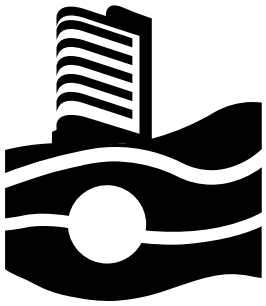
PLAXIS 2026.1

Monopile Designer Manual



Last Updated: September 30, 2026

SEEQUENT

UNDERSTAND THE UNDERGROUND

© 2026. Seequent, The Bentley Subsurface Company

Trademark

Caution:

PLAXIS Monopile Designer is a program for specific geotechnical/structural applications in which soil models are used to simulate the soil behaviour. The PLAXIS Monopile Designer code and its soil models have been developed with great care. Although a lot of testing and validation have been performed, it cannot be guaranteed that the PLAXIS Monopile Designer code is free of errors. Moreover, the simulation of geotechnical problems involves some inevitable numerical and modelling errors. The accuracy at which reality is approximated depends highly on the expertise of the user regarding the modelling of the problem, the understanding of the soil models and their limitations, the selection of model parameters, and the ability to judge the reliability of the computational results. Hence, PLAXIS Monopile Designer may only be used by professionals that possess the aforementioned expertise. The user must be aware of his/her responsibility when he/she uses the computational results for geotechnical and structural design purposes. No warranty, expressed or implied, is offered as to the accuracy of results from PLAXIS Monopile Designer, its documentation or its fitness for a particular purpose. Plaxis bv nor its officers or employees can be held responsible or liable for design errors that are based on PLAXIS Monopile Designer calculations or documentation.

Windows® is a registered trademark of the Microsoft Corporation.

PLAXIS is a registered trademark of the PLAXIS company (Plaxis bv).

Copyright PLAXIS program by:

Plaxis bv P.O. Box 572, 2600 AN DELFT, Netherlands

Fax: +31 (0)15 257 3107; Internet site: www.bentley.com

These manuals may not be reproduced, in whole or in part, by photo-copy or print or any other means, without written permission from Plaxis bv.

ISBN-13: 978-90-76016-27-6

© 2026 Plaxis bv, Bentley Systems, Incorporated

Published in the Netherlands

Contents

Trademark.....

1 General Information Manual..... 6

1.1 Preface..... 6
1.2 PLAXIS product, licencing and services..... 9
1.3 Short overview of features..... 10
1.4 Manuals..... 11
1.5 First time installation..... 11

2 Reference Manual.....14

2.1 Introduction..... 14
2.1.1 The PISA Project..... 14
2.1.2 The design methodology..... 15
2.1.3 Interoperability with PLAXIS 3D..... 16
2.1.4 The 1D FE Model..... 16
2.1.5 The components of PLAXIS Monopile Designer..... 18
2.1.6 Graphical user interface..... 19
2.1.7 Useful terminology..... 19
2.2 General Information..... 20
2.2.1 Using PLAXIS Monopile Designer with and without PLAXIS 3D..... 20
2.2.2 Program layout..... 21
2.2.3 New Project..... 22
2.2.4 Open Project..... 23
2.2.5 Menus in the Menu bar..... 23
2.2.6 Units and sign convention..... 24
2.2.7 Automatic saving..... 26
2.2.8 Help facilities..... 27
2.3 Calibration mode..... 27
2.3.1 Calibration procedure for soil reaction curves..... 28
2.3.2 Calibration mode layout..... 28
2.3.3 Calibration Matrix..... 31
2.3.4 Soil Profile tabsheet..... 39
2.3.5 Geometry datasets tabsheet..... 42
2.3.6 Results inspection pane..... 46
2.3.7 Recommended workflow..... 56
2.4 Analysis mode..... 57
2.4.1 Monopile geometry..... 58
2.4.2 Structural properties..... 59
2.4.3 Calculation Parameters..... 60
2.4.4 Soil reaction curves..... 61
2.4.5 Soil Layers..... 62
2.4.6 Pile segments and segment properties..... 63
2.4.7 Modal Analysis..... 65
2.4.8 Expert settings..... 68
2.4.9 1D Analysis..... 70
2.4.10 3D Design Verification..... 71
2.4.11 Results inspection pane..... 75

2.5 Results mode ... 80
2.5.1 Workload and load factor ... 81
2.5.2 Graph tab ... 81
2.5.3 Table tab ... 83
2.5.4 Accuracy metrics η and ρ ... 84
2.6 Remote Scripting Interface ... 85
2.6.1 Anatomy of a command ... 85
2.6.2 Creating scripts ... 86
2.6.3 Running scripts ... 86

3 Tutorial 1 - Numerical Based Design ... 87
3.1 Calibration mode - Input ... 90
3.2 Verification of the calibration procedure ... 98
3.3 Final Design ... 100
3.4 Verification of the final design ... 101

4 Tutorial 2 - Layered Soils ... 103
4.1 Rule based-design ... 105
4.1.1 Analysis - Layered profile ... 105
4.1.2 Design verification ... 110
4.2 Numerical-based design ... 112
4.2.1 Calibration – Layered soil profile ... 112
4.2.2 Analysis – Layered profile ... 114
4.2.3 Design verification ... 115
4.3 Comparison between rule-based and numerical-based design ... 115

5 Scientific Manual ... 117
5.1 Introduction ... 117
5.2 Material Models ... 117
5.2.1 Clay: NGI-ADP material Parameters ... 118
5.2.2 Sand: HSsmall material Parameters ... 119
5.3 PLAXIS 3D Models ... 120
5.3.1 Generating 3D models ... 120
5.3.2 Soft beam properties ... 124
5.3.3 Soil reaction curves ... 125
5.3.4 Results inspection pane ... 126
5.4 Optimization Module ... 127
5.5 Rule based models ... 130
5.5.1 Plain text file format rules ... 131
5.5.2 Conventional p-y curves ... 132
5.5.3 Depth Variation Functions ... 137
5.6 1D FE Model ... 142
5.6.1 Formulation of the model 1D FE Model ... 142
5.6.2 Implementation aspects of the 1D FE model ... 146
5.7 Results Mode ... 150
5.7.1 Realised H ... 150
5.7.2 Realised M ... 150
5.7.3 Load factor ... 150
5.7.4 Accuracy metrics η and ρ ... 151
5.7.5 H-v and M-θ ... 152
5.7.6 v(z) and θ(z) plots ... 152
5.7.7 Soil profile plots ... 152
5.7.8 p(z) plot ... 152

5.7.9 SACS pile models.....153

6 References.....158

7 Appendices.....161

A Warnings and errors.....

A.1 Calibration Mode - Warnings and errors.....162
A.2 Analysis Mode - Warnings and errors.....167

B Scripting Reference.....

B.1 Commands Reference.....169
B.2 Object Reference.....171

1

General Information Manual

1.1 Preface

PLAXIS Monopile Designer is a software program, developed for the analysis and design of monopiles used as foundation elements for offshore wind turbines, under lateral loading conditions.

It is a part of the PLAXIS product range, a suite of finite element programs that is used worldwide for geotechnical engineering and design.

PLAXIS Monopile Designer is based on the results of the Pile Soil Analysis (PISA) research project. The PISA project is aimed at investigating and developing improved design methods for laterally loaded piles, specifically tailored to the offshore wind sector. It is a joint industry project led by DONG Energy (nowadays named Ørsted) and run through the Carbon Trust's Offshore Wind Accelerator programme.

The main aim of the PISA project is to develop a new design methodology for offshore wind turbine monopile foundations, to overcome the shortcomings of the current methods. The project focuses on the use of numerical finite element modelling to develop the new design method, which is validated through a campaign of large scale field tests.

PLAXIS Monopile Designer can be used as a stand-alone tool for the rule-based design method and in connection with PLAXIS 3D for the numerical-based design method, as defined in the PISA research project. The development of PLAXIS Monopile Designer was performed by Plaxis bv, in collaboration with Oxford University (Profs. Burd, Byrne, Houlsby, Martin, McAdam), Imperial College London (Profs. Jardine, Potts, Zdravkovic, and Dr. Taborda) and University

College Dublin (Prof. Gavin). Collaboration with Fugro, as a designer of offshore foundations, is also acknowledged.

Goals and objectives

PLAXIS Monopile Designer is intended to provide a tool for practical analysis and design of monopiles to be used by geotechnical engineers who are not necessarily numerical specialists. Quite often practising professional engineers consider non-linear computations cumbersome and time-consuming. The PLAXIS research and development team has addressed this issue by designing robust and theoretically sound computational procedures, which are encapsulated in a logical and easy-to-use shell. As a result, many geotechnical engineers world-wide have adopted the PLAXIS products and are using them for engineering and design purposes.

Scientific network

The development of the PLAXIS products would not be possible without worldwide research at universities and research institutes. To ensure that the high technical standard of PLAXIS is maintained and that new technology is adopted, the development team is in contact with a large network of researchers in the field of geo-engineering and numerical methods.

Direct support is obtained from a series of research centres:

Name	Organization (Country)
Prof. Michael Hicks, Prof. Bert Sluys	Delft University of Technology, Mathematics & Informatics (NL)
Prof. Kees Vuik	Delft University of Technology, Mathematics & Informatics (NL)
Mr. Mark Post, Dr. Cor Zwanenburg	Deltares (NL)
Dr. Michael Heibaum, Mr. Oliver Stelzer	BundesAnstalt für Wasserbau (DE)
Prof. Helmut Schweiger, Dr. Franz Tschuchnigg	Technical University, Graz (AT)
Prof. Cino Viggiani	Univ. of Grenoble, Laboratoire 3R (FR)
Prof. Harvey Burd, Prof. Byron Byrne, Prof. Ross McAdam	University of Oxford (UK)
Prof. Minna Karstunen, Dr. Mats Olsson, Dr. Anders Kullingsjö	Chalmers University of Technology (S)
Prof. Andrew Whittle	Massachusetts Institute of Technology (USA)
Prof. Juan Pestana	University of California at Berkeley (USA)
Prof. Richard Finno	Northwestern University (USA)
Prof. Youssef Hashash	Univ. of Illinois at Urbana-Champain (USA)
Dr. Anoosh Shamsabadi	California Department of Transportation (USA)
Prof. Steinar Nordal, Prof. Gustav Grimstad	Norwegian Univ. of Science and Tech (NO)
Dr. Lars Andresen, Prof. Hans Petter Jostad, Dr. Nallathamby Sivasithamparam	Norwegian Geotechnical Institute (NO)
Prof. Antonio Gens, Prof. Eduardo Alonso	Technical University of Catalunya (ES)
Prof. Harry Tan	National University of Singapore (SG)

Name	Organization (Country)
Prof. Angelo Amorosi	Sapienza University of Rome (IT)
Prof. Yasser El-Mossallamy	Ain Shams University, Cairo (EG)
Prof. Ivo Herle	Technical University, Dresden (DE)
Prof. David Mašin	Charles University, Prague (CZ)
Prof. Tim Lansivaara	University of Tampere (FI)
Prof. Tom Schanz y, Prof. Günther Meschke	Ruhr University, Bochum (DE)
Prof. George Gazetas, Dr. Nikos Gerolymos	National Technical University, Athens (GR)
Prof. Steven Kramer, Prof. Pedro Arduino	University of Washington (USA)
Prof. Christophe Geuzaine	University of Liege (BE)
Prof. Yves Renard	INSA-Lyon (FR)
Prof. Mahdi Taiebat	University of British Columbia (CA)
Prof. Daniela Boldini	University of Bologna (IT)

This support is gratefully acknowledged.

The editors

1.2 PLAXIS product, licencing and services

Updated versions and new releases of PLAXIS, containing various new features, are released frequently. In addition, courses and user meetings are organised on a regular basis. Registered users receive detailed information about new developments and other activities. Valuable user information is provided on the Bentley website and Bentley Communities.

Products

In addition to PLAXIS Monopile Designer, PLAXIS offers powerful products for specific geotechnical analysis. These software packages are listed below:

- PLAXIS 2D
- PLAXIS 3D
- PLAXIS 2D Output Viewer and PLAXIS 3D Output Viewer
- PLAXIS Designer
- PLAXIS 2D LE
- PLAXIS 3D LE

Note:

1. PLAXIS Monopile Designer can be used as a stand-alone tool for the rule-based design method.
2. PLAXIS Monopile Designer can perform the numerical-based design method, as defined in the PISA research project, in connection with PLAXIS 3D. In this case, soil reaction curves, used in the one-dimensional finite element kernel of PLAXIS Monopile Designer, are derived and calibrated from the results of a series of 3D finite element calculations performed in PLAXIS 3D.
3. For more information about PLAXIS products and the different geotechnical analysis that are possible please visit the PLAXIS 3D General Information Manual.

Licencing

PLAXIS products are offered under a particular licencing schemes. For detailed information visit Bentley Communities.

In the case of PLAXIS Monopile Designer, currently, there is one comprehensive licence is necessary to apply the rule-based design method; however, a PLAXIS 3D distribution and Geotechnical SELECT Entitlements (see Bentley Communities) are required to access functionalities such as *autogeneration of 3D models* and *scripting*.

Services: Geotechnical SELECT Entitlements [GSE]

Geotechnical SELECT Entitlements [GSE] is an additional subscription system on top of the professional software licenses. Geotechnical SELECT subscribers benefit from the latest releases of their PLAXIS software maintenance, support from PLAXIS technical experts and extended features. Functionalities part of Geotechnical SELECT Entitlemets are mainly focused on interoperability with other Bentley Systems software (e.g., CAD or ISM import), scripting and satellite tools based on scripting. Further information about [GSE] features can be checked on Bentley Communities.

1.3 Short overview of features

PLAXIS Monopile Designer is a software package intended for the design of monopiles as foundation elements for offshore wind turbines under lateral loading conditions. As a design tool, it includes a highly efficient one-dimensional finite element calculation model based on the Timoshenko beam theory to model the monopile, and non-linear soil reaction curves to model the soil response. PLAXIS Monopile Designer also facilitates the efficient generation and calculation of a series of PLAXIS 3D models for the calibration of soil reaction curves and for checking final monopile designs. The calibration process is fully automated. A brief summary of the important features of PLAXIS Monopile Designer is given below.

Input of soil stratigraphy

Based on a preliminary selection of either clay-type or sand-type soils, PLAXIS Monopile Designer facilitates the efficient input of basic soil properties in layers.

Generation of PLAXIS 3D models

Monopiles can be defined by only a few geometric parameters. Based on geometric data sets, together with the soil stratigraphy, PLAXIS 3D finite element models are automatically generated and calculated, with the purpose to extract the soil response under lateral loading conditions. This requires the presence of a compatible PLAXIS 3D [GSE] subscription (see PLAXIS 3D General Information Manual) .

Visualisation option

Convenient visualisation option is available to preview and check each generated model in PLAXIS 3D before starting the calculation process.

Modification of generated models

Generated models can be modified in PLAXIS 3D, if desired, provided that the modified model represents the same situation as originally created in PLAXIS Monopile Designer. It is possible to change the soil constitutive models used in PLAXIS 3D. Any constitutive model can be used in place of the default selection, including user-defined soil models.

Calibration of soil reaction curves

Soil reaction curves, used in PLAXIS Monopile Designer design calculations are automatically calibrated based on the extracted soil response from the PLAXIS 3D models. In addition to conventional non-linear p-y curves for lateral loading, PLAXIS Monopile Designer provides additional moment-rotation reactions along the pile shaft as well as shear and moment reactions at the pile base, according to the PISA design methodology.

Efficient 1D design calculations

Using the calibrated (or user-defined) soil reaction curves, PLAXIS Monopile Designer enables a quick design calculation and optimisation of monopile dimensions under lateral loading conditions; both for SLS and ULS design. Calculations are based on Timoshenko beam theory, encapsulated in the built-in one-dimensional finite element model. PLAXIS Monopile Designer can run as a stand-alone package to perform 1D design calculations without the need to have other PLAXIS software installed.

Presentation of results

PLAXIS Monopile Designer facilitates the presentation of various results in both graphical and tabulated format. Results can be copied to clipboard and printed.

1.4 Manuals

To obtain a quick working knowledge of the main features of PLAXIS Monopile Designer, it is suggested that users work through the example problems contained in the 3 Tutorial 1 - Numerical Based Design (p. 87) and 4 Tutorial 2 - Layered Soils (p. 103).

The Reference Manual is intended for users who want more detailed information about program features. This manual covers topics that are not discussed exhaustively in the Tutorial manuals. It also contains practical details on how to use PLAXIS Monopile Designer for the design of monopiles according to the PISA method.

Also the Reference Manual is arranged according to the modes 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 or the corresponding sections of the PLAXIS 3D manuals.

1.5 First time installation

If you install PLAXIS Monopile Designer for the first time, download the installer from PLAXIS software downloads at Bentley Communities. You will need to create an account and log in to access the download site.

Software and hardware requirements

Please visit System Requirements on Bentley Communities website to find the software and hardware requirements.

Installation

During installation the following software is installed:

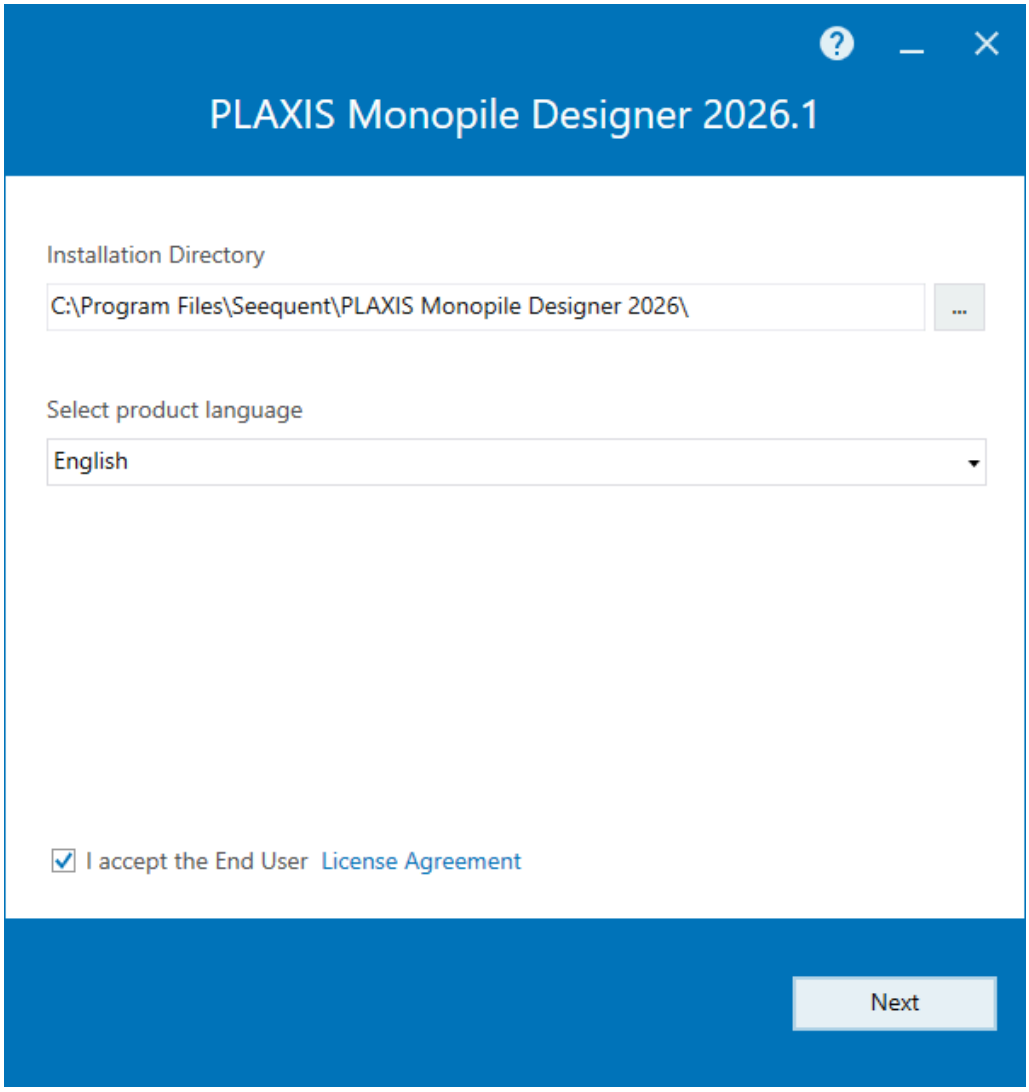
- PLAXIS Monopile Designer
- CONNECTION Client
- PLAXIS 3D
- PLAXIS Python distribution
- Manuals for PLAXIS Monopile Designer

To install PLAXIS Monopile Designer:

1. Go to the Installer executable that you have downloaded and double-click the file.

   The PLAXIS Monopile Designer Installation Wizard opens.

2. (Optional) To change the location where PLAXIS Monopile Designer is installed, either:
   - Type a path in the *Installation Directory* field or
   - Click the *Browse* button (...) and browse to the folder you want to install PLAXIS Monopile Designer.
   - Click *Ok*.



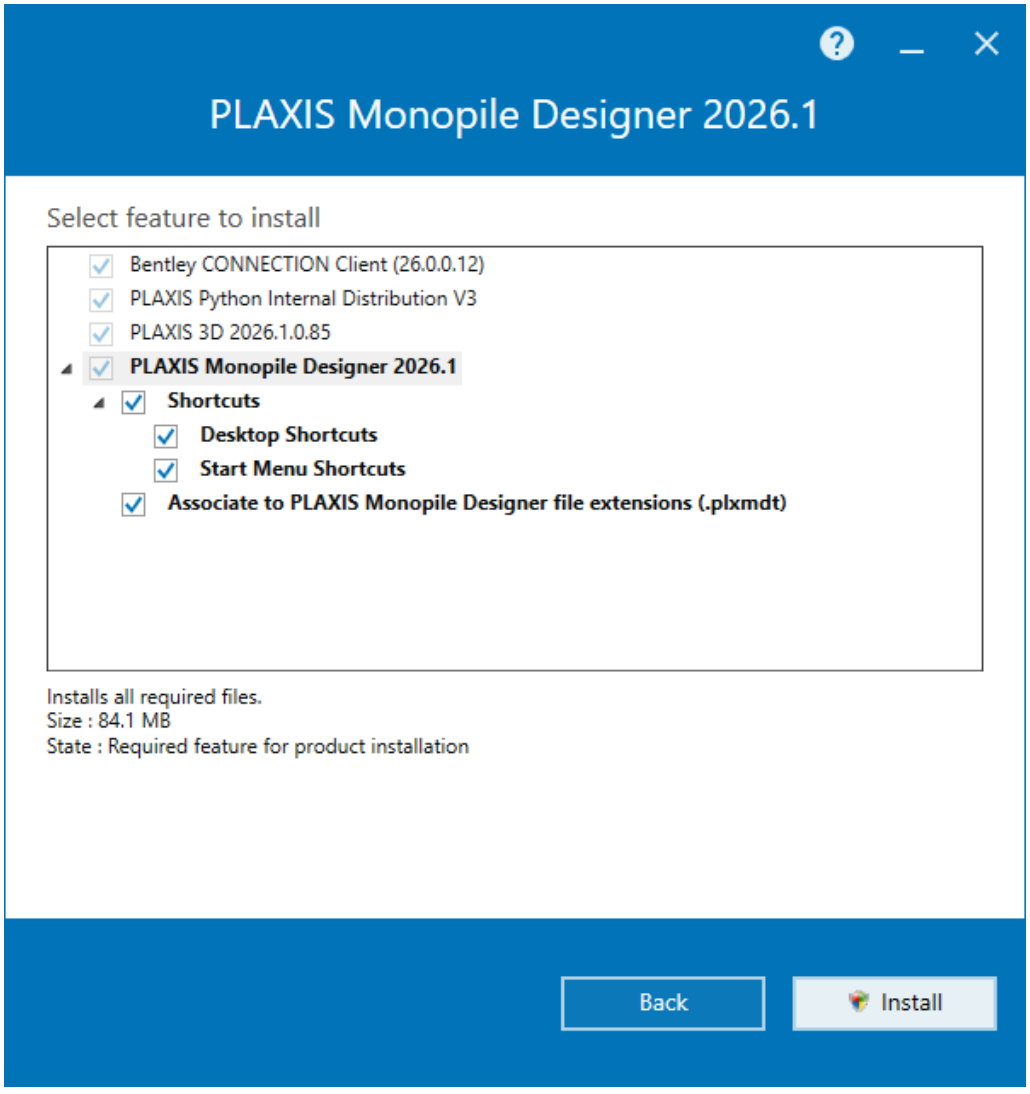
3. To select the installation language use the drop-down menu in *Select product language*.
4. To read the End-User License Agreement (EULA), click the *License Agreement link*.

   The EULA opens in a web browser.

   - After reading the license agreement, check the *I accept the End User License Agreement* box to acknowledge that you understand and agree to the EULA.

     This step is required to install the software.
   - Click *Next*.

5. (Optional) Select the features which you want to install.
   a. On PLAXIS Monopile Designer:



   b. If the user wants to enhance the capabilities PLAXIS Monopile Designer and allow the *autogeneration of 3D models* the installation of PLAXIS 3D is suggested (please visit the PLAXIS 3D General Information Manual for more information).

6. Click *Install* to start the installation.

> **Note:** The installation requires administrator rights. If Windows prompts you with a User Account Control dialog, click *Yes* to proceed.

7. Once the installation has finished the Install Wizard will notify you.

   >Installed PLAXIS 2026.1

   - Click *Finish* to close the Wizard.

2

Reference Manual

2.1 Introduction

PLAXIS Monopile Designer is a PLAXIS-based specific tool providing an enhanced design methodology for monopile foundations under lateral loading. Monopile design can be performed efficiently by using one-dimensional (1D) finite element (FE) analyses of a laterally loaded pile. The adopted design methodology is based on the PIle Soil Analysis (PISA) joint industry research project.

The monopile is modelled by means of the Timoshenko beam theory whereas the soil reaction is modelled using calibrated or user-defined soil reaction curves. The calibration of the soil reactions is based on three-dimensional (3D) finite element calculations using PLAXIS 3D. In addition to the 1D design analysis, the design tool facilitates the generation and calculation of the PLAXIS 3D models, and the derivation of the soil reactions based on the calculation results. A real installation site can be represented with finite element models in PLAXIS 3D and a site-specific 1D model can be calibrated and used for the design of monopile foundations.

