 358)).

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.

Note:

Note that water conditions at the mesh boundary and pore pressures in stress points are not updated as the mesh is updated.

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 358)).

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.

7.5 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.

7.5.1 *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 353)), 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 373)).

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:

$$f_{applied} = f_0 + \Sigma Mstage(f_{defined} - f_0) \qquad Eq.\ [87]$$

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.

7.5.2 *Minimum excess pore pressure*

 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.

7.5.3 *Degree of consolidation*

 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)*:

$$\text{Minimim pore pressure} = (100 - U)\text{Pmax} \qquad \textit{Eq. [88]}$$

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 373)). 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.

7.5.4 *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 348)) 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.

7.5.5 *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 348)). 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:

$$SF = \frac{\text{available strength}}{\text{strength at failure}} = \text{value of } \Sigma\text{Msf at failure} \qquad \textit{Eq. [89]}$$

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 481)). 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.

7.6 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 3D 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.

7.6.1 *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 351)) and the water conditions of the clusters (Defining water conditions (on page 75)). 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 volumes (on page 378)) or the cluster should be *Non-porous* (General tabsheet (on page 193)).
- 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)

The steady-state water pressure calculation by *Steady state groundwater flow* is based on the input of hydraulic boundary conditions (Hydraulic conditions (on page 176) and Water pressure generation and previewing (on page 380)). 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 volumes (on page 378)) or the cluster should be *Non-porous* (General tabsheet (on page 193)).
- 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)

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 (Surface flow boundary conditions (on page 180)) and water levels (Water levels (on page 375)) 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 249)).

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.

7.6.2 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 375).

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 3D 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 3D 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 235 (on page 352)).

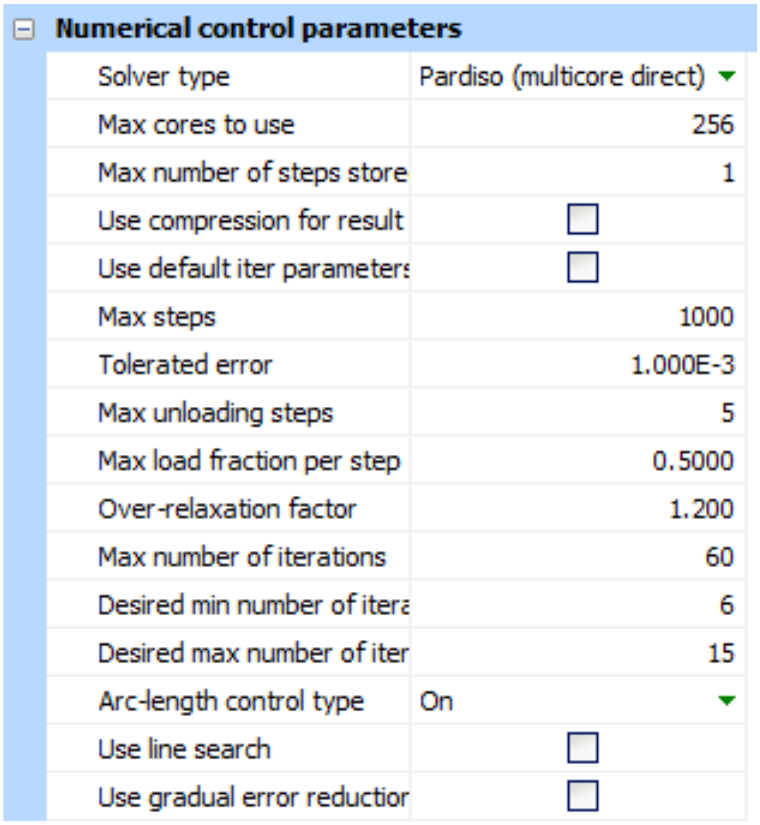


*Figure 235: Numerical control parameters subtree in the **Phases** window*

The automatic load stepping procedure is controlled by a number of calculation control parameters (Numerical control parameters (on page 362)). 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 362)). 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 362)). 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 362)). 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 3D performs a trial calculation step and determines a suitable step size on the basis of this trial.
- PLAXIS 3D 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 352)). 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 362)) and the current stiffness parameter CSP is less than 0.015 (see Automatic error checks (on page 407) 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.

Another problem related to the solution process can occur due to insufficient direct computer memory (RAM). In such cases, the iterative solver cannot store the minimum amount of data necessary to have sufficient accuracy. As a result, the iteration process converges very slowly, or does not reach the accuracy condition. To solve this problem, either the problem size should be reduced, or internal memory capacity should be increased.

Note:

A list of warning and error messages appearing in PLAXIS 3D is available in Appendix H.

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 352)).

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 3D is available in Appendix H.

7.7.4 *Automatic time stepping (consolidation or fully coupled analysis)*

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 362). 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* 

When the *Calculation type* is set to *Dynamic* or *Dynamic 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- *Chapter 7 - Bounds on the system response*). The proper critical time step for dynamic 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 dynamic 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 236 (on page 356)).

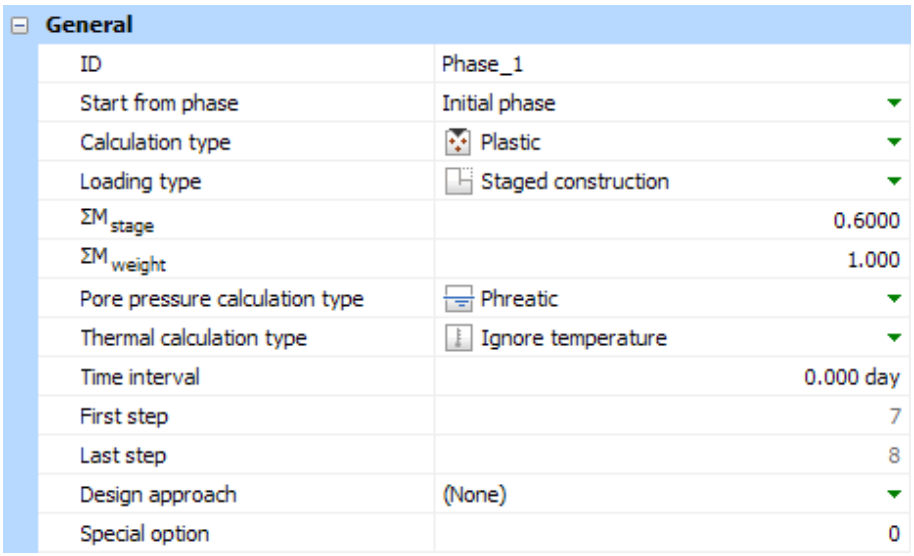


*Figure 236: General subtree in the **Phases** window*

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 3D is given in Types of analysis (on page 328).

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 3D is given in Loading type (on page 345).

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 ΣM_{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 dynamic 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 dynamic 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 dynamic calculation phase, expressed in seconds [s].

A dynamic analysis uses a different time parameter than other types of calculations. The time parameter in a dynamic 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.

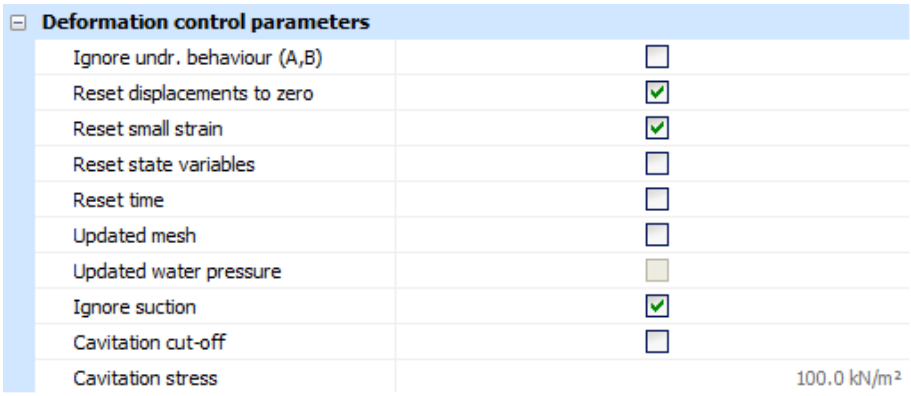


*Figure 237: Deformation control parameters subtree in the **Phases** window*

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 *Dynamic 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 3D 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.

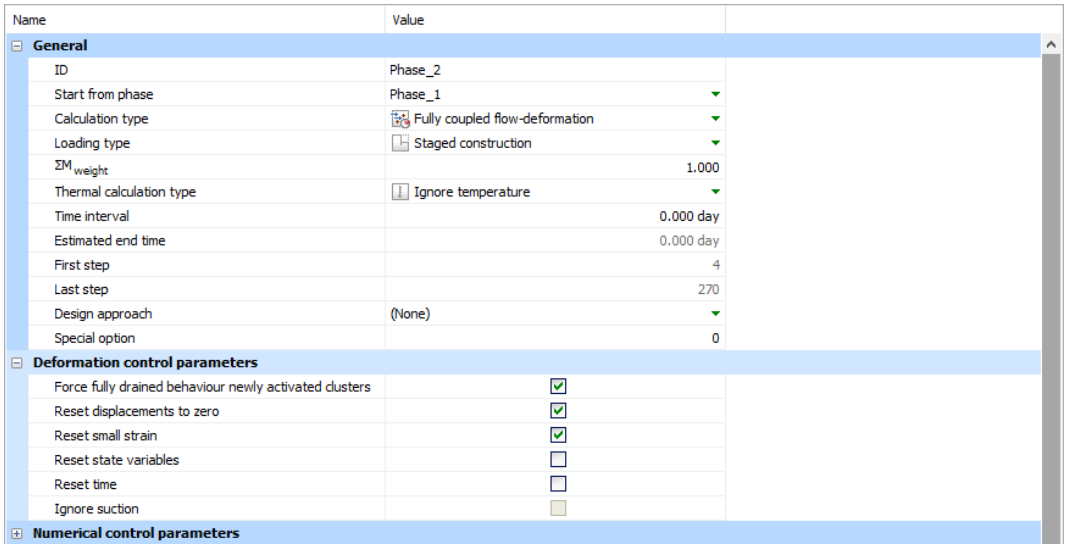


Figure 238: Deformation Control Parameters: Force Fully Drained Behaviour

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 GSE

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 *Dynamic 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 *Dynamic 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 17 (on page 361).

Table 17: 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 343).

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*

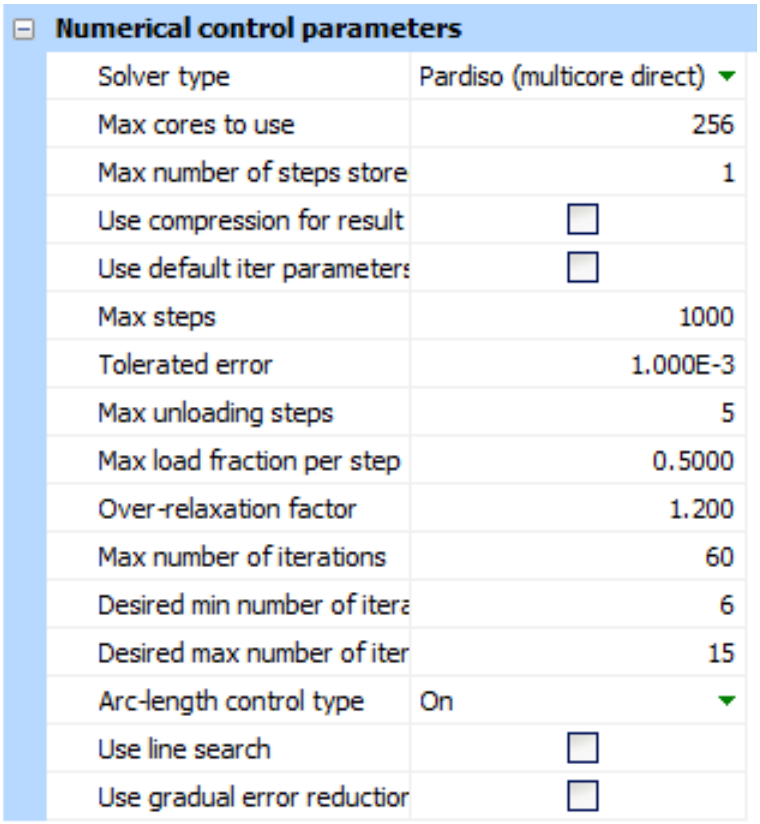


*Figure 239: Numerical control parameters subtree in the **Phases** window*

Solver type

In PLAXIS 3D the assembly and solution of sparse linear equation systems is carried out by solvers. The available solvers are:

Picos (multicore iterative)	It stands for *Plaxis Iterative Concurrent Solver*. *Picos* is an efficient iterative solver that solves the system of equations in parallel on multi-core processors. It is generally the fastest way to perform calculations.
Pardiso (multicore direct)	It stands for *Parallel Sparse Direct Solver*. *Pardiso* is a direct solver that solves the system of equations in parallel on multi-core processors. It is generally the most robust way to solve equations, but it has also the highest memory consumption (Schank & Gärtner (2006) (on page 512) and Schank, Wächter & Hagermann (2007) (on page 512)).
Classic (single core iterative)	This is the iterative solver that has been used in previous PLAXIS 3D versions. It solves the system of equations using only a single core on the processor.

Max cores to use GSE

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 that the option of using more than two cores is a PLAXIS GSE feature.The *Max cores to use* parameter has no effect when the *Classic* solver is used.

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.

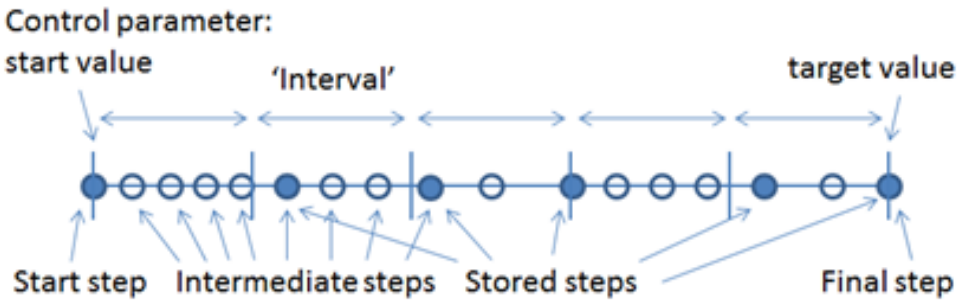


Figure 240: Max number of steps stored

The calculation control parameter and the 'Interval' depend on the type of calculation which is described in table below (Figure 240 (on page 364)).

Table 18: 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)

Calculation type	Control parameter	'Interval'
Dynamic analysis	Dynamic time	Dynamic 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 3D 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 *Dynamic* 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 *Dynamic* 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 *Dynamic* 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 or fully coupled analysis) (on page 355)). 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 - *Chapter 4 - Critical Time Step*. The *Min time step* and *Max time step* parameters are the minimum and maximum time step values.

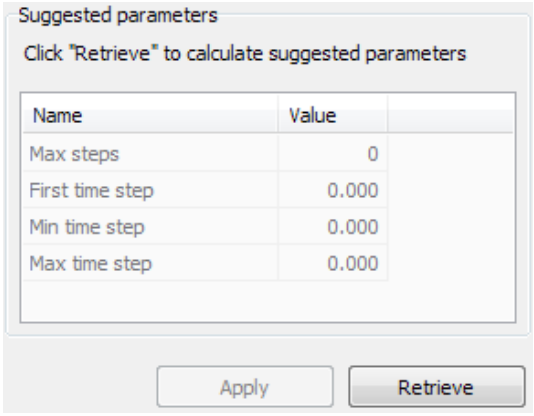


Figure 241: Dialog box displaying the retrieved value for the First time step parameter

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 *Dynamic* calculation

The time step used in a *Dynamic* calculation is constant and equal to $\delta t = \Delta t /(m \cdot n)$, where Δt is the duration of the dynamic loading (*Dynamic 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 dynamic 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 dynamic calculation. If the *Time step determination* is set to default option *Automatic*, PLAXIS 3D 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 (dynamic 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 3D 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 242 (on page 367)).



Figure 242: 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 243 (on page 368). 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 407)). The error limits adopted in PLAXIS 3D 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.

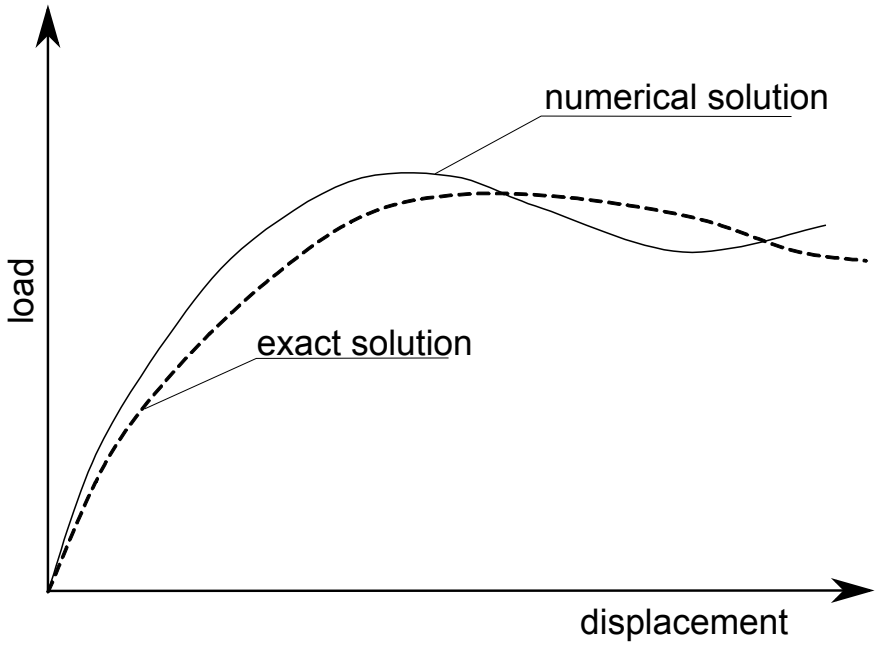


Figure 243: Computed solution versus exact solution

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 3D see Automatic error checks (on page 407).

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 3D 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 3D makes use of an over-relaxation procedure as indicated in Figure 246 (on page 371). 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.

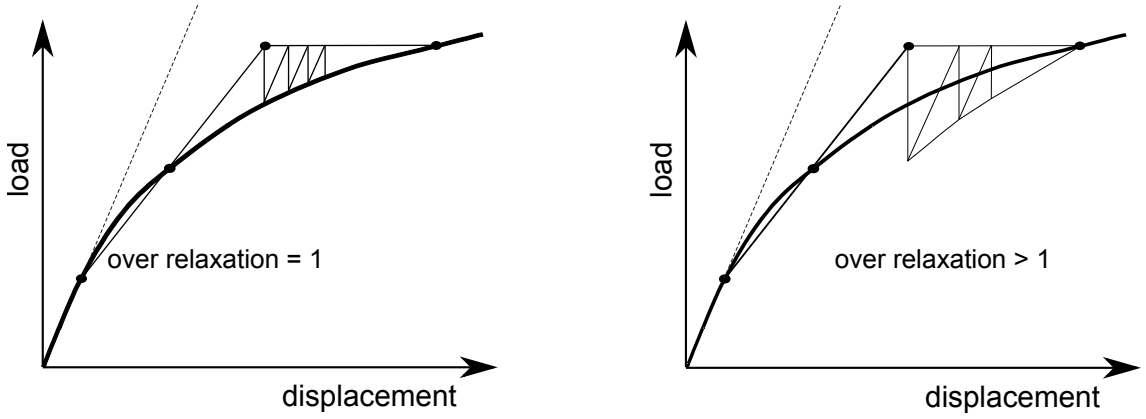


Figure 244: Influence of over-relaxation

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 3D makes use of an automatic step size algorithm (Automatic error checks (on page 407)). 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 352).

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:

$$\text{Desired minimum} = 3 \quad \text{Desired maximum} = 7$$

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:

$$\text{Desired minimum} = 8 \quad \text{Desired maximum} = 20$$

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 3D to obtain reliable collapse loads for load-controlled calculations. The iterative procedure adopted when arc-length control is not used is shown in Figure 245 (on page 370) 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.

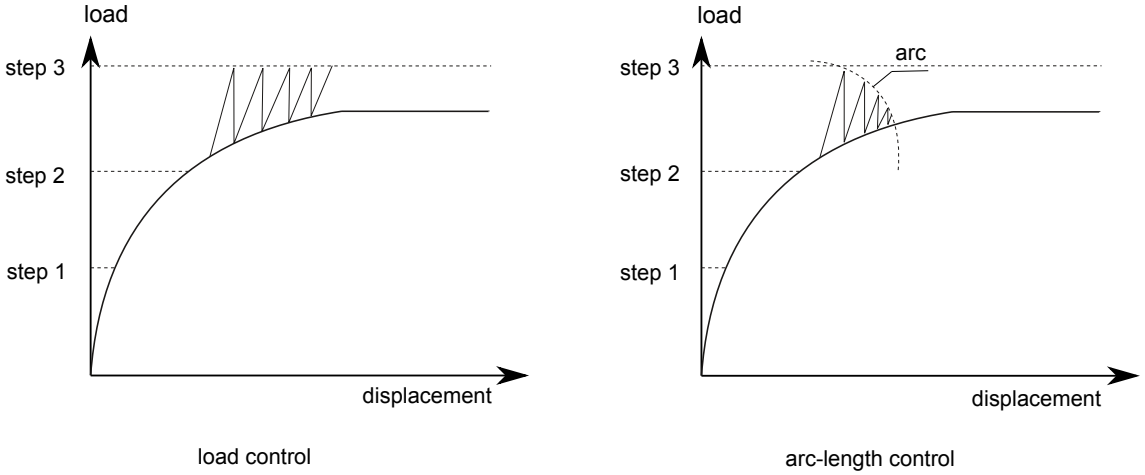


Figure 245: Influence of Arc-length control

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 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 3D. 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, the usage of line search is advised for problems without severe non-linearity, like

serviceability calculations. On the other hand, 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 3D. 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 3D 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).

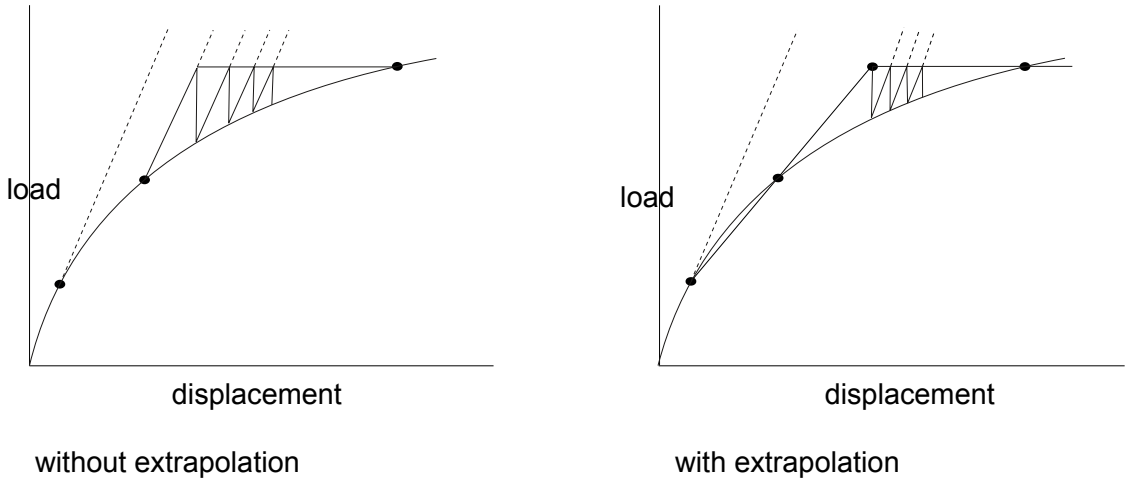


Figure 246: Difference between elastic prediction (a) and extrapolation from previous step (b)

After the first iteration, subsequent iterations are based on the elastic stiffness matrix, as in the initial stress method (Zienkiewicz, 1977 (on page 512)). 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*



*Figure 247: 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 362)), 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 3D 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 3D 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 3D needs more steps, higher values than 1.5 might result in faster convergence. The maximum possible value is 2.0.

7.8.5 *Dynamic control parameters*

The parameters controlling a *Dynamic* and *Dynamic with consolidation* analysis can be defined in the *Dynamic 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 248: Dynamic 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 dynamic calculations, PLAXIS 3D 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 dynamic 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 dynamic calculations (*0≤ Mass matrix ≤ 1*), allowing for a mixed matrix to be used.

7.8.6 *Reached values*

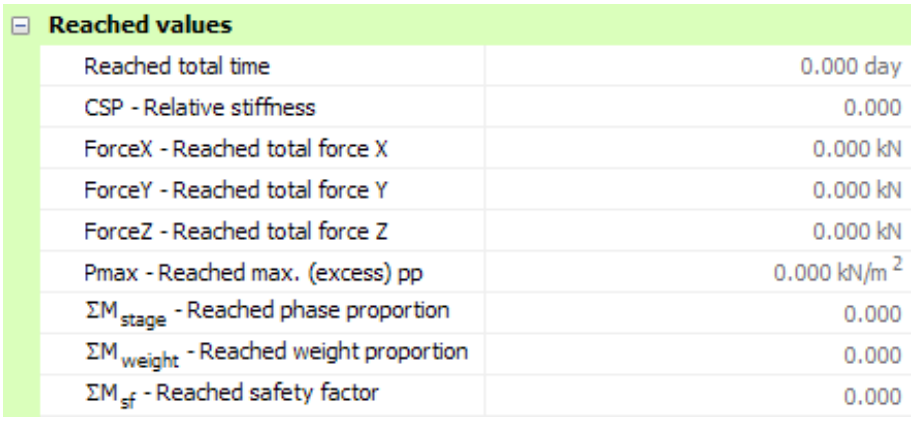


*Figure 249: Reached values subtree in the **Phases** window*

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 407)). 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 345)).
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 356)). 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 337)).

7.9 Flow conditions

PLAXIS 3D is generally used for effective stress analysis in which total stresses are divided into effective stresses, σ', and active pore pressures, p_{active}.

$$\sigma = \sigma' + p_{active} \qquad Eq.\ [90]$$

Active pore pressure (p_{active}) is defined as the effective saturation, S_{eff}, times the pore water pressure, p_{water}.

$$p_{active} = S_{\text{eff}} \cdot p_{water} \qquad Eq.\ [91]$$

Pore water pressure differs from active pore pressure when the degree of saturation is less than unity. PLAXIS 3D 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 249).

In the pore water pressure a further division is made between steady state pore pressure, p_{steady}, and excess pore pressure, p_{excess}.

$$p_{water} = p_{steady} + p_{excess} \qquad Eq.\ [92]$$

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 *Dynamic analysis*, excess pore pressures can occur in clusters for which the *Drainage type* is set to *Undrained A* or *Undrained B*. In *Consolidation analysis*, *Dynamic 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:

$$p_{excess} = p_{water} - p_{steady} \qquad Eq.\ [93]$$

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 volume clusters. These features will be described in more detail below.

7.9.1 *Water levels*

Water levels can either be generated according to the information specified in boreholes (Defining water conditions (on page 75)) 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 250 (on page 376)). 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* and non-horizontal water levels.