2.1.1 The PISA Project

The PISA project was a research study (2013-2016) on the development of new design procedures for monopile foundations for offshore wind turbine applications. The project consisted of field testing, numerical modelling and design model development. The research was conducted by an Academic Work Group drawn from the University of Oxford, Imperial College London and University College Dublin and was conducted in collaboration with a range of project partners. Ørsted (then DONG Energy) took the lead role for the partners. The broad scope of the PISA study is summarised in conference publications (e.g. Byrne et al., 2018, Byrne

et al., 2017, Burd et al., 2017, Byrne et al., 2015a, Byrne et al., 2015b, Zdravkovic et al., 2015) and more recently in a themed issue of the journal *Géotechnique* (Volume 70 Issue 11, editorial in Byrne, 2020). One outcome of this study is a one-dimensional (1D) design model, based on the use of Timoshenko beam theory, that overcomes certain limitations of existing methods (Burd et al., 2020a, 2020b, Byrne et al., 2020).

PLAXIS Monopile Designer provides a means of implementing the PISA design method in a daily engineering context, for the design of monopile foundations for offshore wind turbines including large diameter monopiles in clay (Panagoulias et al., 2018a, Panagoulias et al., 2018b), sand (Brinkgreve et al., 2020, Panagoulias et al., 2020), and layered soils (Panagoulias et al., 2019).

2.1.2 The design methodology

The PISA project resulted in a new design methodology, which employs rapid, 1D design calculations, based on the use of the Timoshenko beam theory to model the behaviour of an embedded monopile under lateral loading. Additional components of soil reaction are integrated in the design model to enhance its performance. The pile self weight and any additional vertical loads are not taken into account as primarily lateral loading and not vertical loading is considered.

The proposed design method consists of two alternative design procedures (Byrne et al., 2017), both incorporated in the design tool. PLAXIS Monopile Designer can be used as a stand-alone tool for the rule-based design method and in connection with PLAXIS 3D for the numerical-based design method.

Rule-based design

In the rule-based design approach, soil reaction is defined via mathematical functions, the parameters of which are determined via standard soil investigation data. According to this design procedure, the 1D model calibration data can be imported from previous numerical-based calibrations on other projects, from standard publications or supplied by a consultant. It should be noted that, in this case, the soundness of the pile response prediction depends on the difference in the soil profiles, the considered pile geometries and the loading conditions between the original calibration case and the target design study. Thus, the rule-based design approach is likely to be used for concept or preliminary design.

Numerical-based design

The numerical-based design approach involves 3D FE models for site-specific, and possibly more accurate, calibration of the soil reaction. Detailed 3D FE calculations are employed along with high quality soil data, potentially obtained via site investigation and laboratory/field testing for the calibration of the used soil constitutive models. Subsequently, the 1D design model is calibrated based on the results of the 3D FE analyses. In this way, the numerical-based design approach is likely to be used for detailed design.

Note that PLAXIS Monopile Designer deals with the calibration of the advanced soil constitutive models employed in PLAXIS 3D, based on limited input soil data, via predefined empirical correlations. The user may fine-tune the derived values of the material parameters if necessary. The reader may refer to 2.3.4 Soil Profile tabsheet (p. 39) and 5.2 Material Models (p. 117) for more information.

Each 3D FE model represents a design scenario for the considered design study. It is suggested to choose the variation on the monopile geometry configurations, and consequently the number of the employed calibration 3D FE models, such that an appropriate coverage of the calibration space is achieved. Based on experience (Panagoulias et al., 2018a, Panagoulias et al., 2018b) 8 to 10 calibration models are generally sufficient to calibrate the soil reaction curves, although good results can be achieved with as few as 4 calibration models (Kaltekis et al., 2019).

It is highly recommended that the results of the 1D FE model for the final design configuration are checked against an equivalent 3D FE model to validate the soundness of the 1D analysis.

The numerical-based design philosophy provides a well-based means of continuous advancement of the soil reaction curves, towards a global database of site-specific and calibration space-specific curves. The database could be effectively extended as new site investigation data together with soil testing data are obtained from specific offshore locations. In addition, improvements on the used numerical methods and/or the employed constitutive models could be used to enrich the database and possibly enhance existing soil reaction data sets.

2.1.3 Interoperability with PLAXIS 3D

PLAXIS Monopile Designer reaches its full potential when used in connection with PLAXIS 3D. The latter offers a complete, well proven and robust finite element solution for offshore or onshore structures. The coupling of the two software packages facilitates the numerical-based design, via the automatic calibration of the soil reaction curves for the specific design case.

The design tool provides automatic generation and calculation of 3D FE calibration models in PLAXIS 3D based on specific soil input data and value ranges of the monopile geometry components (length, diameter, wall thickness and height above the seabed where the load is applied). Soil reaction curves are extracted from the 3D FE models and turned into parameterised functions based on the defined soil properties and geometrical parameters.

2.1.4 The 1D FE Model

PLAXIS Monopile Designer facilitates the execution of rapid 1D design calculations. A 1D FE model is integrated in the design tool, formulated by means of the Timoshenko beam theory. The adopted formulation embodies, in an approximate way, the influence of the shear strains to the overall pile response. This influence is likely to increase with decreasing length-to-diameter ratios (Byrne et al., 2015, Burd et al., 2017).

If the rule-based design approach is followed, the 1D model makes direct use of the user-imported soil reaction data. If the numerical-based design is employed, the calibration of the 1D model involves a limited set of 3D numerical calculations, which span an assumed design space for the monopile foundation. Soil reaction curve data are derived from the 3D models; they are then used in the 1D FE model. The latter is used to conduct a range of (rapid) design calculations to optimise the monopile geometry, based on the assumed soil conditions at the site and the monopile design space.

The main components of the 1D design model are depicted in Figure 2–1 (p. 17). Under a horizontal force H and a moment M applied to the pile at a certain height above the ground level, four components of soil reaction are acting on the embedded part of the pile:

- the distributed lateral load p
- the distributed moment m
- the base horizontal force H_B
- the base moment M_B

The distributed lateral load p acts along the pile shaft and it is consistent with the approach adopted by the conventional p-y method. The additional component of the distributed moment m along the pile shaft results from the vertical shear tractions induced at the soil-pile interface, due to local pile rotation. Besides, if the pile is loaded close to failure, considerable shear tractions are likely to be developed on the passive side of the pile due to the induced wedge-type failure mechanism (Burd et al., 2017). Two separate soil reaction components are acting on the base (toe) of the pile, namely the base shear force H_B and the base moment M_B. The effect

of the additional components on the pile response becomes more dominant as the length-to-diameter ratio reduces (Burd et al., 2017).

In line with the conventional p-y design method, all components of soil reaction are applied to the embedded beam elements via the Winkler approach (Winkler, 1867). This implies that the soil reaction components mentioned above are linked to local pile displacements and rotations. Despite any limitations of this approach, mainly related to the uncoupling between adjacent elements, it constitutes a direct and computationally efficient formulation approach for the 1D design model.

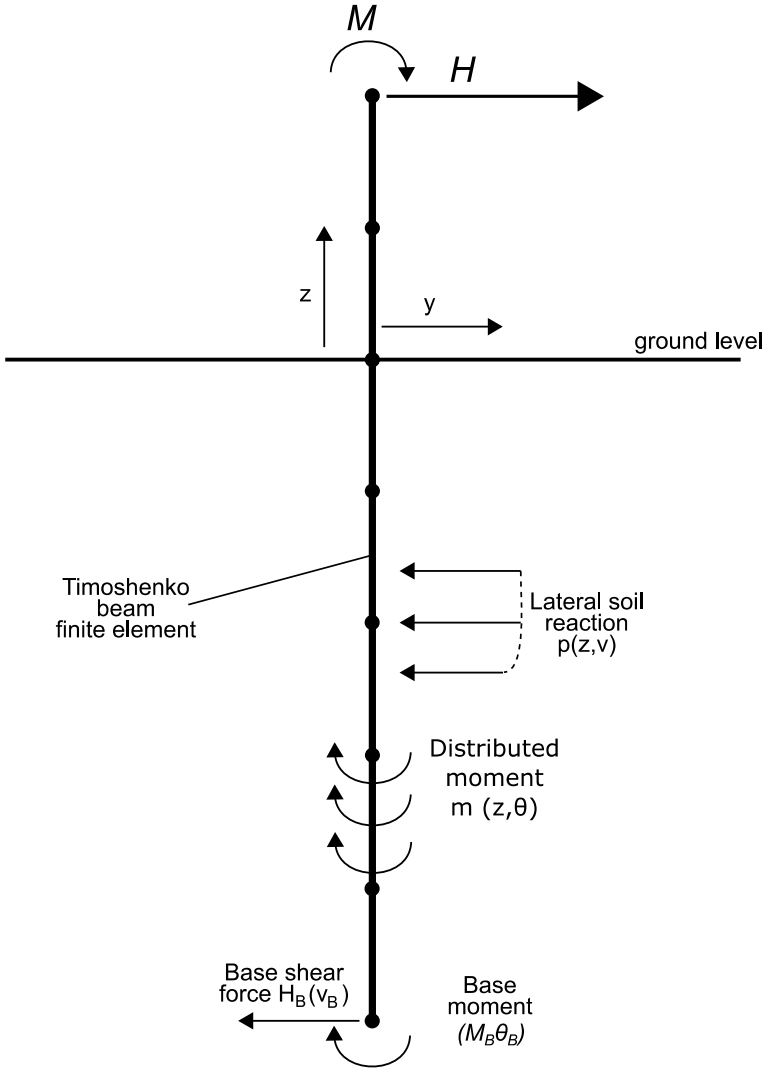


Figure 2–1: Components of the 1D FE model (based on Byrne et al., 2015b)

2.1.5 The components of PLAXIS Monopile Designer

The tool consists of three main individual components, which communicate via the Graphical user interface (GUI). Each component deals with different parts of the calculation process (Figure 2–2 (p. 18)).

Component 1: the 1D FE model

This component is based on the use of Timoshenko beam theory to model the behaviour of an embedded monopile. The soil response is modelled via soil reaction curves, applied along the shaft and at the base of the monopile.

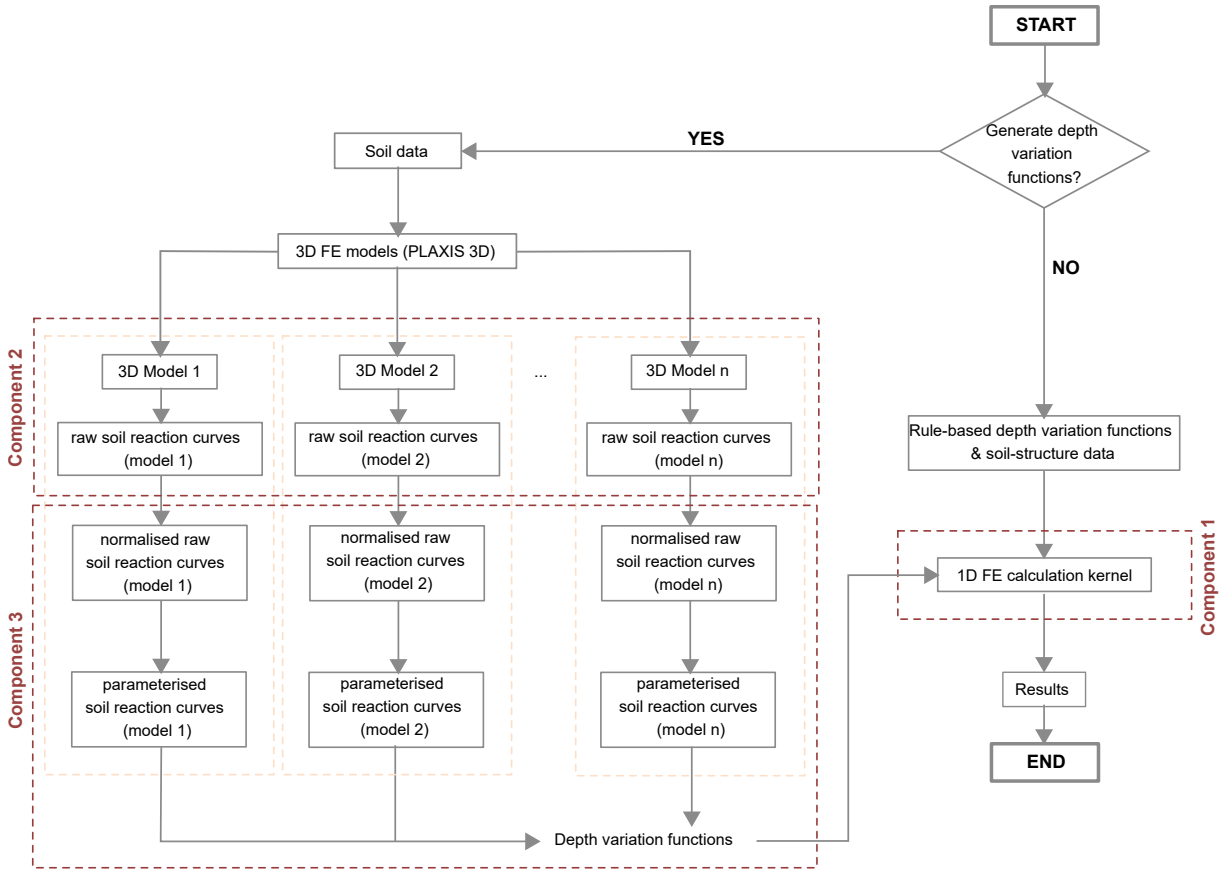


Figure 2–2: PLAXIS Monopile Designer workflow

Component 2: a set of 3D FE models

This component facilitates the automatic generation and calculation of a set of 3D FE calibration models in PLAXIS 3D, to obtain sets of raw soil reaction curves.

Component 3: the Optimisation Module

This component deals with the parameterisation of the soil reaction curves derived from the PLAXIS 3D models, i.e. the transformation of the raw soil reaction curves to mathematical functions which are subsequently used by the 1D FE model.

2.1.6 Graphical user interface

The Graphical User Interface (GUI) deals with the exchange of data among the three main individual components. Moreover, it presents the calculation results from the 3D (if employed) and the 1D FE analyses. The GUI consists of there operational modes, namely the **Calibration mode**, the **Analysis mode** and the **Results mode**.

If the rule-based design is followed, only the last two modes of the design tool are used, i.e. the **Analysis mode** and the **Results mode**. The soil reaction data are imported in the **Analysis mode** to run rapid 1D FE calculations. The **Results mode** provides the results of the 1D analysis.

If the numerical-based design is adopted, all three modes of the tool are used sequentially. The **Calibration mode** is used to: define the site-specific soil layers and soil data, define the various monopile geometric configurations, to generate and calculate the PLAXIS 3D models based on the data coming from this mode, likewise to extract and parameterise the soil reaction curves. Relevant results from the 3D FE analyses are presented in the **Calibration mode** as well.

The parameterised soil reaction curves are imported in the **Analysis mode** to run the 1D FE analysis, whereas the **Results mode** provides the obtained results.

2.1.7 Useful terminology

Basic terminology adopted throughout the design tool and this manual, is presented below.

Design space

See 2.3.5.2 Design Space (p. 44) .

Soil reaction curves

The raw soil reaction curves represent the functions which relate the non-linear soil reactions (force or moment) to the local pile deformation (displacement or rotation). They are based on the data extracted directly from the PLAXIS 3D models. Four types of raw soil reaction curves are considered to simulate the behaviour of an embedded monopile under lateral loading, namely:

- Distributed lateral load p - lateral displacement v
- Distributed moment m - rotation θ.
- Base horizontal force H_B - lateral displacement v_B
- Base moment M_B - base rotation θ_B

Parameterisation procedure

The parameterisation procedure is conducted in the **Calibration mode**, if the numerical-based design is followed, by the Optimisation Module (Figure 2–2 (p. 18)). It consists of several sub-processes, including the normalisation of the raw soil reaction curves, the calibration of the mathematical function which approximates the non-linear soil reaction curves and the optimisation of the derived fitting parameters.

Depth variation functions

Each type of the non-linear soil reaction curves is approximated with a mathematical function during the parameterisation procedure. The mathematical function itself constists of certain fitting parameters. The depth variation functions define the variation of each one of the fitting parameters as a function of depth.

dvf file

A file with a specific format used to define the parameterised soil reaction curves. It also includes relevant data for the site-specific soil conditions and the design (calibration) space based on which the soil reaction curves were generated. The file is used as input to the 1D design model to run the 1D FE analysis. It can either be user-defined (rule-based design) or produced via the parameterisation procedure (numerical-based design) in the **Calibration mode**.

Monopile head and toe

The term Monopile head refers to the level at distance h above the seabed level, at which either a prescribed displacement (**Calibration mode**) or a lateral load H and/or a bending moment M (**Analysis mode**) are applied to the monopile. Note that this level may not necessarily coincide with the actual monopile head. If h is zero, then the supposed head meets the mudline. The term Monopile toe refers to the base of the monopile at distance L below the seabed level.

2.2 General Information

Information in this chapter applies to all modes of the design tool.

2.2.1 Using PLAXIS Monopile Designer with and without PLAXIS 3D

Functionality without PLAXIS 3D [GSE]

Without PLAXIS 3D [GSE], the user can access the last two modes (**Analysis** and **Results mode**), which are related to the 1D calculation. The functionality to generate, calculate, parameterise and visualise PLAXIS 3D models (i.e. elaborated in 2.3.5 Geometry datasets tabsheet (p. 42)) is not available unless PLAXIS 3D [GSE] is installed. Nevertheless, existing calibrated soil reaction curves can be used in the **Analysis mode** to perform monopile design calculations.

For more information on PLAXIS [GSE], please communicate with our PLAXIS Sales Department at contact us.

Functionality with PLAXIS 3D [GSE]

If PLAXIS 3D [GSE] is present, the full functionality provided by the design tool is available.

2.2.2 Program layout

To carry out analysis and design calculations using PLAXIS Monopile Designer, the user has three modes to work with: **Calibration mode**, **Analysis mode**, and **Results mode**. Each mode appears as a coloured tabsheet in PLAXIS Monopile Designer.

After starting the program, the user chooses whether to open an existing project or start a new one. A new project (Figure 2–3 (p. 21)) must first be saved before being worked with.

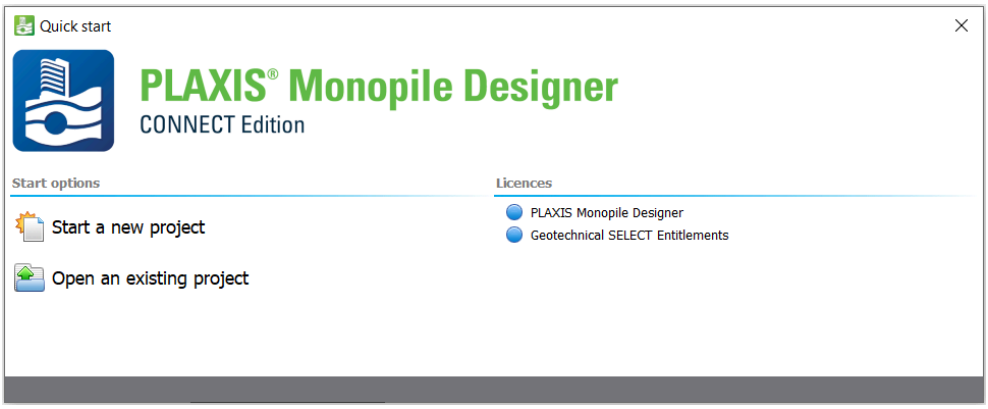


Figure 2–3: Start Screen

The general layout of the program is shown in:

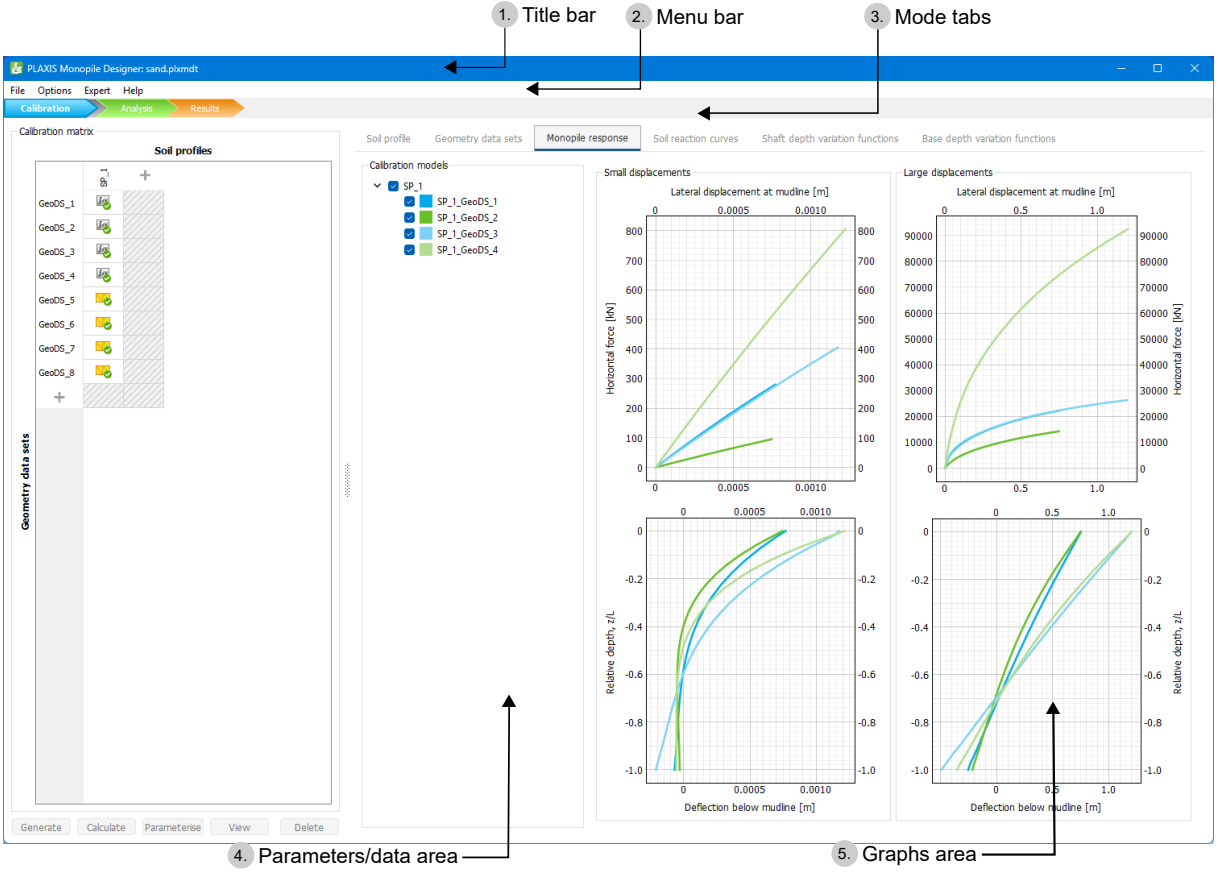


Figure 2–4: Layout of the program

The contents of the window differ for the different modes, all of which are described in their sections. The main and common items are as follows:

Title bar

The name of the program and the title of the project is displayed in the title bar. Unsaved or unelaborated modifications in the project are indicated by an asterisk ('*') next to the project name.

Menu bar

It contains a File, Options, and Help menu.

Mode tabs

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

Calibration	Optional mode for the users with access to PLAXIS 3D, to define the soil stratigraphy, to generate and calculate 3D FE models, the soil reaction curves which will be extracted and parameterised.
Analysis	To run the 1D FE Analysis (Static and Modal).
Results	To view the results of the 1D FE Analysis.

Note: After analysis in the **Analysis** and **Results modes** and then modifying the data in the **Calibration mode**, the last two modes are marked by an asterisk. This is to indicate that the *.dvf file used in the analysis might not be valid anymore.

Parameters area

Each mode has different fields and different parameters the user can set. In the data area, the user can add soil layers, add geometric data sets (GeoDS), set structural properties, and much more.

Graphs/tables area

This area represents the results graphically. The graphs can be customised by changing axes and plot options. They are available in **Calibration**, **Analysis**, and **Results mode**.

2.2.3 New Project

How to access:

- **File** > **New project**
- **CTRL + N**

At the start of the program, the user sees the **Calibration mode** with an empty data set. The user chooses whether to open an existing project or start a new one. A new project (Figure 2–3 (p. 21)) must first be saved before being worked with.

In the numerical-based design, the first step the user should take is to add soil layers and configure the material data. For more information on the 2.3.4 Soil Profile tabsheet (p. 39).

In the rule-based design, the first step the user should take is to switch to **Analysis mode** and upload a **.dvf* file containing the soil reaction curves (either calibrated or user-defined). For more information on the 2.4 Analysis mode (p. 57)

2.2.4 Open Project

How to access:

- **File** > **Open project**
- **CTRL+O**

The user may open an existing project by searching for it in Windows® requester.

2.2.5 Menus in the Menu bar

The menu bar of the program contains drop-down menus covering most options for handling files and setting options.

File menu	
New project	To start a new project.
Open project...	To open an existing project.
Save project	To save the current project under the existing name.
Save project as...	To save the current project under a new name.
Exit	To leave the program.

Option menu	
Display numbers using:	
4 significant digits	To display numbers using 4 significant digits.
5 significant digits	To display numbers using 5 significant digits.
6 significant digits	To display numbers using 6 significant digits.

Note: The default global number of significant digits is 4.

Expert menu	
Run Python script	To configure remote scripting server and open Python scripts to run them.
Run Python tool	To access and run stored and commonly used Python scripts.

Help menu	
Manuals	To display the manuals.

Help menu		
Learning content	Tips and tricks	To display a comprehensive directory of "Tips and Tricks" and Frequently Asked Questions (FAQs) for PLAXIS Monopile Designer.
	Videos	To reach the Bentley communities website where instruction videos to get started with PLAXIS Monopile Designer.
License configuration...	To view, configure and update licences.	
About	Disclaimer	To display the complete disclaimer text.
	About PLAXIS Monopile Designer	To display information about the program version and licence.
Contact Support	To send a request for support.	

2.2.6 Units and sign convention

Standard units

PLAXIS Monopile Designer uses a consistent system of units. The basic units are:

- Length: m
- Force: kN
- Moment: kNm
- Stress: kN/m^2
- Unit weight: kN/m^3

All input data should conform to the adopted system of units, and the output data should be interpreted using the same system. Every example used in the manual is defined using these standard units.

Sign convention used in PLAXIS 3D

The following applies to the PLAXIS 3D FE models generated by PLAXIS Monopile Designer.

- Positive x-, y-, z-direction as displayed in Figure 2–5 (p. 25).
- Positive moment: right-handed coordinate system.
- Compressive stress: negative (solid mechanics convention).

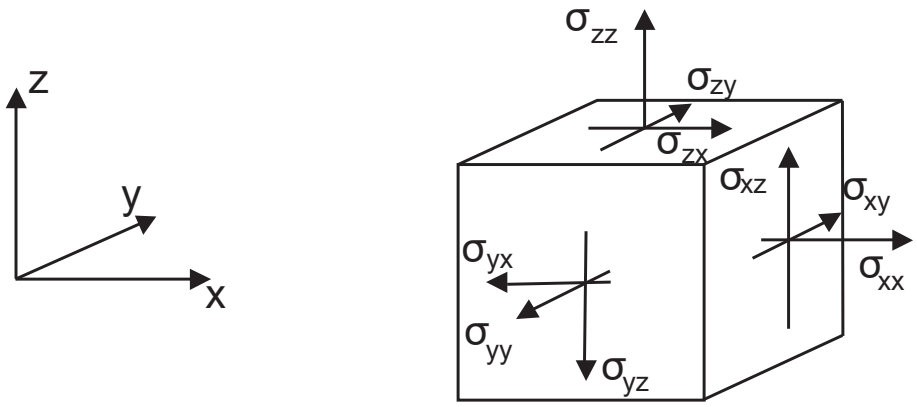


Figure 2–5: Coordinate system and the indication of positive stress components

Sign convention used in 1D model

The following applies to the 1D FE model.

- Positive y-, z-direction as displayed on Figure 2–1 (p. 17) positive lateral load p and moment m as displayed in Figure 2–6 (p. 26).

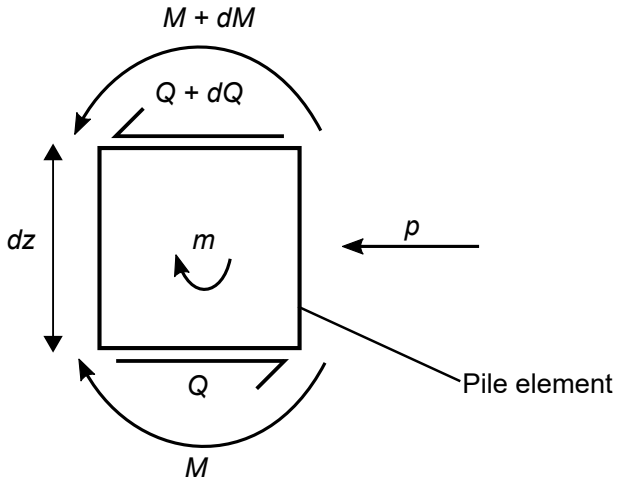


Figure 2–6: Sign convention in 1D FE model

With this sign convention the variables are related by:

$$p = -\frac{dQ}{dz} \qquad m - \frac{dM}{dz} = Q \tag{2–1}$$

where

Q = Pile shear force.

M = Pile bending moment.

2.2.7 Automatic saving

When creating a new project, the user needs to choose a name and location for the project. The project data can become very large, so PLAXIS Monopile Designer performs automatic saving before certain actions. The program lets the user know when the project is automatically saved by displaying warnings and the save icon on the corresponding buttons of the UI.

Note:

The actions before which the project will be automatically saved are all encountered in the **Calibration mode**:

- Adding a new GeoDS
- Deleting a GeoDS
- Generating a model
- Calculating a model

2.2.8 Help facilities

PLAXIS Monopile Designer provides extensive help facilities for the users. In the Help menu (2.2.5 Menus in the Menu bar (p. 23)), there is a link to the Monopile designer manuals in PDF form.

Knowledge base

Additional information can be found on Bentley Communities.

Customer support [GSE]

Need Help? Tell us about your issue and find the best support option on Bentley Communities/support.

2.3 Calibration mode

The **Calibration mode** is intended for users who want:

- To follow the numerical-based design approach
- To define the soil stratigraphy for the PLAXIS 3D models
- To use PLAXIS 3D to generate and run a set of 3D models
- To extract the soil reaction curves
- To parameterise soil reaction curves and generate (soil-type and design-space dependent) depth variation functions.

> **Note:**
>
> - For the numerical design approach, the **Calibration mode** should be used before the **Analysis mode**.

As mentioned the **Calibration mode** is used to define the soil stratigraphy for the PLAXIS 3D models that are generated to calibrate soil reaction curves. Hence, the user must first choose which is the (dominant) material type in the subsoil for the considered homogeneous soil profile. Depending on the material type (clay or sand), a different set of soil parameters needs to be specified. These parameters are employed in the soil models that are used in the PLAXIS 3D model (see 5.2 Material Models (p. 117)). Some parameters are also used to normalise the soil reaction curves. Although only one particular soil type can be selected, the user may define as many sub-sections (Soil layers) as necessary to accurately represent a measured stiffness profile (G_0) or shear strength profile (s_u) in depth.

> **Tip:** Analysis of layered soils according to the PISA method is based on the hypothesis that soil reaction curves calibrated using homogeneous soil profiles can be employed, directly, to conduct 1D analyses of monopiles embedded in a layered soil (Burd et al., 2020b). The depth variation functions parameterised for ideal homogeneous profiles in the **Calibration mode** can subsequently be used to configure layered stratigraphies in the **Analysis mode**.

For the numerical-based design approach the **Calibration mode** makes use of PLAXIS 3D to generate and run a set of 3D models, to extract the soil reaction curves, parameterise them and generate (soil-type and design space dependent) depth variation functions. The **Calibration mode** should be used before the **Analysis mode** as its results constitute an input for the **Analysis mode**.

The **Calibration mode** is used to define the monopile geometric dimensions for the PLAXIS 3D models that are generated to calibrate soil reaction curves. The monopile geometry is defined by the height above the ground level h at which a horizontal displacement is applied, the embedded length *L*, the diameter *D* and the wall thickness t. For each geometric data set (GeoDS), the target displacement at the mudline is specified as a fraction of the pile diameter (v_g/D_{out}).

2.3.1 Calibration procedure for soil reaction curves

The procedure to calibrate soil reaction curves consists of three steps.

1. Generating PLAXIS 3D models based on the soil profile and the GeoDS defined in the **Calibration mode**. This step will not only create the geometry model in PLAXIS 3D but also the 3D finite element mesh and the necessary calculation phases.
2. Calculating the selected finite element models in PLAXIS 3D. Note that this step can be quite time-consuming since several 3D finite element calculations are performed. The result of this step is a set of raw soil reaction curves obtained from each of the finite element model calculations.
3. The parameterisation of the raw soil reaction curves obtained from the 3D finite element calculations.

2.3.2 Calibration mode layout

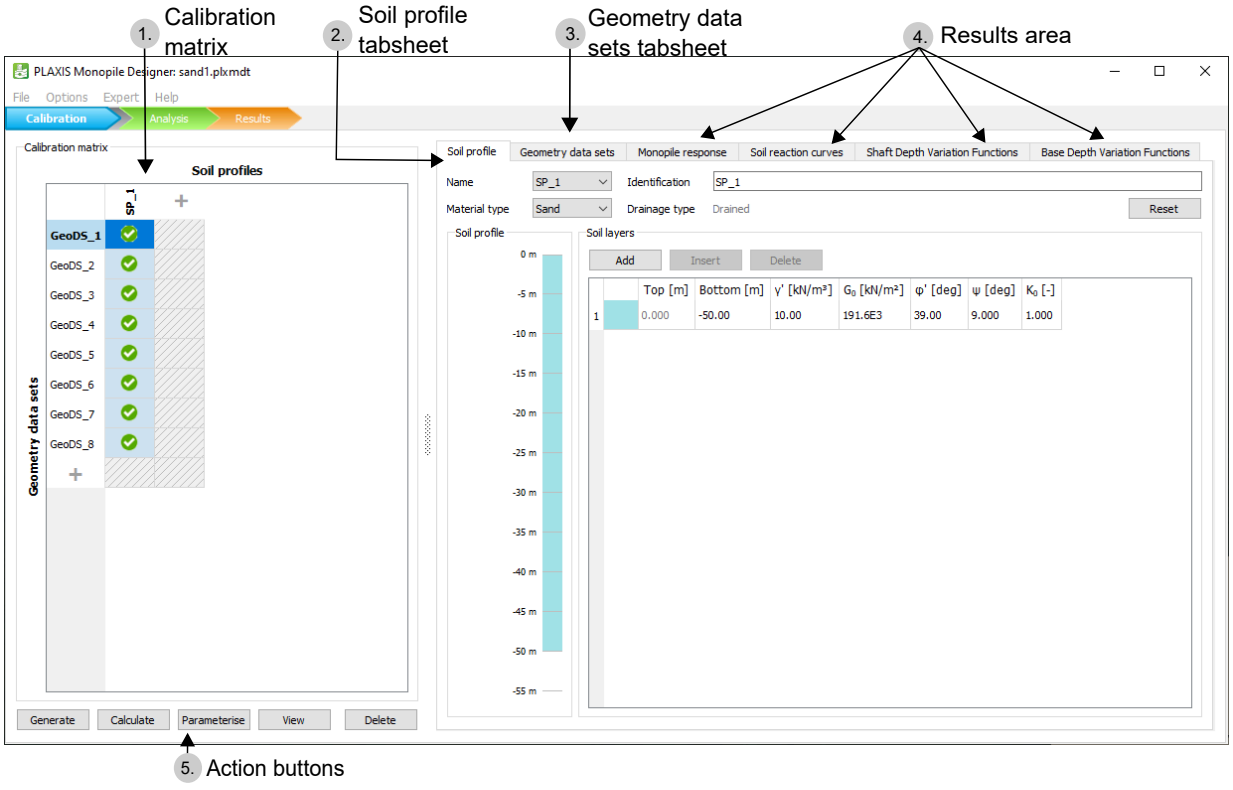


Figure 2–7: Layout of the Calibration mode

Geometry datasets

Tabular overview of the geometrical configuration of different monopiles per soil layer grouped in data sets. Each data set is identified by the name GeoDS_#, where # is the number of the data set.

Adding/deleting sets can be done inside the **Calibration Matrix** area using the **+** and **delete** buttons.

Editing the properties of each GeoDS can be done inside the **Geometry datasets** tabsheet.

> **Note:**
>
> The name of each geometry data set is assigned automatically and cannot be changed. However, it is possible to specify a custom identification by selecting Rename. Previously deleted numbers are not reused unless the deleted numbers were the highest. The same name is used for the corresponding finite element model generated in PLAXIS 3D. The selection here determines which structural, pile properties and results are shown. Multiple data sets may be selected simultaneously. Actions are performed on all selected data sets.

- **Structural properties:**

  Overview and editing of monopile material parameters.

- **Pile properties:**

  Static schematic 3D model of a monopile and the calculated geometrical and mechanical properties (*A*, *I*, *EA*, *EI* and *GA*, for more information, see Table 2–3 (p. 45)).

Calibration Matrix

The Calibration matrix allows defining multiple soil profiles at the same time that allows the navigation and status visualization for different Calibration models.

Soil profile

The Soil profile tabsheet is an overview of the soil stratigraphy allowing to define the material, drainage type, layering and properties of each layer of a soil column.

Results area

View of results after calculation/parameterisation. These are presented in the 2.3.6 Results inspection pane (p. 46) .

> **Note:**
>
> - The 2.3.6 Results inspection pane (p. 46) appears when any of the following tabsheets are active: **Monopile response**, **Soil reaction curves**, **Shaft Depth Variation Functions**, and **Base Depth Variation Functions**.

Action buttons - Generate, Calculate, Parameterise

Executing the corresponding action on the data selected in the table of GeoDS. One or multiple datasets can be selected.

View button

Visualisation of the 3D model that corresponds to the selected dataset in PLAXIS 3D, where the user is allowed to modify the models manually. For example, the user can change default material parameters. Note that this should be done with caution.

Delete button

Allows to delete one or multiple geometry datasets selected from the Calibration Matrix.

Note:

- The delete function is also available by doing Right click on the geometry dataset(s). Notice that by doing Right click is also possible to rename any geometry dataset.

2.3.3 Calibration Matrix

The **Calibration matrix** (see Figure 2–8 (p. 31)) provides navigation and status visualisation for the different Calibration models. Soil profiles and geometry data sets are combined via the Calibration matrix.

The **Calibration matrix** allows for multiple soil profiles to be defined. However, each soil profile must consist of a single type of soil, i.e., either sand or clay, for which one set of depth variation functions will be parameterised.

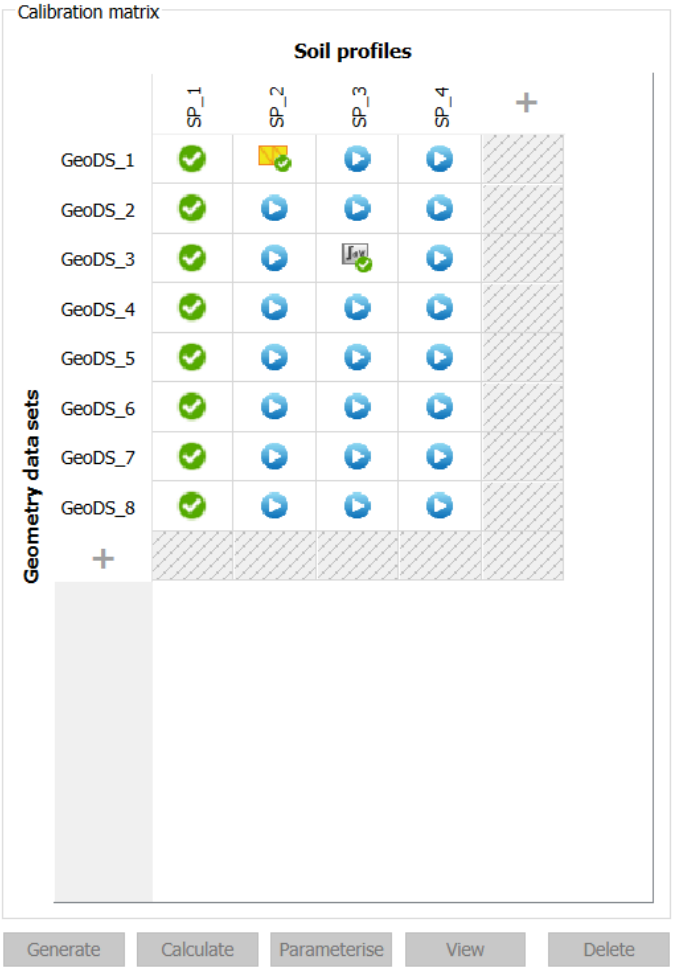


Figure 2–8: Calibration matrix with 4 Soil Profiles and 8 GeoDS per soil profile

Hint:

- Different soil profiles are represented as columns in the matrix.
- Different geometry data sets are represented as rows in the matrix.

It is important to remark that a combination of a soil profile and a geometry data set defines an individual PLAXIS 3D Calibration model.

Add and delete Soil profiles and GeoDS

The defined Soil profiles and GeoDS are listed in the **Calibration matrix** as displayed in Figure 2–9 (p. 32).

To add a new Soil Profile, click on the **+** button in the columns area. A new data soil profile is added to the matrix, to the right of the last soil profile.

To add a new GeoDS for the soil profiles available, click the **+** button in the rows area. A new data set is added to the matrix, below the last data set.

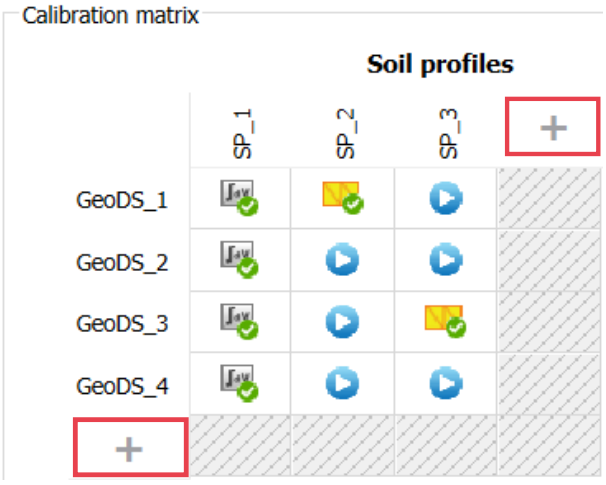


Figure 2–9: Adding Soil profiles data sets (GeoDS)

Select one or more soil profiles or data sets and click the **Delete** button to delete them. Any generated projects will be deleted as well.

> **Hint:** The user can select a single calibration model by clicking anywhere on the matrix. The user can also select single Soil profiles or GeoDS by clicking on its row/column. Multiple elements maybe selected by using Shift+click (consecutive elements) or Control+click (single elements).

> **Note:** Adding or deleting a Soil Profile or GeoDS performs an autosave. Any changes in the project are saved and cannot be undone.

State icons

GeoDSs have different states, depending on the actions that were carried out on the GeoDS and the success or failure of these actions.

The following states exist and are represented by the corresponding icons.

The model is not generated yet.

The model is successfully generated, but not calculated yet.

The model is not successfully generated. An error occurred during generation or meshing.

 The model is successfully generated and calculated.

 The model is not successfully calculated (but it is already successfully generated). An error occurred during calculation.

 The model is partially calculated (but it is already successfully generated). Calculation stopped before target displacement was reached, but partial data could still be recovered.

 The model is successfully included in the parameterisation process.

 The model was changed since it was last generated, calculated or parameterised.

Tip: In case the generation or calculation fails ( or ): The user should open PLAXIS 3D by using the View button and check the error encountered during the generation or calculation of the model.

Generate

When the GeoDS and their Parameters (p. 43) have been added, you can generate the PLAXIS 3D models. To generate a model, PLAXIS Monopile Designer will:

- Generate the soil layers (as it is specified in the 2.3.4 Soil Profile tabsheet (p. 39) of PLAXIS Monopile Designer).
- Generate the soil materials and the model parameters based on the values specified in the 2.3.4 Soil Profile tabsheet (p. 39).
- Generate the structure (monopile and corresponding interfaces) based on the settings of GeoDS, from the **Geometry data set** tabsheet.
- Divide the pile into slices to extract the raw soil reactions at different depths.

  Note: all the needed material parameters are calculated based on predetermined relationships. See 5.2.1 Clay: NGI-ADP material Parameters (p. 118) and 5.2.2 Sand: HSsmall material Parameters (p. 119). Calculated values can be manually modified via the Materials menu in PLAXIS 3D.

- Generate the plate material and assign structural properties specified in the **Geometry data set** tabsheet.
- Generate the calculation phases and adjust the numerical settings to values suitable for accurate and fast calculations (for the specific type of models generated by PLAXIS Monopile Designer).
- Generate a finite element mesh and select precalculation curve points in Output.

Note: Adding or deleting a GeoDS performs an autosave. Any changes in the project are saved and cannot be undone.

> **Note:** The selected pre-calculation curve points may be used by the user to check additional results (in PLAXIS 3D Output) but they are not directly used by the PLAXIS Monopile Designer workflow and calculations.

> **Note:** The calculation phases are always generated before the mesh. In case the mesh generation fails, the model still contains properly defined calculation phases. The user can open the PLAXIS 3D model and try to generate the mesh manually. Changes on the default mesh settings may be needed to mesh successfully. Afterwards, the user should save the PLAXIS 3D project and close it. The calculation must be done via PLAXIS Monopile Designer. Note that any manual changes to the model will be copied to the next one added in the GeoDS menu of the tool (see 2.3.7 Recommended workflow (p. 56)).

A user may modify a 3D model that is automatically generated by PLAXIS Monopile Designer in PLAXIS 3D However, the modified model must represent the same situation (i.e. soil profile and monopile geometry) that has been defined in PLAXIS Monopile Designer. Open the corresponding model in PLAXIS 3D.

This should preferably be done for the first GeoDS that is defined in PLAXIS Monopile Designer since subsequent models are based on the previously generated (and modified) 3D model. In this way, subsequent models can be automatically generated by PLAXIS Monopile Designer, taking into account the modifications of the first model. The user remains responsible for a correct representation of the 3D finite element models when modifying these models in PLAXIS 3D.

Initial generation

During generation, the following calculation phases are created:

- *Initial phase:* generation of initial stress state.
- *Phase 1:* pile wished-in-place.
- *Phase 2:* small displacements calculation.
- *Phase 3:* large displacements calculation.

The large displacements calculation is intended to capture the pile response in the large displacements region, under which the lateral displacement at the mudline is about $D_{out}/10$. Note that this calculation is not a large deformations calculation (updated mesh analysis).

Note that all these calculations are performed in the framework of small deformation theory.

To generate the model(s):

1. In the table, select one or several data sets for which the model is to be generated.
2. Click the *Generate* button.

> **Note:** The model generation works only if there is no feedback (warning) or if the user chooses to ignore it. The checks and feedback messages for the generate action are described in A.1.1 Generate (p. 162).

Regeneration

The regeneration of models is different than the initial generation. See A.1.2 Regenerate (p. 165) for more information concerning the modification on the projects that each action triggers.

The purpose is to maintain as many as possible manual modifications that the user did on the PLAXIS 3D models after the initial generation. The modifications that are allowed can be found in Table A–4 (p. 165).

After one or more models have been calculated, it is possible to add new geometry configurations to PLAXIS Monopile Designer, generate the corresponding models and perform the corresponding calculations.

Calculate

Model calculation includes the following:

- Calculation of all 4 phases (as described in Initial generation (p. 34)).
- Extraction of the raw soil reaction curves.
- Extraction of data to be displayed in the results area (monopile response, raw soil reaction curves).

To calculate the model(s):

1. In the table, select one or several data sets for which the model is to be calculated.
2. Click the Calculate button.

> **Note:** The model calculation works only if there is no feedback (warning) or if the user chooses to ignore it.

During calculation, a window pops up showing the calculation progress, see Figure 2–10 (p. 35). To stop all calculations, click the Stop button (x).

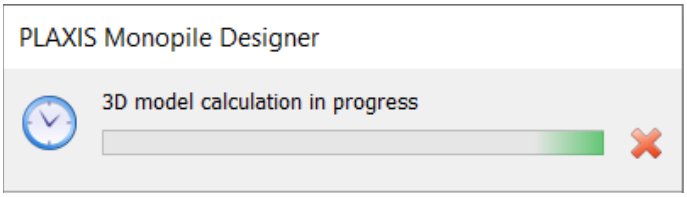


Figure 2–10: Model calculation progress dialog

> **Note:** If multiple calculations are selected to be performed at once and one or more fail, the others continue. Stopping a single calculation can only be done by stopping the calculation of the phases in PLAXIS 3D (calculation progress dialogue). Then the project is saved, and the calculation of the following GeoDS starts. After each calculation finishes, the project is automatically saved.

The checks and feedback messages for the calculate action are described in the A Warnings and errors (p. 162).

Parameterise

The parameterisation process does the following:

- The normalisation of the raw soil reaction curves extracted from the 3D model.
- Fitting of a mathematical function (see 5.4 Optimization Module (p. 127)) to each type of the raw soil reaction curves, at the shaft (monopile slices) and at the base separately. The input raw data come from all the selected geometry data sets.
- Derivation of the functions that describe the variation of the parameters of the mathematical function along the shaft and at the base of the pile, these are called *depth variation functions* (dvf).
- Generation of data to be displayed at the results inspection pane (parameterised soil reaction curves, shaft and base depth variation functions).
- Creation of the calibrated.dvf file containing the depth variation functions for the site-specific soil conditions and design space. This file can then be saved under a different name and/or in a different location and imported for future analysis without requiring a new calibration.

To parameterise the model(s):

1. In the table, select one or more successfully calculated GeoDS for which the model is to be parameterised.
2. Click the Parameterise button.

The checks and feedback messages for the parameterise action are described in A Warnings and errors (p. 162).

The parameterised soil reaction curves are based on the selected models. It is possible to select only a subset of the calculated models for parameterisation.

> **Note:** Based on experience, the parameterisation works best when eight to ten 3D calibration models are used to define the design space. However, a smaller or larger number can also be used.

The following further suggestions are provided here for a successful parameterisation:

- The models should reasonably cover the intended design space.
- The design space (i.e. the variations between the models) should not be extremely large.
- The reached displacement at mudline should be such that for each monopile geometry the failure mechanism is completely developed.

> **Note:** The parameterisation does not trigger automatic saving of the project. The user needs to save the project manually if desired.

View button

This button is used to launch PLAXIS 3D and view or inspect the 3D model.

> **Note:** Only one GeoDS can be selected and visualized at once. The model has to be generated before viewing it.

It is useful in the following situations:

- The user can inspect what went wrong, in case generation of the model fails.
- The user is always advised to inspect the generated models, even if the generation was successful.

- The user can inspect output results via PLAXIS Output, after successful calculation.
- The user can modify the model. The message shown in Figure 2–11 (p. 37) appears (unless switched off by the user via the 'Do not show this message again' option).

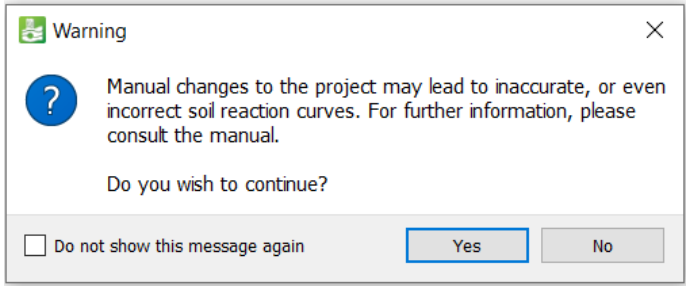


Figure 2–11: Warning about manual changes

Note: After visualising the model, the user should close it directly from the PLAXIS 3D user interface by either clicking on the Close (x) button or by selecting *File -> Exit*.