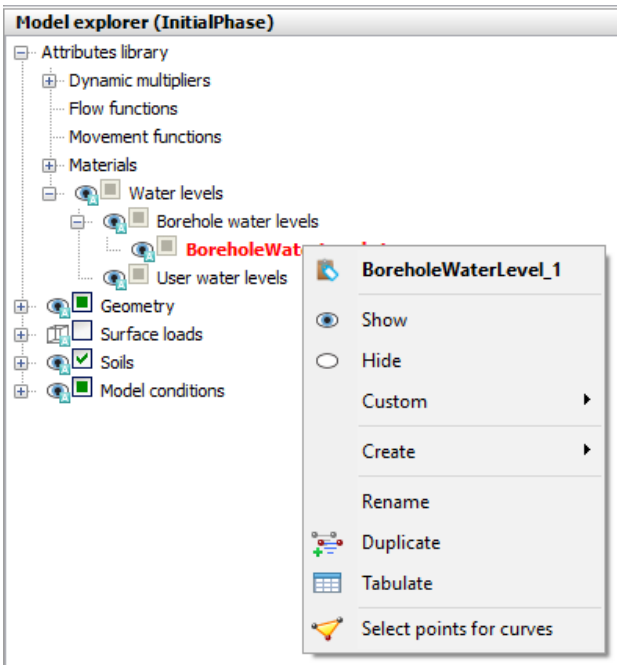


Figure 250: Copying water levels in Model explorer

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 3D 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 75)). 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 75).

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. As a result,a horizontal water level is created through this point, extending to the model boundaries. At the same time, the **Waterlevel points** window appears with a table of the point coordinates and a pressure increment with depth.

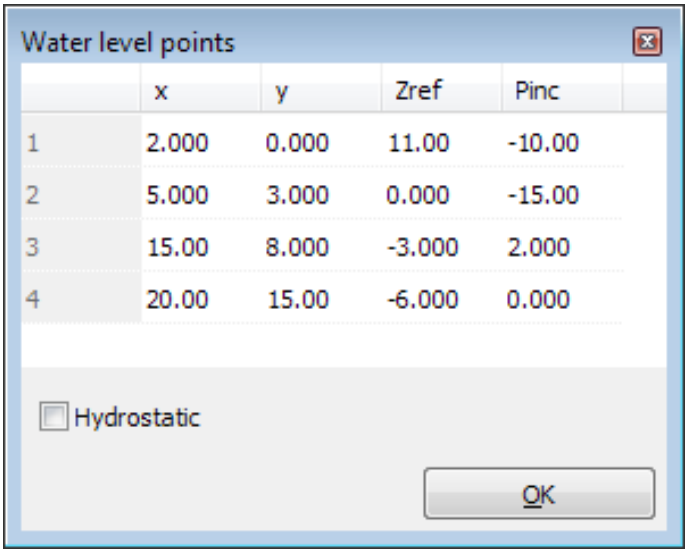


*Figure 251: **Waterlevel points** window*

A non-horizontal water level can be created by defining more points in the geometry, while the *Create water level* option is still active. The creation of water levels is basically the same as the creation of surfaces (Create surface (on page 105)). If necessary, the point coordinates or the pressure increment with depth (*Pinc*) may be changed in the table. Unlike surfaces, user water levels can only be moved in vertical direction by dragging them with the mouse. 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 *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 volumes*

Pore pressures in soil volumes are generated on the basis of the water conditions assigned to these volumes. By default, the water conditions as defined in the **Modify soil layers** window are assigned to all soil volumes 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, volumes 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 volumes cluster above and the volumes 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 (z_{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 (z_{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 volumes based on the pore pressures of the adjacent soil volumes (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 volumes is interpolated linearly in vertical direction, starting from the value at the bottom of the soil volumes above and ending at the value at the top of the soil volume below. The *Interpolate* option can be used repetitively in two or more successive soil volumes (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 volumes 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 volumes 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 volumes, 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*

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 (Surface flow boundary conditions (on page 180)). 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 Surface flow boundary conditions (on page 180) 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*

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 252 (on page 381)).

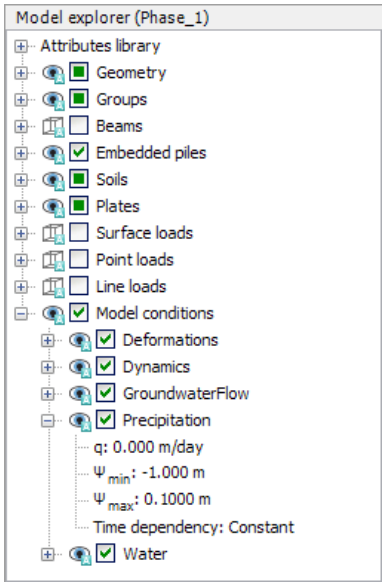


*Figure 252: The expanded **Precipitation** subtree in the Model explorer*

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 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 (on page 185).

7.10 Geometry configuration - Staged Construction mode

In PLAXIS 3D, it is possible to change the geometry and load configuration by activating or deactivating loads, soil volume 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 352)).

7.10.1 *Changing geometry configuration*

Loads, soil volume 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 elements in a selected mode in *Phases explorer*, click the *Activate* button and click the objects to be activated in the drawing area. All the elements assigned to the geometry selected after clicking *Activate* will be activated.

To deactivate elements in a selected mode in *Phases explorer*, click the *Deactivate* button and click the objects to be deactivated in the drawing area. All the elements assigned to the geometry selected after clicking *Deactivate* will be deactivated.

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 253 (on page 383)).

- ☐ Not active object or group of objects.
- ☑ Active object or group of objects.
- ■ Partly active group of objects.

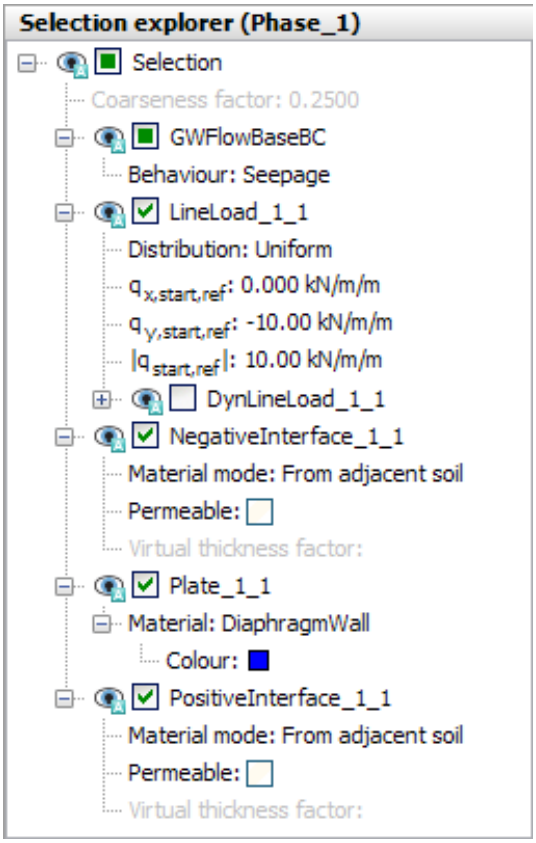


*Figure 253: Selection explorers in **Staged construction** mode*

The activation or deactivation of soil volume 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.

- The visibility of deactivated objects is defined in the **Visibility** tabsheet of the **Visualisation settings** window

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 *Definition of connections*

When structural elements are connected, by default they share all degrees of freedom (rotations and translation degrees of freedom) in the connected node, which implies that the connection is rigid.

In the **Calculation** modes it is possible to customize the connection between two plates or between a plate and a beam by explicitly defining a *Connection* object. In this way it is possible to define line hinges (between two plates) and point hinges (between a plate and a beam), as well as dilation joints (displacement discontinuities) between two plates and between a plate and a beam.

A connection consists of:

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

A connection requires the presence of interface elements (Interfaces (on page 147)). Interfaces are features that can only be assigned to surfaces. Interface elements are composed of pairs of nodes in which the nodes in a pair have the same coordinates (Scientific Manual **- Chapter 5**). This ensures that extra nodes are available for the extra degrees of freedom that are associated with the creation of a hinge or dilation joint.

When an invalid connection is defined a warning message appears before calculating the project (Figure 254 (on page 385)). If a connection is defined on a node where no interface is applied, the problem can be overcome by defining interfaces larger than the reference plate, defining interfaces on both sides of the reference plate or by assigning interfaces to both of the connected plates. Note that a *Beam* cannot be the *Reference part* of the connection as interface elements cannot be assigned to beams.

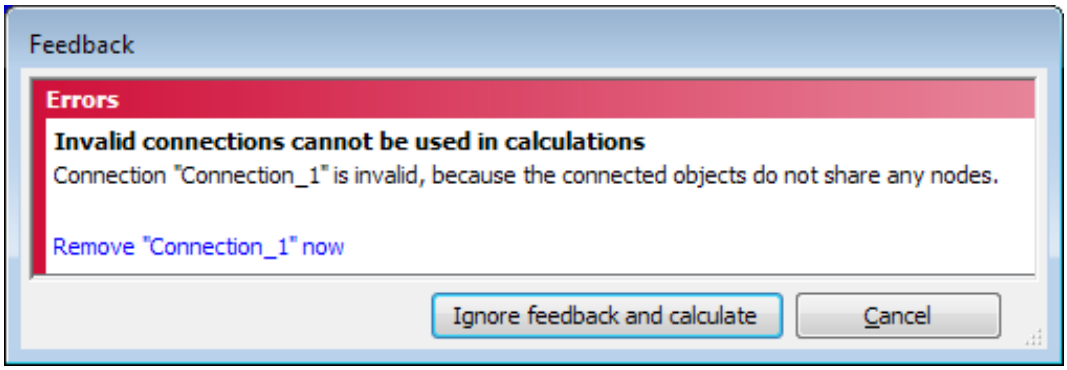


Figure 254: Warning regarding invalid connections

A connection is active when both parts are active in a calculation phase. Modifications done in connections are valid for all the phases where the connection is active. It is not possible to assign different properties to a connection in different phases.

If a connection has been defined, by default, the rotation degree of freedom of the 'child's' node(s) involved in the connection is disconnected (free) from the corresponding degree of freedom in the 'Parent's' node(s), whereas the translational degrees of freedom are tied (fixed). This means that the default connection is a hinge. The user can modify a connection by defining for each 'child' degree of freedom whether it is free or fixed. Distinction is made between rotation around the first local axis and translation in the three individual local directions. In order to define a dilation joint, at least one translational degree of freedom should be set free.

Connection between two plates

To create a connection between two plates:

- Make sure that an interface as well as a plate is assigned to the surface representing the *Reference part* in the connection.
- In the **Staged construction** mode select all plates involved in the connection.
- Right click the model and select the *Create custom connection* option from the appearing menu. The newly created connection is indicated by a yellow line in the model. The local axes of the connections are displayed if the *Show local axes* option is selected in the *Options* menu. A new connection is listed in the *Connections* subtree of the *Model explorer* (Figure 255 (on page 387)).
- Right-click on the model and select the *Create custom connection* option from the appearing menu.

To create a line hinge between two plates:

- Set the required option for rotation to *Free*.

To create a dilation joint between two plates:

- Set the required option for translation to *Free*.

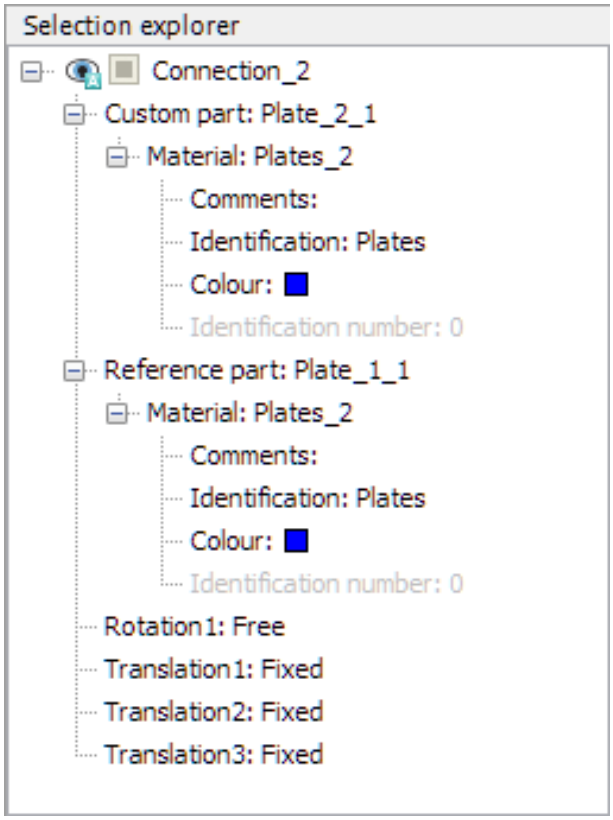


Figure 255: Connections subtree displaying a plate-plate connection

Note:

- The order of selection effects the part of the connection assigned to each plate. The first selected plate will be assigned as the *Custom part* of the connection whereas the second selected part will be assigned as the *Reference part* of the connection.
- The role of the parts of the connection can be interchanged by right clicking the connection in the *Object explorer* and selecting the *Invert connected part* option from the appearing menu.
- The order of selection is not important in a plate-beam connection. In that case the plate will always be the reference part of the connection.

Connection between a plate and a beam

To create a connection between a plate and a beam:

- Make sure that an interface as well as a plate is assigned to the surface representing the *Reference part* in the connection.
- In the **Staged construction** mode multi select the plate and the beam, the connection between which is to be customised.
- Right click the model and select the *Create custom connection* option from the appearing menu. The newly created connection is indicated by a yellow point in the model. The local axes of the connections are displayed if the *Show local axes* option is selected in the *Options* menu. A new connection is listed in the *Connections* subtree of the *Model explorer* (Figure 256 (on page 388)).

To create a point hinge between a plate and a beam:

- Set the required option for rotation to *Free*.

To create a dilation joint between a plate and a beam:

- Set the required option for translation to *Free*.

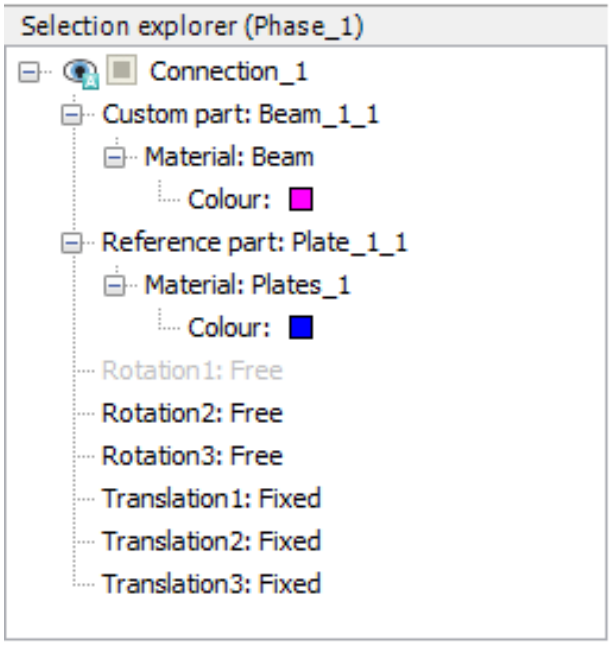


Figure 256: Connections subtree displaying a plate-beam connection

7.10.5 *Reassigning material data sets*

It is possible in a calculation phase to assign new material data sets to soil volume 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 3D 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 volume and clicking the checkboxes corresponding to *VolumetricStrain* and to *Apply* (Figure 257 (on page 389)).

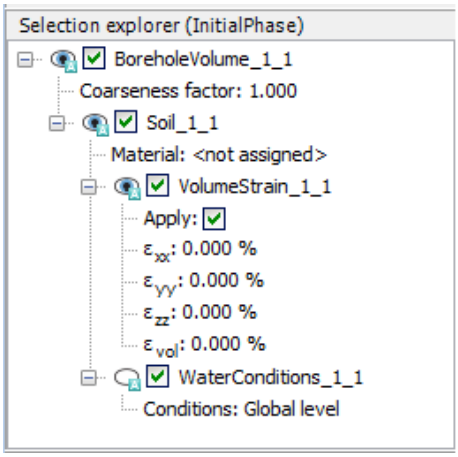


Figure 257: Volumetric strain in the Selection explorer

Values can be defined for the three strain components in *x*-, *y*- and *z*-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 (on page 172)).

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 3D automatically imposes a set of standard fixities to the boundaries of the geometry model. These conditions are generated according to the following rules.

Soil volume:

- Vertical model boundaries with their normal in *x*-direction (i.e. parallel to the *yz*-plane), are fixed in *x*-direction (u_x=*0*) and free in *y*- and *z*-direction.
- Vertical model boundaries with their normal in *y*-direction (i.e. parallel to the *xz*-plane), are fixed in *y*-direction (u_y = *0*) and free in *x*- and *z*-direction.
- Vertical model boundaries with their normal neither in *x*- nor in *y*-direction, are fixed in *x*- and *y*-direction (u_x = u_y = *0*) and free in *z*-direction.
- The model bottom boundary is fixed in all directions (u_x = u_y= u_z = *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*
BoundaryZMax	*Free*, *Normally fixed*, *Horizontally fixed*, *Vertically fixed*, *[Fully fixed]*
BoundaryZMin	*[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.

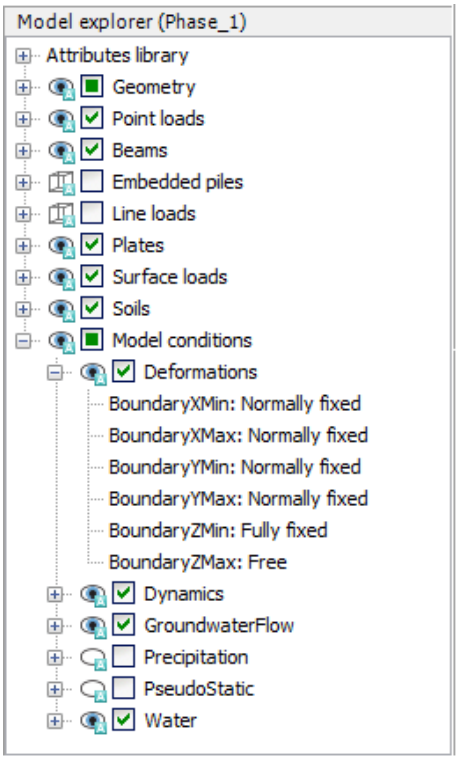


Figure 258: Boundary conditions

Structures: Structures that extend to the model boundary (XMin, XMax, YMin, YMax, ZMin, ZMax), where at least one displacement direction is fixed, obtain at least two fixed rotations in the points at the boundary.

- At vertical model boundaries with a normal in *x*-direction: $\varphi_y=\varphi_z=0$ (φ_x=free).
- At vertical model boundaries with a normal in *y*-direction: $\varphi_x=\varphi_z=0$ (φ_y= free).
- At vertical model edges and at the bottom boundary: $\varphi_x=\varphi_y=\varphi_z=0$.
- At the 'ground surface': $\varphi_x=\varphi_y=\varphi_z$= free.

Note:

- Note that the standard displacement fixities (Model conditions (on page 390)) apply to soil volumes and structures.
- User-defined prescribed displacements always prevail over the automatically defined fixities.

Dynamics - Dynamic boundary conditions

For dynamic 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 3D provides the option to use *Viscous* boundaries but other options are available as well.

When a *Dynamic* 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 259 (on page 392)).

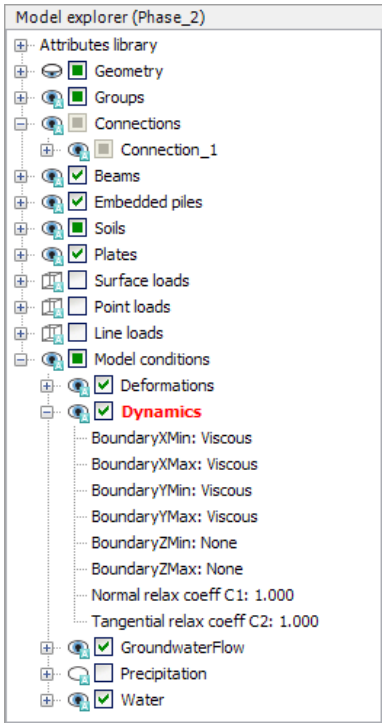


Figure 259: Boundary conditions for Dynamic calculations

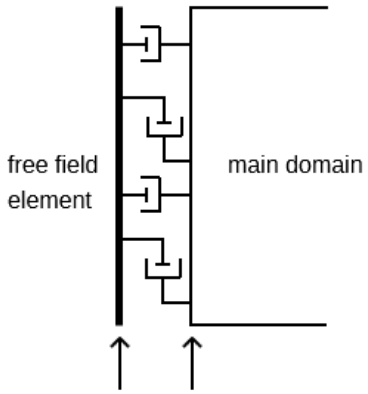
The options available for the model boundaries are (defaults in square brackets):

BoundaryXMin	*None, [Viscous], Free-field*
BoundaryXMax	*None, [Viscous], Free-field*
BoundaryYMin	*None, [Viscous], Free-field*

BoundaryYMax	*None*, *[Viscous]*, *Free-field*
BoundaryZMin	*[None]*, *Viscous*, *Compliant base*
BoundaryZMax	*[None]*, *Viscous*
All nodes fixities	*[None]*, *Fixed in x-direction*, *Fixed in y-direction* , *Fixed in z-direction*

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 511). This option is suitable for problems where the dynamic 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 - *Chapter 5 - 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 260 (on page 394)). 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 input is applied along the model bottom boundary. The input motion is then also applied at the base of the free-field elements.  *Figure 260: 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 511)) 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 dynamic 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.
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.
Fixed in z-direction	All nodes at the boundary are fixed in z 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.

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* condition 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 GSE

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 261 (on page 396)).

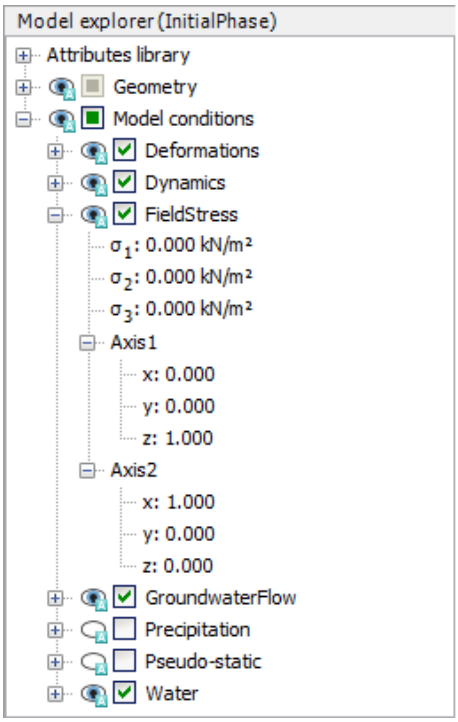


Figure 261: Boundary conditions for groundwater flow

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 329) for a more elaborate description of the Field stress option.

Groundwater flow boundary conditions

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 262 (on page 397)).

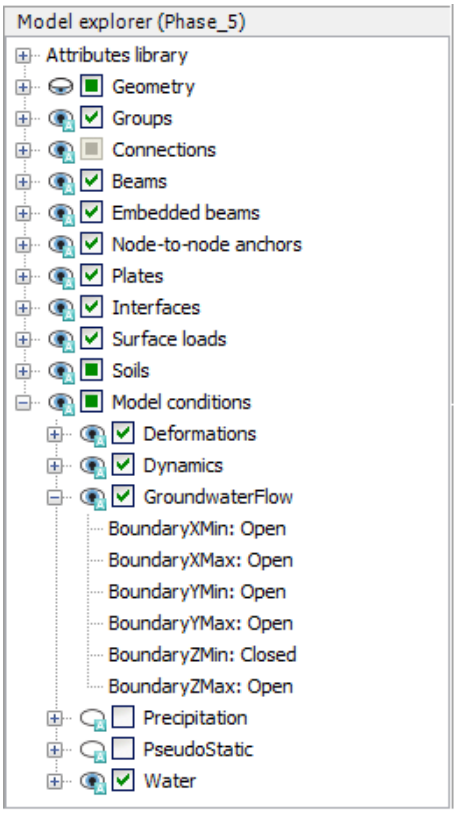


Figure 262: Boundary conditions for groundwater flow

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 (on page 380).

Precipitation

The *Precipitation* subtree in the *Model explorer* (Figure 263 (on page 398)) 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 (on page 381).

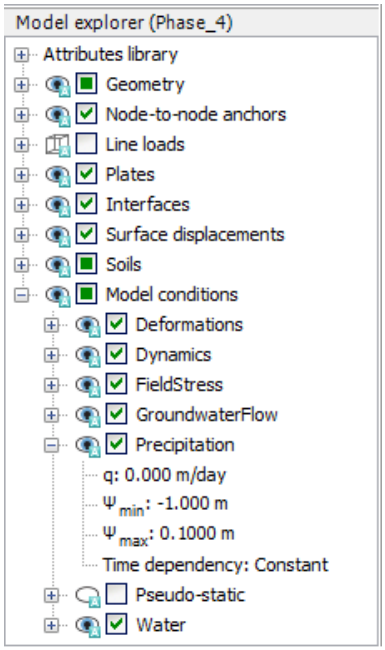


Figure 263: The Precipitation subtree in the Model explorer

Pseudo-Static

Dynamic forces can be considered in a pseudo-static way by specifying a global acceleration (Figure 264 (on page 398)). 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.

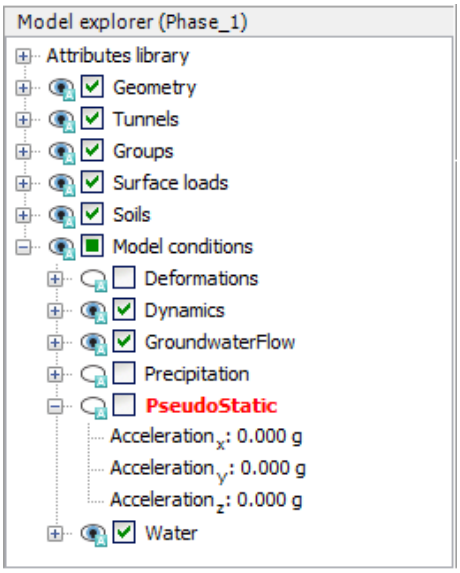


Figure 264: The PseudoStatic subtree in the Model explorer

Water

The global water level in a calculation phase can be defined in the *Water* subtree (Figure 265 (on page 399)). More information on *Global water level* is available in Assigning water conditions to soil volumes (on page 378).

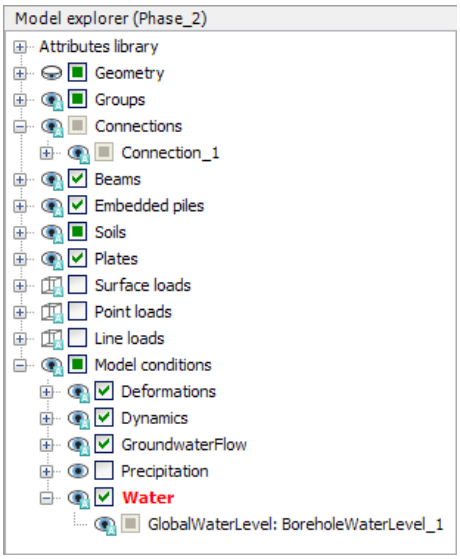


Figure 265: The Water subtree in the Model explorer

7.11 Special option

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 410) 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 project geometry is defined and the mesh is generated, 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 .

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 481).

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 3D finite element deformation analysis, information about the calculation process is presented in a separate window (Figure 266 (on page 401)). The phase being calculated is indicated in the *Phase* tabs.