Model changes outside PLAXIS Monopile Designer

The user can open the model in PLAXIS 3D and modify it. However, PLAXIS Monopile Designer does not reflect any manual changes applied inside PLAXIS 3D.

Warning: Carefully follow the rules explained below to avoid inconsistencies and problems with calculations.

The user may apply only the following safe modifications if needed:

- Change the constitutive model, and even use a user-defined soil model.
- Change the default soil material parameters.
- Change the default mesh settings and regenerate the mesh.
- Change the default numerical settings.

Note: After any modifications, the user should save the PLAXIS 3D model manually and close it afterwards.

The user should not perform any modifications that could interfere with the analysis, including but not limited to:

- Delete the project manually (The project should only be edited via PLAXIS Monopile Designer).
- Modify the name of the project (it will not be recognised anymore by PLAXIS Monopile Designer).
- Modify the soil layers manually in PLAXIS 3D by adding/removing boreholes, adjusting the top and bottom layer boundaries.
- Delete or modify the structure ('tunnels'). The structure (monopile geometry) is created by the PLAXIS 3D *Tunnel Designer*.
- Modify the material parameters of the plate elements (this should be done via PLAXIS Monopile Designer).

- Delete or modify the interface elements around the structure (monopile modelled with plate elements) and at its bottom.
- Add/remove calculation phases.
- Change the calculation type (e.g. to dynamic).
- Modify the boundary conditions.
- Perform calculations directly in PLAXIS 3D.

Calculations must always be performed in PLAXIS Monopile Designer since it immediately extracts the raw soil reaction curves from the finite element results; this is not done if the user performs the calculations directly in PLAXIS.

Warning: If the default soil parameters are modified manually in PLAXIS, the parameters displayed in the **Soil mode** of PLAXIS Monopile Designer should be updated to reflect the new parameters in PLAXIS. This is very important as it ensures that the parameterisation takes into account the correct values of the soil parameters.

2.3.4 Soil Profile tabsheet

2.3.4.1 Soil profile layout

To define the soil stratigraphy, the user needs to choose a material type and determine the soil layers in the **Soil profile** tabsheet.

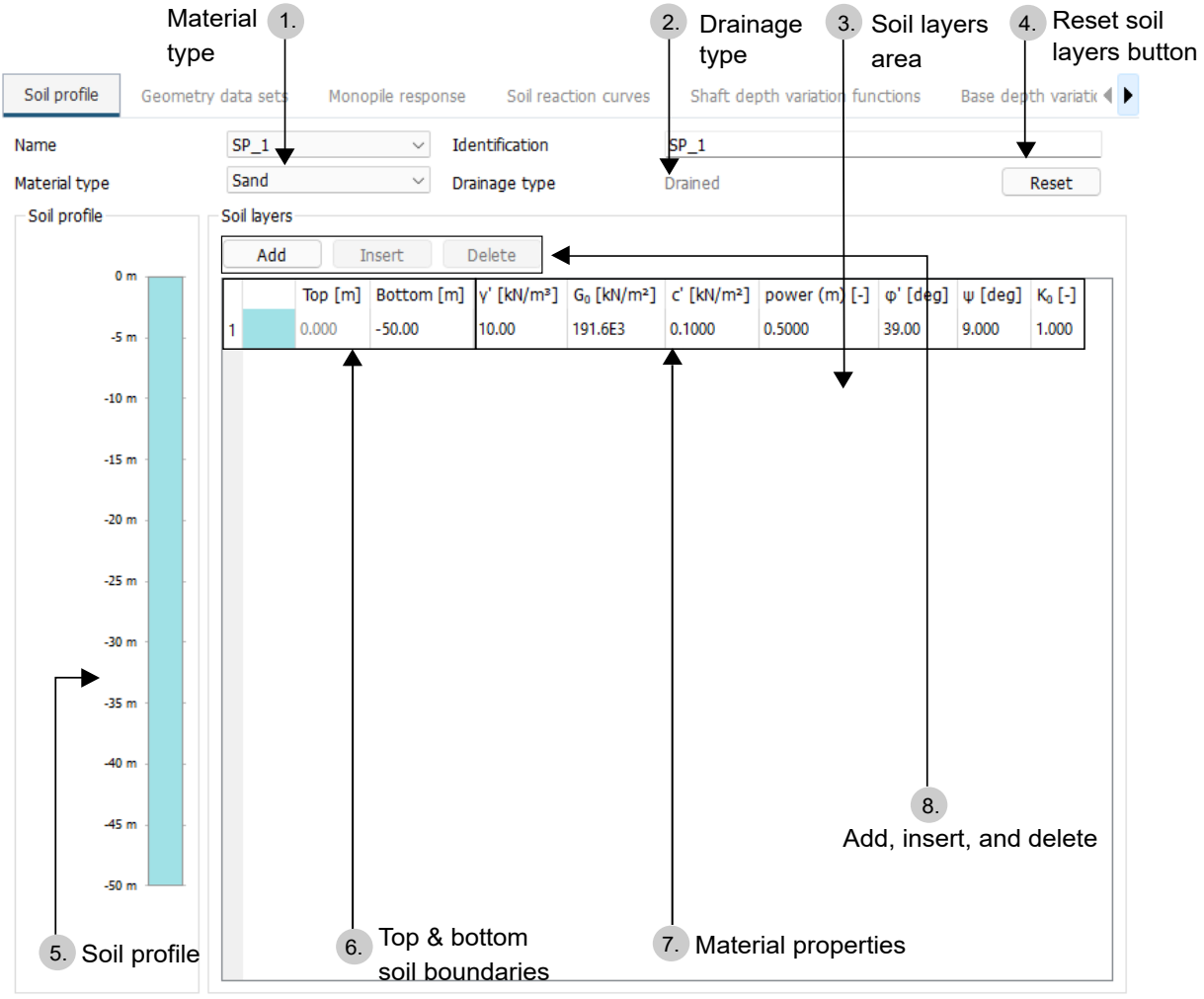


Figure 2–12: View of the project - Soil profile in the Calibration mode

Reset button

When using the Reset button, the program shows a warning. If the user confirms the action, the **Soil profile** is reverted to the default (initial) state:

- Default material type is Clay.
- Soil layers are deleted.

2.3.4.2 | Material types

The user chooses between two available material types: Clay or Sand.

> **Note:** When changing the soil material type after creating a soil layer, the layer boundaries (top and bottom) are retained, but the rest of the parameters are reset.

Clay

The clayey soil material type is formulated using the NGI-ADP model (for more information, see Brinkgreve et al., 2021). The material behaviour (drainage type) is assumed to be undrained. To read more about the parameters of this material model, see 5.2.1 Clay: NGI-ADP material Parameters (p. 118). The following input parameters need to be defined per soil layer:

Parameter	Definition	Unit
γ'	Submerged unit weight	[kN/m^3]
G_0	Small strain shear stiffness modulus in the middle of the soil layer	[kN/m^2]
$s_{u,top}$	Undrained shear strength at the top of the soil layer	[kN/m^2]
$s_{u,bottom}$	Undrained shear strength at the bottom of the soil layer	[kN/m^2]
K_0	Lateral earth pressure coefficient at rest	[-]

Sand

The sandy soil material type is formulated using the HSsmall model (for more information, see Brinkgreve et al., 2021). The material behaviour (drainage type) is assumed to be drained. To read more about the parameters of this material model, see Section 5.2.2 Sand: HSsmall material Parameters (p. 119). The following input parameters need to be defined per soil layer:

Parameter	Definition	Unit
γ'	Submerged unit weight	[kN/m^3]
G_0	Small strain shear stiffness modulus in the middle of the soil layer	[kN/m^2]
c'	Effective cohesion. Default value: 0.1 kPa	[kN/m^2]
power (m)	Power for stress-level dependency of stiffness. Default value: 0.5	[-]
φ'	Effective angle of internal friction	[°]
ψ	Angle of dilatancy	[°]
K_0	Lateral earth pressure coefficient at rest	[-]

Tip: In both cases, the user may change the constitutive model or use a user-defined soil constitutive model via PLAXIS 3D.

2.3.4.3 | Creating soil layers

The user creates soil layers using buttons above the soil layers area:

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

General rules for adding, inserting, and deleting soil layers

1. The thickness of a newly added layer is zero by default.
2. The top boundary of an underlying layer is defined by the lower boundary of the overlying layer.
3. To change the thickness of a layer, the user modifies the bottom boundary.
4. A newly added soil layer appears as the lowest soil layer.
5. A newly inserted soil layer is inserted right above a selected layer.
6. A layer's bottom boundary cannot be less than the underlying layer's bottom boundary.
7. When deleting a layer, a confirmation window pops up.

Soil profile

The user can inspect the soil profile not only by looking at layer boundaries in the table but also in the graphical representation of the Soil profile (Figure 2–13 (p. 41)), which is visible in the left panel of the **Soil profile** tabsheet. This visual representation of the inserted soil layers shows the top and bottom boundaries.

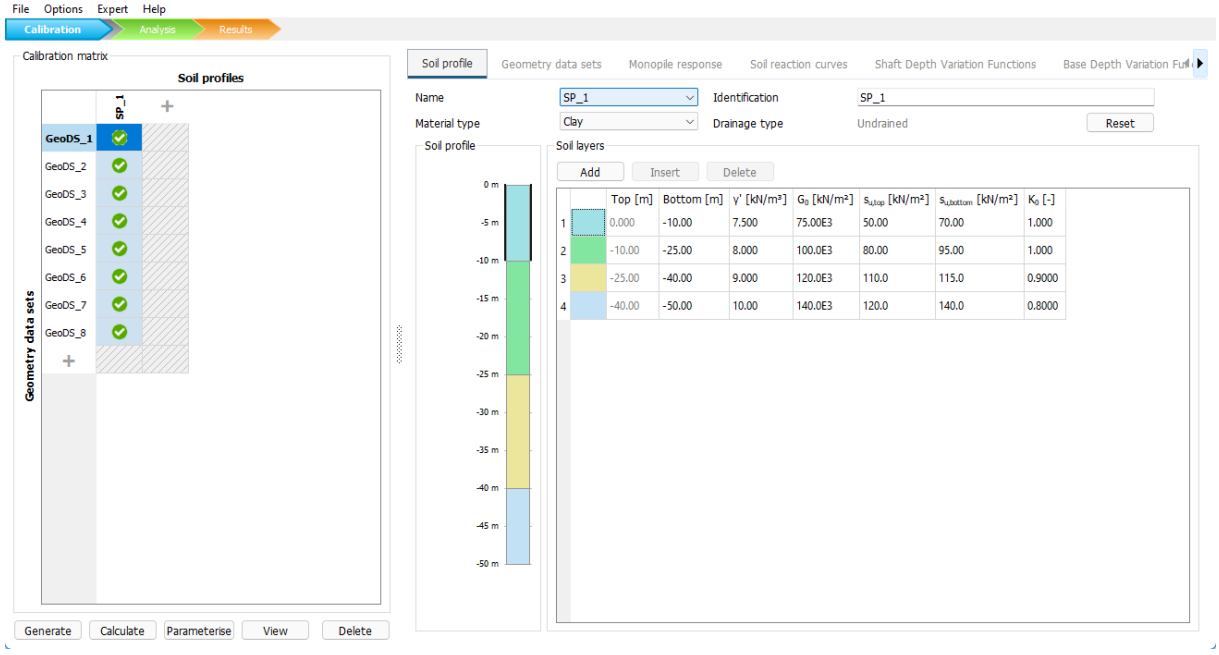


Figure 2–13: Soil profile in the Soil mode

Tip: For easier reference and navigation, the selected soil layer is also highlighted in the Soil profile panel each time the users click on the tabulated data.

2.3.5 Geometry datasets tabsheet

A geometry data set (GeoDS) defines the geometrical characteristics of a monopile. The combination of a soil profile and a GeoDS corresponds to a particular PLAXIS 3D calibration model. The soil stratigraphy in the model comes from the 2.3.4 Soil Profile tabsheet (p. 39) of PLAXIS Monopile Designer.

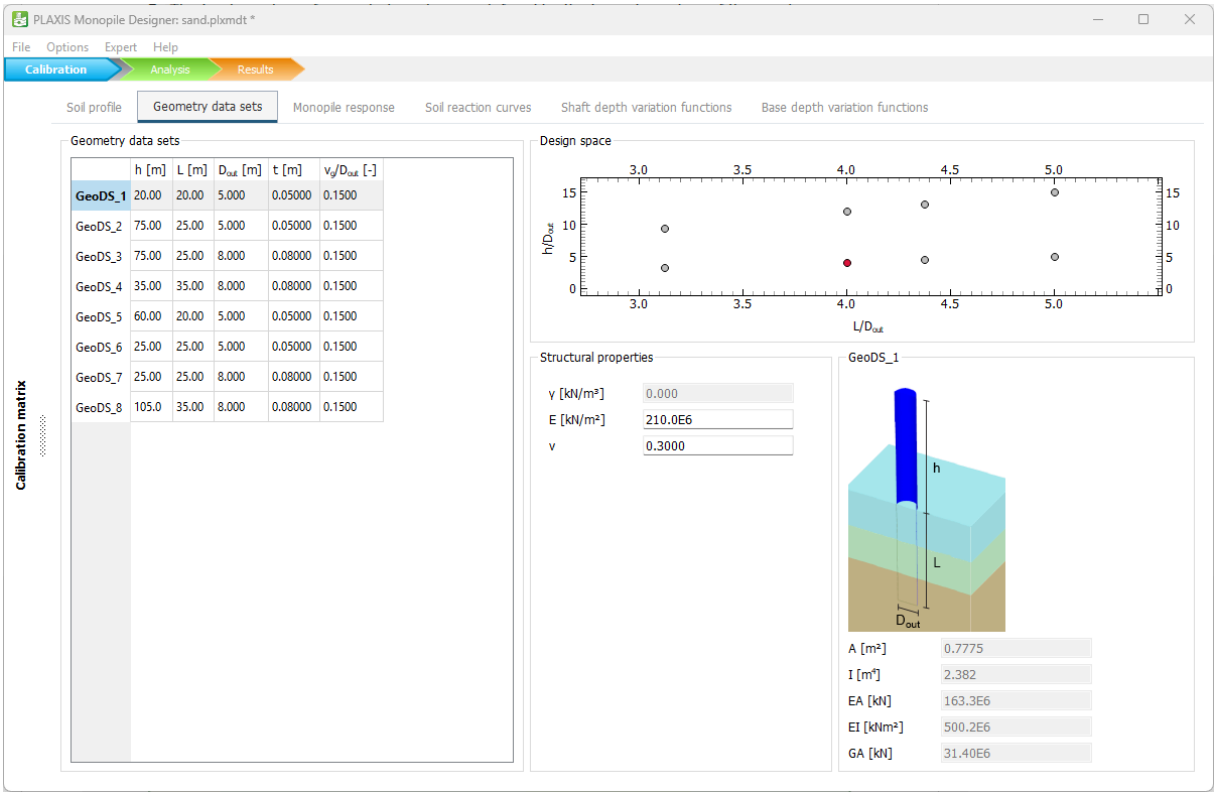
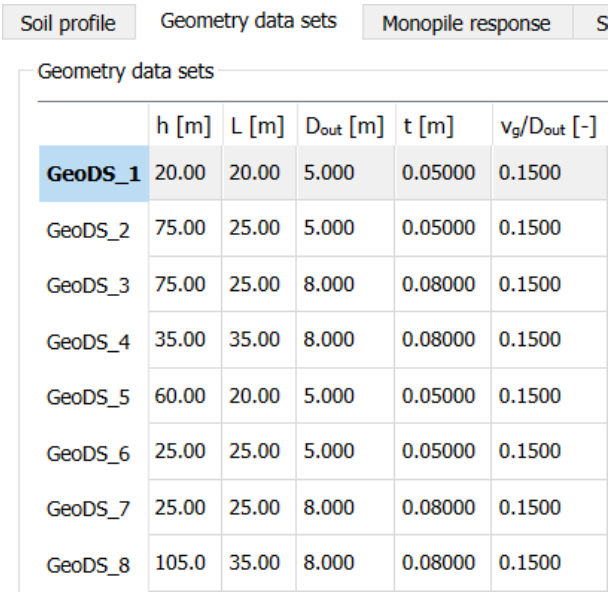


Figure 2–14: Geometry data sets tabsheet

2.3.5.1 Geometry definition of a GeoDS

To define the input parameters that describe the geometry of a data set the user needs to:

1. Select the **Geometry dataset tabsheet**.
2. Select a GeoDS.
3. Fill the geometry data in the Figure 2–15 (p. 43).



	h [m]	L [m]	D_{out} [m]	t [m]	v_g/D_{out} [-]
GeoDS_1	20.00	20.00	5.000	0.05000	0.1500
GeoDS_2	75.00	25.00	5.000	0.05000	0.1500
GeoDS_3	75.00	25.00	8.000	0.08000	0.1500
GeoDS_4	35.00	35.00	8.000	0.08000	0.1500
GeoDS_5	60.00	20.00	5.000	0.05000	0.1500
GeoDS_6	25.00	25.00	5.000	0.05000	0.1500
GeoDS_7	25.00	25.00	8.000	0.08000	0.1500
GeoDS_8	105.0	35.00	8.000	0.08000	0.1500

Figure 2–15: Geometry data set parameters table

Note: To define the geometry of a monopile, a Soil profile and a GeoDS need to be generated beforehand using the 2.3.3 Calibration Matrix (p. 31).

Parameters

Table 2–1: Geometry data sets parameters

Parameter	Description	Unit
h	Monopile height above mudline at which the prescribed lateral displacement is applied	[m]
L	Monopile embedded length	[m]
D_{out}	Monopile outer diameter	[m]
t	Monopile wall thickness	[m]
v_g/D_{out}	Target relative displacement at mudline	[-]

All other parameters can be changed (within the max/min boundaries). To change a value click in the cell and edit the value.

Tip: The length L is limited by the max soil depth minus $0.15 \cdot D_{out}$. There is an error message displayed on the screen if this condition is not met. See A Warnings and errors (p. 162) for more information. If there are no soil layers defined, the user is not able to fill in L.

2.3.5.2 | Design Space

Located at the right side of the **Calibration mode** > **Geometry data sets**, the design space (or calibration space/variation between models) defines the space covered by the variation of the geometrical parameters assigned to the calibration set of the 3D FE models. The parameters that span the design space are the embedded length, the diameter, the wall thickness of the pile, as well as the height above the ground level where the excitation is applied.

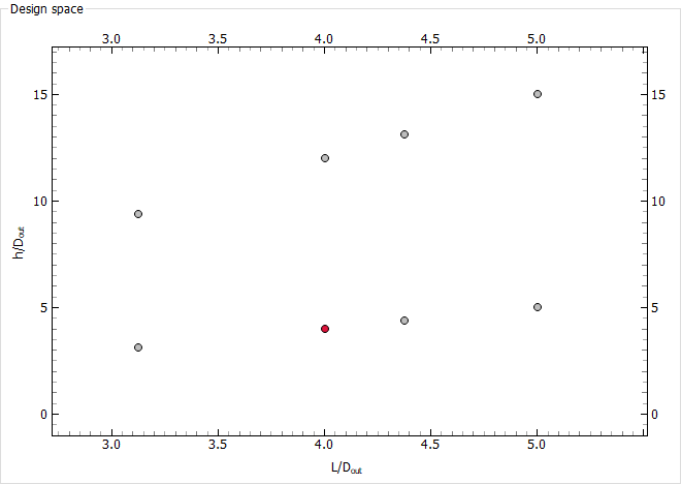


Figure 2–16: Design space window

2.3.5.3 | Structural and material properties

Located at the right side of the **Calibration mode** > **Geometry data sets**. It shows the structural and material properties of a monopile with its schematic representation (see Figure 2–17 (p. 44)). The information corresponds to the GeoDS that is selected in the GeoDS table. Note that the schematic is for general illustration purposes. The dimensions are not updated when selecting a geometry data set.

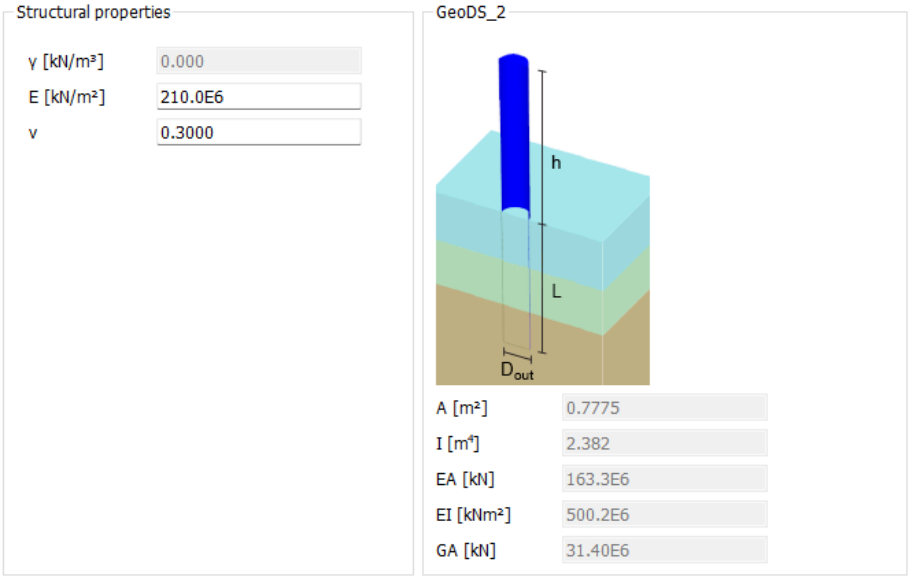


Figure 2–17: Schematic view of a monopile and properties

Structural properties

In the structural properties area, the Young's modulus E and Poisson's ratio *v* of the steel can be modified. The pile unit weight is set to zero by default and cannot be changed. For more information on the pile unit weight, see Generating 3D models > Monopile.

The structural properties are described in Table 2–2 (p. 45)

Table 2–2: Structural properties

Property	Description	Unit	Default
γ	pile unit weight	[kN/m^3]	0.000 (fixed)
E	Young's modulus	[kN/m^2]	$210 \cdot 10^6$
v	Poisson's ratio	[-]	0.300

Equivalent pile properties

Based on the values from the GeoDS and structural properties, the equivalent pile properties are calculated. The pile properties are described in Table 2–3 (p. 45)

Table 2–3: Equivalent pile properties

Property	Description	Formula	Unit
A	cross section area	$A = \frac{\pi({D_{out}}^2 - {D_{in}}^2)}{4}$	[m^2]
I	moment of inertia	$I = \frac{\pi({D_{out}}^4 - {D_{in}}^4)}{64}$	[m^4]
EA	axial stiffness	$EA = E \cdot A$	[kN]
EI	bending stiffness	$EI = E \cdot I$	[kNm^2]
GA	shear stiffness	$GA = \frac{0.5\kappa EA}{(1+\nu)}$ $where\ \kappa = 0.5$	[kN]

Also, see 2.3 Calibration mode (p. 27).

2.3.6 Results inspection pane

The results inspection pane is on the right side in the **Calibration mode**, and it may be used to quickly get an insight into the calculated projects and to identify potential errors in the calculation or parameterisation.

The user can inspect in detail:

- the load-displacement curves at the ground level for both small and large displacements.
- the monopile lateral deflection at the ground level for both small and large displacements.
- the soil reaction curves (shaft and base) extracted from the 3D FE calculations (only for large displacements).
- the depth variation functions of the 16 fitting parameters for the shaft and the base.

The result tab include **Monopile response**, **Soil reaction curves**, **Shaft depth variation functions** and **Base depth variation functions**.

The graphs have the following functionality:

- To show values at particular points of the curve, hover over the curve, see Figure 2–18 (p. 46)

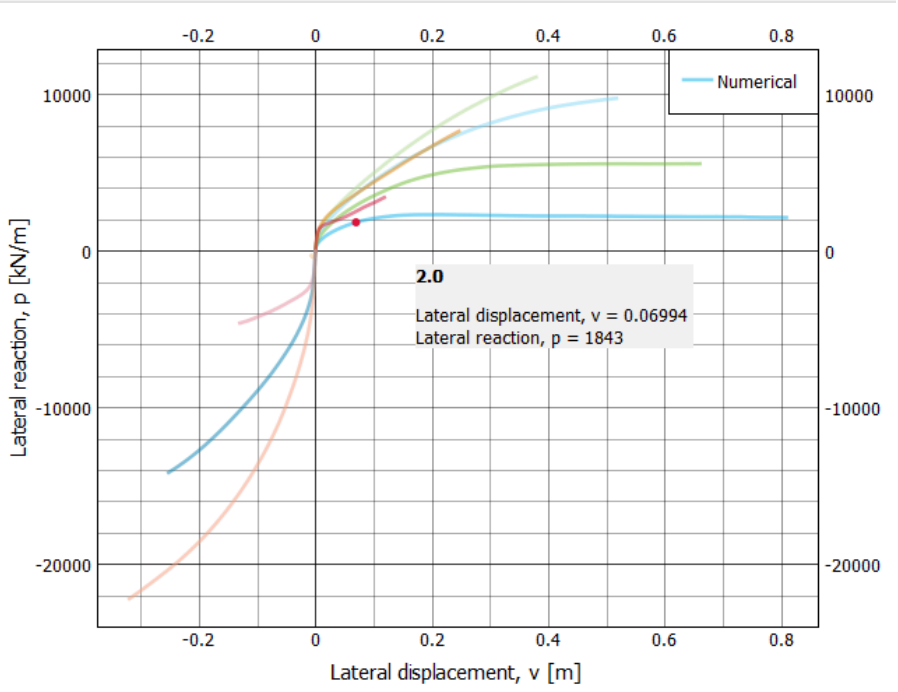


Figure 2–18: Hover over curve to show values

- To zoom in/out, use the scroll-wheel or click-and-drag from top-left to bottom-right or click-and-drag from bottom-right to top-left.
- To pan click and drag.
- For more options, right-click the graph to open the context menu (Figure 2–19 (p. 47)). This allows users to adjust the appearance of the graph and to export it as an image or vector graphic.