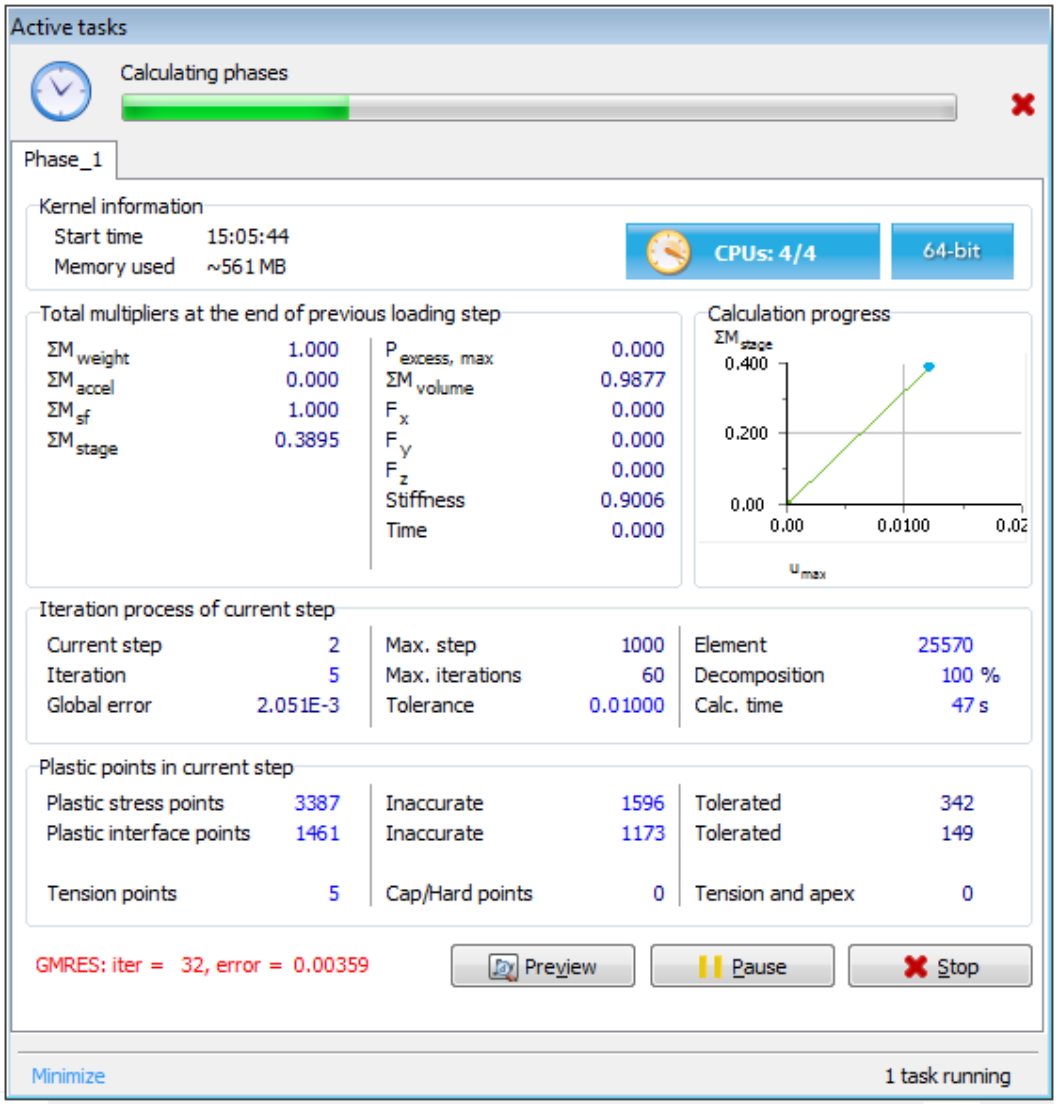


*Figure 266: The **Active tasks** window*

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 3D 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 337).
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.
ΣM_{volume}	It indicates the proportion of the total volume of soil clusters in the geometry model that is currently active. If all soil clusters are active, then the value is *1.000*.
ΣF_x, ΣF_y , ΣF_z	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 407) 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, $P_{excess,max}$ 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 407).
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 407).
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:

Reading data...	Reading input data from the disk.
Renumbering...	Optimising the node numbering and determining matrix properties.
Profile...	Determining the profile of the global stiffness matrix or pre-conditioner.
Forming matrix...	Forming the global stiffness matrix.
Forming pre-conditioner...	Forming the pre-conditioner for the iterative solution procedure.
Solving equations...	Solving the global system of equations to obtain the displacement increments.
Calculating stresses...	Calculating the strain increments and constitutive stresses.
Reaction forces...	Calculating the reaction forces and the out-of-balance force.
Writing data...	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 427)) 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 503)).

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 406).

7.12.6 *Selecting calculation phases for output*

After the calculation process has finished, the *Phases explorer* and **Phases** window are 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.

In the *Phases explorer* individual phases can be selected. When a calculation phase is selected, the *View results* button can be clicked after which the results are presented 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 **Geometry** modes, the model is automatically regenerated in the **Calculation** modes. The projects should be re-meshed. Note that the phases in the *Phases explorer* are marked for calculation automatically. The deleted geometry entities will not be any more available in the calculation phases. However, the program preserves the definition of the phases. If significant changes in the geometry are made then all settings need to be redefined, since PLAXIS 3D is not able to properly regenerate the settings automatically.

Modification of feature properties

The properties assigned to soil layers or features in the **Geometry** modes are the default properties for these in the calculation phases. When a property is changed (or modified) in the **Geometry** modes, the program will automatically change this property in the defined phases. However, if the property is already redefined in a phase (it is not the default any more), it will not be changed. The property as defined in the calculation phase will remain valid.

When a property is modified in a reference phase in the **Phase definition** modes, it will not be automatically modified in the following phases. The properties defined in the reference phase can be copied by using the *Regenerate* option in the right mouse button menu appearing when the feature is right clicked in the *Object explorer*. Regeneration consists of two steps:

- Determining which properties in which phases should be reset
- Finding out what the new values should be and applying those values

In the determination step by definition the properties of the target feature in the target phase are included. The properties of the target feature are examined in all child phases of the target. If the properties of target feature in a given child phase are identical to those in target phase, those properties will be included in the regeneration, and the process will repeat for all child phases of the child phase. Otherwise, the properties of target feature in the child phase will not be included in the regeneration process.

When establishing the new values, the properties of target feature in the parent phase of the target phase are examined. If the parent phase is the initial phase (which has no previous phase), the properties as defined in the geometry will be taken as source. These properties are then applied to everything that was selected in the determination process.

Note that the regeneration process is implicitly applied when switching from the **Geometry** modes to the **Calculation** modes, in order to maintain staged model integrity even if the geometry was modified. This procedure is mainly useful for quick propagation of staged construction changes to child phases.

7.12.8 *Automatic error checks*

During each calculation step, the PLAXIS 3D 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 3D 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:

$$\text{Global error} = \frac{\Sigma \| \text{ Out of balance nodal forces} \|}{\Sigma \| \text{ Active loads} \| \; + CSP \cdot \| \text{ Inactive loads} \|} \qquad \textit{Eq. [94]}$$

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.

CSP is the current value of the *Stiffness* parameter, defined as:

$$\text{Stiffness} = \int \frac{\Delta\varepsilon \cdot \Delta\sigma}{\Delta\varepsilon D^{e} \Delta\varepsilon} \qquad \textit{Eq. [95]}$$

which is a measure for the amount of plasticity that occurs during the calculation. See the Material Models Manual - *Chapter 2 - 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:

$$\text{AEC for structures} = \frac{\max \| \text{ Out of balance structure nodal moments} \|}{\max \| \text{ Structures nodal moments} \|} \qquad \textit{Eq. [96]}$$

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.

For embedded beams, an additional error check (AEC) is defined for the skin friction:

$$\text{AEC for skin friction} = \frac{\Sigma \| \text{ Out of balance nodal forces} \|}{\Sigma \| \text{ Nodal forces} \|} \qquad \textit{Eq. [97]}$$

Local error check

Local errors refer to the errors at each individual stress point. To understand the local error checking procedure used in PLAXIS 3D 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

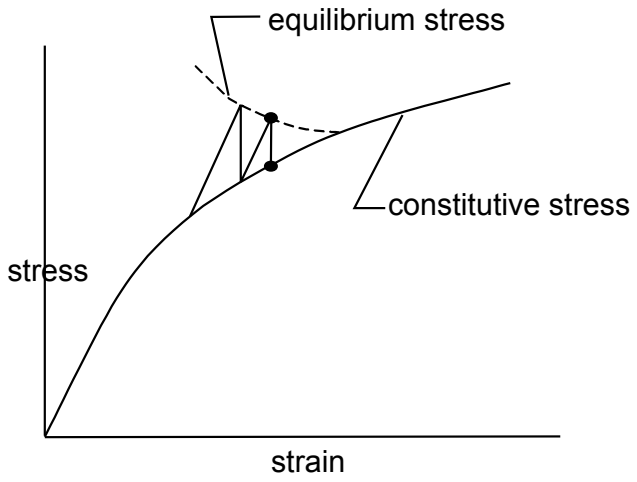


Figure 267: Equilibrium and constitutive stresses

At the end of each iteration, two important values of stress are calculated by PLAXIS 3D. The first of these, the 'equilibrium stress', is the stress calculated directly from the stiffness matrix (e.g. point **A** in Figure 267 (on page 408)). 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 267 (on page 408).

The dashed line in Figure 267 (on page 408) 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:

$$\text{Local error} = \frac{\| \ \sigma^e - \sigma^c \|}{T_{\max}} \qquad \textit{Eq. [98]}$$

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:

$$\| \sigma^e - \sigma^c \| = \frac{\sqrt{\left(\sigma_{xx}^e - \sigma_{xx}^c\right)^2 + \left(\sigma_{yy}^e - \sigma_{yy}^c\right)^2 + \left(\sigma_{zz}^e - \sigma_{zz}^c\right)^2 + \left(\sigma_{xy}^e - \sigma_{xy}^c\right)^2 + \left(\sigma_{yz}^e - \sigma_{yz}^c\right)^2 + \left(\sigma_{zx}^e - \sigma_{zx}^c\right)^2}}{} \qquad \textit{Eq. [99]}$$

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:

$$T_{\max} = \max\left(\frac{1}{2}\left(\sigma_3' - \sigma_1'\right),\ c \cdot \cos\varphi\right) \qquad \textit{Eq. [100]}$$

When the stress point is located in an interface element the following expression is used:

$$\text{Local error} = \frac{\sqrt{\left(\sigma_n^e - \sigma_n^c\right)^2 + \left(\tau^e - \tau^c\right)^2}}{c_i - \sigma_n^c \tan\varphi_i} \qquad \textit{Eq. [101]}$$

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 362)).

Termination of iterations

For PLAXIS 3D 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 512).

$$\begin{aligned} &\text{Maximum global error} \le \text{Tolerated error} \\ &\text{No. of inaccurate soil points} \le 3 + \frac{\text{No. of plastic soil points}}{10} \\ &\text{No. of inaccurate interface points} \le 3 + \frac{\text{No. of plastic interface points}}{10} \\ &\text{No. of inaccurate structure points} \le 3 + \frac{\text{No. of plastic structure points}}{10} \end{aligned} \qquad \textit{Eq. [102]}$$

where *Maximum global error* is the maximum between the *Global error*, the *Additional error check for skin resistance* 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 3D 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 421). In addition to the 3D view, it is possible to view output in individual work planes by selecting corresponding tabsheets in the **Output** window.

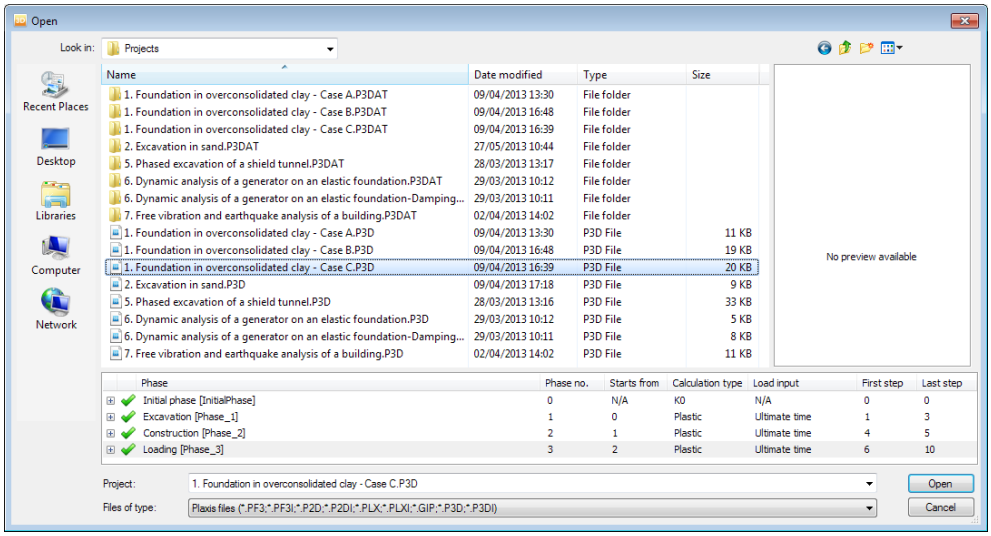


Figure 268: File requester of Output program

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:

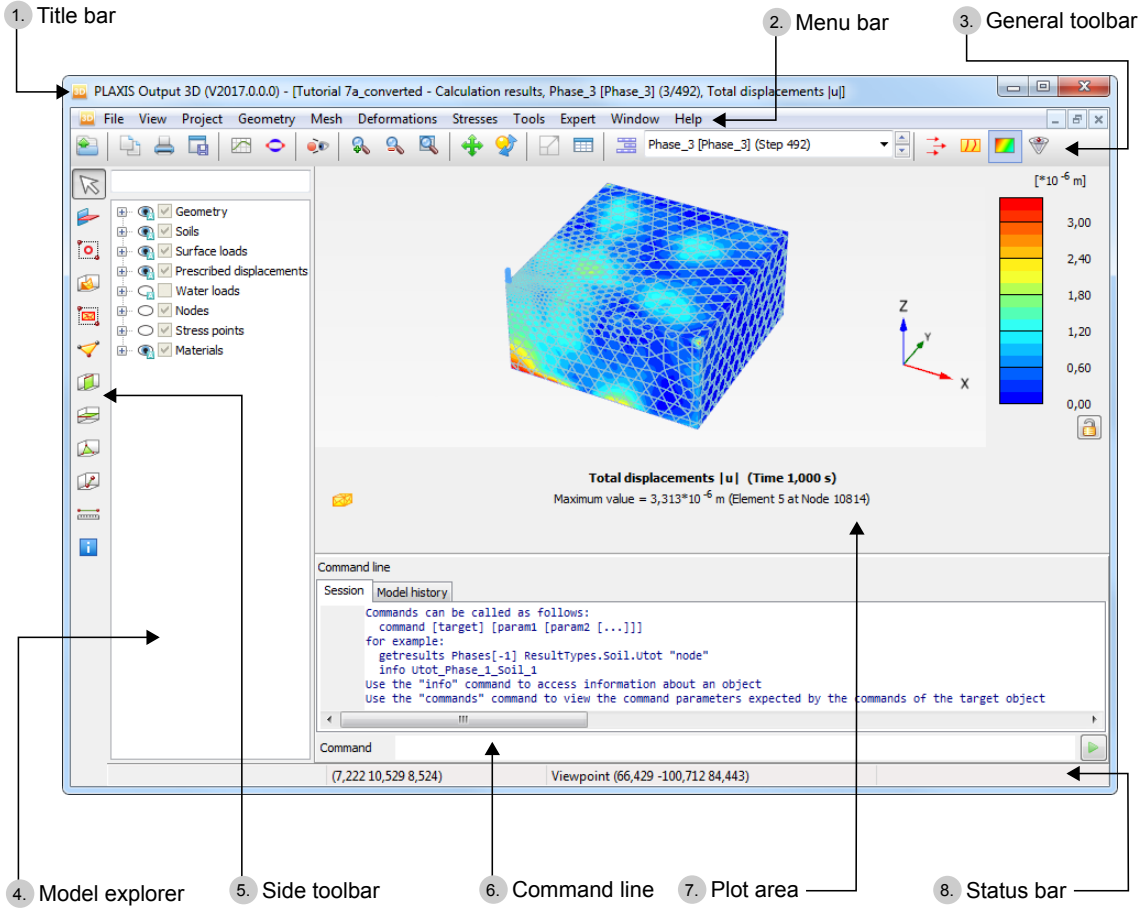


*Figure 269: **Main** window of the Output program*

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 412)).

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 434).

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 3D 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 62).

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 3D 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 3D project files are stored.
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).
Export to Paraview 	To export PLAXIS results to the visualization tool Paraview
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.
Viewpoint...	To change the viewpoint of the 3D projection of the model. This allows the input of a particular viewpoint or a selection from several predefined viewpoints.
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.
Title	To toggle the display of the title of the active plot in the caption.
Model explorer	To toggle the model explorer panel.
Legend	To toggle the display of the legend of contours or 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.
Shrink	To shrink the soil volumes in the model. This option is available in the **Model** view.
Expand	To expand the structures in the model. This option is available in the **Model** view.
Explode	To disassemble the components of the project geometry. This option is available in the **Model** view.
Implode	To assemble the components of the project geometry. This option is available in the **Model** view.
Settings	To set various graphical attributes, such as object and background colours, symbol size, font size and diffuse shading.

Note:

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 currently phase and active part of the structure.

8.2.3 *Project menu*

Node fixities	View the table of the node fixities.
Load information	View the table of the active loads and bending moments in the current step.
Water load information	View the table of the external water loads on the geometry boundaries in the current step.
Virtual interface thickness	View the table of the virtual interface thickness.
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.
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.
Rigid body configurations	View the rigid body configurations.
Rigid body results	View the rigid body results.

8.2.4 *Geometry menu*

Phreatic level	Toggle the display of the phreatic level in the model.
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 spheres	The quality of the elements in the mesh is defined as inner circle divided by the outer spheres of the soil or plate element where an equal sided tetrahedralis normalised at 1.0. The quality of a plate element in the mesh is defined as the inner circle divided by the outer circle of the plate element where an equilateral triangle is normalised at 1.0. For soil elements the displayed mesh elements in the model vary according to the quality value selected as the yellow bar is dragged through the legend.
Quality SICN	A signed inverse condition number to check the quality of the elements. This parameter takes into account the intermediate nodes for quality measurement hence it gives a better understanding of the quality for higher order elements. For soil elements the displayed mesh elements in the model vary according to the quality value selected as the yellow bar is dragged through the legend.
Table of quality	View the table of the quality of the soil or plate elements according to different criteria.
Volume	View the distribution of the volume of the soil elements.
Table of element volumes	View the table of the distribution of the volumes of the soil elements.
Connectivity plot	View the connectivity plot (Connectivity plot (on page 445)).
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 the assigned design approaches.

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 dynamic analysis) in the finite element model (Deformations (on page 446)). 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 451)). 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 462)).

8.2.9 Options

Autocomplete 	To select the *Significant digits* for the project.
Licencing information	To switch on or off showing the licence information at startup.

8.2.10 *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 481)).
Table	To open a new form with a table of numerical values of the presented quantity.
Vertical cross section	To select a user-defined vertical cross section with a distribution of the presented quantity. The cross section is defined by specifying two points in the corresponding window. To define the cut line in the plot, press the left mouse button at one end of the cross section; hold the mouse button down while moving the mouse to the other end of the line; release the mouse button. The cross section is presented on a new form.
Horizontal cross section	To select a user-defined horizontal cross section with a distribution of the presented quantity. The cross section must be selected by defining the *z*-coordinate. The cross section is presented on a new form.
Free cross section	To select a user-defined cross section with a distribution of the presented quantity. The cross section is defined by specifying three points in the corresponding window. The cross section is presented on a new form.
Line cross section	To define a line on which the distribution of the presented quantity is displayed. The line is defined by specifying two points in the corresponding window. The results on the defined line are presented on a new form.
Structural forces in volumes	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 piles (on page 469).
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 431)).
Remove annotations	To remove the annotations either partially (current view) or completely from the plots displayed in the output.

8.2.11 Expert menu

Run commands GSE	To open the **Commands runner** window.
Macro library GSE	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 (on page 542) and HTTP REST API (on page 568)).
Run Python script	To configure remote scripting server and open python script to run them (Appendices Python HTTP REST API wrapper (on page 542) and HTTP REST API (on page 568)).
Run Python tool	To configure remote scripting server and run python scripts (Appendices Python HTTP REST API wrapper (on page 542) and HTTP REST API (on page 568)).
Configure remote scripting server GSE	To specify an available port and open it for connections by local or remote clients (Appendices Python HTTP REST API wrapper (on page 542) and HTTP REST API (on page 568)).

8.2.12 *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.

8.2.13 *Help menu*

Manuals	To display the manuals.
Command reference	To display information about commands in the program.
Instruction movies	To reach the Bentley Geotechnical Engineering 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 3D website.
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.

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 3D 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 3D Output program. An extended overview and description of all available commands is accessible via the *Help* menu in the user interface of the PLAXIS 3D Output program.

More information is available in General information on command line (on page 56).

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 58)).

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 can be displayed by clicking the *View mesh* button in the **Mesh** mode in the Input program.
Pore pressures	The element configuration and pore pressure distribution in the in the *Phases explorer* can be displayed by clicking the *Preview phase* button in the **Flow conditions** and **Staged construction** modes in the Input program.
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 445).

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 **Staged construction** mode in the Input 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 413)).

8.4.2 *Exporting output data*

The PLAXIS 3D 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 270 (on page 422)).

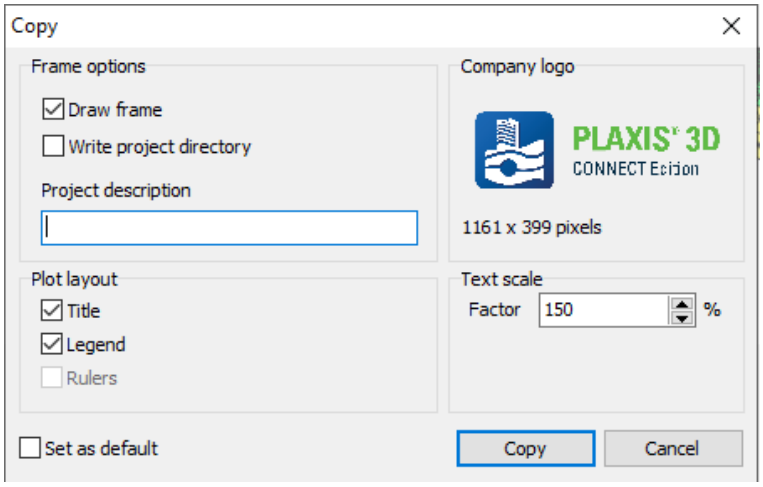


*Figure 270: **Copy** window*

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 271 (on page 423)).

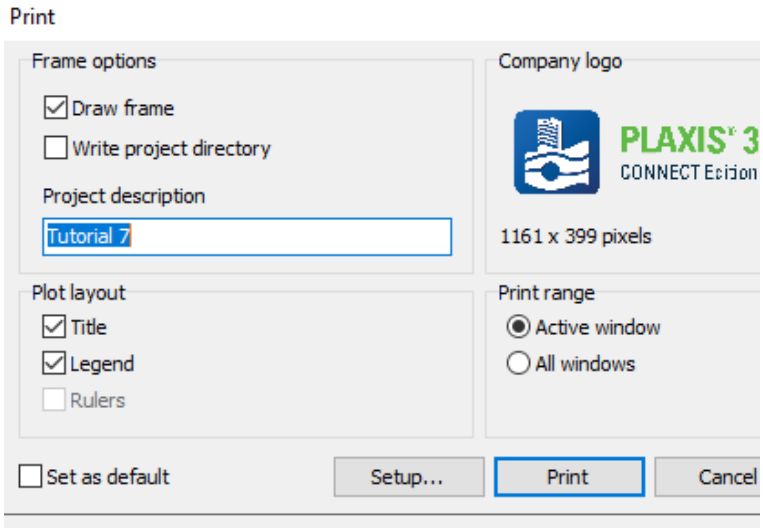


*Figure 271: **Print** window*

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.

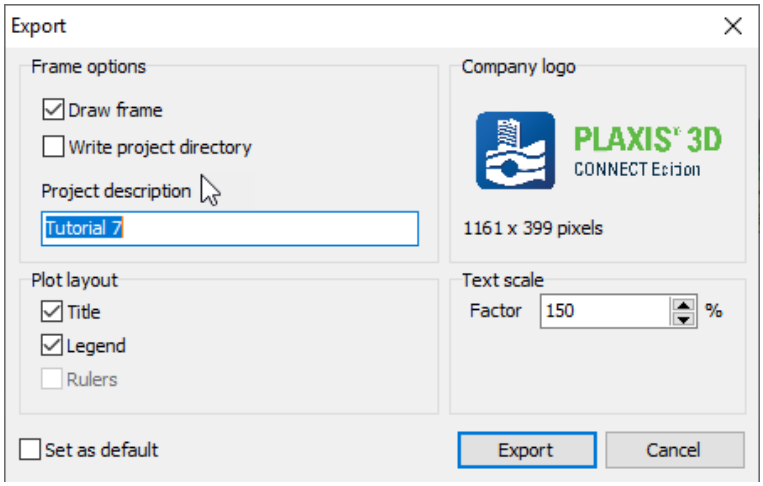


*Figure 272: **Export** window*

Instead of the PLAXIS 3D 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 481).

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 273 (on page 424) window which can be beneficial when the report is generated. Report generation is described in detail in Report generation (on page 438).

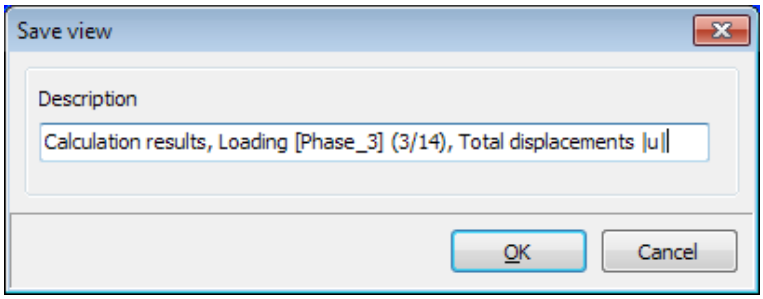


*Figure 273: **Save view** window*

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 413) 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 413) enables restoring the original plot.

8.4.6 Relocation and reorientation of the plot

The plot can be moved or rotated using the mouse. The default behaviour is selected by clicking on either the *Move* button or the *Rotate* button. Holding down the *Ctrl* key switches from *Move* to *Rotate* (or vice versa).

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.

When this button is clicked, the model will be rotated by clicking on the plot area and dragging it while keeping the left mouse button pressed.

The viewpoint changes as the model is moved or rotated. The location of the viewpoint can be defined by selecting the *Viewpoint* option in the *View* menu (View menu (on page 413)). When the option is selected, the **Viewpoint** window pops upFigure 274 (on page 425). A viewpoint can be set by either defining it in the **Viewpoint** window, or by selecting one of the default viewpoint options.