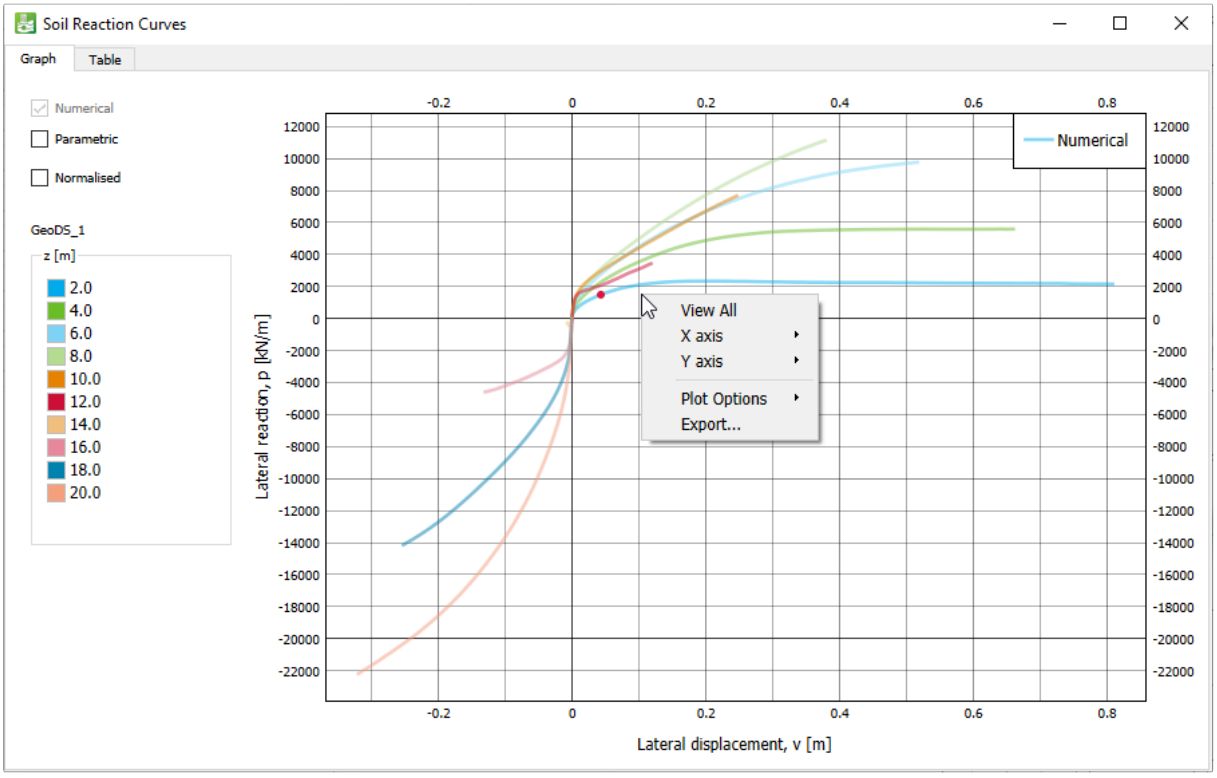


Figure 2–19: Context menu for curves

2.3.6.1 | Monopile response

The **Monopile response tab** displays the results for all 3D models that were successfully calculated and selected on the GeoDS menu. See Figure 2–20 (p. 48).

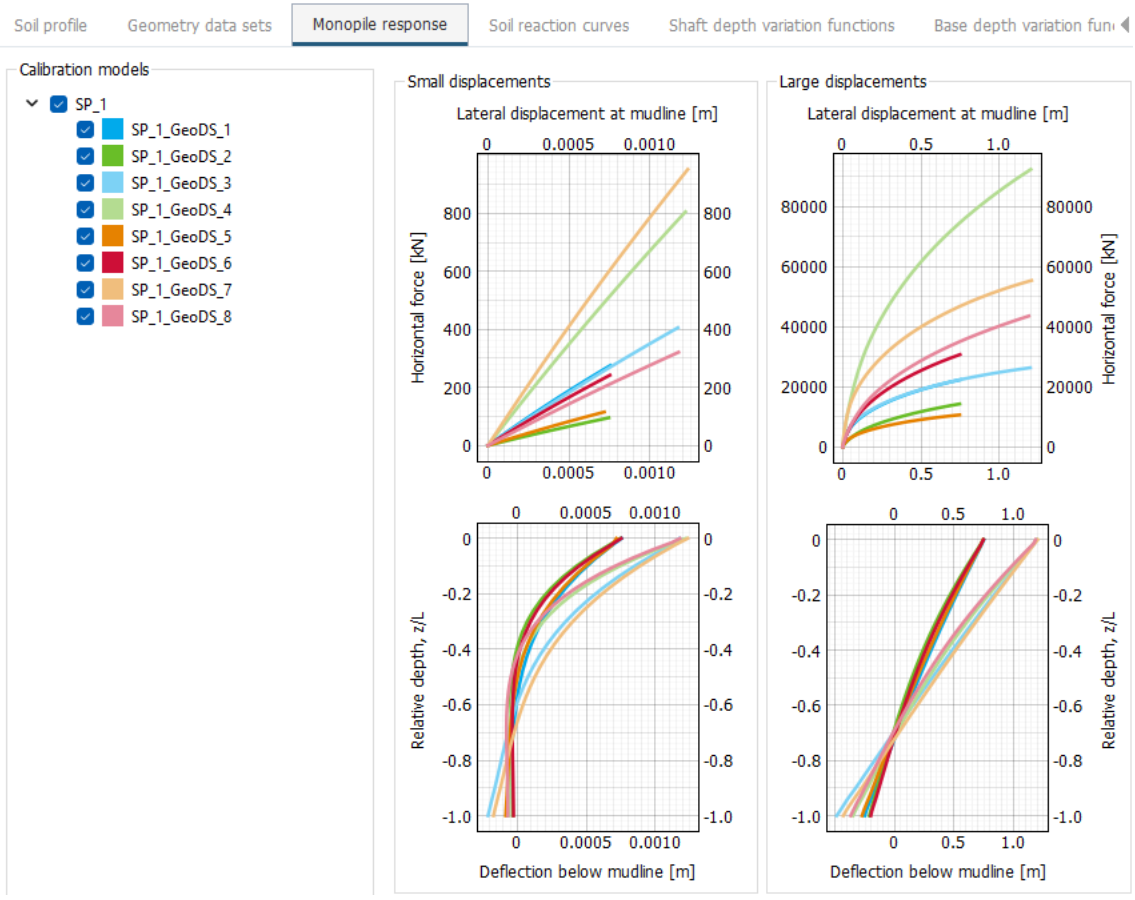


Figure 2–20: Monopile response tab

The results shown are:

- the monopile lateral reaction force in relation to the monopile lateral displacement at the mudline (top)
- the depth z over the embedded monopile length L in relation to the monopile deflection below the mudline (bottom).

The graphs are presented for small displacements (left) and large displacements (right).

Small displacements: the small displacement response at mudline is taken from the results of Phase 2 of the PLAXIS 3D calculation. In this case, the maximum displacement is intended to be around D/10000.

Large displacements: the large displacement response at mudline is taken from the results of Phase 3 of the PLAXIS 3D calculation. The maximum displacement in this case is intended to be around D/10.

Double-clicking one of these graphs opens a separate window, which displays only a larger version of the graph and a table with all the values from which the chart was generated. For information on this view, see 2.3.6 Results inspection pane (p. 46).

> **Note:** Charts are automatically updated after successful calculation of a selected model.

2.3.6.2 Soil reaction curves tab

The **Soil reaction curves tab** displays the results for 3D models that were successfully calculated and selected in the GeoDS table. The results are updated if a model is recalculated successfully. For more information on soil reaction curves, see 5.3.3 Soil reaction curves (p. 125).

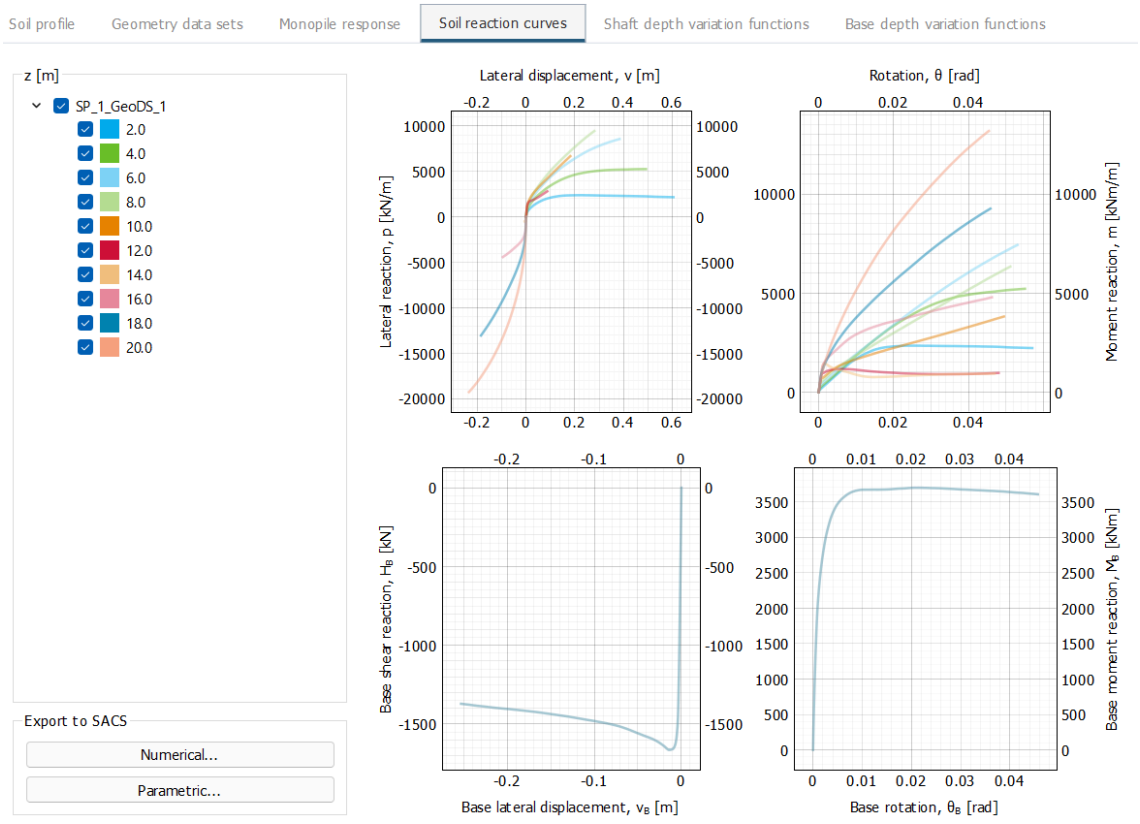


Figure 2–21: Soil reaction curves tab

The results of only a single model are shown at a time. If more than one model is selected in the table, this will be the data from the model that was selected first.

> **Note:** Charts are automatically updated if the selection in the table changes or after successful calculation of a selected model.

The results at different depths (z) are shown:

1. The z_{target} is calculated based on the following pre-determined depths: {z_{target} = 0.1*L*, 0.2*L*, 0.3*L*, 0.4*L*, 0.5*L*, 0.6*L*, 0.7*L*, 0.8*L*, 0.9*L*, 1.0*L*}.

> **Tip:** Although the graph only displays the curves at 10 predefined depths, soil reaction curves are in general computed at finer intervals. The full set of results are available in the Table tab.

2. The soil layer that the aimed depth (z_{target}) corresponds to is found via the *groundfile.dat file*. If this is the exact boundary of two soil layers, the bottom one is selected.
3. The z_{target} is rounded to half a meter (up or down) assuring that it remains within the targeted soil layer, as determined above.
4. If the rounded z_{target} coincides with the boundary between two soil layers, then the soil properties are updated considering the bottom one.
5. If the 1D data are available, the 1D data which correspond to the rounded z_{target} are displayed.
6. If the 3D data are available, the correct monopile slice is selected, within the updated selected soil layer (from step 4), based on the rounded z_{target} and the top and bottom boundary of the slice (Figure 2–22 (p. 50)). If the rounded z_{target} corresponds to the exact boundary of two slices, the bottom slice is selected.

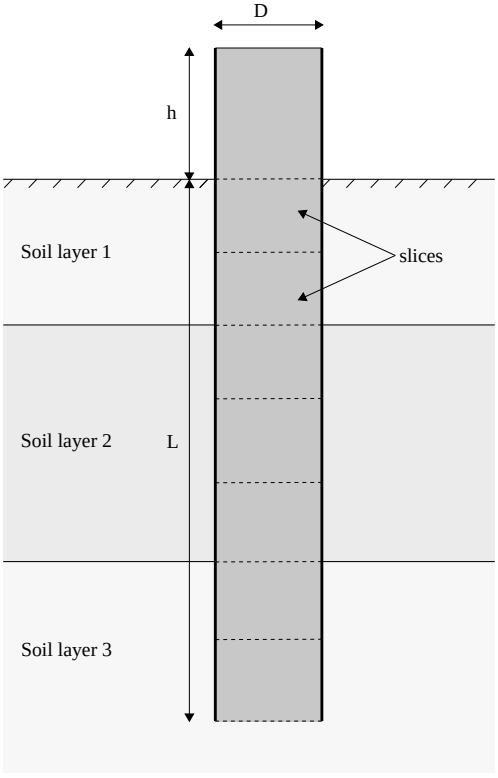


Figure 2–22: Slices of a monopile

> **Tip:** As a result of the process presented above, certain target depths (z_{target}) may be selected more than once, for example in case of short piles. In this case, the resulting plot lines could be less than 10 as coinciding lines are not plotted.

The displayed soil reaction curves are the raw (not normalised, not parameterised) soil reaction curves.

The four types of soil reaction curves (p - ν, H_B - ν_B, m - θ_B and M_B - θ_B) are displayed in four graphs.

- The top graphs show the p - ν and m - θ_B soil reaction curves along the monopile at the predefined depths mentioned above.
- The bottom graphs show the H_B - ν_B and M_B - θ_B soil reaction curves at the base of the monopile (at depth L).

The displayed raw soil reaction curves (normalised or unnormalised) derived from the 3D calculations follow the sign convention presented in 2.2.6 Units and sign convention (p. 24).

The parameterised (normalised or unnormalised) soil reaction curves derived from the parameterisation process have positive signs, irrespective of the adopted sign convention. This is related to the needed preprocessing of the raw data before the parameterisation is executed.

Double-clicking on a graph pops up a new window, which displays this particular graph enlarged and a table with the data. See 2.3.6 Results inspection pane (p. 46) for more information.

The enlarged graph allows choosing to show the parameterised and/or normalised values. There are two checkboxes for this purpose. Once the Parameterised box is checked, absolute values are shown. This is done to compare the raw soil reaction curves with the parameterised soil reaction curves, either in normalised (normalised checkbox checked) or unnormalised (normalised checkbox unchecked) format.

> **Note:** The comparison might give the impression that the soil reaction curves do not match. This is expected as the goal of the parameterisation procedure is to optimise the parameters of the fitting function and generate the depth variation function for each one of those parameters. This procedure might lead to local inaccuracies in the interest of the overall performance.

2.3.6.3 | Depth variation functions

Once the parameterisation is completed, the depth variation functions per fitting parameter are automatically generated or updated. The Shaft depth variation functions and Base depth variation functions show the variation of the fitting parameters along the shaft and at the base of the monopile accordingly.

In addition, when a model that was used for the parameterisation is deleted, the graphs are cleaned up as they are not valid anymore.

> **Note:** The curve data is based on all models included in the preceding parameterisation.

- **Shaft depth variation functions:** Eight graphs (Figure 2–23 (p. 52)) illustrate the variation of the fitting parameters along the shaft of the monopiles. The fitting parameters shown in the graphs are defined as in Table 2–4 (p. 52). The vertical axis (z) is normalised. The normalisation context changes per graph. For further detail on the normalisation, see Table 5–2 (p. 129). In case that the parameter is constant over depth, then the outer diameter is used for the normalisation.

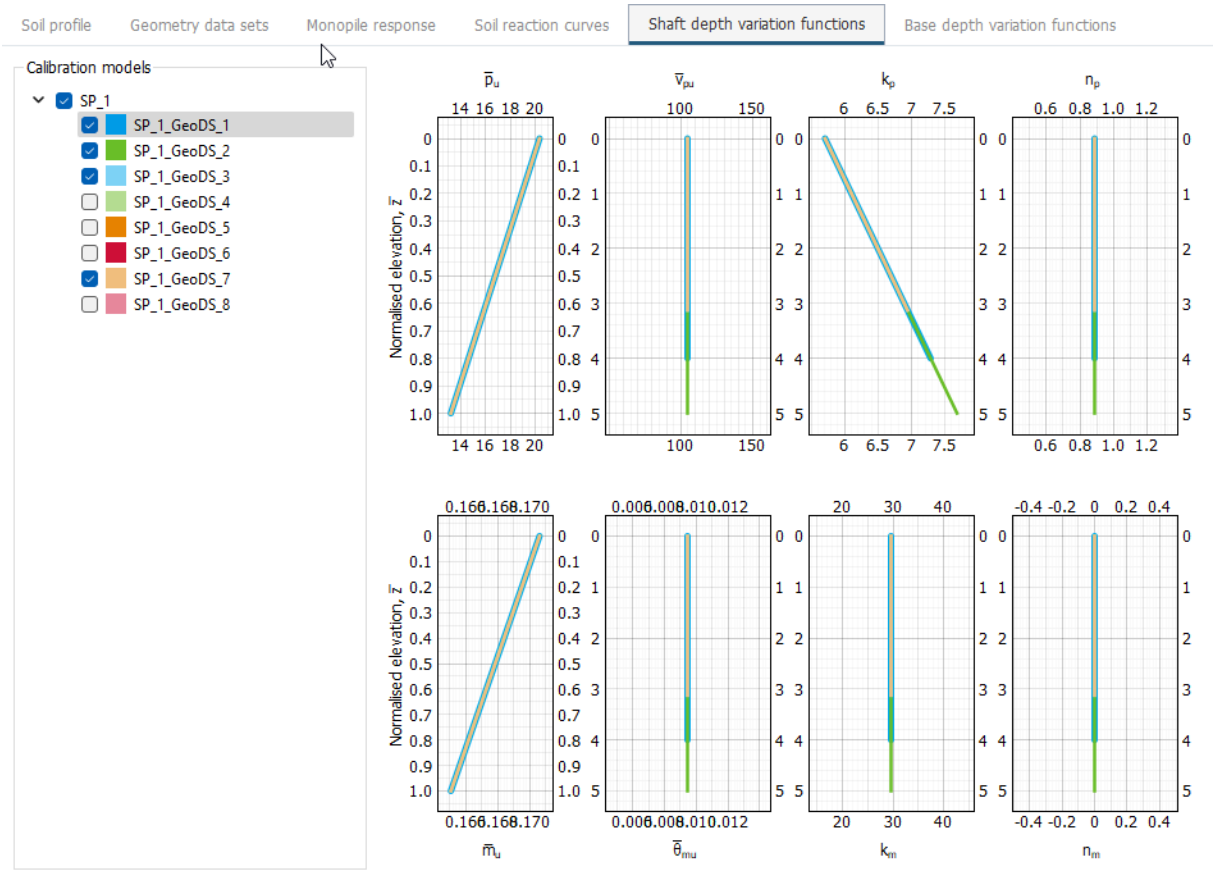


Figure 2–23: Shaft depth variation functions

Table 2–4: Shaft depth variables

Variable on x-axis	Definition
$\bar{v}_{pu}$	normalised ultimate lateral displacement
$\bar{p}_u$	normalised ultimate lateral soil reaction

Variable on x-axis	Definition
k_p	normalised initial stiffness of the lateral soil reaction
n_p	normalised curvature of the lateral soil reaction
$\bar{\theta}_{mu}$	normalised ultimate rotation
$\overline{m}_u$	normalised ultimate moment reaction
k_m	normalised initial stiffness of the moment reaction
n_m	normalised curvature of the moment reaction

- **Base depth variation functions:** Eight graphs (Figure 2–24 (p. 53)) illustrate the variation of the fitting parameters at the base of the monopiles. The fitting parameters shown in the graphs are defined in Table 2–5 (p. 53), and originate from equations used in Table 5–2 (p. 129). Note that the vertical axis (L/D_{out}) is normalised.

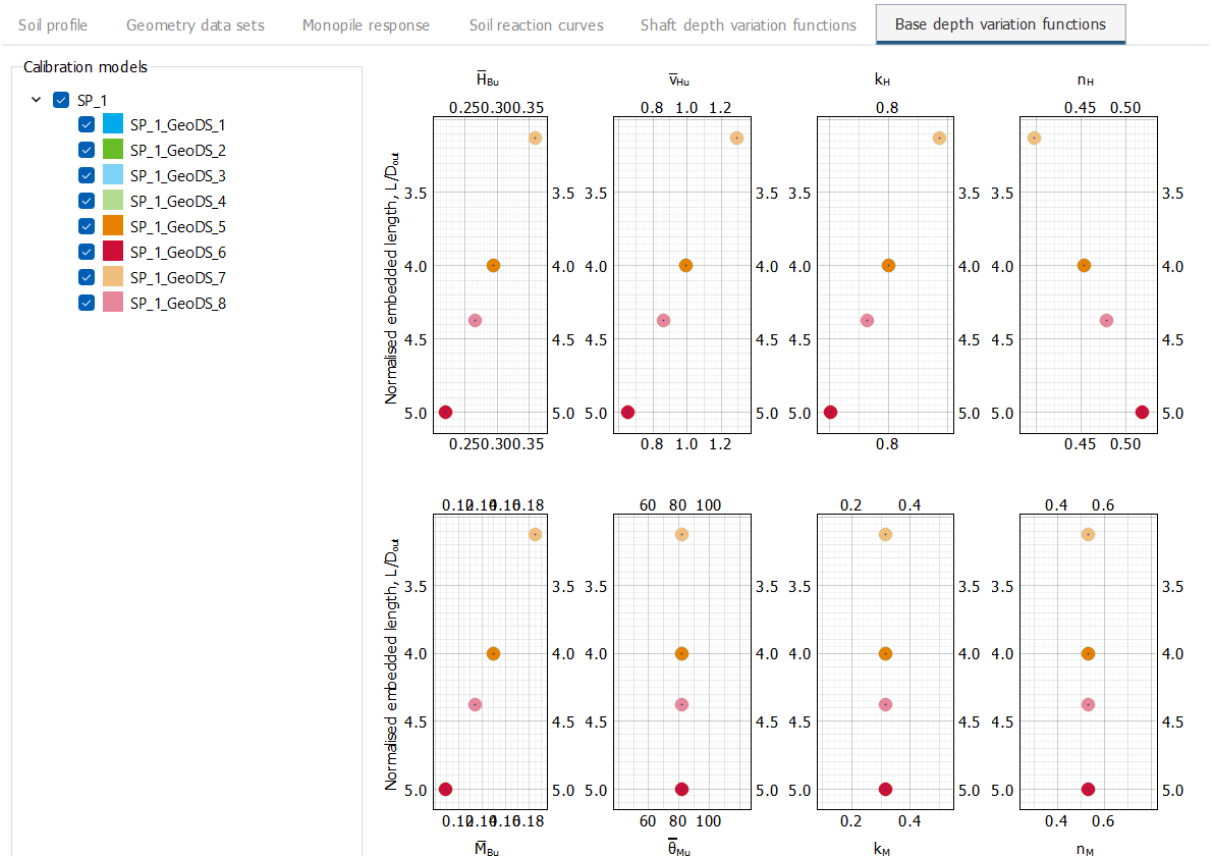


Figure 2–24: Base depth variation functions

Table 2–5: Base fitting parameters

Fitting parameter	Definition
$\bar{v}_{Hu}$	normalised ultimate lateral base displacement
$\overline{H}_{Bu}$	normalised ultimate lateral base soil reaction
k_H	normalised initial stiffness of the lateral base soil reaction
n_H	normalised curvature of the lateral base soil reaction

Fitting parameter	Definition
$\bar{\theta}_{Mu}$	normalised ultimate base rotation
$\overline{M}_{Bu}$	normalised ultimate moment base reaction
k_M	normalised initial stiffness of the base moment reaction
n_M	normalised curvature of the base moment reaction

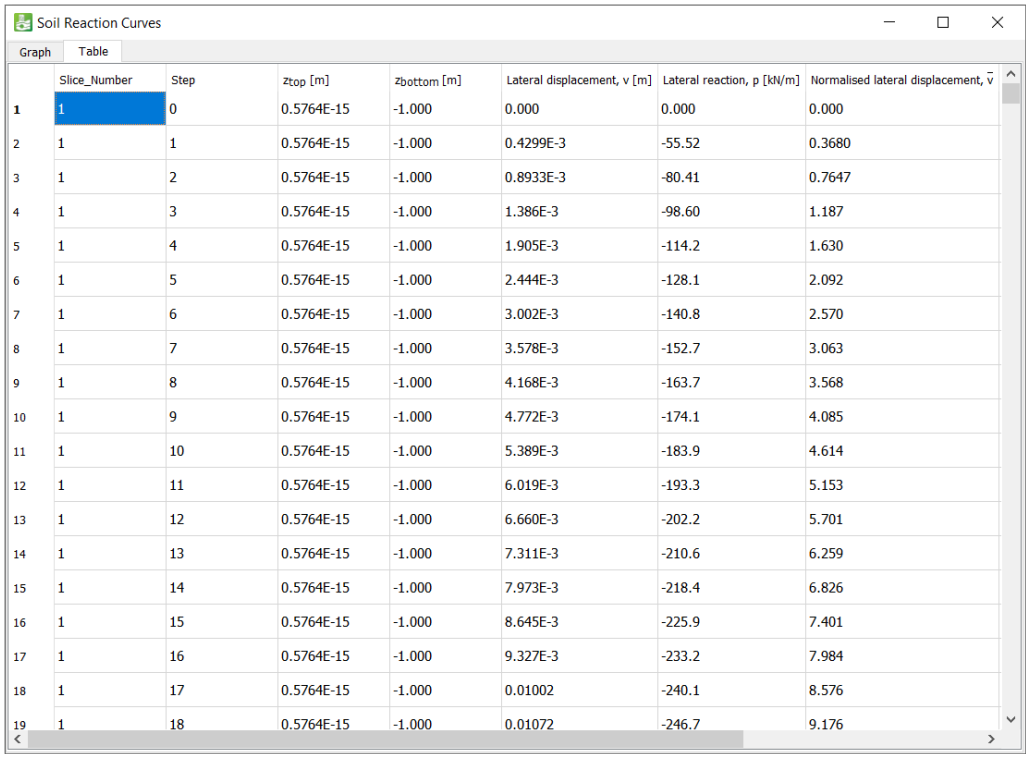
2.3.6.4 | Graph and table windows

When double-clicking a graph a separate window opens, which displays only this particular larger version of the graph and a table with all the values from which the chart was generated.

For the soil reaction curves, the user can choose to show the normalised and parameterised curves. There are checkboxes to control this behaviour. By default both boxes are unchecked. The displayed soil reaction curves are the raw ones.

The *Parameterised* checkbox is available only if the parameterisation has been done and the selected model was included in the parameterisation.

Switch to the **Table** tab to get a tabular overview of the values for the curve.



	Slice_Number	Step	ztop [m]	zbottom [m]	Lateral displacement, v [m]	Lateral reaction, p [kN/m]	Normalised lateral displacement, v̄
1	1	0	0.5764E-15	-1.000	0.000	0.000	0.000
2	1	1	0.5764E-15	-1.000	0.4299E-3	-55.52	0.3680
3	1	2	0.5764E-15	-1.000	0.8933E-3	-80.41	0.7647
4	1	3	0.5764E-15	-1.000	1.386E-3	-98.60	1.187
5	1	4	0.5764E-15	-1.000	1.905E-3	-114.2	1.630
6	1	5	0.5764E-15	-1.000	2.444E-3	-128.1	2.092
7	1	6	0.5764E-15	-1.000	3.002E-3	-140.8	2.570
8	1	7	0.5764E-15	-1.000	3.578E-3	-152.7	3.063
9	1	8	0.5764E-15	-1.000	4.168E-3	-163.7	3.568
10	1	9	0.5764E-15	-1.000	4.772E-3	-174.1	4.085
11	1	10	0.5764E-15	-1.000	5.389E-3	-183.9	4.614
12	1	11	0.5764E-15	-1.000	6.019E-3	-193.3	5.153
13	1	12	0.5764E-15	-1.000	6.660E-3	-202.2	5.701
14	1	13	0.5764E-15	-1.000	7.311E-3	-210.6	6.259
15	1	14	0.5764E-15	-1.000	7.973E-3	-218.4	6.826
16	1	15	0.5764E-15	-1.000	8.645E-3	-225.9	7.401
17	1	16	0.5764E-15	-1.000	9.327E-3	-233.2	7.984
18	1	17	0.5764E-15	-1.000	0.01002	-240.1	8.576
19	1	18	0.5764E-15	-1.000	0.01072	-246.7	9.176

Figure 2–25: Table of curve values

To copy or save data:

- Select values by cell or complete row, if specific values are needed. To copy/save: **Right-click > Copy All/Save All** or **Right-click Copy/Save Selection**, see Figure 2–26 (p. 55).

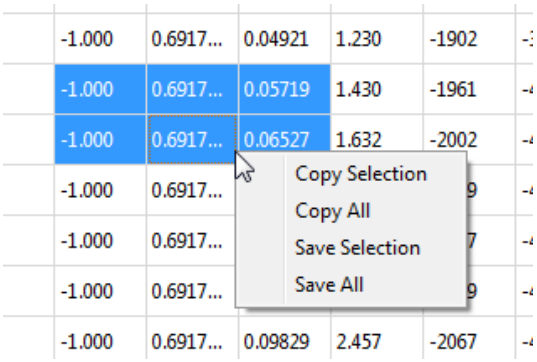


Figure 2–26: Context menu for the table of values

- To copy to the clipboard, use **CTRL+C** shortcut.

2.3.6.5 Export to SACs/OpenWindPower

It is possible to export the geometry and soil reaction curves of any monopile analysed in PLAXIS Monopile Designer to a SACS Pile Input file, which can be used as input for various analysis types in SACS and OpenWindPower Fixed Foundation.

When a GeoDS is selected, the Export to SACS menu appears in the Soil reaction curves tab. Two options are displayed, which will be active only if the corresponding results have been obtained:

- Numerical: exports 3D results, obtained directly from the calculated GeoDS in PLAXIS 3D.
- Parametric: exports 1D results, obtained from the depth variation functions for the Parameterised calibration set.

Exporting with any of the options will open a '*Save as*...' dialogue, where the destination folder and file name can be specified. Selecting '*Save*' automatically generates the SACS Pile Input file in the destination folder, which for the selected GeoDS and curve type (Numerical or Parametric) includes:

1. Soil profile and monopile geometry.
2. All 4 sets of horizontal and rotational soil reaction curves (p-v, m-θ, H_B-v_B , and M_B-θ_B), filtered to a small number of points (fewer than 30) while preserving their global shape.
3. Vertical soil reaction curves (t-w, V_B -w_B) constructed according to the API standard (API, 2014) and the parameters of the soil profile.

For more information on the format and usage of the SACS Pile Input file, refer to the SACS Pile-Structure Interaction Manual (Bentley Systems, 2021).

> **Note:** Use of distributed rotational soil reaction curves (m-θ) requires SACS CONNECT Edition V14.2 or higher; use of pile base curves (H_B-v_B , M_B-θ_B) requires SACS CONNECT Edition V14.3 or higher. It is still possible to use the exported Pile Input file in previous versions of SACS by deleting the lines associated with the unsupported soil curves.

2.3.7 Recommended workflow

The following workflow is recommended for the **Calibration mode** of this tool.

1. Add a Soil Profile in the 2.3.3 Calibration Matrix (p. 31)
2. Add a GeoDS in the 2.3.3 Calibration Matrix (p. 31).
3. Add the layers and properties to the corresponding soil profile.
4. Generate the GeoDS.
5. Visualise the GeoDS in PLAXIS 3D to ensure that the geometry, the material parameters and the mesh are as expected. If needed, any modifications mentioned in 2.3.5 Geometry datasets tabsheet (p. 42) may be made.
6. Calculate it using the Calculate button in PLAXIS Monopile Designer and inspect the results in PLAXIS Monopile Designer.
7. Visualise the calculated project in PLAXIS 3D to ensure that the calculation results are as expected.
8. Only after that add more data sets.
9. When adding a dataset, not only the values of the previous dataset are copied, but also the associated PLAXIS 3D project.
10. During the generation of an added dataset with different geometry configuration, PLAXIS Monopile Designer modifies this copy to fit the changed geometry and/or prescribed displacement applied to the top of the monopile, thereby keeping any modified material parameters, altered constitutive models, changed mesh settings and/or numerical phase settings.

> **Tip:** If the user needs to make modifications to the PLAXIS 3D models, the modification guidelines in 2.3.5 Geometry datasets tabsheet (p. 42) have to be followed.

> **Tip:** If generation of an added dataset fails, select the failed GeoDS and generate it again. In this second attempt, PLAXIS Monopile Designer will start the dataset from an empty model and any user modifications will need to be reapplied.

2.4 Analysis mode

The **Analysis mode** (Figure 2–27 (p. 57)) is used to run fast and robust 1D FE calculations to obtain the monopile response under lateral monotonic loading. The monopile is modelled by means of the Timoshenko beam theory whereas the soil reaction is modelled using calibrated or user-defined soil reaction curves (2.1 Introduction (p. 14)). The analysis of monopiles under static loading can be done in the **1D analysis** area using the **Static** option (see Figure 2–27 (p. 57)). PLAXIS Monopile Designer can also evaluate the natural frequencies and modes of monopile-tower-turbine systems when the 2.4.7 Modal Analysis (p. 65) is activated (as displayed in Figure 2–38 (p. 66)).

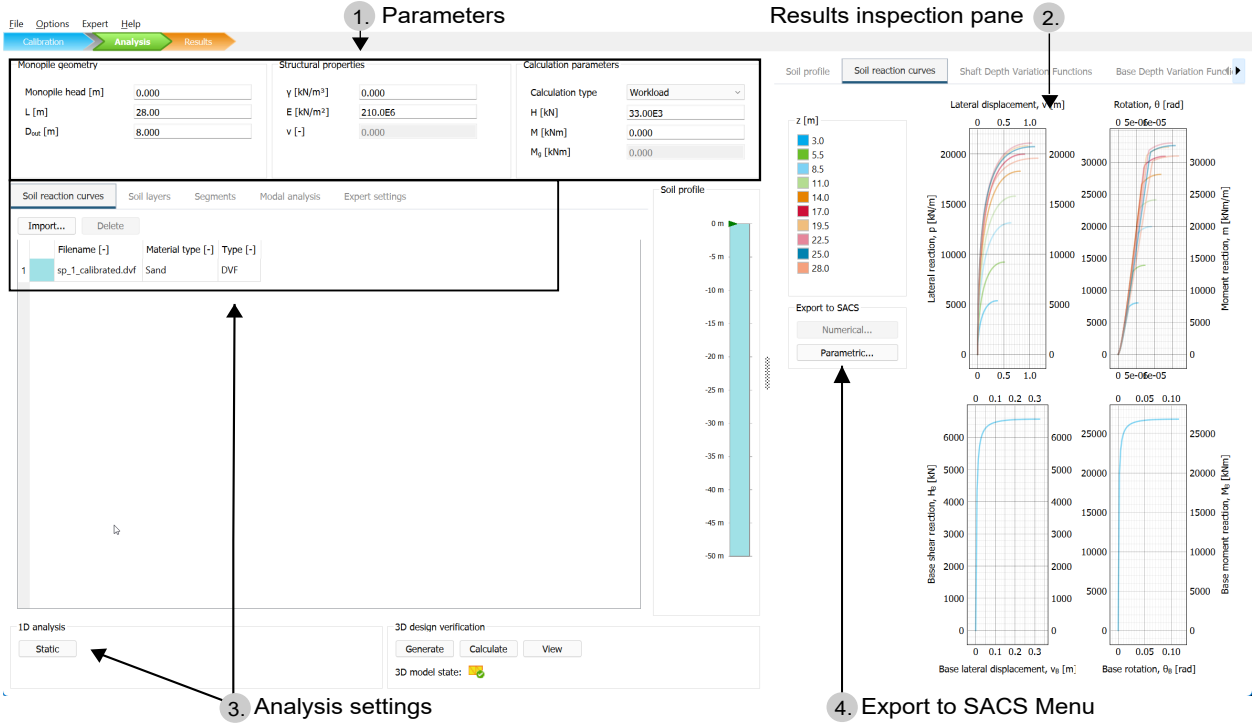


Figure 2–27: Analysis mode layout - Static analysis

To obtain reliable results, the monopile geometrical and structural properties, as defined in this mode, should fall within the 'design space' as considered by the 3D calibration models from which the soil reaction curves were obtained.

In case that the numerical-based design approach has been followed via PLAXIS Monopile Designer this information is included in the produced *dvf* file. If the rule-based design is adopted, then this information should be specified by the users in the imported *dvf* file. In case that the selected monopile properties fall outside the considered design space, the user is notified via a warning message that the calculation results may be invalid.

In contrast to the **Calibration mode**, the monopile may consist of different segments with different wall thickness; thereby allowing for further optimisation of the geometry.

The results of each calculation may be inspected in the **Results mode** of PLAXIS Monopile Designer.

2.4.1 Monopile geometry

The *Monopile geometry* tabsheet allows to define the geometrical configuration of the monopile foundation inside the soil. The available geometry parameters are the same as in the **Calibration mode** with the remark that the *Monopile head* of the **Analysis mode** corresponds to the monopile height (h) in the **Calibration mode**.

The monopile geometry properties are shown in Table 2–6 (p. 58).

Table 2–6: Monopile geometry parameters

Parameter	Description	Unit
Monopile head	Height of the monopile head above the mudline	[m]
L	Monopile embedded depth	[m]
D_{out}	Monopile outer diameter	[m]

The Monopile geometry embedded in some soil layers can be visualized in the *Soil Profile* window. Figure 2–28 (p. 58) displays one example.

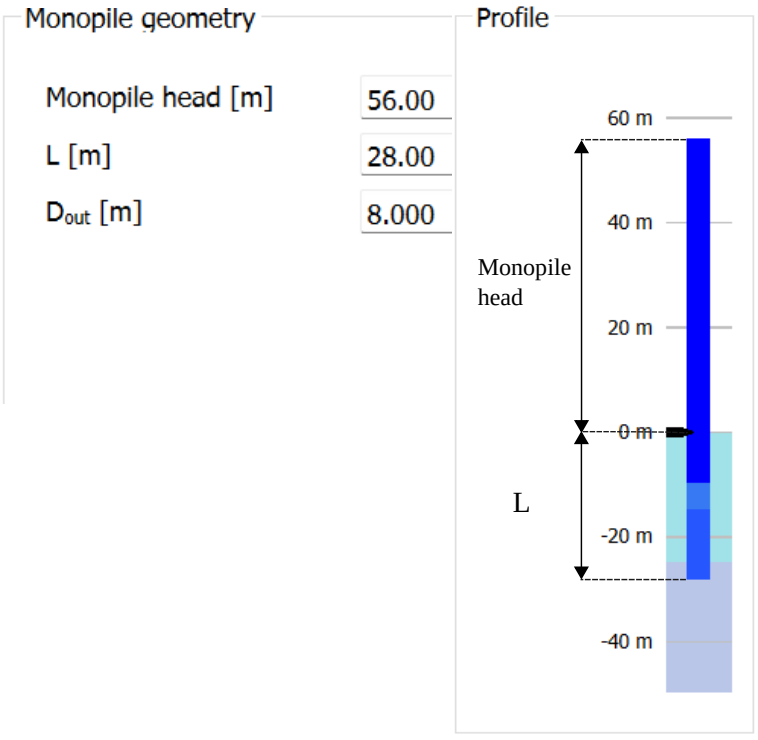


Figure 2–28: Representation of a monopile embedded in soil at a 28 m depth

2.4.2 Structural properties

In the *Structural properties* (Figure 2–29 (p. 59)) panel it is possible to add/change values of the Young's modulus *E*. The user cannot edit values of *γ* and *v*. These are automatically set to zero by the software.

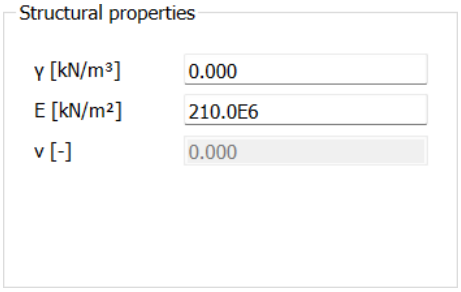


Figure 2–29: Structural properties - *Analysis mode*

The *Structural properties* are shown in Table 2–7 (p. 59).

Table 2–7: Structural properties list

Property	Description	Unit
γ	Pile distributed weight	[KN/m^3]
E	Young's modulus	[m]
v	Poisson's ratio	[-]

Note:

- Poisson's ratio is fixed to 0. The reason being that it is modelled as a 1D beam. But in reality, it is a tube. Hence even if there is a Poisson effect, it would be negligible on the whole. But if it were a solid beam, then there would be a Poisson effect.
- The pile unit weight *γ* is set to 0 as a default. The reason is that the analysis is done considering the monopile a 1D beam, but in reality, it is a tube. Take into account that *γ* can take values different to the default in dynamic analysis of Monopiles (see Modal analysis for more information).

2.4.3 Calculation Parameters

2.4.3.1 Workload

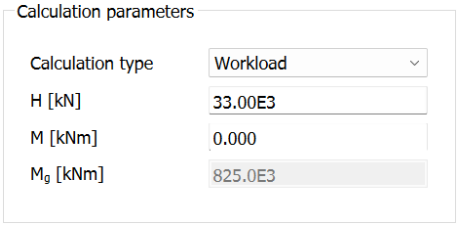


Figure 2–30: Workload - Analysis mode

A horizontal load *H* and a bending moment *M* may be applied to the monopile head at a height *h*. If *h* is zero, then the head coincides with the mudline. The equivalent bending moment at ground level M_g is calculated based on the input values of *H*, *M* and *h* as follows:

$$M_g = H \cdot h + M \tag{2–2}$$

Table 2–8: Parameters of the Workload calculation type

Parameter	Description	Unit
h	Monopile head (Elevation of the head of the monopile)	[m]
H	Horizontal force acting on the head of the monopile.	[kN]
M	Moment acting on the head of the monopile.	[kNm]
M_g	Bending moment at the ground level	[kNm]

Figure 2–31 (p. 60) shows a monopile profile with the application of a workload.

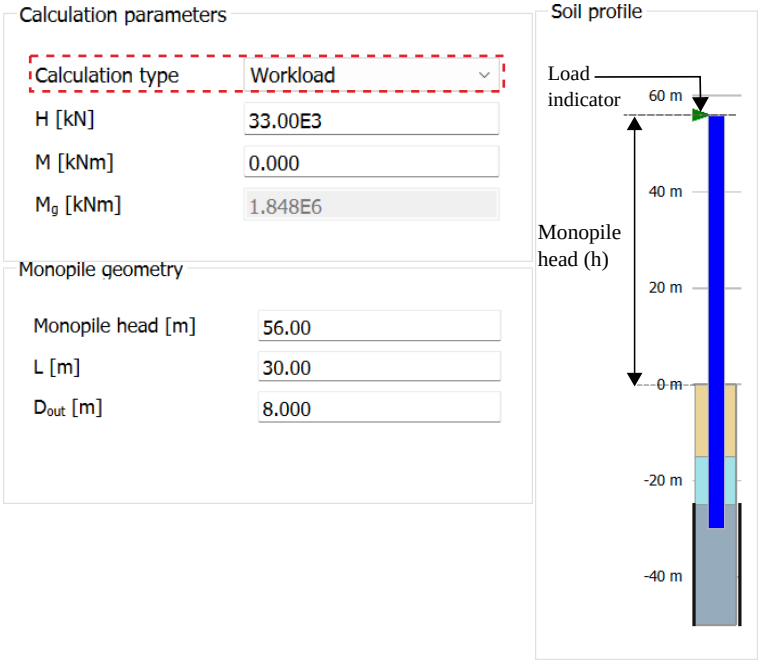


Figure 2–31: Horizontal load applied on the head of the Monopile

2.4.3.2 Target displacement

The analysis stops when a prescribed target displacement at mudline is reached. The target displacement at the mudline is specified as a fraction of the pile diameter (v_g/D_{out}) as displayed in Figure 2–32 (p. 61).

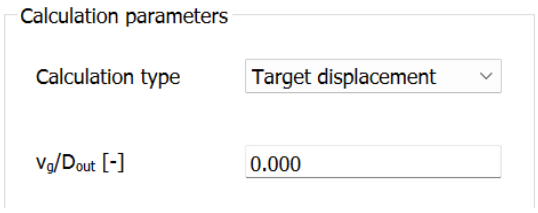


Figure 2–32: Calculation Parameters - Target Displacement

2.4.4 Soil reaction curves

The soil reaction curves that are used in the 1D calculation are defined via depth variation functions. These functions can be derived from the **Calibration mode**, or they can be user-defined. The file format for importing user-defined functions is *.*dvf* (a plain text file with the *.dvf* extension). Other supported formats are **.cpy* and **.spy* for conventional p-y curves for clays and sands, respectively. For descriptions of the different file formats, refer to 5.5 Rule based models (p. 130).

If the **Calibration mode** has been used, then a numerical-based *.dvf* file will have been generated and saved in the project folder's location and it will also appear in this tab as '*calibrated.dvf*'. The user may import any other user-defined *dvf*, *cpy*, or *spy* file as long as it complies with the required format. Once a file has been imported, it is displayed in this tab as well.

> **Note:** The '*calibrated.dvf*' name is reserved for the file that is generated in the **Calibration mode** and it will be overwritten each time the calibration set is Parameterised. To preserve the existing .dvf file, save a copy under a different name before running the new parameterisation. To use a *.dvf* file generated in a different project, rename it before importing.

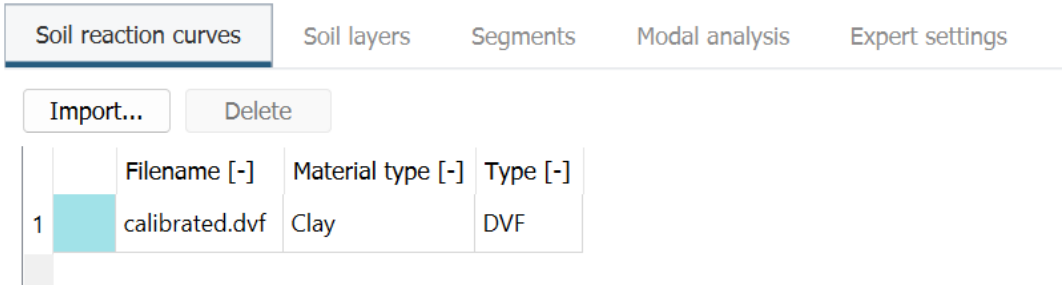


Figure 2–33: Calibrated option for depth variation functions

2.4.5 Soil Layers

The *Soil layers* tab is used to specify the soil profile used in the 1D analysis and the 3D design verification.

The user creates soil layers using buttons above the soil layers area:

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

The soil reaction curve sets imported in the *Soil reaction curves* tab can be assigned to the different layers. Each *dvf* file may contain one or more sub-layers with varying material parameters. To create these *.dvf* files please refer to 5.5.3.3 Format: Depth variation functions (p. 140).

The *1D analysis*: uses both the profile depth variation function coefficients and the material parameters of each sub-layer to determine the mechanical characteristics of the soil springs at different depths.

The *3D design verification*: is generated using only the material parameters.

The user can inspect the soil profile by looking at layer boundaries in the *Soil layers* tab (see Figure 2–34 (p. 62)), which is visible in the central panel of the **Analysis mode**. The *Soil layers* section displays 1) the visual representation of the inserted soil layers and their top and bottom boundaries, and 2) the Material Parameters of the layer according to the depth.

> **Note:** Material parameters are only visible when a layer is selected

In the **Analysis mode** there are available by default some of the published PISA rule-based models such as Cowden till, Bothkennar clay and Dunkirk sand models, also the API p-y models (clay and sand) (see 5.5 Rule based models (p. 130) for more information).

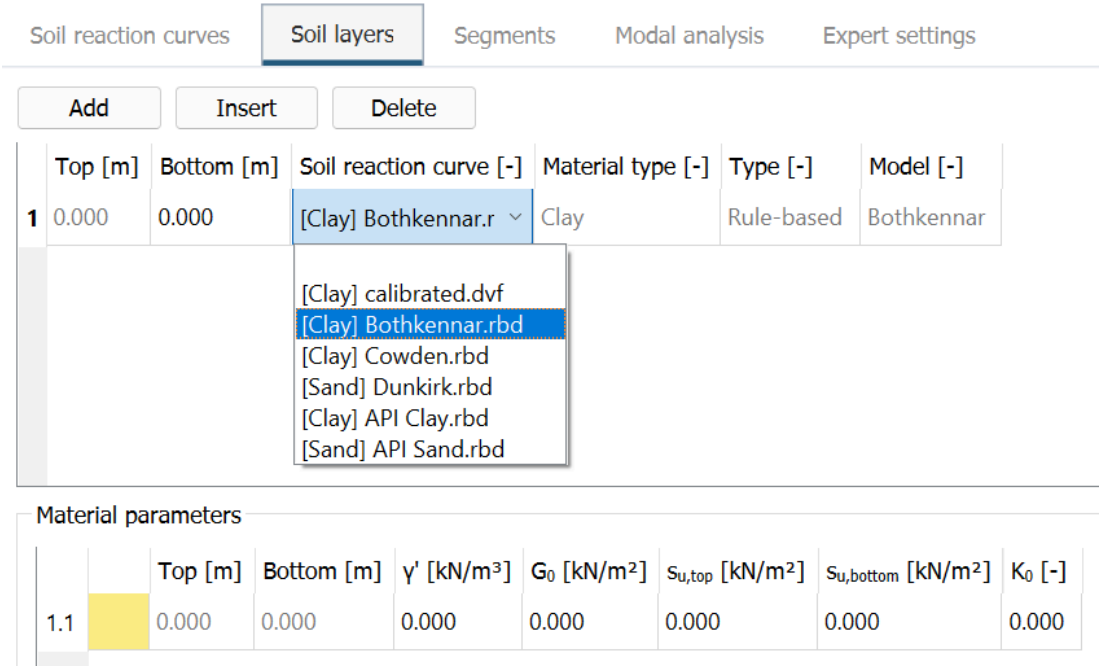


Figure 2–34: Soil layers - Analysis Mode

2.4.6 Pile segments and segment properties

The Monopile Designer allows the creation of pile segments and the definition of their properties. Figure 2–35 (p. 63) presents a conceptual representation of a monopile with different segments and thickness variation.

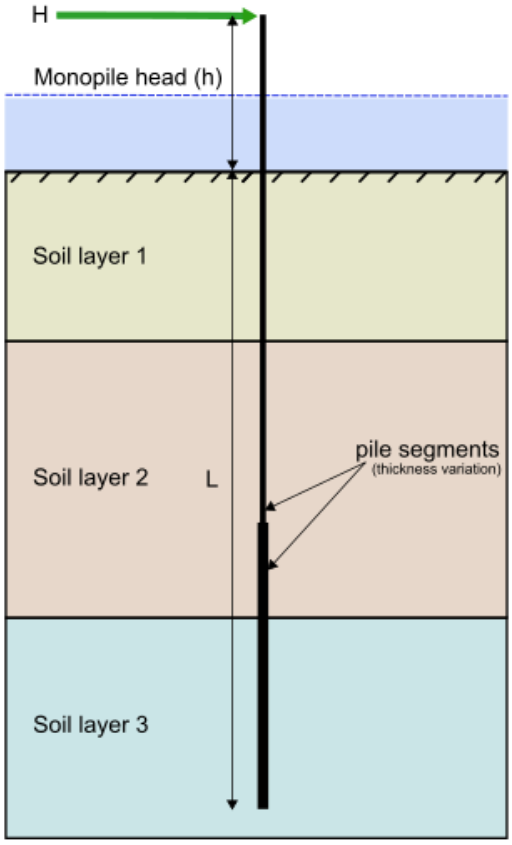


Figure 2–35: Schematic representation of the pile with two different segments with thickness variation used in 1D model

Pile segments can be defined done in the *Segments* tabsheet as displayed in Figure 2–36 (p. 63).