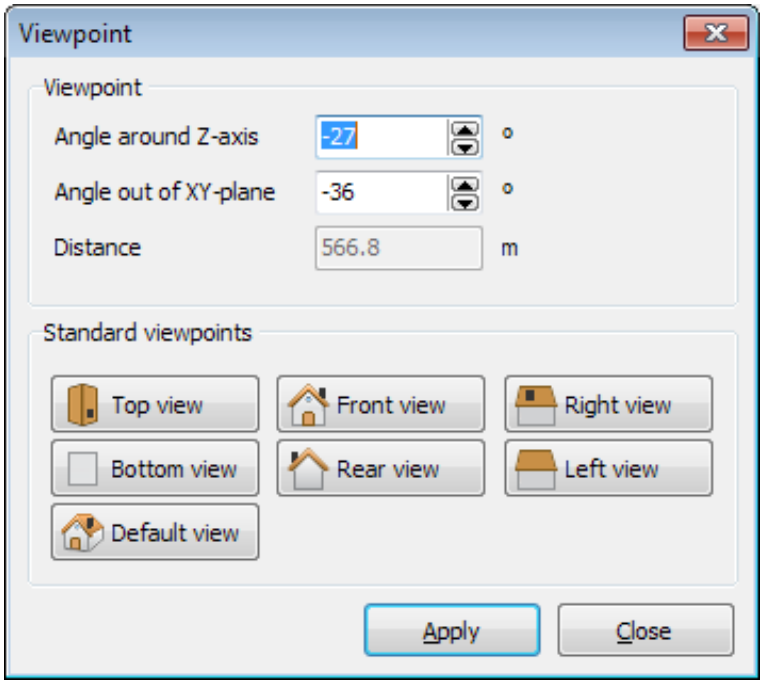


*Figure 274: **Viewpoint** window*

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 427)), 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.

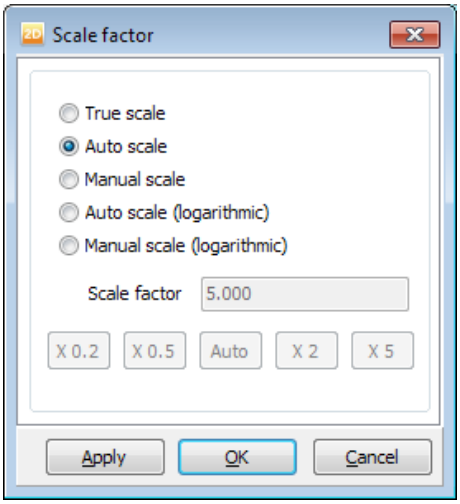


*Figure 275: **Scale factor** window*

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 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 are displayed as iso-areas.
-  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, geogrids or plates is displayed. 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 deformed shape of line structures (e.g. beam) is displayed. The relative deformation is indicated by arrows. Scaling of the results is possible.
-  The distribution of the results in line structures (e.g. beam) is displayed. Scaling of the results is possible.
-  The wireframe distribution of the results in line structures (e.g. beam) 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 459)). 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.

Note:

To quickly view structures inside the 3D model, the *Materials* option may be deselected from the *Geometry*.


Output for structures and interfaces can be obtained by clicking the *Select structures* button and then double clicking the desired object in the 3D 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 beams is selected, only other beams can be added to the selection and no embedded beams, plates or geogrids .

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 276 (on page 429).

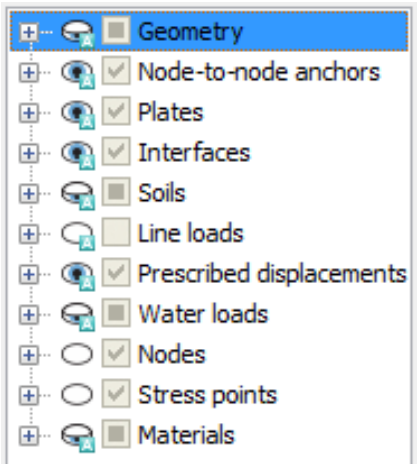


Figure 276: Model explorer in Output

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 3D 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 3D for all types of stresses and displacements in soil elements as well as structural elements and interfaces. The user can define vertical, horizontal, arbitrary and line cross sections.


The user may define vertical cross sections by clicking on the *Vertical cross section* button in the toolbar or by selecting the corresponding option from the *View* menu. Note that this option is also available in the right mouse click pop-up menu. Upon selection of this option, the 3D model is presented in top view with the *x*-axis pointing to the right and the *y*-axis pointing upwards. The cross section can be specified precisely by clicking on one end of the cross section line in the plot and moving the cursor to the other end while holding down the mouse button. Cross sections exactly in *x*- or *y*-direction may be drawn by holding down the *Shift* key on the keyboard while drawing the cross section. Alternatively, both cross section coordinates can be entered precisely in the **Cross section points** window that appears at the upper left hand corner.

 In addition to the vertical cross section, the user may define horizontal cross sections by clicking on the *Horizontal cross section* button in the toolbar or by selecting the corresponding option from the *View* menu. Upon selection of this option, the **Cross section height** window pops up in which the *z*-coordinate can be defined.

 Free cross sections can be defined by clicking on the *Free cross section* button in the toolbar or by selecting the corresponding option from the *View* menu. Upon selection of this option, the **Cross section points** window pops up in which the three cross section coordinates can be defined.

 A line cross section can be defined by clicking the *Line cross section* button in the side button bar or by selecting the *Line cross section* option in the *View* 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 3D 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 3D 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 *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 277 (on page 432)).

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).

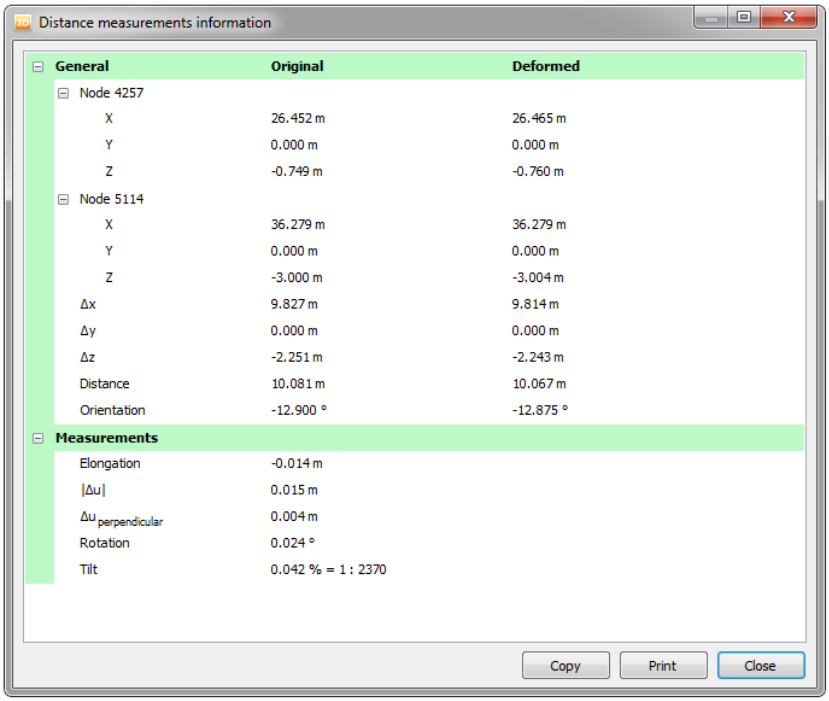


Figure 277: Distance information

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.
Δ z	The *Original* and *Deformed* z-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 278 (on page 433)). 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 horizontal (*XY*) plane.. Orientation is positive if the vector describing the line points upwards (*Z>0*) and negative if it points downwards (*Z<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 278 (on page 433)). This angle is always between *0* ° and *180* °.
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: $$\text{Tilt} = \tan(\alpha_{\text{Tilt}}) = \frac{\Delta u_{\text{perpendicular}}}{\text{v}}$$

Note:

- A normal vector to the measurement line is used to calculate the *Rotation*, *Tilt* and $\Delta u_{perpendicular}$.
- If the original measurement line is nearly vertical (i.e. an angle with the vertical of *+/- 5* degrees or *180 +/- 5* degrees), then the normal vector *N* is defined as the vector product between the y-axis and the original vector formed by the two selected points.
- Otherwise, the original measurement line is considered mostly horizontal. The normal vector *N* is defined as the vector product between the original vector formed by the two selected points and the vector N_{plane}. N_{plane} is defined as the vector product between the z-axis and the original vector obtained by the two selected points.

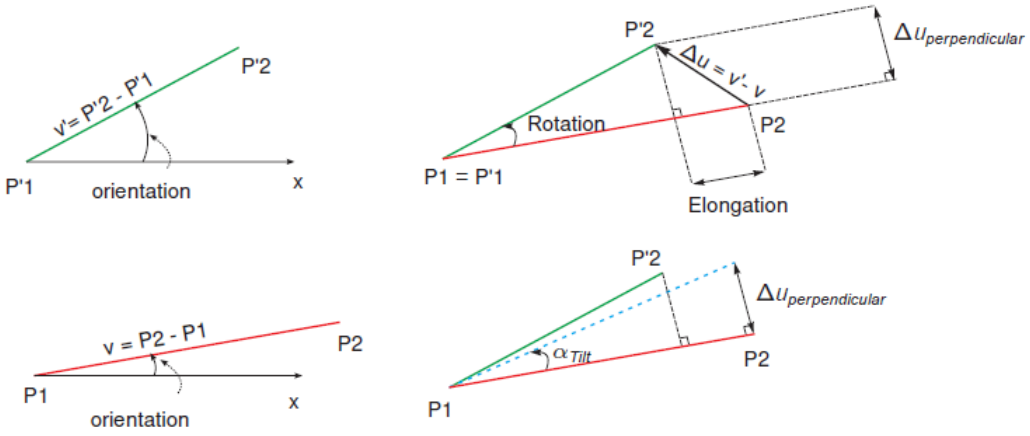


Figure 278: Deformation measurement

Draw scanline

When the *Contour lines* option is selected for the plot in cross sections or plates, 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 481).

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 413)).

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 413).

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 279 (on page 435)). 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.

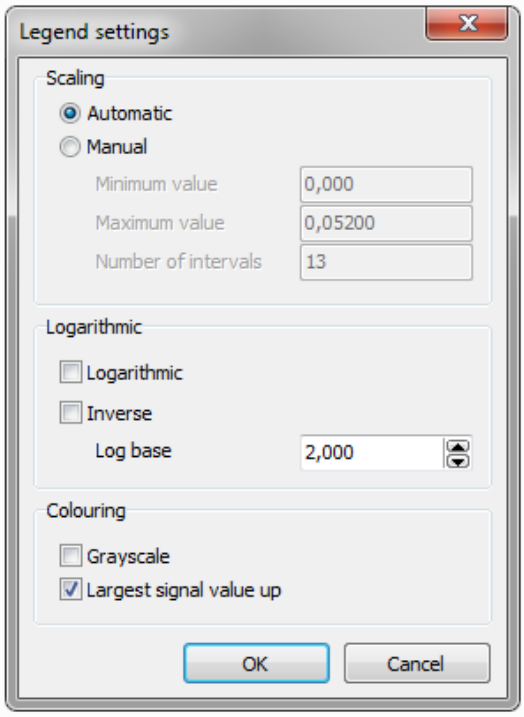


Figure 279: ***Legend settings*** *window*

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 280 (on page 436)).

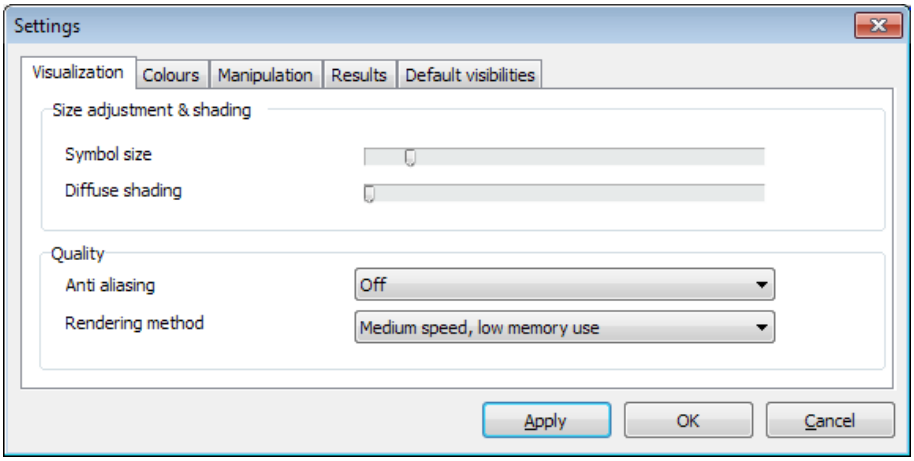


*Figure 280: **Visualization** tabsheet of the **Settings** window*

Symbol size	To modify the size of the symbols in the display for nodes, forces, etc.
Diffuse shading	To make the appearance of the 3D 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 281 (on page 437)).

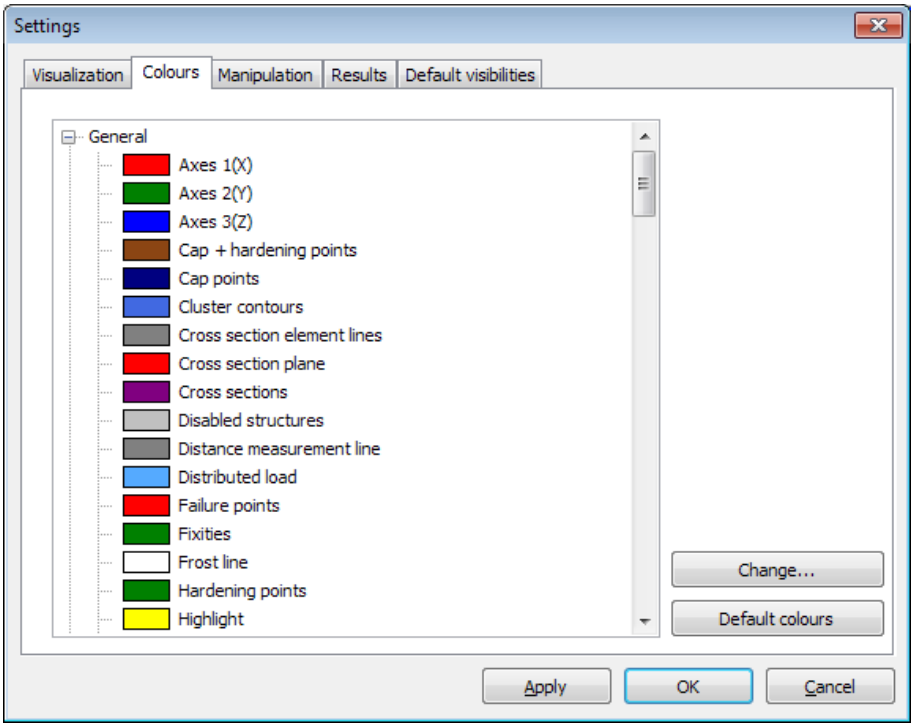


*Figure 281: **Colours** tabsheet of the **Settings** window*

The function of the left and the middle mouse buttons can be defined in the **Manipulation** tabsheet of the **Settings** window (Figure 282 (on page 437)).

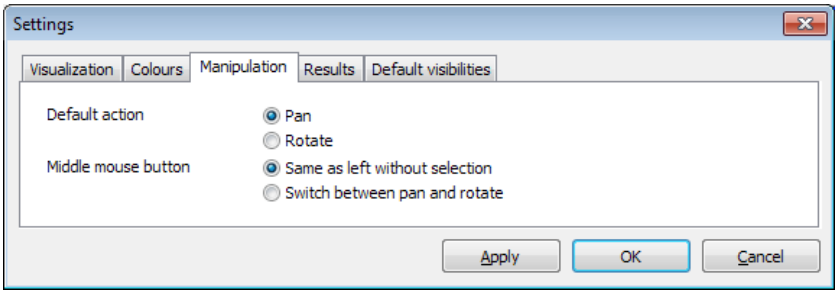


*Figure 282: **Manipulation** tabsheet of the **Settings** window*

The display of particular results can be toggled on/off in the **Results** tabsheet of the **Settings** window (Figure 283 (on page 437)).

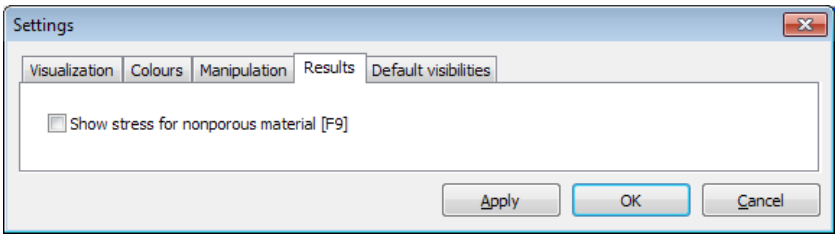


*Figure 283: **Results** tabsheet of the **Settings** window*

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 284 (on page 438)).

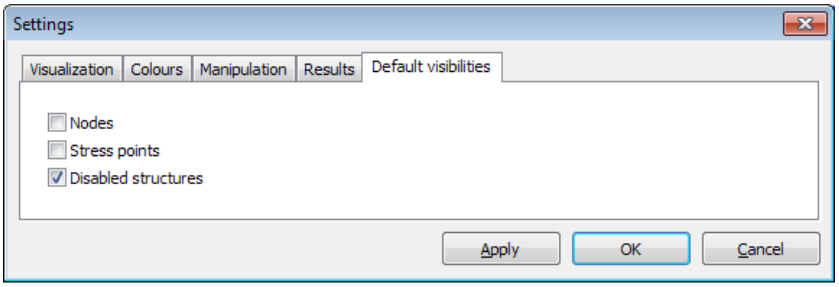


*Figure 284: **Default visibilities** tabsheet of the **Settings** window*

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 *Line cross section* view the results in the defined line cross section are displayed.
- In the *Vertical cross section* view the results in the defined vertical cross section are displayed.
- In the *Horizontal cross section* view the results in the defined horizontal cross section are displayed.
- In the *Free cross section* view the results in the defined free cross section are displayed.

8.7 Report generation



To document project input data and computational results, a *Report Generator* facility is available in the PLAXIS 3D 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 285 (on page 439)).

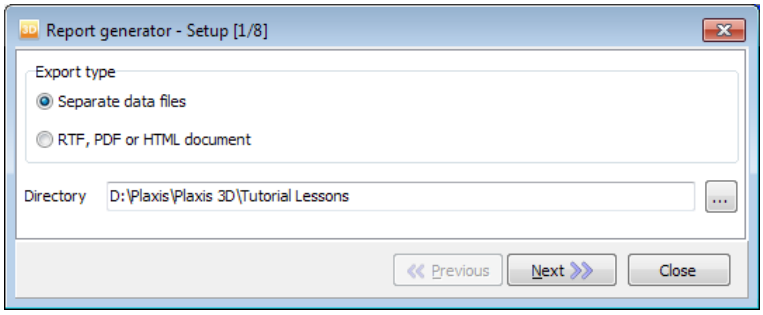


Figure 285: Report generator - Setup

Step 2: Select the phases for which results will be included in the report (Figure 286 (on page 439)).

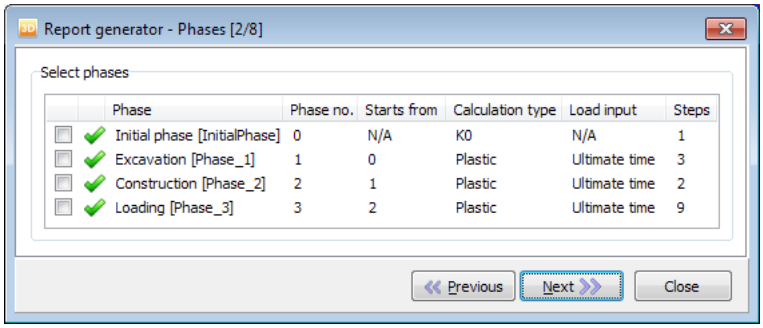


Figure 286: Report generator - Phases

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 287 (on page 440)).

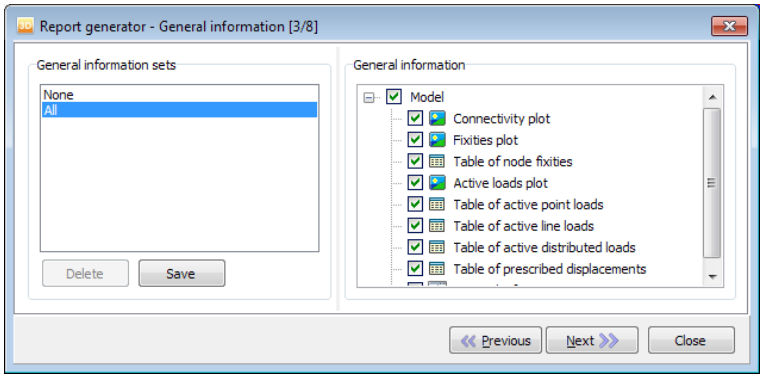


Figure 287: Report generator - General information

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 288 (on page 440)).

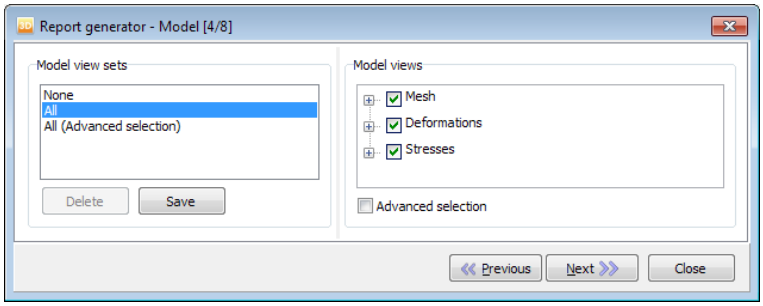


Figure 288: Report generator - Model

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 289 (on page 440)).

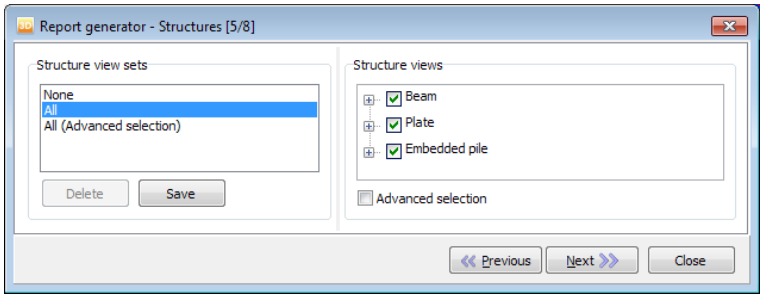


Figure 289: Report generator - Structures

Step 6: Select saved views to be included in the report (Figure 290 (on page 441)). For more details on saved views, see Store the view for reports (on page 424).

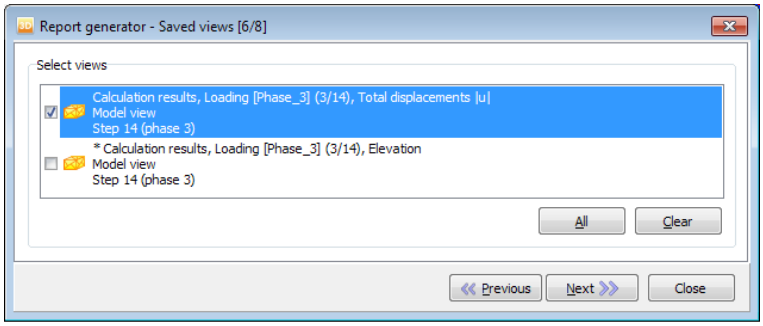


Figure 290: Report generator - Saved views

Step 7: Select generated charts to be included in the report (Figure 291 (on page 441)).

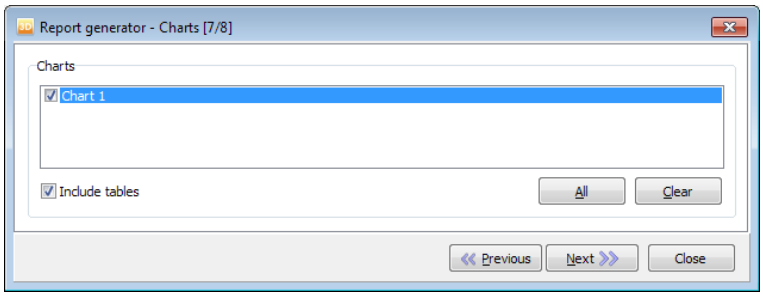


Figure 291: Report generator - Charts

Step 8: A summary of the number of rows and the number of figures in the report is given (Figure 292 (on page 441)). The report is created as the *Export* button is clicked. A progress bar appears displaying the number of the remaining rows and images.

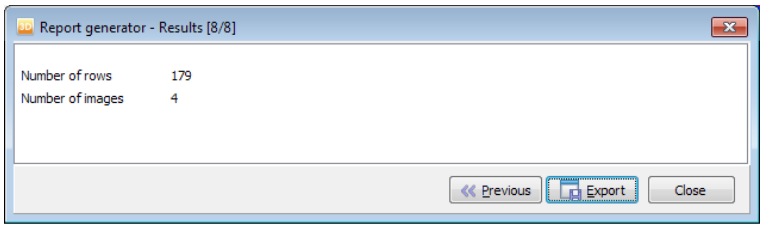


Figure 292: Report generator - Results

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 293 (on page 442)) 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.

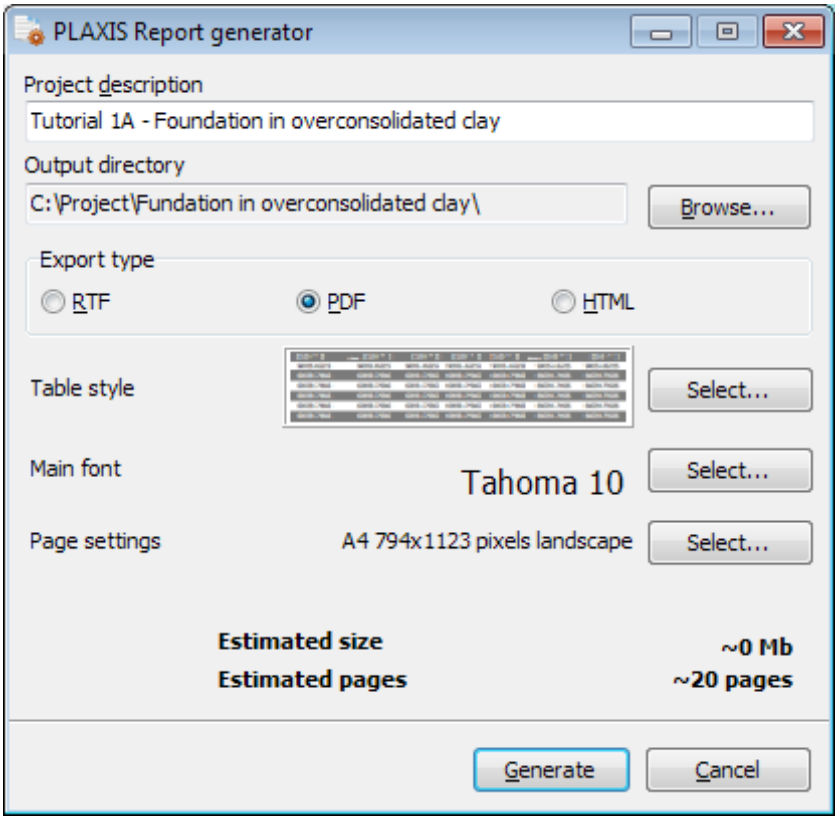


Figure 293: Document properties

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 294 (on page 443)). 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.