Soil reaction curves | Soil layers | **Segments** | Modal analysis | Expert settings

Add | Insert | Delete

	Top [m]	Bottom [m]	t [m]	$D_{out,top}$ [m]	$D_{out,bottom}$ [m]	γ [kN/m³]	A [m²]	I [m⁴]	EA [kN]	EI [kNm²]	GA [kN]
1	56.00	-10.00	0.07000	8.000	8.000	0.000	1.744	13.71	366.2E6	2.879E9	91.55E6
2	-10.00	-15.00	0.08000	8.000	8.000	0.000	1.991	15.61	418.0E6	3.278E9	104.5E6
3	-15.00	-30.00	0.09000	8.000	8.000	0.000	2.236	17.49	469.7E6	3.674E9	117.4E6

Figure 2–36: Segments tabsheet

The user can create/delete pile segments using the buttons on the Table 2–9 (p. 64)

Table 2–9: Segment creation buttons

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

Tip: After adding or inserting a segment, it becomes the current (selected) segment.

Properties of segments are displayed in Table 2–10 (p. 64).

Table 2–10: Segment property definition

Property	Description	Unit
Top/bottom	Top and bottom boundary of the monopile segment	[m]
t	Thickness of the monopile segment	[m]
$D_{out,top}$, $D_{out,bottom}$	Outer diameter at the top and bottom of the monopile segment.	[m]
γ	Unit weight of the monopile segment	[KN/m^3]

Note:

- The pile unit weight γ is set to 0 as a default because the analysis is done considering the monopile as a 1D beam, but in reality, it is a tube. Take into account that γ can take different values for dynamic analysis of Monopiles (see Modal analysis for more information).
- $D_{out,top}$/$D_{out,bottom}$ can take different values in monopile segments not embeeded in the mudline.

Adding, inserting, and deleting pile segments general rules:

1. The thickness of a newly added segment is zero by default.
2. The top boundary of an underlying segment is defined by the bottom boundary of the overlying segment.
3. To change the thickness of a segment, the user modifies the bottom boundary.
4. A newly added segment appears as the last segment.
5. A segment's bottom boundary cannot be lower than the next segment's bottom boundary.
6. When deleting a segment, a warning pops up.

Based on the values of the geometric parameters and structural properties, the cross-sectional area (*A*), moment of inertia (*I*), axial stiffness (*EA*), flexural rigidity (*EI*) and shear stiffness (*GA*) are automatically calculated and displayed.

2.4.7 Modal Analysis

The Monopile Designer does not only allow the modelling of a monopile foundation subjected to a lateral load (H), also offers the capability of evaluating the dynamic response of a monopile - superstructure (tower and turbine) system.

Some considerations taken in the **Modal Analysis** of a Monopile-Superstructure (Transition piece, tower and rotor-nacelle assembly [RNA]) include:

- The system is constructed as a 1D beam-spring model to consider the distribution of masses and the superstructure.
- The transition between monopile - superstructure is modelled as a part of the monopile.
- The tower can be defined by segments with diameter and thickness varying along the z-axis to represent a cone truncated structure .
- The RNA is modelled as lumped mass at the top of the tower.

To activate the **Modal Analysis** please click on the **Include modal analysis** check box as displayed in Figure 2–37 (p. 65).

Once the Modal Analysis is activated:

- The main parameters to run the Modal Analysis can be defined. These parameters are briefly described in Table 2–11 (p. 65).

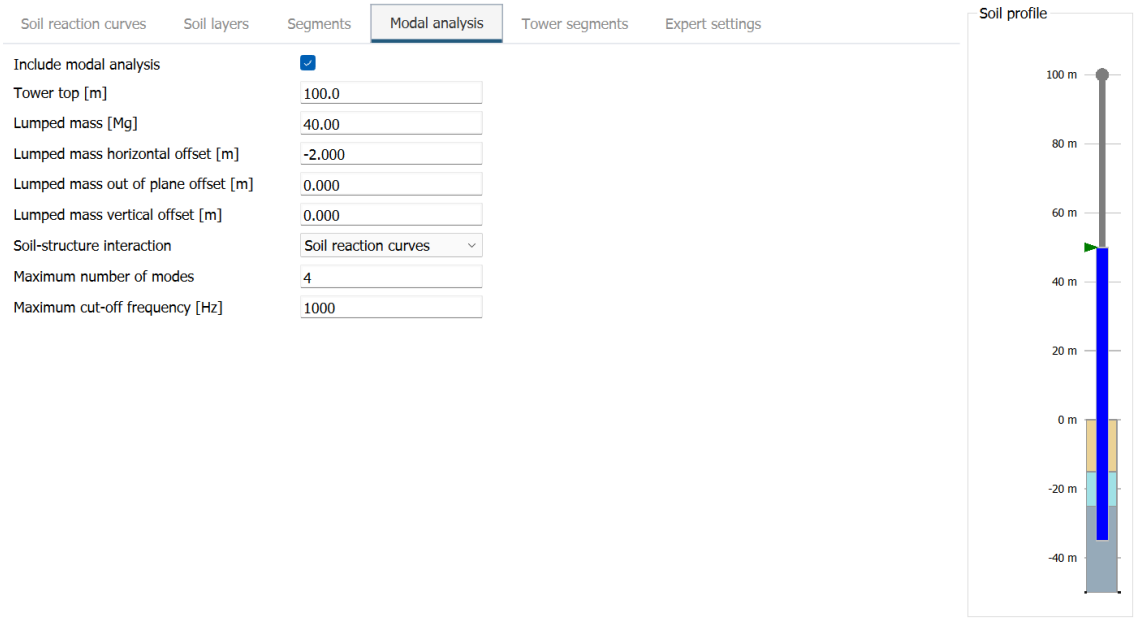


Figure 2–37: Modal analysis and tower activation

Table 2–11: Parameters of the Modal Analysis

Parameter	Description	Unit
Tower top	Tower elevation from the mudline.	[m]
Lumped mass	Rotor - nacelle mass concentrated in a point for modal analysis.	[Mg]

Parameter	Description	Unit
Lumped mass horizontal offset	Displacement of the center of gravity of the lumped mass in the horizontal direction regarding the center of mass of the monopile - tower system.	[m]
Lumped mass out of plane offset	Displacement of the center of gravity of the lumped mass in the out-of-plane direction regarding the center of mass of the monopile - tower system.	[m]
Lumped mass vertical offset	Displacement of the center of gravity of the lumped mass in the vertical direction regarding the center of mass of the monopile - tower system.	[m]
Soil-structure interaction	**1- Soil reaction curves**: Flexible foundation, with small-strain stiffness determined by the assigned soil reaction curves.	-
	2- Clamped: Rigid foundation, fully fixed at ground level.	
Maximum number of modes	Maximum number of natural frequency modes in the monopile-tower-RNA system.	-
Maximum cut-off frequency	Maximum natural frequency for convergence	[Hz]

- The Tower segments tabsheet becomes available (see Figure 2–37 (p. 65)) to input the characteristics of the tower-turbine system. For more information visit Tower segments.

> **Note:** To visualize the tower as shown in Figure 2–37 (p. 65) remember to provide, in the Modal analysis tabsheet, a tower elevation higher than the elevation of the head of the monopile.

- In the *1D analysis* section of the **Analysis Mode** the **Modal** button becomes available to run the Modal Analysis (see Figure 2–38 (p. 66)).

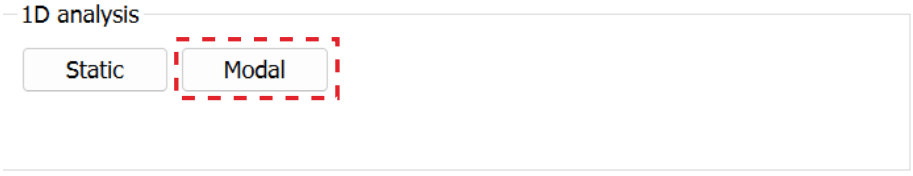
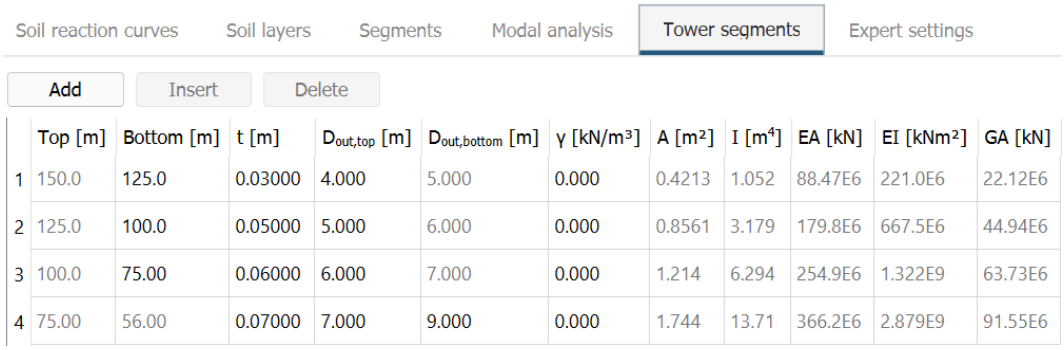


Figure 2–38: *1D analysis* - Modal Analysis run button

2.4.7.1 Tower segments

To run a **Modal analysis** on a tower-turbine system, the geometric characteristics of the tower need to be defined **Tower Segments** tabsheet as displayed in Figure 2–39 (p. 67) and briefly explained in Table 2–12 (p. 67).



	Top [m]	Bottom [m]	t [m]	$D_{out,top}$ [m]	$D_{out,bottom}$ [m]	γ [kN/m³]	A [m²]	I [m⁴]	EA [kN]	EI [kNm²]	GA [kN]
1	150.0	125.0	0.03000	4.000	5.000	0.000	0.4213	1.052	88.47E6	221.0E6	22.12E6
2	125.0	100.0	0.05000	5.000	6.000	0.000	0.8561	3.179	179.8E6	667.5E6	44.94E6
3	100.0	75.00	0.06000	6.000	7.000	0.000	1.214	6.294	254.9E6	1.322E9	63.73E6
4	75.00	56.00	0.07000	7.000	9.000	0.000	1.744	13.71	366.2E6	2.879E9	91.55E6

Figure 2–39: Modal Analysis - Tower segments tabsheet

Table 2–12: Parameters of tower segments tabsheet

Property	Description	Unit
Top/bottom	Top and bottom boundary of the tower segment	[m]
t	Thickness of the tower segment	[m]
$D_{out,top}/D_{out,bottom}$	Outer diameter at the top and bottom of the tower segment	[m]
γ	Unit weight of the tower segment	[KN/m^3]

Note:

A modal analysis can be run for a tower-turbine system without the presence of a monopile. To do this:

1. In the **Analysis mode** go to the *Monopile geometry* area. Assign zero values to *Monopile head*, *L* and D_{out}.
2. In the *Modal Analysis* tabsheet activate the Modal analysis and define the main characteristics of the system. Ensure that the Soil-structure interaction is selected as *Clamped*.
3. In the *Tower segments* tabsheet define the geometric characteristics of the system.
4. In the 1D analysis area, click on the **Modal** button.

2.4.8 Expert settings

The *Expert settings* tabsheet (Figure 2–40 (p. 68)) has parameters as described below.

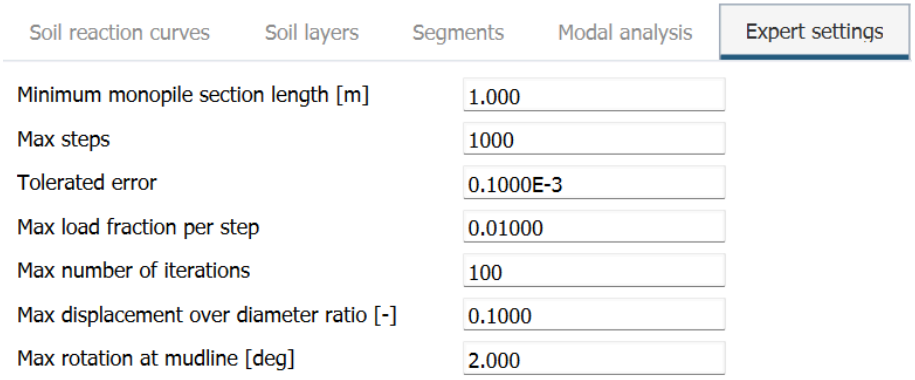


Figure 2–40: Expert settings tabsheet

Table 2–13 (p. 68) gives the minimum, maximum and default values for the expert settings parameters.

Table 2–13: Minimum, maximum and default values for expert settings parameters

Property	Min	Max	Default
Minimum monopile section length (m)	-	*L*	1.000
Max steps	1	10000	1000
Tolerated error	1	0.500	0.0001
Max load fraction per step	-	1.000	0.01000
Max number of iterations	2	250	100
Max displacement over diameter ratio	-	-	0.1000

Minimum monopile section length

The minimum monopile section length (in metres) divides the monopile into *N* beam elements according to this relation:

$$N = \frac{L}{< \text{Minimum monopile section length} >}$$

Note: This relation is used as a first approximation. The length of each beam element is automatically adjusted to accommodate transitions between soil layers and monopile segments.

Max steps

This parameter specifies the maximum number of calculation steps (load steps) that are performed during the 1D calculation. The *Max steps* parameter should be set to an integer number representing the upper bound of the required number of steps for a calculation.

Note: The user should make sure that the specified number of steps suffices to reach the applied load. If the number of steps is not enough a warning is displayed after the calculation (2.4 Analysis mode (p. 57)).

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 is relatively quick but may be inaccurate. If a low tolerated error is adopted, then computation time may increase.

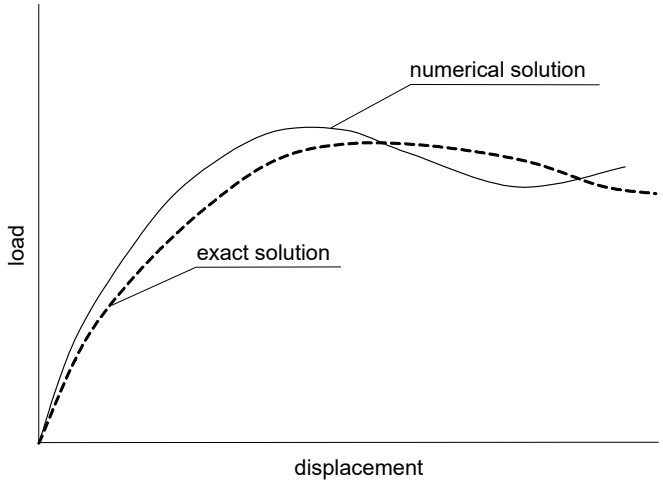


Figure 2–41: Computed solution versus exact solution

Max load fraction per step

This value controls the size of the load step during the calculation. Since this is a fraction, it determines what maximum part of a calculation can be solved in one step. For instance, a value of 0.5 means that the applied load or unbalance is solved at least in 1/0.5 = 2 steps. More steps are possible if convergence is slow, but not fewer.

The user might want to use small values (like 0.02 to force at least 50 steps) to observe the kinematics of the deformation process or to prevent divergence in case of high nonlinearity.

Max number of iterations

This value represents the maximum allowable number of iterations within any individual calculation step. In general, the solution procedure restricts the number of iterations that take place. This parameter is required only to ensure that computation time does not become excessive due to errors in the specification of the calculation.

If the maximum allowable number of iterations is reached in the final step of a calculation phase, then the final result may be inaccurate. Such a situation occasionally occurs when the solution process does not converge.

Max displacement over diameter ratio

Max displacement over diameter ratio is used to end the analysis in case of diverging iterations. This is another stopping criterion for the calculation apart from the input workload. Displacement refers to the lateral displacement of the monopile at mudline.

Max displacement over diameter ratio

Max displacement over diameter ratio is used to end the analysis in case of diverging iterations. This is another stopping criterion for the calculation apart from the input workload or target displacement. Displacement refers to the lateral displacement of the monopile at mudline.

Max rotation at mudline

Max rotation at mudline is used to end the analysis in case of diverging iterations. This is another stopping criterion for the calculation apart from the input workload or target displacement. Refers to the larotation of the monopile at mudline.

> **Note:** In 1D analysis calculations the analysis will now stop either when “max displacement over diameter ratio” OR “max rotation at mudline” is reached.

2.4.9 | 1D Analysis

The user can:

- Click on the **Static** button to start the calculation for a monopile under static loading conditions specifying the monopile geometry, loading, and importing the depth variation functions. For more information on importing depth variation functions refer to 2.4.4 Soil reaction curves (p. 61).

  Static button triggers the following two actions:

  1. The generation of parameterised and normalised soil reaction curves per 0.5 m intervals, and generation of shaft and base variation functions, based on the imported **.dvf* file.
  2. The calculation of the 1D model of the pile. The 1D calculation takes only a few seconds. The results are presented in the **Results mode**.
- (If activated) Click on the **Modal** button to run the Modal Analysis.

After the calculation starts, a calculation window appears with a progress bar. Once the 1D calculation is completed either a green checkmark or a red cross icon appear to indicate a successful or failed calculation accordingly. A message is also displayed next to the icon to provide more information. The reader may refer to A.2 Analysis Mode - Warnings and errors (p. 167) for the complete list of the messages.

The graphs are automatically updated when the calculation is completed successfully.

> **Note:** After calculation in the **Analysis mode**, the results presented in the **Results mode** are automatically updated.

To **zoom in/out**, use the scroll or click-and-drag option from top-left to bottom-right or **click-and-drag** from bottom-right to top-left.

2.4.10 | 3D Design Verification

The 3D design verification module makes use of PLAXIS 3D to generate and run a 3D model that replicates the conditions of the 1D analysis. Both models are generated using the same input, which includes soil layers and material parameters of each sub-layer, monopile geometry including thickness variation per segment, structural properties, and workload. This enables comparing the results between the 3D model and 1D model.

State icons

3D design verification models have different states, depending on the actions that were carried out on the model and the success or failure of these actions.

The following states exist and are represented by the corresponding icons:

 The model is successfully generated, but not calculated yet.

 The model is not successfully generated. An error occurred during generation or meshing.

 The model is successfully generated and calculated.

 The model is not successfully calculated (but it is already successfully generated). An error occurred during calculation.

 The model is partially calculated (but it is already successfully generated). Calculation stopped before target displacement was reached, but partial data could still be recovered.

 The model was changed since it was last generated, calculated or parameterised.

Generate

To generate a 3D design verification model, PLAXIS Monopile Designer will:

- Run or update the 1D analysis (as described in 2.4.9 1D Analysis (p. 70)).
- Generate the soil layers and sub-layers (as specified in the *Soil layers* tab and the *Soil reaction curves* files).

> **Note:**
>
> All the needed material parameters are calculated based on predetermined relationships. See Clay: NGI-ADP material Parameters and Sand: HSsmall material Parameters.
>
> Calculated values can be manually modified via the Materials menu in PLAXIS 3D.

- Generate the structure (monopile and corresponding interfaces) based on the settings of the Monopile geometry and the pile segments defined in 2.4.6 Pile segments and segment properties (p. 63).
- Divide the pile into slices to extract the raw soil reactions at different depths.
- Generate the plate material and assign structural properties specified in the 2.4.2 Structural properties (p. 59) area.
- Generate the calculation phases and adjust the numerical settings to values suitable for accurate and fast calculations (for the specific type of models generated by PLAXIS Monopile

Designer). The following calculation phases are created, based on the settings of Workload (monopile head):

- Initial phase: generation of initial stress state.
- Phase 1: pile wished-in-place.
- Phase 2: workload.

- Generate a finite element mesh and select precalculation curve points in Output.

> **Note:**
>
> - The model generation works only if there is no feedback (warning) or if the user chooses to ignore it.
> - Generating or calculating a 3D design verification model performs an autosave. Any changes in the project are saved and cannot be undone.
> - The selected pre-calculation curve points may be used by the user to check additional results (in PLAXIS 3D Output) but they are not directly used by the PLAXIS Monopile Designer workflow and calculations.
> - The calculation phases are always generated before the mesh. In case the mesh generation fails, the model still contains properly defined calculation phases. The user can open the PLAXIS 3D model and try to generate the mesh manually. Changes on the default mesh settings may be needed to mesh successfully. Afterwards, the user should save the PLAXIS 3D project and close it. The calculation must be done via PLAXIS Monopile Designer.
>
>   A user may modify a 3D model that is automatically generated by PLAXIS Monopile Designer in PLAXIS 3D. However, the modified model must represent the same situation (i.e. soil profile and monopile geometry) that has been defined in PLAXIS Monopile Designer. The user remains responsible for a correct representation of the 3D finite element models when modifying these models in PLAXIS 3D.

Calculate

Model calculation includes the following:

- Calculation of all 3 phases (as described in Generate (p. 71)).
- Extraction of the raw soil reaction curves.
- Extraction of data to be displayed in the **Results mode**.

> **Note:** The model calculation works only if there is no feedback (warning) or if the user chooses to ignore it.

During calculation, a window pops up showing the calculation progress. To stop calculation, click the Stop button (x).

View button

This button is used to launch PLAXIS 3D and view or inspect the 3D model.

> **Note:** The model has to be generated before viewing it.

It is useful in the following situations:

- The user can inspect what went wrong, in case generation of the model fails.
- The user is always advised to inspect the generated models, even if the generation was successful.
- The user can inspect output results via PLAXIS Output, after successful calculation.
- The user can modify the model. The message shown in Figure 2–11 (p. 37) appears (unless switched off by the user via the 'Do not show this message again' option).

> **Note:** After visualising the model, the user should close it directly from the PLAXIS 3D user interface by either clicking on the Close (x) button or by selecting File -> Exit.

Model changes outside PLAXIS Monopile Designer

The user can open the model in PLAXIS 3D and modify it. However, PLAXIS Monopile Designer does not reflect any manual changes applied inside PLAXIS 3D.

> **Warning:** Carefully follow the rules explained below to avoid inconsistencies and problems with calculations.

The user may apply only the following safe modifications if needed:

- Change the constitutive model, and even use a user-defined soil model.
- Change the default soil material parameters.
- Change the default mesh settings and regenerate the mesh.
- Change the default numerical settings.

> **Note:** After any modifications, the user should save the PLAXIS 3D model manually and close it afterwards.

The user should not perform any modifications that could interfere with the analysis, including but not limited to:

- Delete the project manually.
- Modify the name of the project (it will not be recognised anymore by PLAXIS Monopile Designer).
- Modify the soil layers manually in PLAXIS 3D by adding/removing boreholes, adjusting the top and bottom layer boundaries.
- Delete or modify the structure ('tunnels'). The structure (monopile geometry) is created by the PLAXIS 3D *Tunnel Designer*.
- Modify the material parameters of the plate elements (this should be done via PLAXIS Monopile Designer).

- Delete or modify the interface elements around the structure (monopile modelled with plate elements) and at its bottom.
- Add/remove calculation phases.
- Change the calculation type (e.g. to dynamic).
- Modify the boundary conditions.
- Perform calculations directly in PLAXIS 3D.

Warning: If the default soil parameters are modified manually in PLAXIS 3D, the parameters in the *Soil reaction curves* file should be updated to reflect the new parameters in PLAXIS 3D.

2.4.11 Results inspection pane

The results inspection pane can be used to view and analyse data which have been obtained by processing the imported dvf file. This is done as part of the 1D calculation procedure which is triggered via the *Calculate* button. The three different tabsheets which are available for results are *Soil reaction curves*, *Shaft depth variation functions* and *Base depth variation functions*.

Soil reaction curves

The *Soil reaction curves* tab (Figure 2–42 (p. 75)) displays the parameterised soil reaction curves derived from the processing of the imported **.dvf* in the **Analysis mode**. The soil reaction curves are computed at *0.5 m* intervals.

The results at predefined depths are shown. The embedded length *L* is divided into 10 equal segments to define the depths (*0.1·L*, *0.2·L*, *0.3·L*, *0.4·L*, *0.5·L*, *0.6·L*, *0.7·L*, *0.8·L*, *0.9·L*, *1.0·L*). The selected depths are rounded to half a meter (up or down). A legend at the left side of the panel indicates the various depths at which the *Soil reaction curves* are displayed.

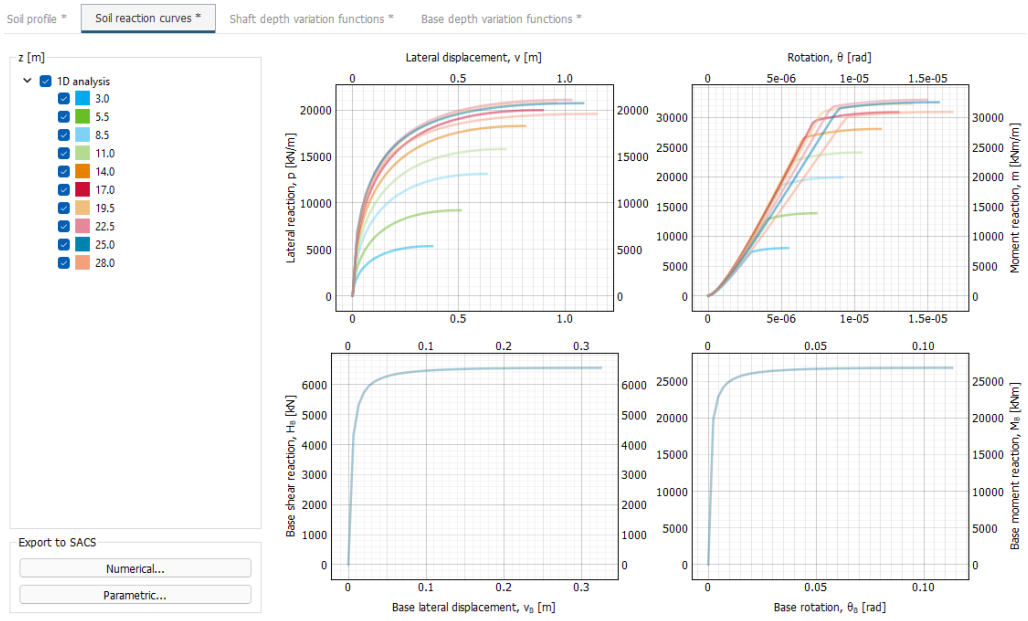


Figure 2–42: Soil reaction curves tabsheet

> **Tip:** Charts are updated after a successful calculation.