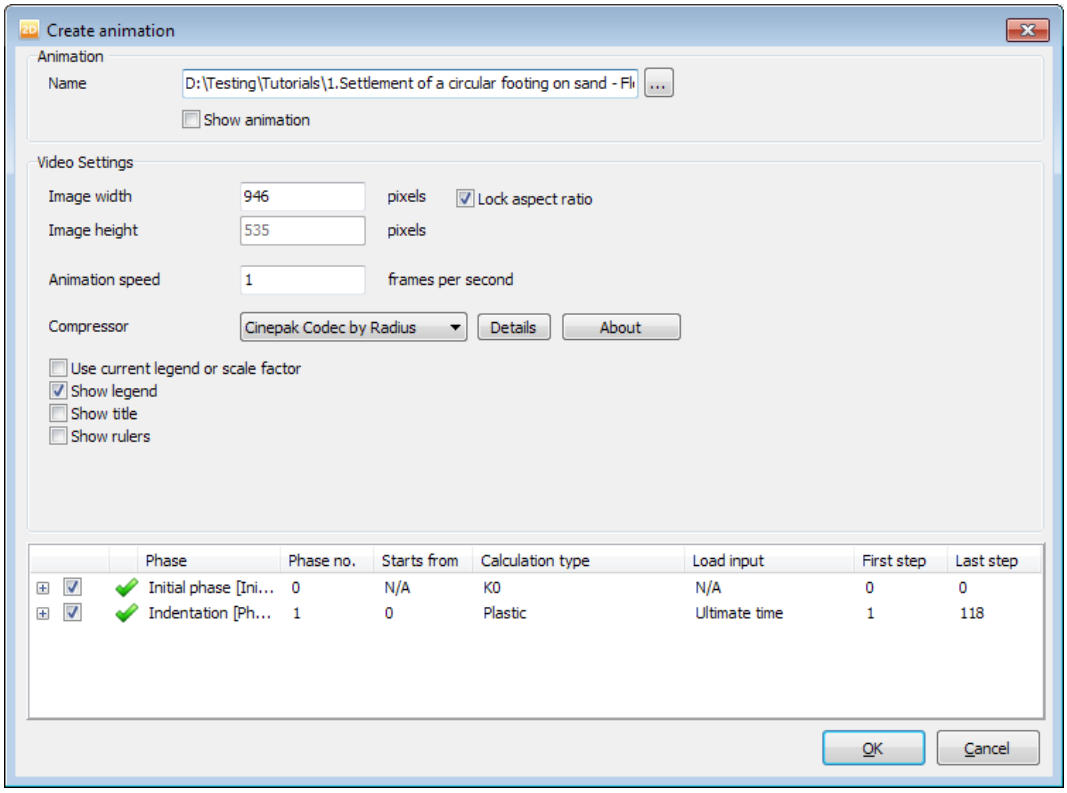


*Figure 294: Selection of phases from **Create animation** window*

8.9 Paraview Export

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 3D. The tool can be accessed in the **Output Program** and clicking the **File** menu (See Figure 295 (on page 443)).

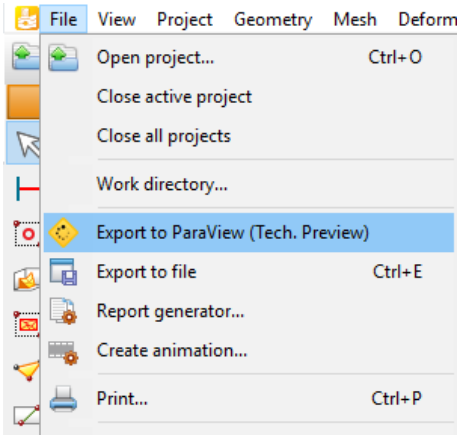


Figure 295: Export to ParaView

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 296 (on page 444)). Finalize the process by selecting the necessary results and click in ***Export***.

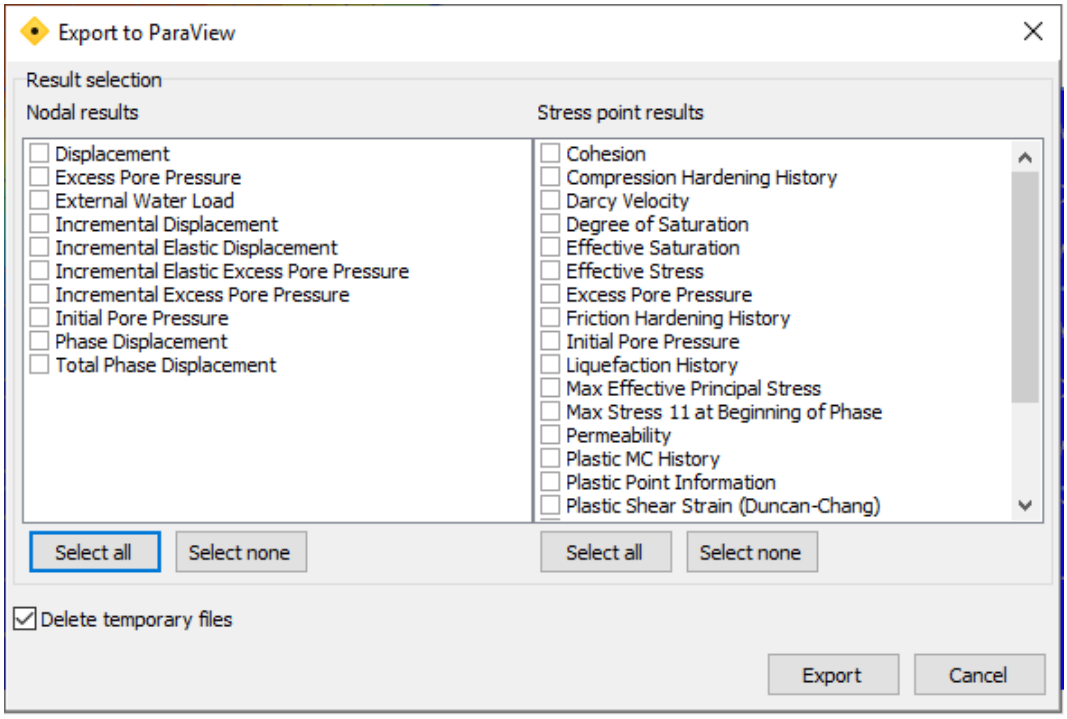


Figure 296: Export to ParaView -Result selection

Note:

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 297 (on page 446).

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, provided that the *Hide items without strength reduction* option in the **View** menu has been selected. 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 435).

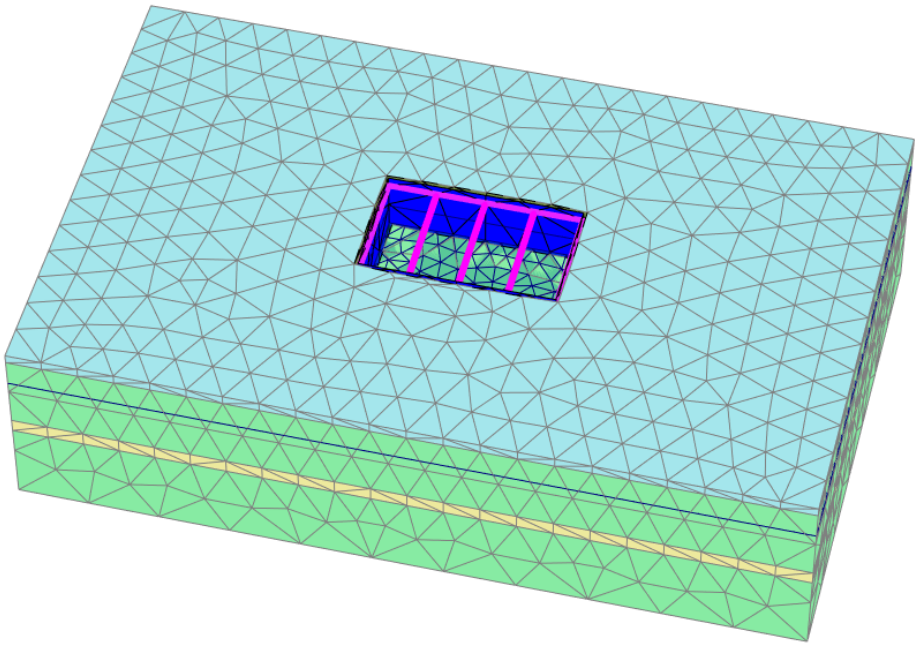


Figure 297: Example of the ***Connectivity plot***

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 Elevation

The height of the soil volume is displayed when the *Elevation* option is selected in the *Deformations* menu.

9.2.2 *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 425)) may be used. The deformed mesh plot may be selected from the *Deformations* menu.

9.2.3 *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, u_y and u_z. The total displacements may be presented as *Arrows*, *Contour Lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

9.2.4 *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, Pu_y and Pu_z. The phase displacements may be presented as *Arrows*, *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

9.2.5 *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 298 (on page 448)(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 298 (on page 448)(b)). The more construction layers are used, the smoother the settlement profile is (Figure 298 (on page 448)(b)).



Figure 298: 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.

9.2.6 *Incremental displacements* DYN

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, Δu_y and Δu_z. The displacement increments may be presented as *Arrows*, *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)). The contours of displacement increment are particularly useful for the observation of localisation of deformations within the soil when failure occurs.

9.2.7 *Velocities* DYN

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, $|v|$, and the individual velocity components, v_x, v_y and v_z. The velocities may be presented as *Arrows*, *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

9.2.8 *Accelerations*

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, $|a|$, and the individual acceleration components, a_x, a_y and a_z. The accelerations may be presented as *Arrows*, *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

9.2.9 *Accelerations in 'g'*

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, $|a('g')|$, and the individual acceleration components, $a_x('g')$, $a_y('g')$ and $a_z('g')$. The accelerations may be presented as *Arrows*, *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

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 six individual Cartesian strain components ε_{xx}, ε_{yy}, ε_{zz}, γ_{xy}, γ_{yz} and γ_{zx}. The individual strain components may be presented as *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

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 six individual Cartesian strain components $P\varepsilon_{xx}$, $P\varepsilon_{yy}$, $P\varepsilon_{zz}$, $P\gamma_{xy}$, $P\gamma_{yz}$ and $P\gamma_{zx}$.

The individual strain components may be presented as *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar Display type of results (on page 427)).

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 six individual Cartesian strain components $\Delta\varepsilon_{xx}$, $\Delta\varepsilon_{yy}$, $\Delta\varepsilon_{zz}$, $\Delta\gamma_{xy}$, $\Delta\gamma_{yz}$ and $\Delta\gamma_{zx}$.

The individual strain components may be presented as *Contour lines*, *Iso surfaces* or *Shadings* by clicking the appropriate button in the toolbar (Display type of results (on page 427)).

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 , $(\varepsilon_1 + \varepsilon_3)/2$, $(\varepsilon_1 - \varepsilon_3)/2$, 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:

  $$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} \qquad \textit{Eq. [103]}$$

  In *Updated mesh* calculations:

  $$\varepsilon_v = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} + \varepsilon_{xx}\varepsilon_{yy} + \varepsilon_{xx}\varepsilon_{zz} + \varepsilon_{yy}\varepsilon_{zz} + \varepsilon_{xx}\varepsilon_{yy}\varepsilon_{zz} \qquad \textit{Eq. [104]}$$
- The deviatoric strain is calculated as:

  $$\varepsilon_q = \sqrt{\frac{2}{3}\left[\left(\varepsilon_{xx} - \frac{\varepsilon_v}{3}\right)^2 + \left(\varepsilon_{yy} - \frac{\varepsilon_v}{3}\right)^2 + \left(\varepsilon_{zz} - \frac{\varepsilon_v}{3}\right)^2 + \frac{1}{2}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right)\right]} \qquad \textit{Eq. [105]}$$
- The void ratio is calculated as:

  $$e = e_0 + (1 + e_0)\varepsilon_v \qquad \textit{Eq. [106]}$$

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 435)).

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 six individual Cartesian stress components σ'_{xx}, σ'_{yy}, σ'_{zz}, σ_{xy}, σ_{yz} and σ_{zx}.

Figure 299 (on page 451) shows the sign convention adopted for Cartesian stresses. Note that pressure is considered to be negative.

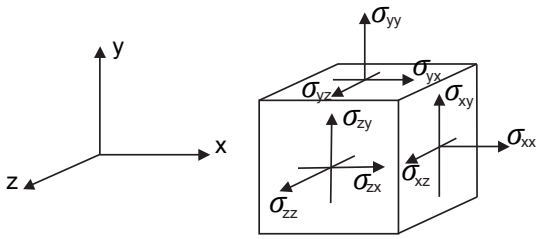


Figure 299: Sign convention for stresses

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 six individual Cartesian stress components σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz} and σ_{zx}. The latter three quantities are equal to the corresponding ones in the *Cartesian effective stress* option, but are repeated here for convenience (Cartesian effective stresses (on page 451)). 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 mean effective stress p', the deviatoric stress q, the relative shear stress τ_{rel} and the mobilised shear strength τ_{mob}.

Note that the effective stress components are arranged in algebraic order:

$$\sigma_1^{'} \leq \sigma_2^{'} \leq \sigma_3^{'} \qquad \textit{Eq. [107]}$$

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:

$$\tau_{\mathrm{rel}} = \frac{\tau_{\mathrm{mob}}}{\tau_{\mathrm{max}}} \qquad \textit{Eq. [108]}$$

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.

$$\tau_{\mathrm{max}} = -\frac{\sigma'_1 + \sigma'_3}{2}\sin\varphi + c\cos\varphi \qquad \textit{Eq. [109]}$$

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:

$$f_{HB} = \sigma'_1 - \sigma'_3 + \overline{f}(\sigma'_3) = 0 \qquad \textit{Eq. [110]}$$

the maximum shear stress is defined by :

$$\tau_{\mathrm{max}} = \frac{1}{2}\overline{f}(\sigma'_3) \quad \text{where} \quad \overline{f}(\sigma'_3) = \sigma_{ci}\left(m_b\frac{-\sigma'_3}{\sigma_{ci}} + s\right)^a \qquad \textit{Eq. [111]}$$

The relative shear stress is correspondingly defined by:

$$\tau_{rel} = \frac{\tau_{mob}}{\tau_{\mathrm{max}}} = \frac{|\sigma'_1 - \sigma'_3|}{\overline{f}(\sigma'_3)} \qquad \textit{Eq. [112]}$$

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 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 451)).

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 300 (on page 453)) 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 301 (on page 454)).

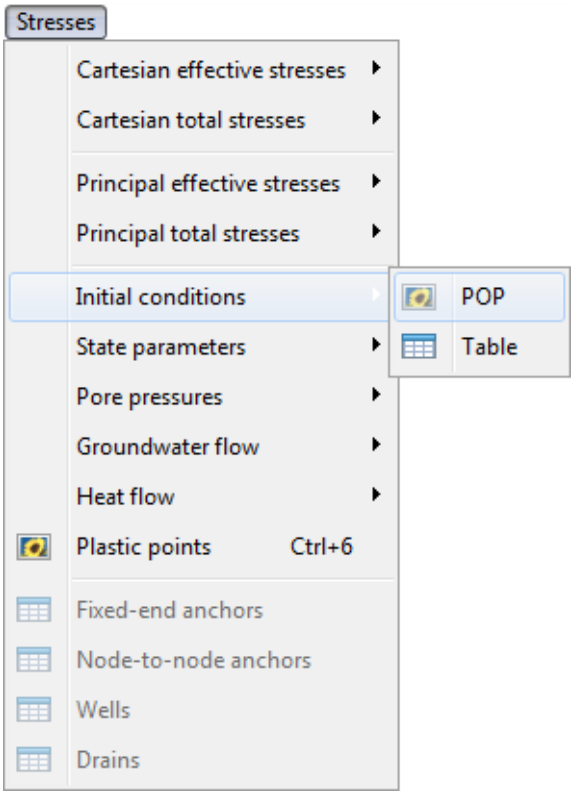


Figure 300: Initial conditions in the Stresses menu (Initial phase)

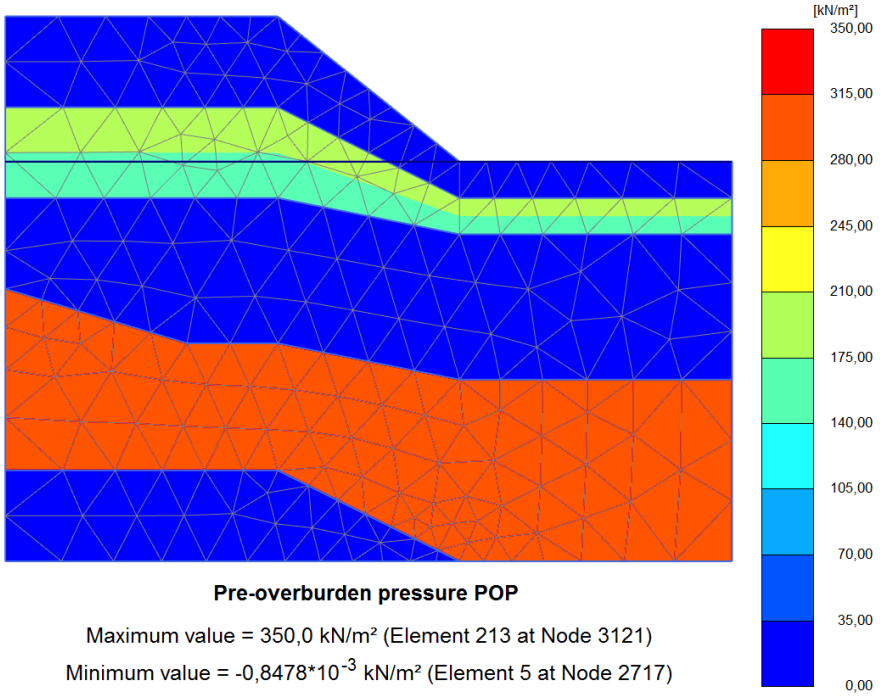


Figure 301: Example of POP distribution

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 ($Permeability_{actual,x}$, $Permeability_{actual,y}$, $Permeability_{actual,z}$) is the relative permeability times the saturated permeability. This value depends on the degree of saturation according to the Van Genuchten (or other) relationship 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, ε_{xy}, ε_{yz} and ε_{zx} 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 **- Chapter 7 - 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

Hardening Soil model

p_{eq}**:** The equivalent isotropic stress p_{eq} is only available in the , Hardening Soil model with small-strain stiffness, Soft Soil model, Soft Soil Creep model, Modified Cam-Clay 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:

$p_{eq} = \sqrt{p^2 + \frac{q^2}{a^2}}$	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.

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. 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. 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 actual Young's modulus *E*

The actual Young's modulus E is the unconstrained elastic stiffness modulus as used during the current calculation step. This option is only available in the Linear Elastic model and Mohr-Coulomb model.

When the Linear Elastic model or the Mohr-Coulomb model is utilised with an increasing stiffness with depth ($E_{increment} > 0$), this option may be used to check the actual stiffness profile used in the calculation. Note that in the Linear Elastic model and the Mohr-Coulomb model the stiffness is NOT stress-dependent.

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.

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 3D 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 3D calculation.

Total stresses are divided into effective stresses, σ', and active pore pressures, p_{active}.

$$\sigma = \sigma' + p_{active} \qquad Eq.\ [113]$$

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}.

$$p_{active} = S_{\text{eff}} \cdot p_{water} \qquad Eq.\ [114]$$

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:

$$h = z - \frac{p_{water}}{\gamma_w} \qquad Eq.\ [115]$$

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}.

$$p_{water} = p_{steady} + p_{excess} \qquad Eq.\ [116]$$

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 3D Output program. Most quantities can be presented as *Contour lines*, *Shadings* or *Iso surfaces*. 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.

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

Any positive value (tension) in the active pore pressure. When multiplied by the tangent of the friction angle, the effective suction gives a kind of 'artificial cohesion' in the soil.

9.3.8 *Groundwater flow*

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, q_z).

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, q_z)

The total discharge through a cross section can be obtained by drawing a cross section through the interface element and selecting *Stresses* → *Groundwater flow*→$|q|/q_x/q_y/q_z$. The value of total discharge is displayed below the plot in the unit of volume per unit of time .

Saturation

The PlaxFlow module within PLAXIS 3D 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, i_y, and i_z. These can be presented as *Contour lines*, *Shadings*, *Arrows*, and *Iso surfaces*.

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 *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 3D mesh, or in the elements around the work planes 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.

Soil

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 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 symmetry plane 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.

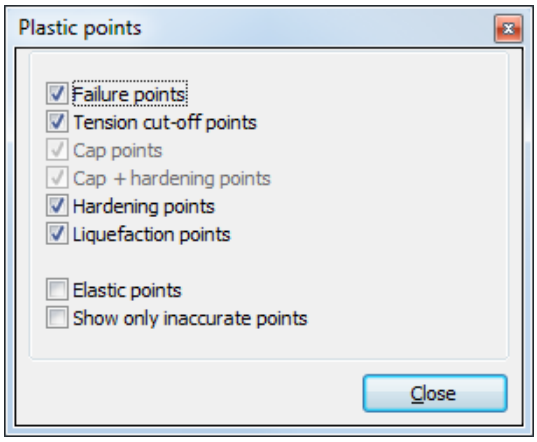


Figure 302: Plastic points window

When *Plastic points* is selected in the *Stresses* menu the *Plastic points* dialog is shown (Figure 302 (on page 460)). 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 407)). When the results are displayed with the *Table* option, an asterisk appears on the status column to identify the inaccurate plastic points.

Structures

The plastic stress points are indicated by small symbols that can have different shapes and colours, depending on the type of structural element.

For elastoplastic beams:

- A red cube (Plastic point) indicates that the stress has reached the plastic value.

For elastoplastic plates:

- A red pyramid (Plasticity in local 1 direction only) indicates that the stress reached the plastic value only in first axial direction.

- A gray upside-down pyramid (Plasticity in local 2 direction only) indicates that the stress reached the plastic value only in second axial direction.
- A purple diamond (Plasticity in local 1 and 2 direction) indicates that the stress reached the plastic value in both directions.

For plates, the user can select which types of plastic points are displayed. For instance, if only plasticity in local 1 direction is checked, points that are plastic in 1 and 2 direction are not shown, since they represent two different type of plasticity.

Conventions used for Plastic point information with Paraview Export 

If *plastic point information* is exported from PLAXIS 3D to ParaView (see Paraview Export (on page 443)), the point information will have associated different status described in Conventions used for Plastic point information with Paraview Export (on page 461).

Table 19: 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

9.3.10 *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.

9.3.11 *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.

9.3.12 *Wells*

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.

9.3.13 *Drains*

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 3D 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 446)). For each item a further selection can be made among the displacement vectors $|u|$, and the individual total displacement components, u_x, u_y and u_z.

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, u_2 and u_3.

The *Rotation* option is available for *Plates* and *Beams* displaying the total rotation (*Rotation*) and the incremental rotation (*ΔRotation*) of the selected plates with respect to the global axes.

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 *Surface 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:

$$C_{ref}^{line} = \frac{\Delta L}{L_0} = \frac{L - L_0}{L_0} \qquad \textit{Eq. [117]}$$

$$C_{ref}^{surf} = \frac{\Delta A}{A_0} = \frac{A - A_0}{A_0} = \frac{(R_0 + \Delta R)^2 - R_0^2}{R_0^2} \cong 2\frac{R - R_0}{R_0} = 2\frac{L - L_0}{L_0} \qquad \textit{Eq. [118]}$$

The formulation above does not only apply to a circular surface but to all closed surfaces.

9.4.3 *Resulting forces in beams*

When a beam is displayed, the options *Axial force N*, *Shear force Q_{12}*, *Shear force Q_{13}*, *Bending moment M_2* and *Bending moment M_3* are available from the *Forces* menu. Note that positive axial forces refer to tension whereas negative forces refer to compression (Figure 303 (on page 464)). The sign of bending moments and shear forces depend on the beam's local system of axes (Figure 304 (on page 464) (a)).

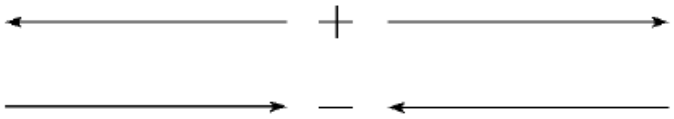


Figure 303: Sign convention for axial forces in beams and plates

The *Structure axes* option from the *View* menu may be used to display the beam's local system of axes (1,2,3). The first direction is always the axial direction. The second and third axis are always perpendicular to the beam axis.

The *Axial force N* is the force in the first (axial) direction (Figure 304 (on page 464) (b)). An axial force is positive when it generates tensile stresses, as indicated in Figure 303 (on page 464).

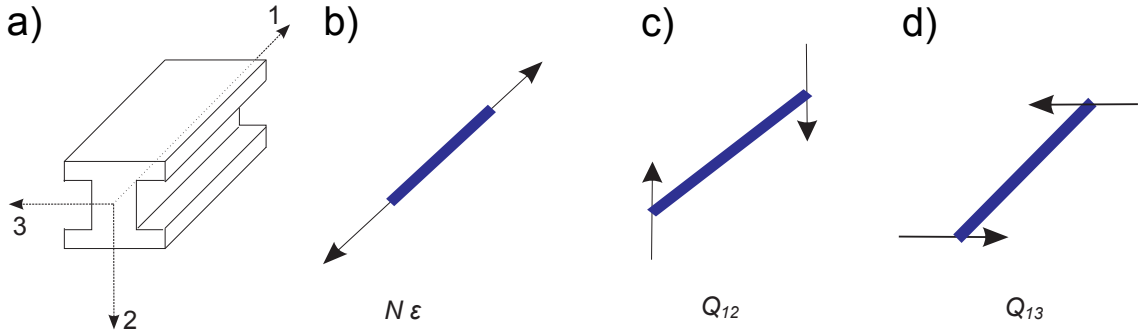


Figure 304: Positive axial force and shear forces in horizontal beams Axial force and shear forces. a) Local axes. b) Axial force N. c) Shear force Q_{12}. d) Shear force Q_{13}

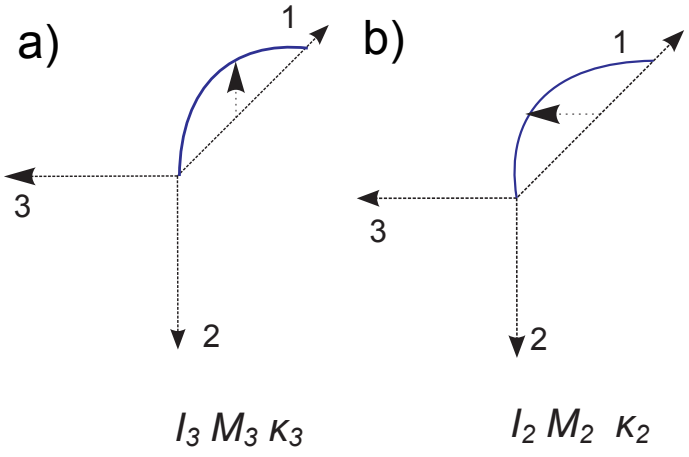


Figure 305: Positive bending moments in beams. a) Bending Moment M_3, b) Bending Moment M2

The *Shear force* Q_{12} is the shear force over the second beam axis (Figure 304 (on page 464) (c)), whereas the *Shear force* Q_{13} is the shear force over the third beam axis (Figure 304 (on page 464) (d)).

The *Bending moment* M_3 is the bending moment due to bending around the third axis (Figure 305 (on page 464) (a)), whereas the *Bending moment* M_2 is the bending moment due to bending around the second axis (Figure 305 (on page 464) (b)). The *Bending moment* M_1 is the *Torsion* (bending around the first axis).

9.4.4 *Resulting forces in embedded beams*

When an embedded beam is displayed, the options *Axial force N*, *Shear force* Q_{12}, *Shear force* Q_{13}, *Bending moment* M_2, *Bending moment* M_3 , *Skin force* T_{skin} (in axial pile direction) and lateral forces T_{lat} and T_{lat2} are available from the *Forces* menu. The latter three options relate to the pile-soil interaction (see below). For a detailed description of structural forces in the embedded beam elements see Resulting forces in beams (on page 463).

Note:

The *Axes* option from the *View* menu may be used to display the pile's local system of axes (1,2,3). The first direction is always the axial direction. The second direction is perpendicular to the pile axis with a component in the global x-direction and the third direction is also perpendicular to the pile with a component in the global z-direction.

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, 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 material data set (Material set (on page 308)).

The interaction force T_{lat} relates to the relative displacement perpendicular to the pile in the pile's second direction whereas the interaction force T_{lat2} relates to the relative displacement perpendicular to the pile in the pile's third direction. These quantities are expressed in the unit of force per unit of pile length. Note that T_{lat} and T_{lat2} are not limited. In fact, when these forces become very large, plasticity will occur outside the elastic zone in the surrounding soil volume elements.

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, 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 material data set (Material set (on page 308)).

9.4.5 *Resulting forces in plates*

When a plate is displayed, axial forces N_1 and N_2, shear forces Q_{12}, Q_{23} and Q_{13} and moments M_{11}, M_{22} and M_{12} are available from the *Forces* menu. These forces represent the actual forces at the end of the calculation step.

Note that axial forces are positive when they generate tensile stresses, as indicated in Figure 283 . The sign of bending moments and shear forces depend on the plate's local system of axes.

The *Structure axes* option from the *View* menu may be used to display the plate's local system of axes (1,2,3). The first and second direction lie in the plane of the plate whereas the third direction is perpendicular to the plate.

The *Axial force* N_1 is the axial force in the first direction (Figure 306 (on page 466) (b)). The *Axial force N2* is the axial force in the second direction (Figure 306 (on page 466) (c)).

The *Shear force* Q_{12} is the in-plane shear force (Figure 307 (on page 466) (a)). The *Shear force* Q_{13} is the shear force perpendicular to the plate over the first direction (Figure 307 (on page 466) (b)), whereas the *Shear force* Q_{23} is the shear force perpendicular to the plate over the second direction (Figure 307 (on page 466) (c)).

The *Bending moment* M_{11} is the bending moment due to bending over the second axis (around the second axis) (Figure 308 (on page 467) (b)). The *Bending moment* M_{22} is the bending moment due to bending over the first axis (around the first axis) (Figure 308 (on page 467) (c)).

The *Torsion moment* M_{12} is the moment according to transverse shear forces (Figure 308 (on page 467) (a)).

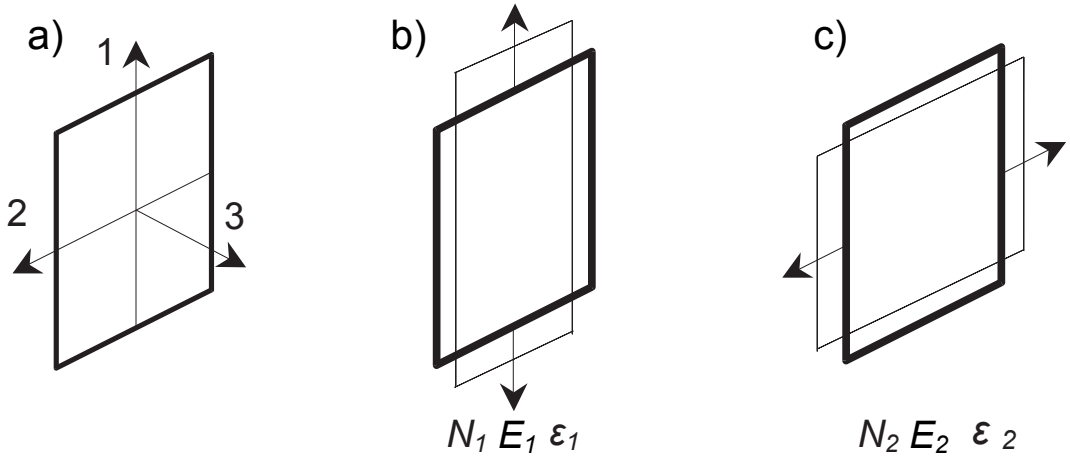


Figure 306: Positive axial forces in plates and geogrids. a) Local plate directions, b) Axial force N_1, c) Axial force N_2

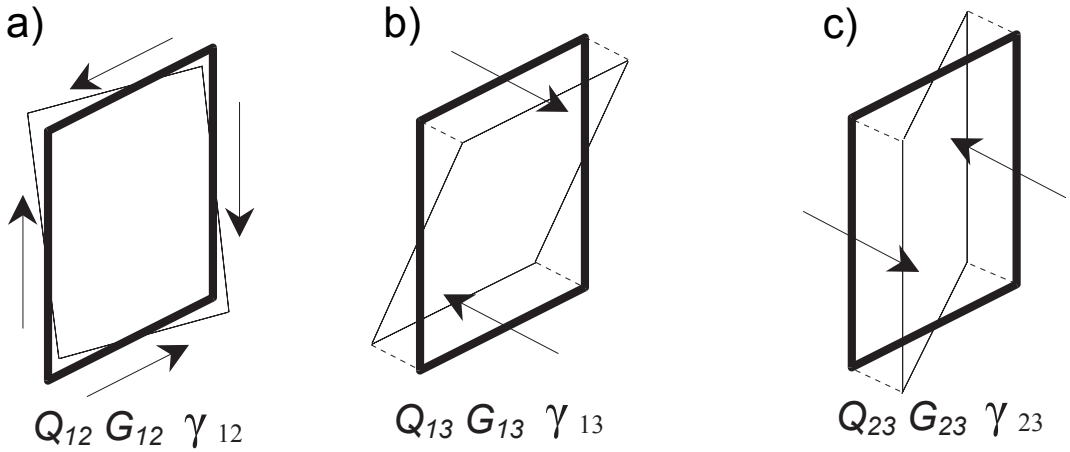


Figure 307: Positive shear forces in plates. a) Shear force Q_{12}, b) Shear force Q_{13}, c) Shear force Q_{23}

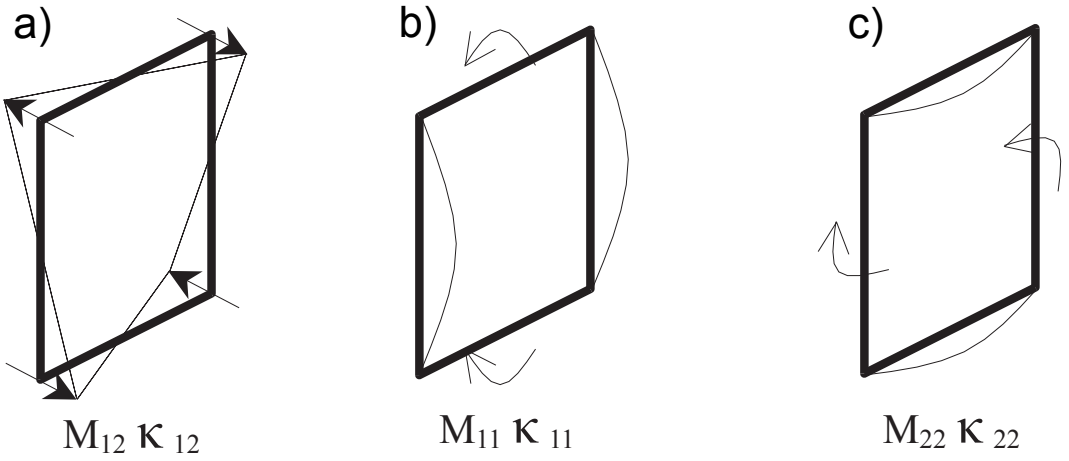


Figure 308: Positive bending moments in plates. a) Torsion moment M_{12}, b) M_{11}, c) Bending moment M_{22}

9.4.6 *Resulting forces in geogrids*

When a geogrid is displayed, the options *Axial force N_1* and *Axial force $_2$* are available from the *Forces* menu (Figure 306 (on page 466)). Forces in geogrids are always positive (tension). Compressive forces are not allowed in these elements.

9.4.7 *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.

9.4.8 *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 *Geometry* menu. Output for interfaces can be obtained by double clicking on the interface elements in the three-dimensional model. The output for interfaces comprises deformations (Deformations (on page 446)) and stresses.

When an interface is displayed, the options *Effective σ'_N*, *Total σ_N*, *Shear τ_1*, Shear τ_2, *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, *Groundwater head* and *Plastic points* 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:

$$\tau_{rel} = \frac{\sqrt{\tau_1^2 + \tau_2^2}}{\tau_{\max}} \qquad Eq.\ [119]$$

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

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. 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.

9.4.9 *Resulting forces in rigid bodies*

Output for rigid bodies can be viewed by selecting *Rigid body* results in the *Project* menu of the Output program. After selection, a window opens with four tabsheets, containing tables of rigid body input data and results. The main results can be found in the tabsheets *Deformations* and *Forces*.

The *Deformations* tab sheet contains the displacements and rotations that have been applied to the rigid bodies or are the result after applying forces / moments. They are shown as incremental, phase and total values as follows.

Symbol	Name
$\|u\|$	Total displacements
φ	Rotations
$\|Pu\|$	Phase displacements
$P\,\varphi$	Phase rotation
$\Delta\, u$	Incremental displacement
$\Delta\, \varphi$	Incremental rotation

The *Prescribed displacements* tab contains input values of displacements and rotations as follows.

Symbol	Name
$\|u\|$	Input prescribed displacement value
φ	Input rotation value

In case a displacement/rotation is prescribed in the reference point (both 0 or non-zero), the total reaction force is based on ALL forces involved, i.e. stresses in soil and interfaces, forces in structural elements, as well as external loads and body forces (self-weight of the rigid body). In case a force/moment is prescribed in the reference point (both 0 or non-zero), the result of the calculation is a displacement (as listed in the *Deformations* tab) as well as a reaction force (i.e. the sum of stresses external loads and body forces, as listed in the *Forces* tab).

The *Forces* tab sheet contains the resulting forces and bending moments that have been applied to the rigid bodies or are the result after applying displacements / rotations. They are shown as total values, together with their input value as follows.

Symbol	Name
$\|F_{reac}\|$	Reaction force of the soil
$\|M_{reac}\|$	Reaction bending moment of the soil
$\|F\|$	Input force value
$\|M\|$	Input bending moment value
$\|F_a\|$	Applied force (based on the *ΣMstage* value)
$\|M_a\|$	Applied bending moment (based on the *ΣMstage* value)
$\|F_{prev}\|$	Calculated force from previous phase
$\|M_{prev}\|$	Calculated bending moment from previous phase

9.4.10 *Structural forces in volume piles*

 The *Structural forces in volume piles* feature is available in the toolbar or as an option in the *Tools* menu in the Output program (Tools menu (on page 417)). 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.

- The generation of the structural forces may take awhile if the generated mesh has many elements.

Creating a cross section line

When selecting the *Structural forces in volumes* option, a cross section line should be drawn in longitudinal direction through the centre of the area that forms the structure.

 Hide the clusters which are not of interest for generation of the structural forces, using the toggle in the *Model explorer*. Select the *Line cross section* button in the side toolbar, the **Cross section points** window pops up. A cross section line needs to be drawn along the structure which is of interest. The points defining the section composing the cross section line can be defined either by directly clicking on the model or by defining the coordinates of the location in the **Cross section points** window. The process of cross section line definition must be completed by clicking the *OK* button in the **Cross section points** window. The **Local axes** window pops up with approximated values. This can be accepted by clicking the *OK* button. Then the structural forces are computed on-the-fly and visualised along the created line (Figure 309 (on page 471)).

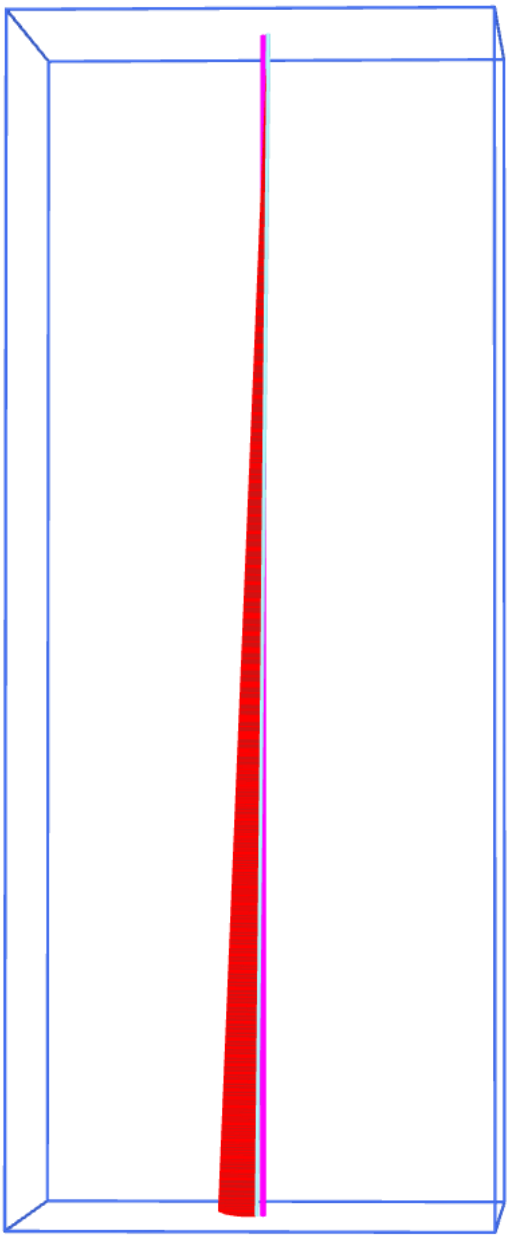


Figure 309: Figure displaying the structural force in a volume pile

Importing a centerline

When selecting the *Structural forces in volumes piles* option, it is also possible to import a centerline of a cylinder.

Select the *Import centerlines* button in the side toolbar. The **Select centerlines to import** window pops up (Figure 310 (on page 472)). The specific volumes can be selected for which structural forces need to be displayed. Note that importing is only possible for whole cylinders.

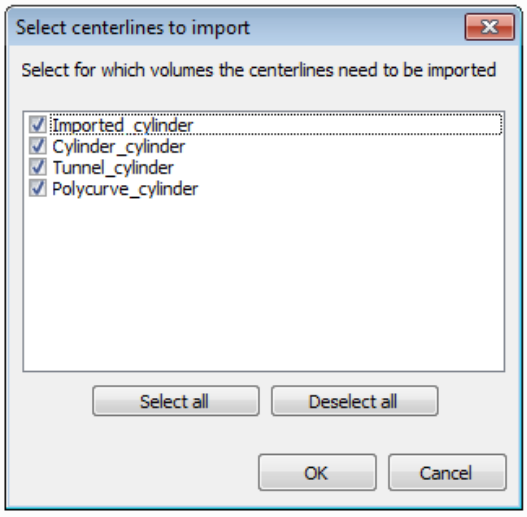


Figure 310: ***Import centerline*** *window*

Automatic center lines wizard

As an alternative to the import of a center line, the center line can be automatically detected using the *Generate center lines* wizard.

The automatic center lines wizard can be used in two situations as described below.

Structural forces in volume piles

By clicking the *Structural forces in volume piles* button, the *Automatic center lines wizard* pops up (Figure 311 (on page 473)). This can now be used to generate center lines specifically only for volume piles.

A user can use the tabsheets as described below to generate center lines using the automatic center lines wizard tool.

Criteria	In the *Criteria* tabsheet (Figure 311 (on page 473)), it is possible to select the materials whose center line is required for calculation of structural forces. Individual volume clusters can be specified in the right-hand panel by clicking on the checkbox *Specify clusters* and selecting or deselecting specific soil volumes. The clusters can be: multi-selected (*Ctrl*), selected all (*Ctrl+A*) and selected in a range (*Shift+Arrowkeys* or *Shift +PageUp/Down*).

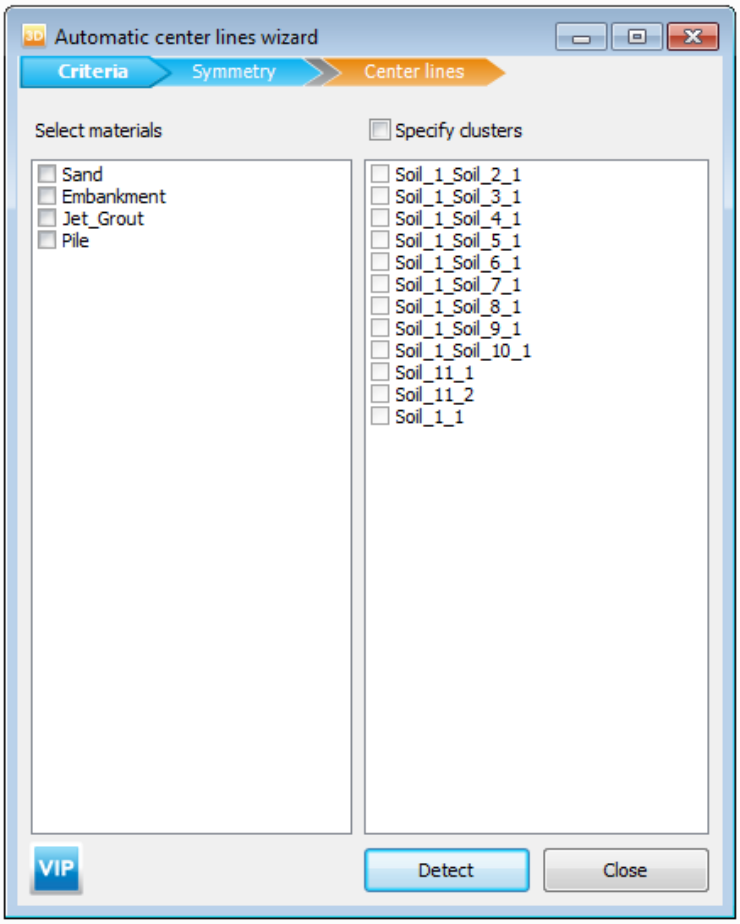


Figure 311: Automatic center lines wizard Criteria tabsheet

Since structures can be modelled as a half or even a quarter of the complex structure (considering planes of symmetry in the model), it is important for a proper visualisation of the structural forces to indicate if such symmetry planes exist.

Symmetry	This blue tabsheet (Figure 312 (on page 474)) allows users to select which horizontal and / or vertical planes should act as a symmetry plane. This can be done, by selecting anyone of the following checkboxes, x_{min}, x_{max}, y_{min} or y_{max}. The user may skip this tabsheet if no symmetry plane is desired.

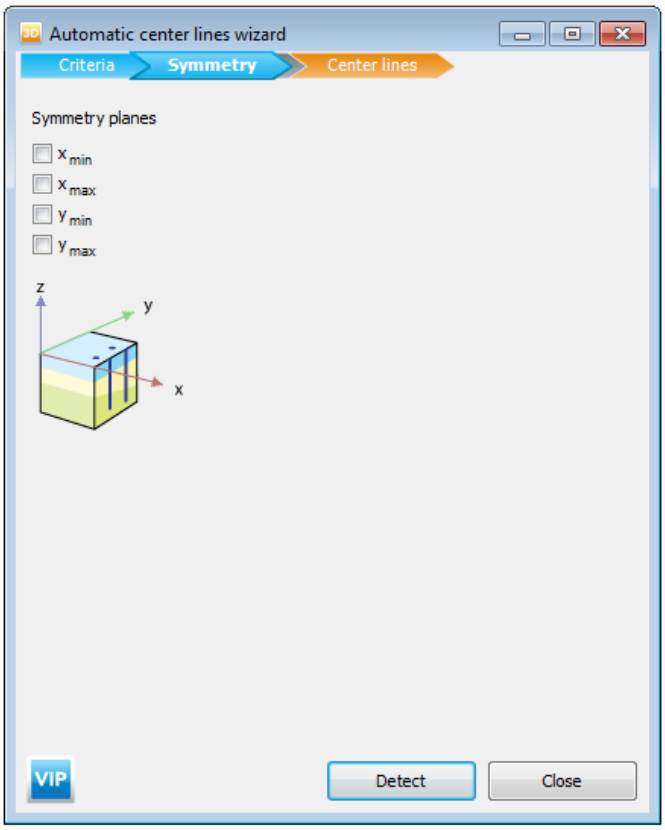


Figure 312: Automatic center lines wizard Symmetry tabsheet

Once the *Criteria* and *Symmetry* have been selected, the user may then proceed to click on the button *Detect* in order to generate the center lines.

Center lines	In this orange tabsheet (Figure 313 (on page 475)), the results for the center lines are shown. Here it is possible to select specific center lines of interest. Each center line can easily be toggled to show or hide it.

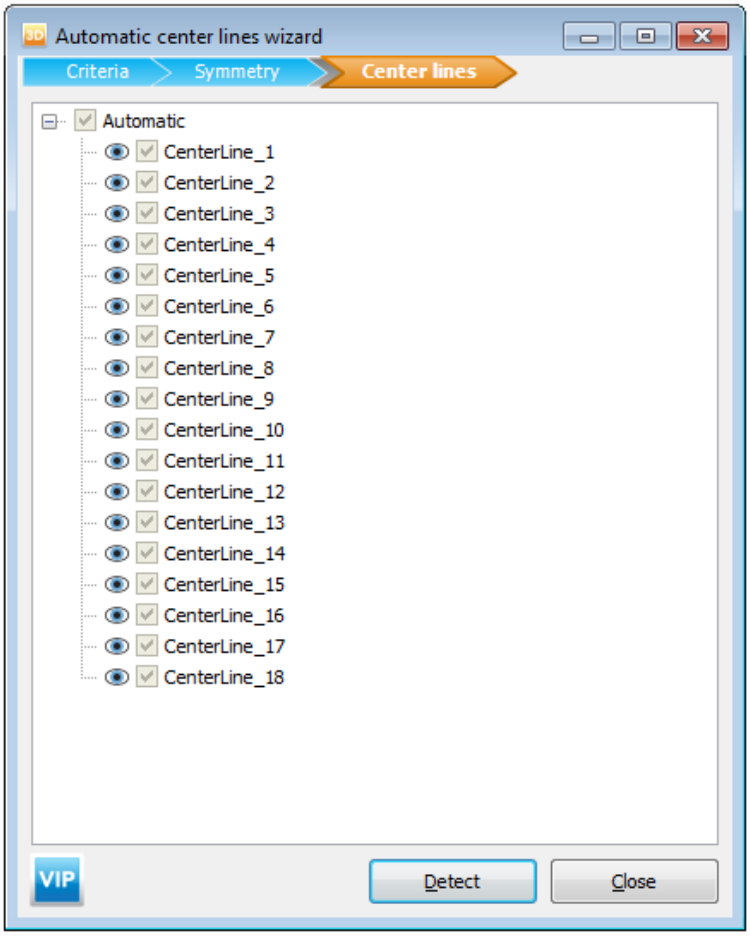


Figure 313: Automatic center lines wizard Center lines tabsheet

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. Automatic local axes determination is based on aligning the local axes with the global axes. For inclined piles, the automatic local coordinate system will be rotated such that the axes remain parallel to the *Global XZ* and *Global YZ* planes. In the case of symmetry, the axes will put on the applicable symmetry plane(s).

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 314 (on page 476)).

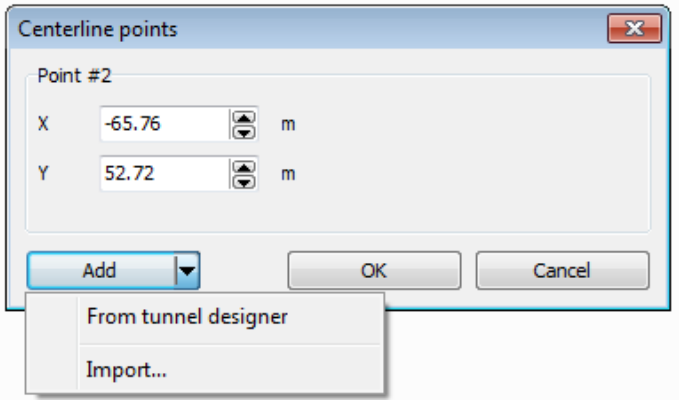


Figure 314: ***Center line points*** *window*

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 (on page 99)). When the thick lining is added to a tunnel, PLAXIS 3D 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.

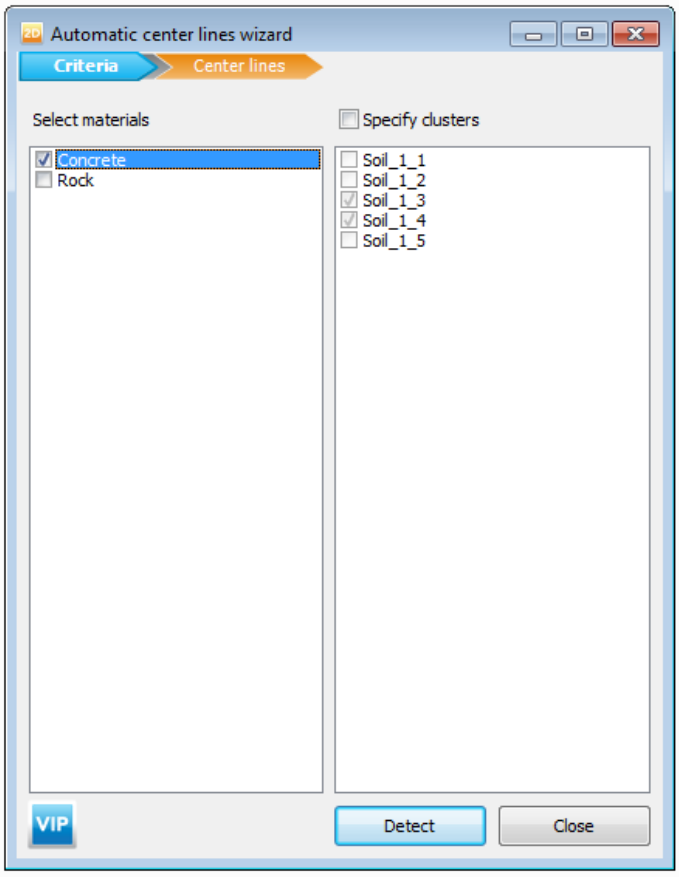


Figure 315: Automatic center lines wizard Criteria tabsheet

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.

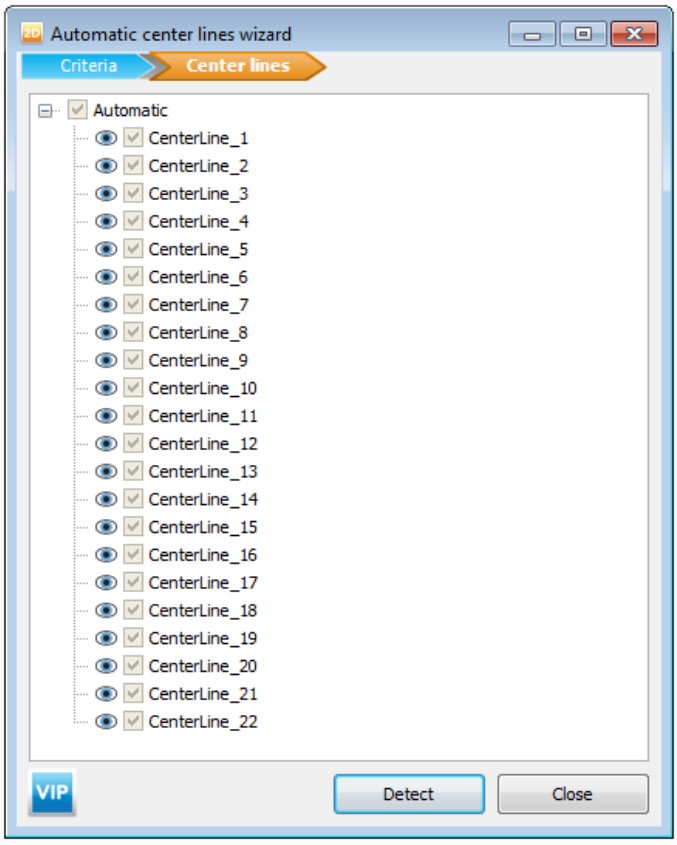


Figure 316: Automatic center lines wizard Center lines tabsheet

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 318 (on page 479)).

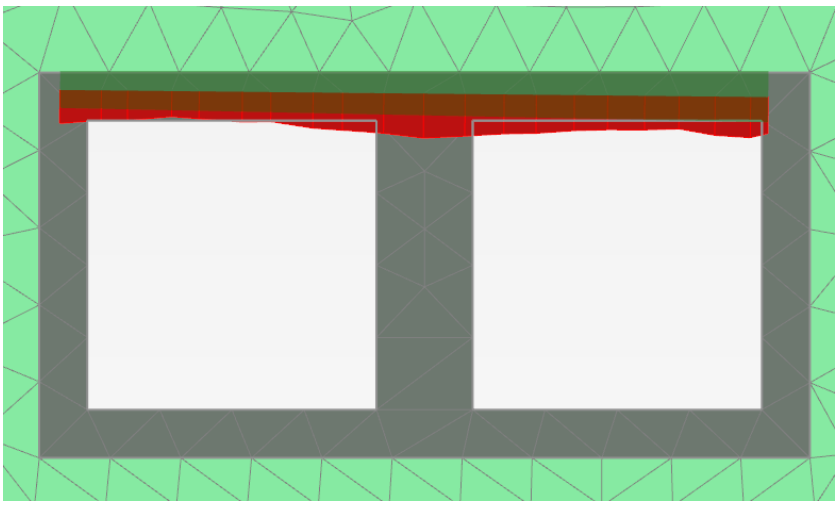


Figure 317: Figure displaying the structural force in a volume and the default radii

It is possible to modify the stress range by clicking on the highlighted area and dragging this area by the mouse (Figure 318 (on page 479)).

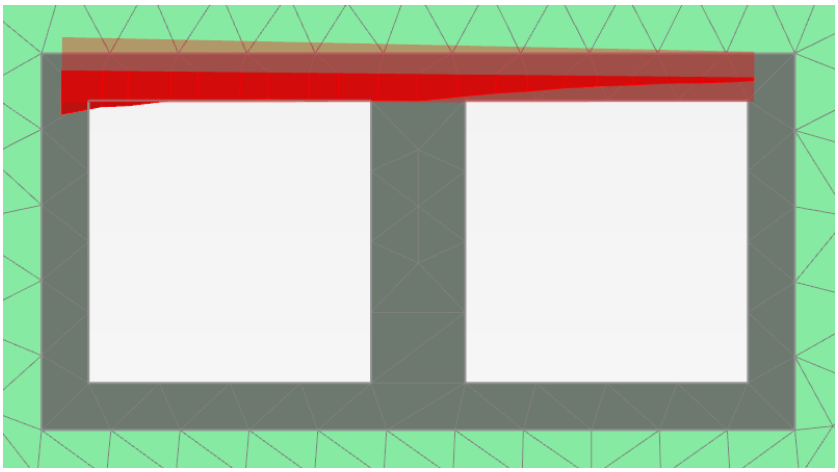


Figure 318: Figure displaying the structural force in a volume and the modified radii

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 319 (on page 480)). 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.

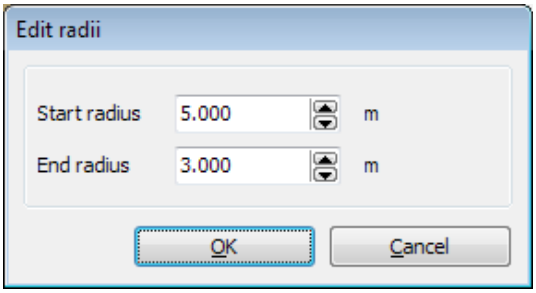


*Figure 319: **Edit radii** window*

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. In the Input program, this button is available in the side toolbar of the **Mesh** and **Staged construction** modes. 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 481).

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 482) and Post-calculation points (on page 483).

10.1.1 *Mesh point selection*

Nodes and stress points can directly be selected by clicking them in the 3D 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 addition, the tabsheets corresponding to the predefined work planes can be used to select nodes or stress points at the work plane levels.

In the **Select points** window (Figure 320 (on page 482)), 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.

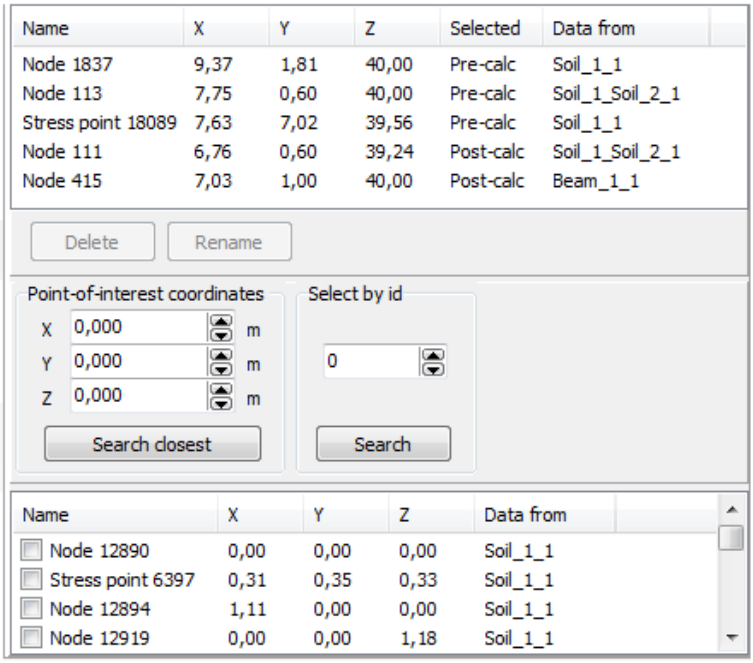


*Figure 320: **Select points** window*

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 429)). 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 321 (on page 484)). 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 481).

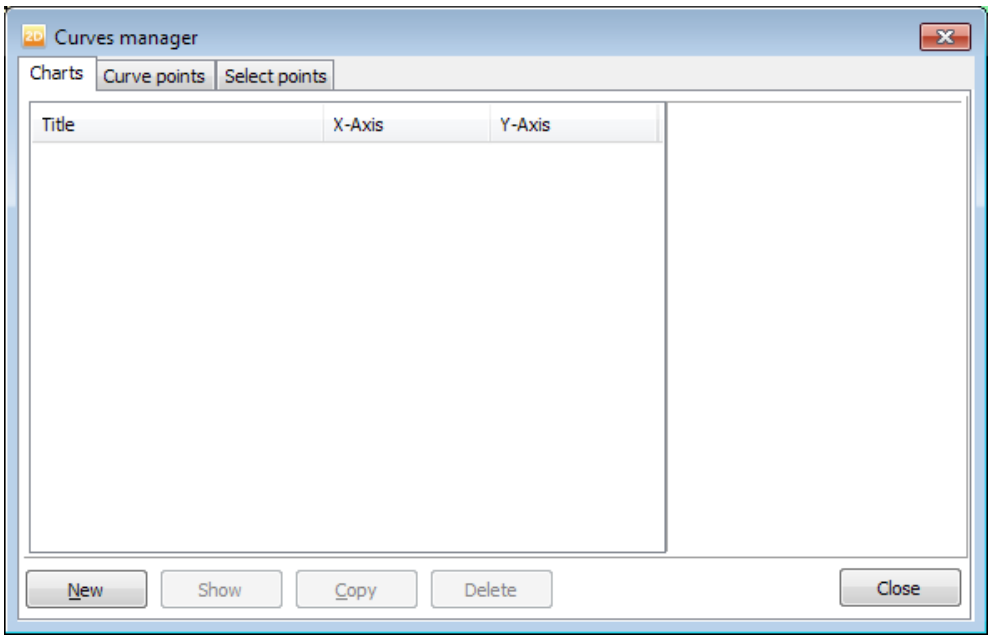


Figure 321: ***Curve manager*** *window*

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 322 (on page 485).

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.

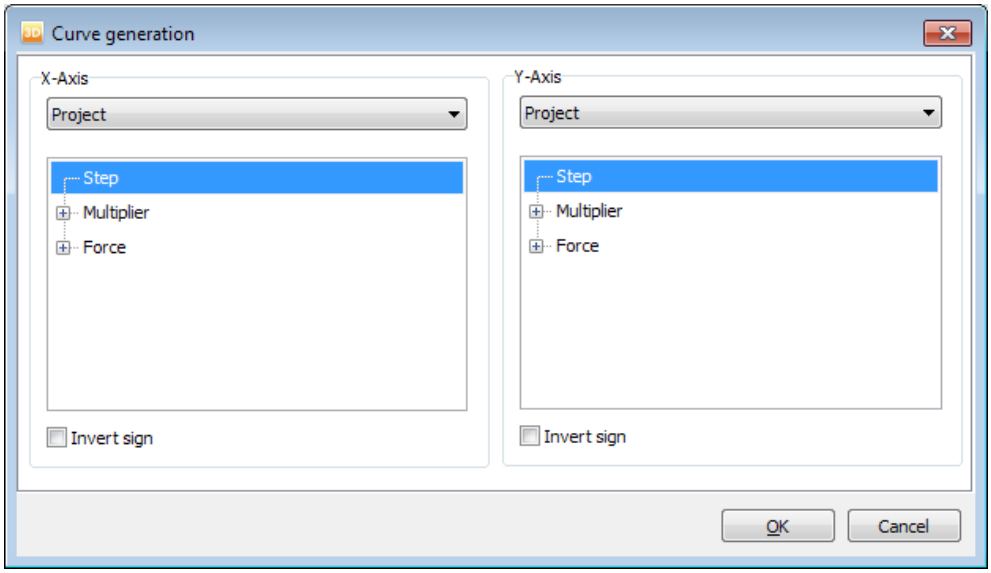


*Figure 322: **Curve generation** window*

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 362)).
- 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 or u_z). The displacements are expressed in the unit of length, as specified in the **General Settings** 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.

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 or u_z). The displacements are expressed in the unit of length, as specified in the **Project properties** 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 462)).

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 451)). However, the *State parameters* option is only available for stress points selected after the calculation Post-calculation points (on page 483)). As a strain quantity all scalar strain quantities available in the *Deformations* menu can be selected (Deformations (on page 446)).

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 Dynamic calculations*

The **Curve generation** window differs when dynamic calculations are executed in the project. The normal tabsheet is similar to the tabsheet when no dynamic calculations are performed. However, the *Dynamic 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 323 (on page 488)).

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 483).

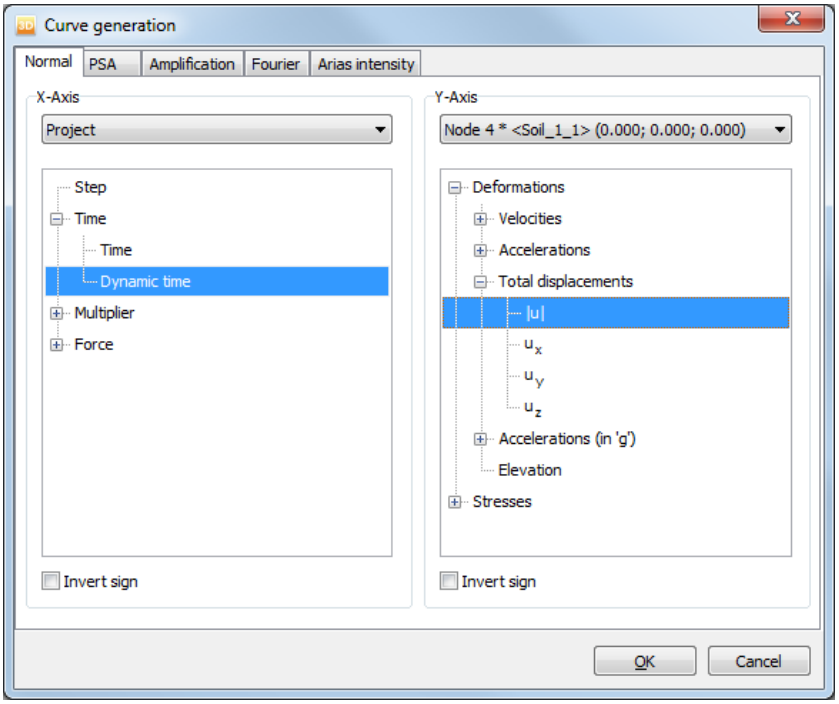


*Figure 323: Options available in **Normal** tabsheet for dynamic calculations*

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).

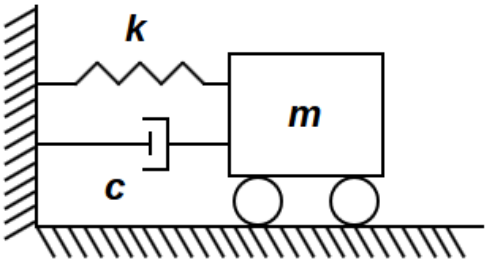


Figure 324: Single Degree Of Freedom System

The dynamic equation of motion is given by:

$$m\ddot{u} + c\dot{u} + ku = -m\ddot{u}_g \qquad \textit{Eq. [120]}$$

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:

$$\xi = \frac{c}{2\sqrt{km}} \qquad \textit{Eq. [121]}$$

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:

$$S_D = \left| u \right|_{\max} \qquad \textit{Eq. [122]}$$

The relative pseudospectral velocity PSV and the absolute pseudospectral acceleration PSA are related to the relative displacement response spectrum according to the following equations:

$$PSV = \omega_0 \cdot S_D \qquad \textit{Eq. [123]}$$

$$PSA = \omega_0^2 \cdot S_D \qquad \textit{Eq. [124]}$$

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:

$$T = \frac{2\pi}{\omega_0} = 2\pi\sqrt{\frac{m}{k}} \qquad \textit{Eq. [125]}$$

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 325 (on page 490)):

$$F_{\max} = m \cdot PSA = \frac{PSA}{g} w \qquad \textit{Eq. [126]}$$

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 325 (on page 490)). 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.

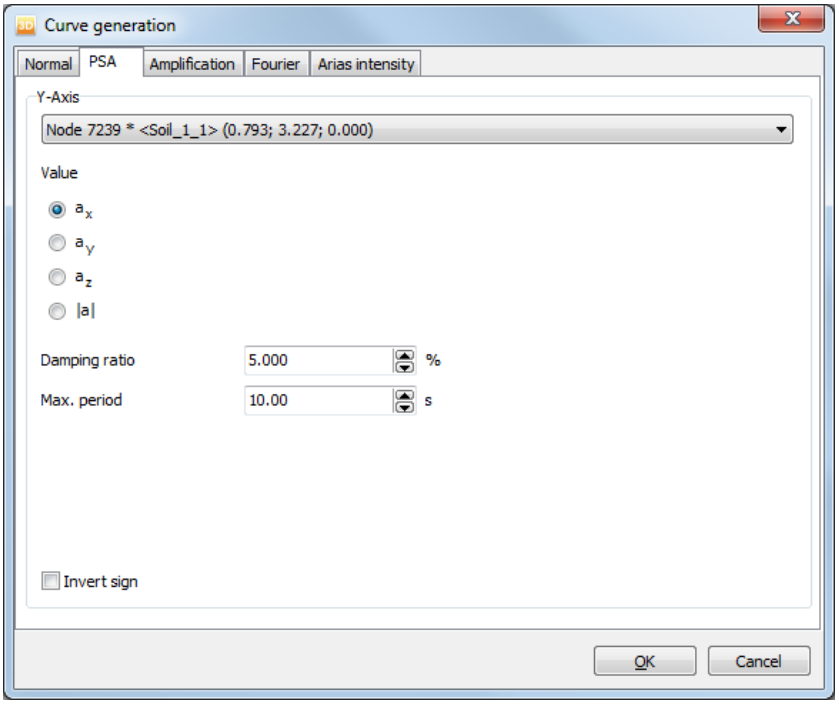


Figure 325: Generate PSA

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.

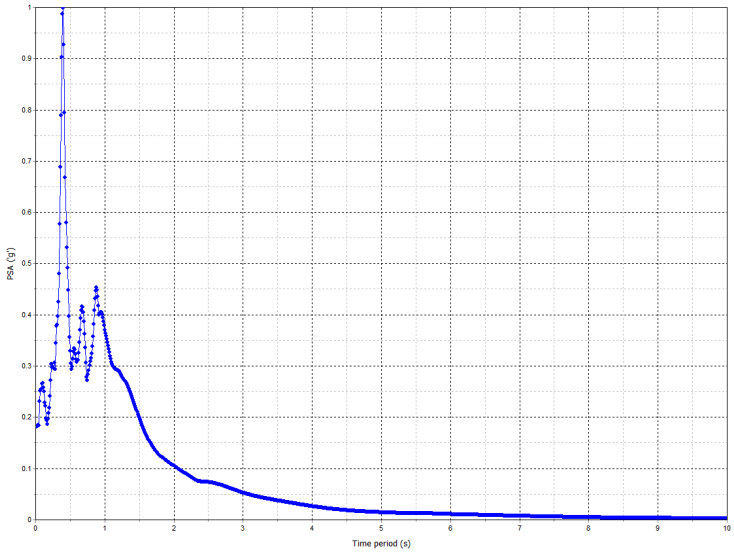


Figure 326: PSA

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 326 (on page 490), 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 327 (on page 491)).

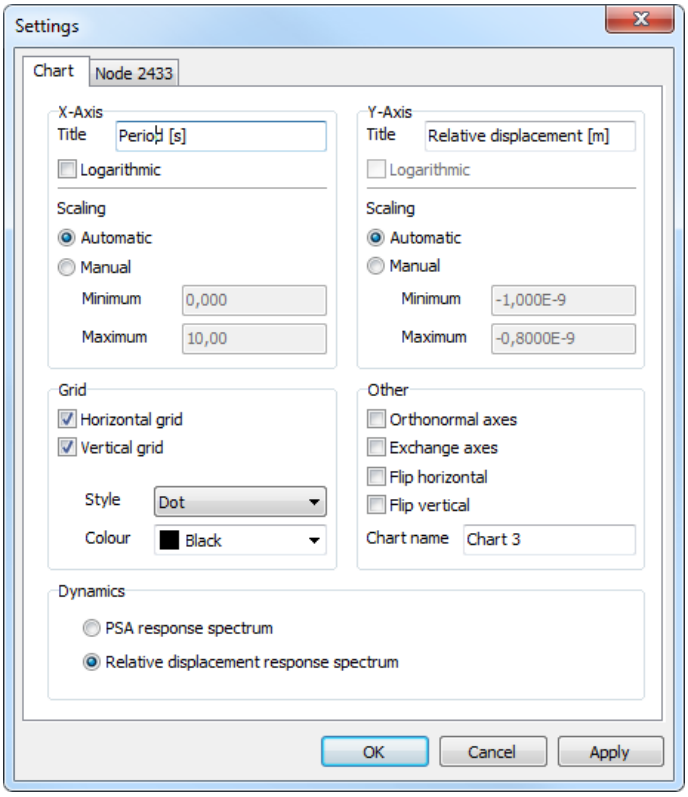


Figure 327: Generate Relative displacement response spectrum

The corresponding chart (Figure 326 (on page 490)) 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.

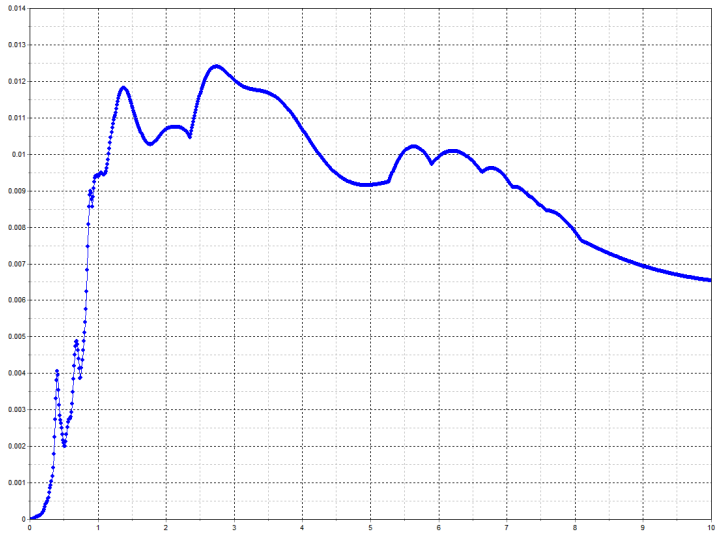


Figure 328: Relative displacement response spectrum

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:

$$F_{\max} = k \cdot S_D \qquad Eq.\ [127]$$

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 329 (on page 493)”). This gives the magnification of the response at one point with respect to given excitation.

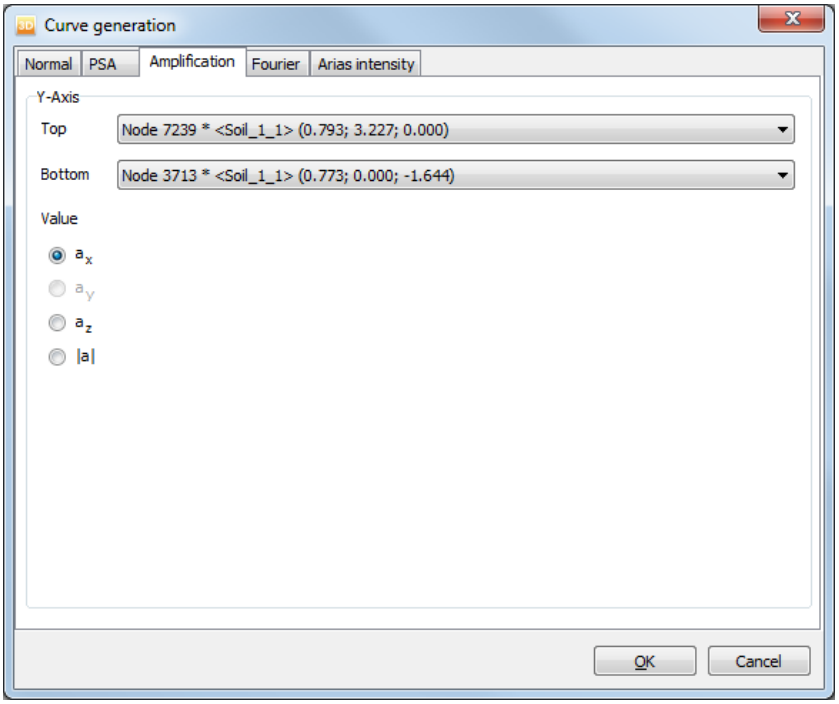


Figure 329: Amplification spectrum generation

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.

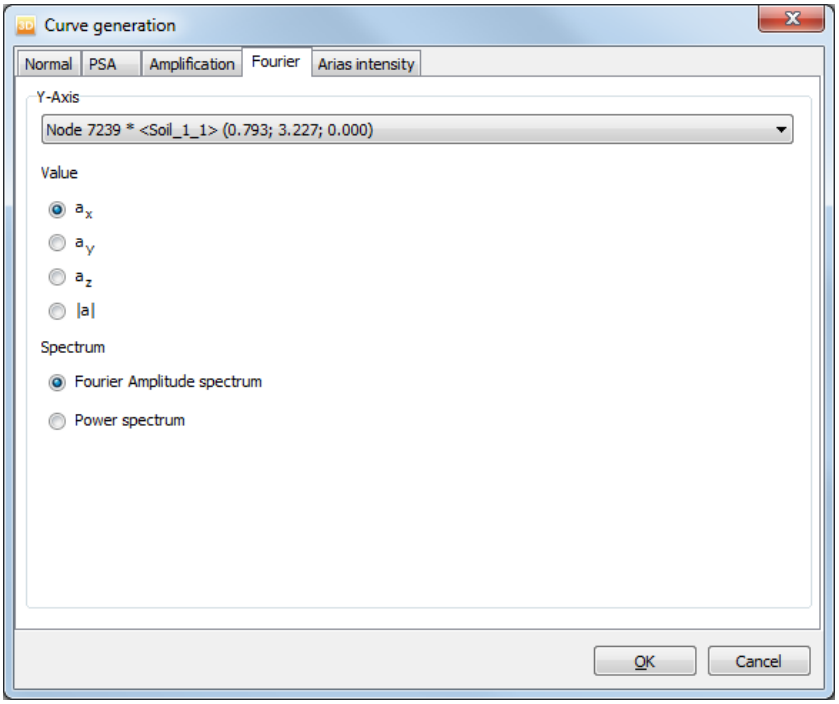


Figure 330: Fourier curve generation

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:

$$I_a = \frac{\pi}{2g}\int_0^T d[a(t)]^2 \, d\, t \qquad \textit{Eq. [128]}$$

where g is the gravity acceleration and T_d is the duration of the motion (i.e. maximum dynamic time in the time history acceleration).

The *Arias Intensity* has the dimension of velocity (Figure 331 (on page 495)). In the **Chart** tabsheet, the user can choose to represent I_a in percentage of the maximum *Arias intensity*.

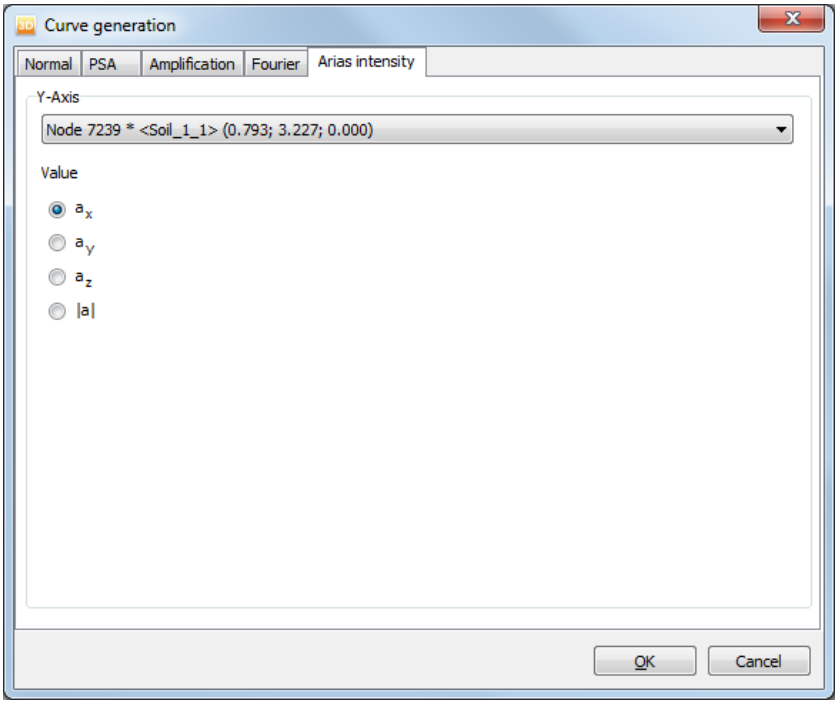


Figure 331: Arias curve generation

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 333 (on page 496)). Clicking the *Extreme values* button, the user can view the *Extreme values* for the curve (Figure 332 (on page 495)).

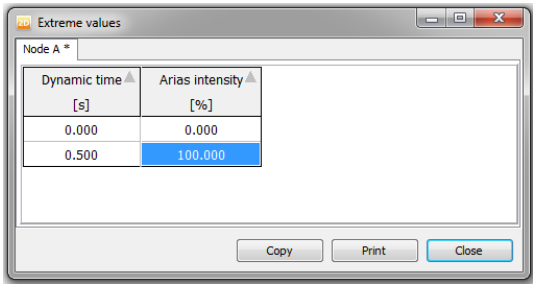


Figure 332: Extreme values for Arias intensity curves

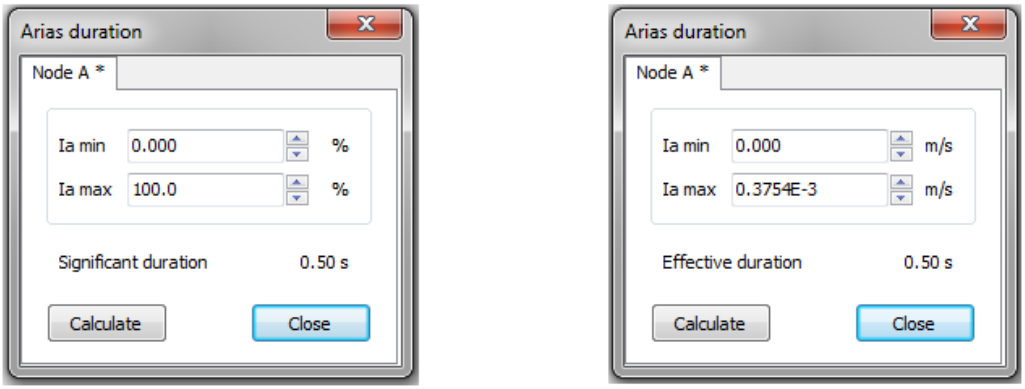


Figure 333: Significant duration (left) and Effective duration for Arias intensity curves (right)

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 334 (on page 497)).

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:

$$\omega = 2\pi f = 2\frac{\pi}{T} \qquad Eq.\ [129]$$

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.

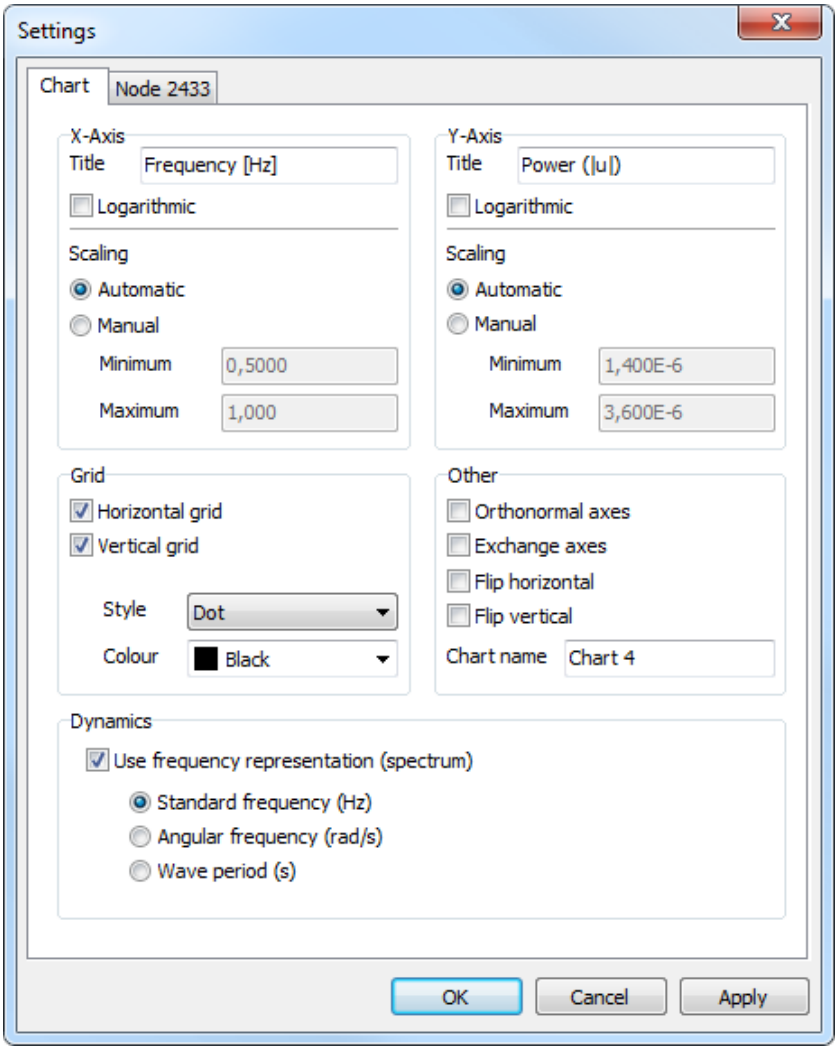


Figure 334: Fast Fourier Transform

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 335 (on page 498).

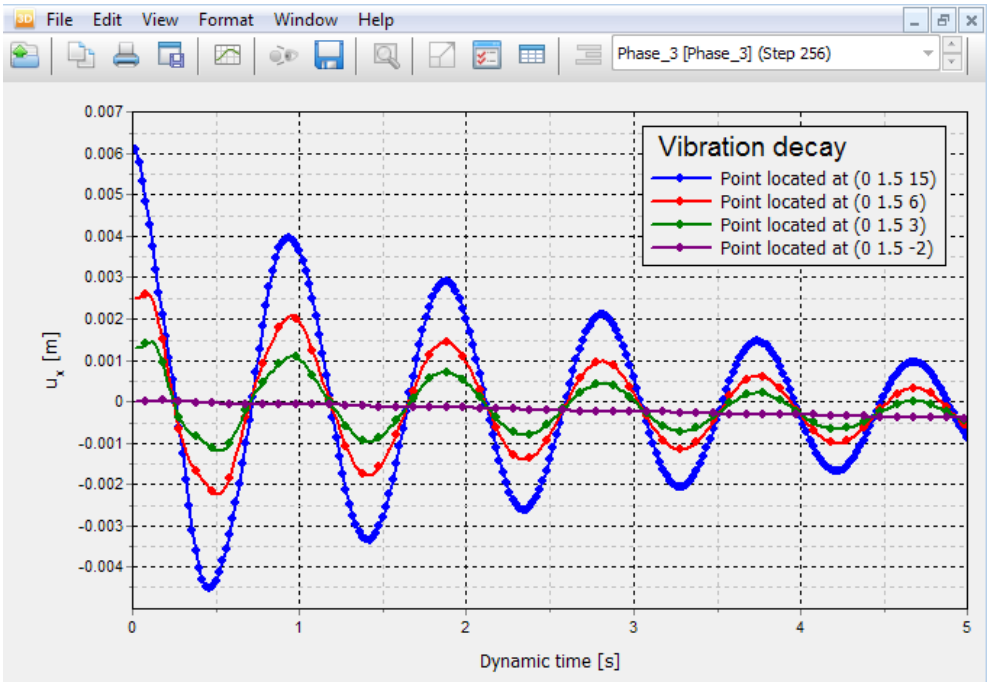


Figure 335: Curves in Output program

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 413).

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 422).
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 499).
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 419) and Help menu (on page 420).

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.

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 501)). The options available in the tabsheets of the curves can be used to customize the plot (Curve settings (on page 502)).

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 336 (on page 501)).

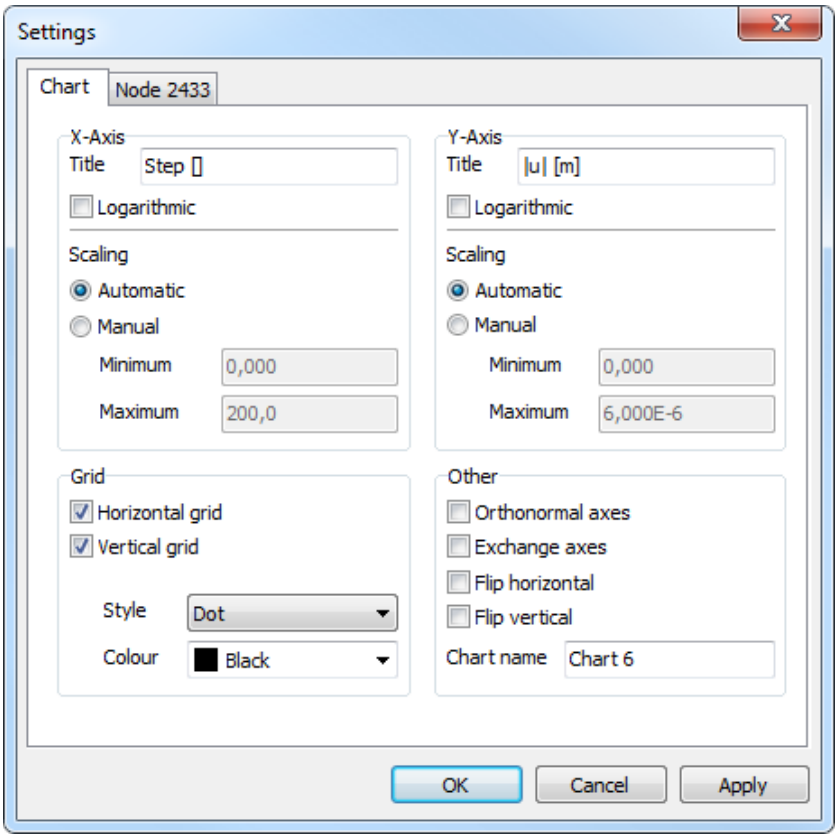


Figure 336: ***Chart settings*** *tabsheet*

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*

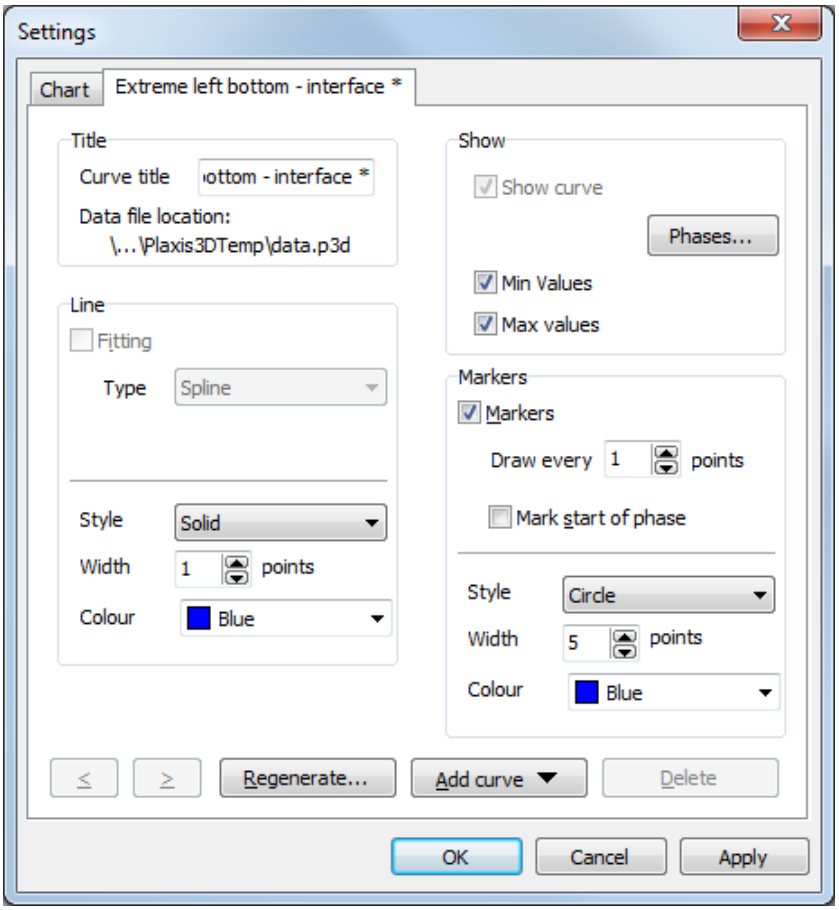


Figure 337: ***Curve settings*** *tabsheet*

The **Settings** window contains for each of the curves in the current chart a tabsheet with the same options (Figure 337 (on page 502)).

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 503)).
Add curve	The *Add curve* button may be used to add new curves to the current chart (Multiple curves in one chart (on page 504)).
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 336 (on page 501)), 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 3D 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 483)). 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 Engineering Applications

11.1 Modelling vacuum consolidation

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 3D.

There are various methods of vacuum consolidation in the real world, but they are all modelled in a similar way in PLAXIS 3D. 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 3D 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.

11.1.1 *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).

11.1.2 *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 (Line drain (on page 178) and Surface drain (on page 179)) 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.

11.1.3 *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.

11.1.4 *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.

11.1.5 *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.

11.2 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 3D.

There are various ways to create ground anchors in the real world, but they can be all modelled in a similar way in PLAXIS 3D. 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.

11.2.1 *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, in which the embedded beam represents the grouted part of the anchor and the node-to-node anchor represents the anchor free length. The behaviour of the embedded beam is set to 'Grout body'. The connection type is internally set to 'Free'. This is required since otherwise the embedded beam 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.

11.2.2 *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, 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. Ground anchors do not have a base resistance, so F_{max} = 0. The interface stiffness factors can be left to their default values.

11.2.3 *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 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).

11.3 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 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 3D.

11.3.1 *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. TLateral 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 elements of PLAXIS 3D can realistically model the piles in these engineering applications.

The user should refer to Embedded beams (on page 143) and Material set (on page 308). No further information is required for the modelling techniques embedded beam elements in abutment pile applications.

11.3.2 *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 elements of PLAXIS 3D 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 3D can easily deal with this complex loading conditions.

The user should refer to Embedded beams (on page 143) and Material set (on page 308). No further information is required for the modelling techniques embedded beam elements in quay wall with relieving platform applications.

12 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. 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.

36. 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.

37. Lysmer, J., Kuhlmeyer, R.L. (1969). Finite dynamic model for infinite media. Engineering mechanics division ASCE, 95, 859–877.

38. Marinos, P., Hoek, E. (2001). Estimating the mechanical properties of heterogeneous rock masses such as flysh. Bulletin of Engineering Geology, 92,60–85.

39. Matsuishi, M., Endo, T. (1968). Fatigue of metals subjected to varying stress. Japan Society of Mechanical Engineers, Fukuoka, Japan, 68(2), 37–40.

40. 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.

41. 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.

42. Owen, D.R.J., Hinton, E. (1982). Finite Elements in Plasticity. Pineridge Press Limited, Swansea.

43. 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.

44. 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.

45. 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.

46. Schank, O., Gärtner, K. (2006). On fast factorization pivoting methods for symmetric indefinite systems. Electronic Transactions on Numerical Analysis, 23, 158–179.

47. 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.

48. Schikora, K., Fink, T. (1982). Berechnungsmethoden moderner bergmännischer bauweisen beim u-bahn-bau. Bauingenieur, 57, 193–198.

49. Sloan, S.W. (1981). Numerical analysis of incompressible and plastic solids using finite elements. Ph.D. thesis, University of Cambridge, U.K.

50. 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.

51. Sluis, J. (2012). Validation of embedded pile row in PLAXIS 2D. Master's thesis, Delft University of Technology.

52. Smith, I.M. (1982). Programming the finite element method with application to geomechanics. John Wiley & Sons, Chichester.

53. Song, E.X. (1990). Elasto-plastic consolidation under steady and cyclic loads. Ph.D. thesis, Delft University of Technology, The Netherlands.

54. 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.

55. van Langen, H. (1991). Numerical analysis of soil structure interaction. Ph.D. thesis, Delft University of Technology, The Netherlands.

56. van Langen, H., Vermeer, P.A. (1990). Automatic step size correction for non-associated plasticity problems. Int. J. Num. Meth. Eng., 29, 579–598.

57. van Langen, H., Vermeer, P.A. (1991). Interface elements for singular plasticity points. Int. J. Num. Analyt. Meth. in Geomech., 15, 301–315.

58. Vermeer, P.A., van Langen, H. (1989). Soil collapse computations with finite elements. In Ingenieur-Archive 59. 221–236.

59. 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.

60. 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.

61. 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.

62. Wyllie, D.C., Mah, C. (2004). Rock slope engineering. CRC Press, Boca Raton.

63. Zienkiewicz, O.C. (1977). The Finite Element Method. McGraw-Hill, London.

64. 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

Program and data file structure

A.1 Program structure

The full PLAXIS 3D program consists of various sub-programs, modules and other files which are copied to various directories during the installation procedure (See General Information). The most important files are located in the PLAXIS 3D program directory. Some of these folders, files and their functions are listed below.

Folders

cef	Chrome embedded framework
Defaults	Default columns for phases explorer
manuals	The manuals of the software and the command reference
python	Python folder
pytools	Python scripts
tools	Tools for expert users
udsm	user-defined soil models

Executables

CurveDataConverter.exe	Curve converter from pre-2017 to current version
PackProject.exe	Pack project
Plaxis3DInput.exe	Input program (pre-processor)
Plaxis3DOutput.exe	Output program (post-processor)
Plaxis3DOutput_2012.02.exe	Output program 2012.02 version (post-processor)
Plaxis3DOutputViewer.exe	Output viewer
Postmesher.exe	Postmesher.exe
ReportGenerator.exe	Report generation module
Soiltest.exe	SoilTest module

A.1 Project data files

The main file used to store information for a PLAXIS 3D project has a structured format and is named <project>.P3D, where <project> is the project title. Besides this file, additional data is stored in multiple files in the sub-directory <project>.P3DAT. The files in this directory may include:

data.###	The files having the extension '###' are created by the calculation kernel. The '###' is the step number; it is at least 3 digits but can be longer when there are 1000 or more steps [1)].
data.ant	Output annotations
data.c##	Contains data for curve generation [2)].
data.centerlines	Centerlines data
data.gpv	Previews of the generated curves
data.gxl	Information about the generated curves
data.h00	Initial settings generated after the mesh generation for the 3D mesh
data.l##	Staged construction settings[2)]
data.m##	File containing information about the materials in the model
data.mesh.plxml	Mesh data
data.mesher3d.log	Additional information from the mesher
data.IMProOutput	Information about the mesh output
data.nsl	Information about the selected nodes and stress points (after Calculation)
data.opi	The information of project phases for output
data.p3d	All the commands executed successfully in the project
p3dlog File	File containing all the commands executed in the project
data.plxmat	Contains all material sets and parameters
PLXML Files	The PLXML file format is essentially a limited binary representation of XML, tweaked for the needs of the Hierarchic Storages
S## Files	Files containing all the monitored values for the integration points
data.svs	Information about the saved views for Output

data.w##	The files having the extension ' W##' contain the current groundwater conditions. The '##' stands for the phase number in which the groundwater condition is first created. A project always has a 'W00' file for the initial conditions.
data.zin	Project model data. Supplement to the INP file
data.zms	Project model data. Supplement to the MSH file. The 'ZMS' file is generated only once; directly after generating the mesh. It contains all data concerning the mesh.
debug.d##	Debug calculation logs generated by the kernel during the calculation[2)].
log.d##	General information about the calculation phase[2)]

[1)]Three digit deformation calculation step number (001, 002,....). Above 999 gives an additional digit in the file extension.

[2)]Two digit calculation phase number (01, 02, ...). Above 99 gives an additional digit in the file extension.

To create a copy of a PLAXISproject under a different name or in a different directory, it is recommended to open the project in the Input program and to save it under a different name using the Save as option in the **File** menu. In this way the required file and data structure is properly created.

During the creation of a project, before the project is explicitly saved under a specific name, intermediately generated information is stored in the TEMP directory as specified in the Windows® operating system using the project name Plaxis3DTemp.

B

The format required for the files to import CPT data

B.1 ASCII files

PLAXIS supports CPT data in CPT format CUR (2004) (on page 511). 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

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 introduces inside square brackets next to the column specifier. Comments can be added behind a '#'. 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.

B.2 ASCII files with CPT extension

The user can assign the extension *.cpt to an ASCII file. In this case the program will recognize the content as CPT data. No header needs to be specified because the file extension indicates its content. If no header is specified or its data is incomplete, the following values are assumed:

1. Missing coordinates are set to 0.0
2. A default header is assumed which considers the number of available columns. A maximum of four columns is considered with specifiers 'd ', 'q ', 'f ' and 'u '. In case of less columns, specifiers are skipped (either 'f ' and/or 'u '). If more than 4 columns are found, the additional columns are skipped.
3. At least 2 columns must be present.
4. If no dimensions are specified, the default project dimensions are used.

B.3 Unit conversion

The conversion of the data units to the project units shown in Table 20 (on page 518) is possible for pressure.

Table 20: Convertable units

Data unit	Project unit	
1 Pa	$1\ N/m^2$	
1 kPa	$1000\ N/m^2$	$1\ kN/m^2$
1 MPa	$1 \cdot 10^6\ N/m^2$	$1000\ kN/m^2$
1 N/mm2	$1 \cdot 10^6\ N/m^2$	$1000\ kN/m^2$
$1\ kN/m^2$	$1000\ N/m^2$	$1\ kN/m^2$

C

Cyclic accumulation Tool - Contour File Formats

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.

C.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 338: Example of a custom contour file

C.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 (%)	τ_a / s_u^c	τ_{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 τ_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 τ_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	

D

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.

E

Conventional and unsaturated soil behaviour in PLAXIS

As of 2013, PLAXIS 3D can handle unsaturated soils. The unsaturated soil behaviour and suction are taken into account using Bishop' s stress, whereas conventional calculations (without suction) are performed using Terzaghi' s effective stress. The way to distinguish between conventional calculations (without suction) and calculations in which unsaturated soil behaviour is taken into account (with suction) is by means of the calculation option Ignore suction.

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) are positive.

E.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 3D 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 535)and Table 23 (on page 535)
- 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 3D 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 3D, χ is supposed to be equal to S_{eff}, so Bishop' s stress in PLAXIS 3D:

$\sigma' = \sigma - S_{eff} \cdot (p_{steady} + p_{excess}) = \sigma - p_{active}$

- 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.

E.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 271), 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.

E.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*, *Safety*) are available, but some specific features or combinations are not available. Limitations are described below. (see Possibilities and limitations (on page 529)).

E.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 3D 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: $p_{excess} = p_{cavitation} - p_{steady}$

E.5 *Unit weight of soil*

Although a proper definition of the unit weight of soil would involve the degree of saturation, PLAXIS 3D 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.

E.6 *Possibilities and limitations*

There are limitations in the use of specific features in the various calculations types in PLAXIS 3D depending on whether suction is ignored or allowed. An overview of the limitations is given below.

E.7 *Steady-state groundwater flow*

- 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.

E.8 *Transient groundwater flow*

- 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.

E.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	Possible, but not recommended since non-hydrostatic pore pressures may lead to non-equilibrium.
Transient groundwater flow	Not 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 (deformations are not considered)
Updated water pressures	Not possible (deformations are not considered)

E.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 *Σ Mweight* (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	Possible
Transient groundwater flow	Not 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

E.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	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.
Transient groundwater flow	Not possible.
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

E.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	Not possible (steady-state pore pressures taken from parent phase)
Transient groundwater flow	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

E.13 *Consolidation analysis*

- 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.
Transient groundwater flow	Not possible
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

E.14 *Fully coupled flow-deformation analysis*

- 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.
Transient 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

E.15 *Dynamic analysis*

- 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)
Transient 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

E.16 *Dynamic analysis with Consolidation*

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 dynamic (drained)	Safety or dynamic (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 ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$K_w = 0$	$K_w = 0$
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 511).

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 ($p_{water} \leq 0$)	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$	$\frac{K_w^{sat}}{n} = \frac{2 \cdot G}{3}\left(\frac{1+\nu_u}{1-2\nu_u} - \frac{1+\nu'}{1-2\nu'}\right)$

Type of Material	Steady - state	Transient
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	$K_w = 0$	$K_w = 0$

F

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

G

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

H

Python HTTP REST API wrapper

The Python wrapper for the PLAXIS HTTP REST API hides the technicalities of dealing with the HTTP communications behind a comfortable object-oriented wrapper.

H.1 Python HTTP REST API wrapper

Prerequisites

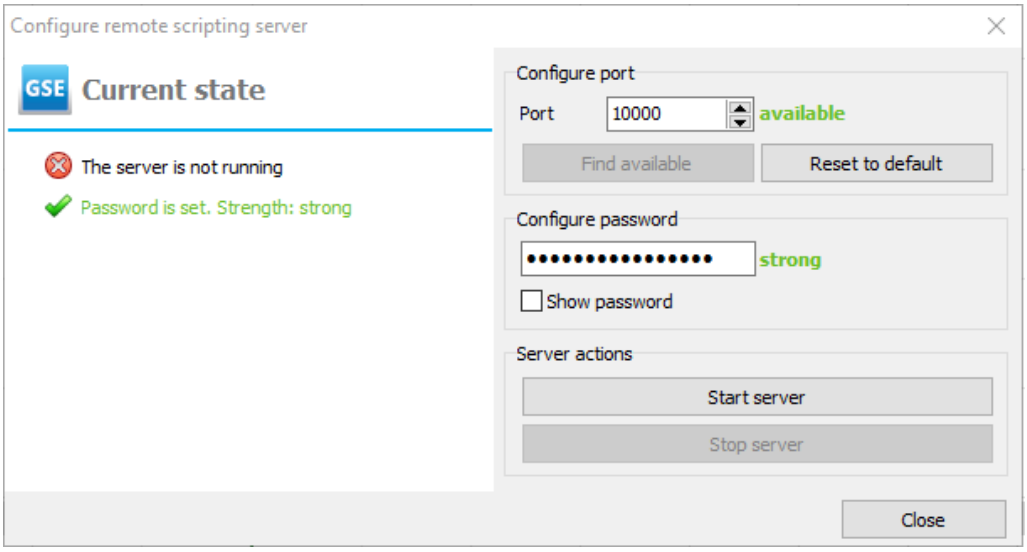
1. An editor for writing Python scripts in. This documentation assumes that the SciTE editor will be used that comes with the PLAXIS installation.
2. At least a very rudimentary knowledge of the Python language is recommended, as this document makes no attempt to teach it. Good resources for this purpose 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.

H.2 Starting the 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):

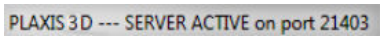
```
netstat -a
netstat -ab
```

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 3D on port 21403:

```
C:\Program Files\Bentley\Geotechnical\PLAXIS 3D 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:



H.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 3D. 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.

H.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 (using a 2D example) 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')
```

H.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.

H.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

H.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))
```

H.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"
```

H.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()
```

H.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.

H.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()
```

H.12 Installing third-party Python modules

When you want to install a third party package that isn‘t already available in the Python installation then you can do so by opening the command prompt from Expert > Python > Command prompt or by selecting ‘Console for PLAXIS 3D’ from the PLAXIS folder in the start menu. From there you can install new packages by either using conda or pip.

For example, if you want to install the dropbox package you would type:

```
pip install dropbox
```

And if you want to install the cffi (C foreign function interface) package you would type:

```
conda install cffi
```

Note that not all packages that are available to pip are available to conda and some complex packages such as numpy might be easier to install using conda than with pip. Lastly installing new packages requires administrator rights on your PC.

H.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 (on page 568)
- The `Token` classes have properties as specified for the HTTP REST API (see HTTP REST API (on page 568)). 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
```

H.14 Sample Applications

This section contains a few simple tasks that highlight different aspects of the Python scripting wrapper. It is assumed PLAXIS 3D is running and the necessary connection boilerplate code has been executed already.

Opening a project, removing all lines containing beams 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)
```

HTTP REST API

I.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.

I.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.

I.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.

I.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 3D. 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 3D 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 3D, 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

J 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 dynamic analysis	The value for a static load has been changed in a dynamic 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 3D 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.

J.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.

J.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.

Index

|
commands 30
integration