The four types of soil reaction curves (p-v, m-θ, H_B-v_B and M_B-θ_B) are displayed in four graphs.

- The top graphs show the p-v and m-θ soil reaction curves along the monopile at the predefined depths
- The bottom graphs show the H_B-v_B and M_B-θ_B soil reaction curves at the base of the monopile (at depth L).
- Double-clicking on a graph pops up a new window, which displays a larger version of this graph and the corresponding table with the results. See the 2.3.6 Results inspection pane (p. 46) for more information.

Depth variation functions

The depth variation functions are automatically generated or updated each time a 1D calculation is performed. The *Shaft depth variation functions* and *Base depth variation functions* show the variation of the fitting parameters along the shaft of the monopile and at the base respectively.

> **Note:** The depth variation functions displayed in the **Analysis mode** will be the same as in the **Calibration mode** if identical monopile geometries are considered.

> **Note:** The vertical (z) axis is not normalised, in contrast to the **Calibration mode**.

- **Shaft depth variation functions**:

  There are eight graphs which illustrate the variation of fitting parameters along the shaft of the monopile. The variables shown in the graphs are defined in Table 5–2 (p. 129).

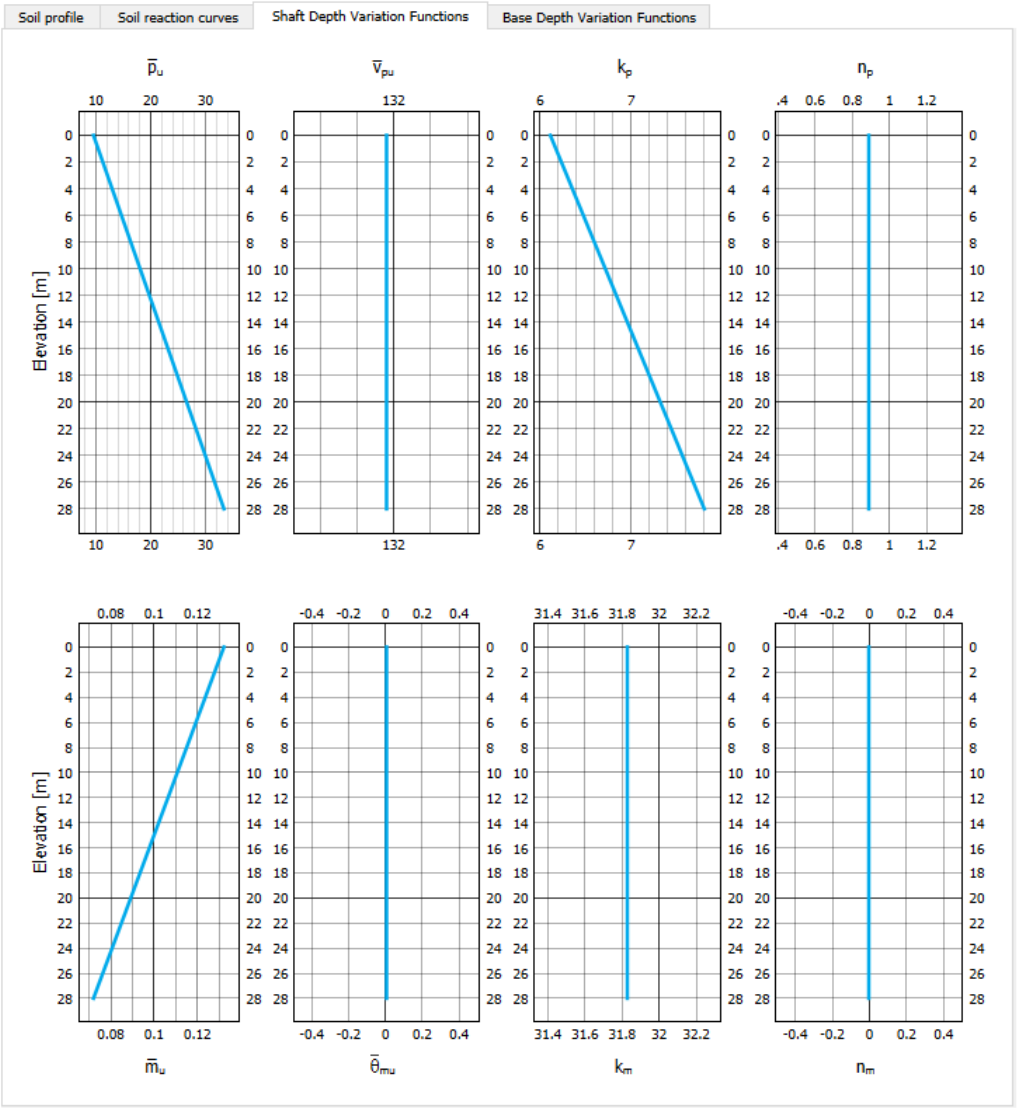


Figure 2–43: Shaft depth variation tab

For more information on the shaft fitting parameters, see Table 2–4 (p. 52).

- **Base depth variation functions:**

There are eight graphs which illustrate the values of fitting parameters at the base of the monopiles. The variables shown in the graphs are defined in Table 5–2 (p. 129). In these graphs, a single point is used rather than an actual variation since the base corresponds to a certain depth which is equal to *L*.

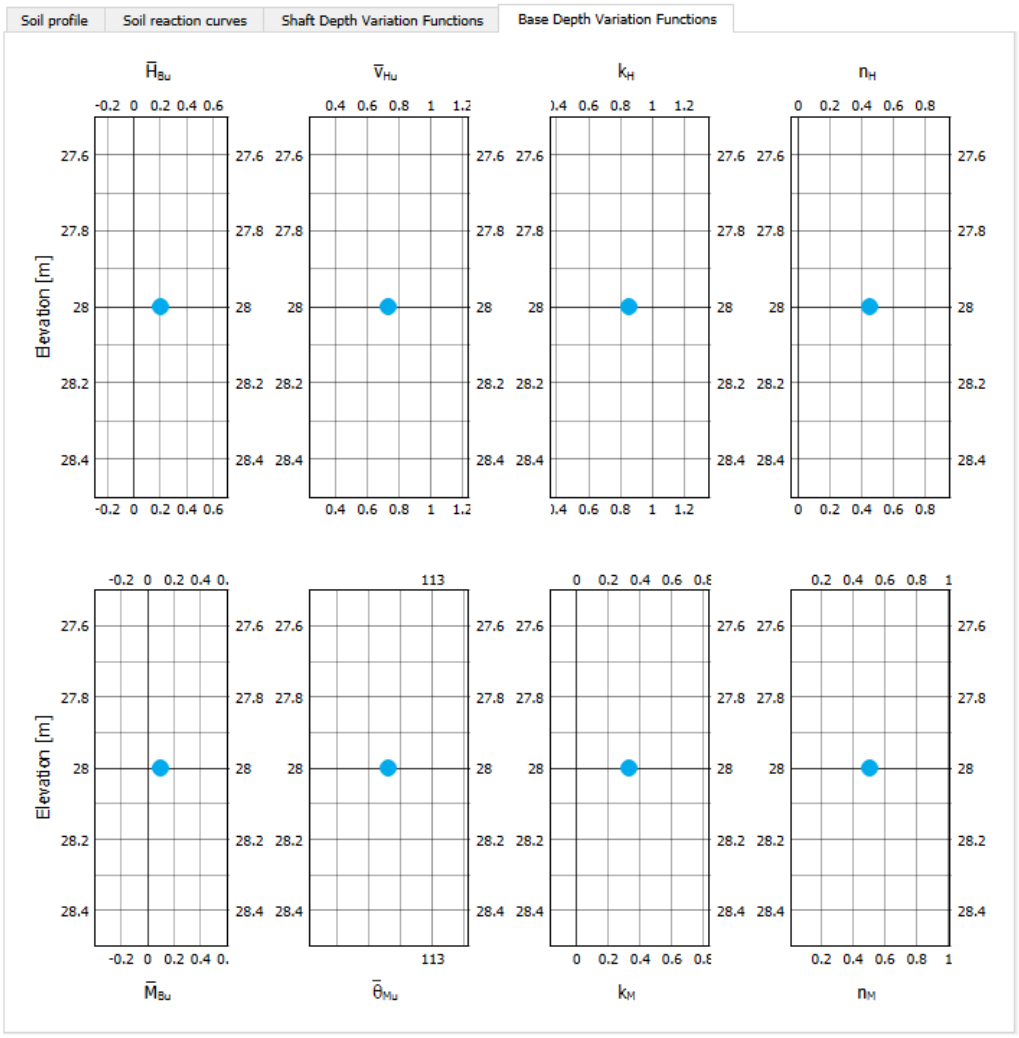


Figure 2–44: Base depth variation tab

For more information on the base fitting parameters, see Table 2–5 (p. 53).

Double-clicking on a graph pops up a new window, which displays a larger version of this particular graph and a table with the data. See 2.3.6 Results inspection pane (p. 46) for more information.

An asterisk (*) is displayed on the tabsheet title when the import option for the dvf file is changed, the dvf file is updated, the embedded length *L* and/or the outer diameter D_{out} of the monopile are modified. The graphs are not valid anymore. Recalculation is required to update them.

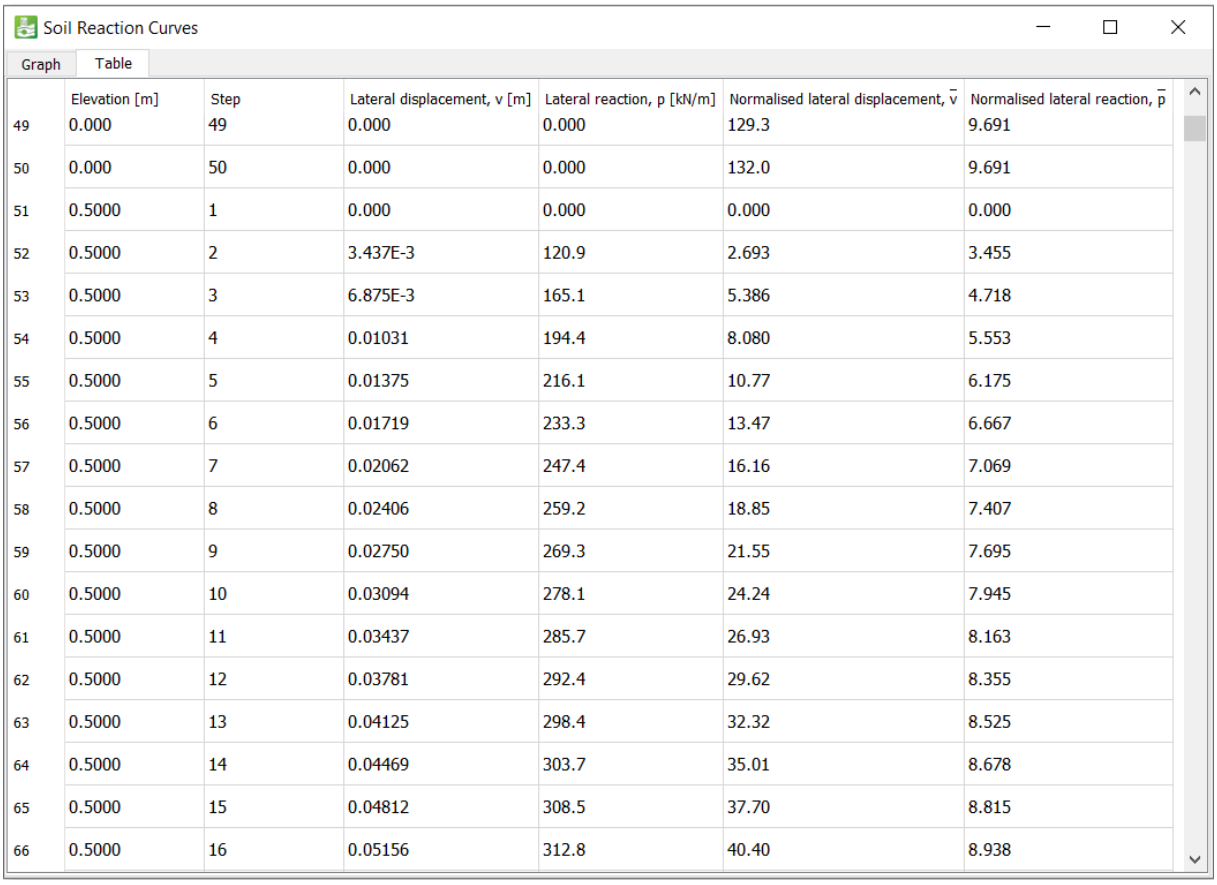
Graph and table windows

When double-clicking a graph a separate window opens, which displays only this particular larger version of the graph and a table with all the values from which the chart was generated.

For the soil reaction curves, the user can choose to show the normalised and parameterised curves. There are checkboxes to control this behaviour. By default both boxes are unchecked. The displayed soil reaction curves are the parametric ones.

The *Numerical* checkbox is available only if the 3D design verification model is *Calculated*.

Switch to the **Table** tab to get a tabular overview of the values for the curve.



	Elevation [m]	Step	Lateral displacement, v [m]	Lateral reaction, p [kN/m]	Normalised lateral displacement, $\bar{v}$	Normalised lateral reaction, $\bar{p}$
49	0.000	49	0.000	0.000	129.3	9.691
50	0.000	50	0.000	0.000	132.0	9.691
51	0.5000	1	0.000	0.000	0.000	0.000
52	0.5000	2	3.437E-3	120.9	2.693	3.455
53	0.5000	3	6.875E-3	165.1	5.386	4.718
54	0.5000	4	0.01031	194.4	8.080	5.553
55	0.5000	5	0.01375	216.1	10.77	6.175
56	0.5000	6	0.01719	233.3	13.47	6.667
57	0.5000	7	0.02062	247.4	16.16	7.069
58	0.5000	8	0.02406	259.2	18.85	7.407
59	0.5000	9	0.02750	269.3	21.55	7.695
60	0.5000	10	0.03094	278.1	24.24	7.945
61	0.5000	11	0.03437	285.7	26.93	8.163
62	0.5000	12	0.03781	292.4	29.62	8.355
63	0.5000	13	0.04125	298.4	32.32	8.525
64	0.5000	14	0.04469	303.7	35.01	8.678
65	0.5000	15	0.04812	308.5	37.70	8.815
66	0.5000	16	0.05156	312.8	40.40	8.938

Figure 2–45: Table of curve values (Default: parametric)

To copy or save data:

- Select values by cell or complete row, if specific values are needed. To copy/save, **Right-click > Copy All/Save All** or **right-click > Copy/Save Selection**.

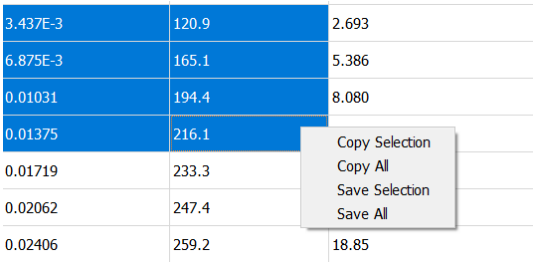


Figure 2–46: Context menu for the table of values

- To copy to the clipboard, use **CTRL+C** shortcut.

Export to SACS/OpenWindPower

It is possible to export the geometry and soil reaction curves of any monopile analysed in PLAXIS Monopile Designer to a SACS Pile Input file, which can be used as input for various analysis types in SACS and OpenWindPower Fixed Foundation.

The Export to SACS menu is located in the Soil reaction curves tab. Two options are displayed, which will be active only if results for the corresponding analysis have been obtained:

- Numerical: exports 3D results, obtained directly from the calculated GeoDS in PLAXIS 3D.
- Parametric: exports 1D results, obtained from the depth variation functions for the parameterised calibration set.

Exporting with any of the options will open a '*Save as*...' dialogue, where the destination folder and file name can be specified. Selecting '*Save*' automatically generates the SACS Pile Input file in the destination folder, which for the selected GeoDS and curve type (Numerical or Parametric) includes:

1. Soil profile and monopile geometry.
2. All 4 sets of horizontal and rotational soil reaction curves (p-v, m-θ, H_B-v_B, and M_B-θ_B), filtered to a small number of points (fewer than 30) while preserving their global shape.
3. Vertical soil reaction curves (t-w, V_B-w_B) constructed according to the API RP 2A-WSD (API, 2014) and the parameters of the soil profile.

For more information on the format and usage of the SACS Pile Input file, refer to the SACS Pile-Structure Interaction Manual (Bentley Systems, 2026).

> **Note:** Use of distributed rotational soil reaction curves (m-θ) requires SACS CONNECT Edition V14.2 or higher; use of pile base curves (H_B-v_B, M_B-θ_B) requires SACS CONNECT Edition V14.3 or higher. It is still possible to use the exported Pile Input file in previous versions of SACS by deleting the lines associated with the unsupported soil curves.

2.5 Results mode

This mode presents the results of the 1D calculation run in the **Analysis mode**. Additional results for a 3D verification model may be presented as well for comparison between the 3D model and 1D model results. To display the results from the 3D model activate the *Design verification* checkbox. This checkbox can only be selected if the 3D verification model is generated and calculated.

The outcome of each analysed model and its calculated data is presented in graphs and tables. The user can select which data to display.

> **Note:** The results presented in this mode are updated only if the 1D calculation in the **Analysis mode** runs again. This includes the data of the 3D verification model.

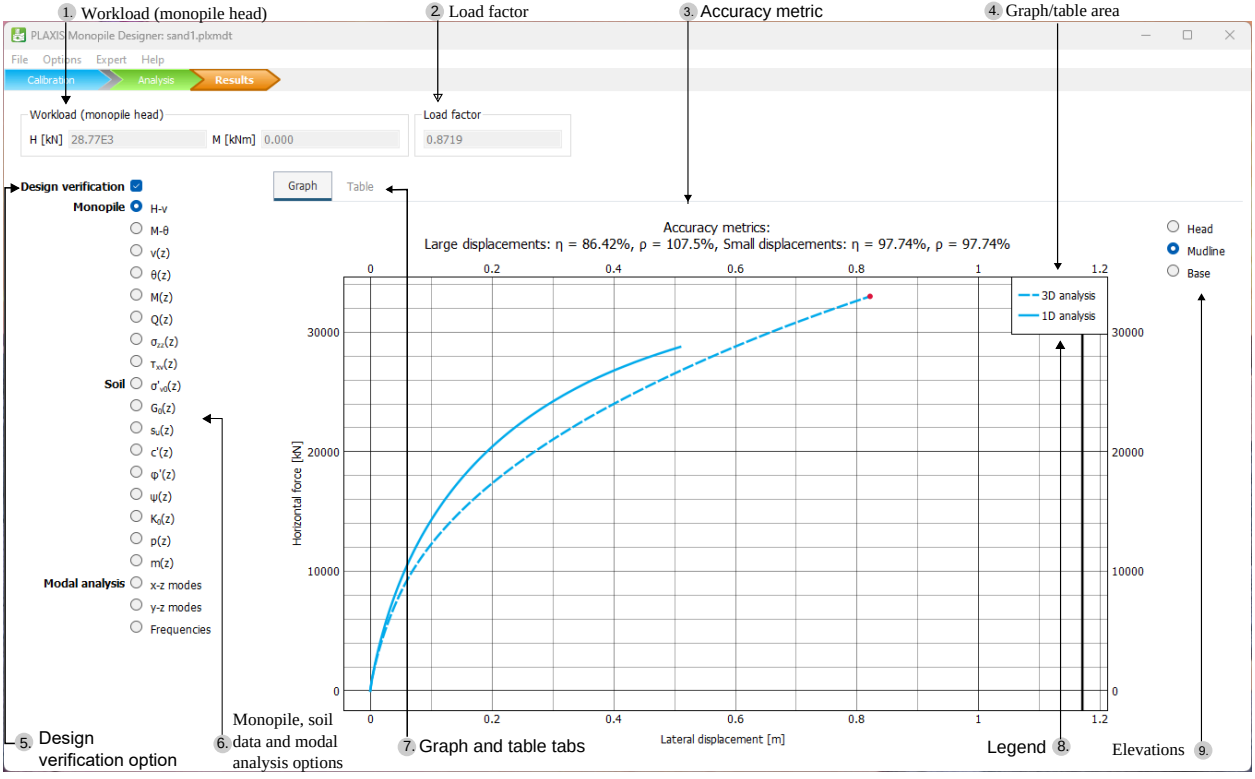


Figure 2–47: View of the Results mode

2.5.1 | Workload and load factor

In the 1D model, the user-defined input load (H and M) are increased by a factor of *3*. At the end of a successful calculation the load factor is the ratio between the reached load and the user-defined load:

- A reached load factor higher than one indicates that the user-defined load can be applied successfully. It also indicates the extra capacity (safety) of the structure.
- Conversely, a load factor less than one indicates that the input load is higher than the capacity of the structure.

Independently of the calculated load factor, the workload always represents the input load specified by the user in the **Analysis mode**. The displayed value is either equal to (load factor ≥ 1.0) or less (load factor < 1.0) than the input value depending on the capacity of the structure.

All results presented in graphs and tables correspond to the applied workload.

2.5.2 | Graph tab

The graph tab shows the graphical results of the 1D model. It can also show the results of the 3D verification model to enable a comparison between the 1D and the 3D models.

The user can display the results of the 3D verification model using the *Design verification* checkbox. The checkbox is available only if the 3D design verification model is calculated.

> **Note:** To update the 3D results after any changes done in the 1D analysis, the user needs to regenerate and rerun the 3D design verification model in the **Analysis mode**. No significant meaning can be derived from comparing different geometries.

The user can select for which properties the graph is to be displayed. Nine Radio buttons control the data displayed on the graphs and the associated tables. The data is represented in the graph and is labelled by the legend.

For the first two plots (*H-v* and *M-θ*), the data on the horizontal axis is provided for three different elevations:

- head (*z=h*)
- mudline (*z=0*)
- base (*z=-L*)

> **Note:** The plotted values of the lateral reaction force *H* and bending moment *M* are always at the mudline. The plotted values of the lateral displacement and cross section rotation can be at head, mudline or base.

> **Note:** The 3D data are only available at mudline for the *H-v* and *M-θ* plots.

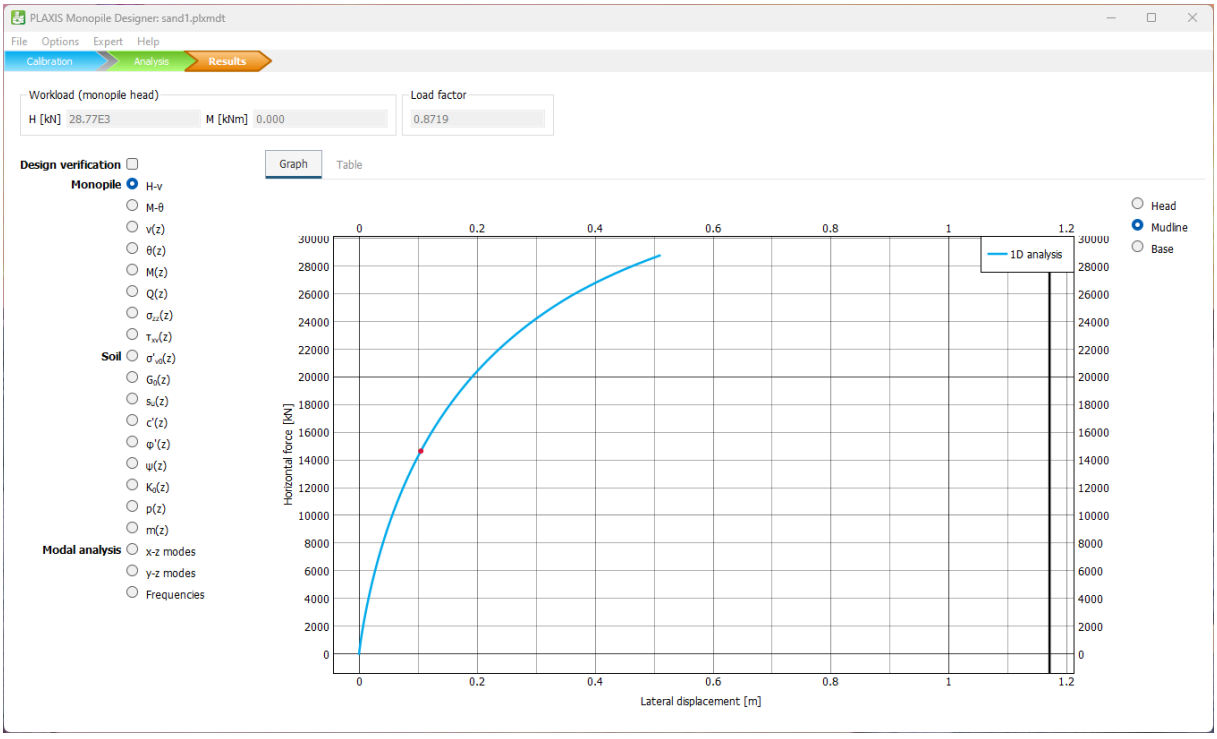


Figure 2–48: Example of H-θ at mudline curve

Monopile results:

H-v	Lateral reaction force (kN) versus lateral displacement (m) at the mudline, base or head.
M-θ	Bending moment (kNm) versus monopile cross section rotation (rad) at the mudline, base or head.
v(z)	Monopile deflection (m) over depth (m).
θ(z)	Monopile cross section rotation (rad) over depth (m).
M(z)	Monopile structural bending moment (kNm) over depth (m).
Q(z)	Monopile structural shear force (kN) over depth (m).

Soil results:

$\sigma'_{v0}(z)$	Initial vertical effective stress (kN/m^2) along depth (m).
$G_0(z)$	Small strain shear stiffness modulus (kN/m^2) along depth (m).
$s_u(z)$	Undrained shear strength (kN/m^2) over depth (m).
$c'(z)$	Effective cohesion (kN/m^2) along depth (m).
$\varphi'(z)$	Effective angle of internal friction (°) along depth (m).
$\psi(z)$	Angle of dilatancy (°) along depth (m)
K_0	Lateral earth pressure coefficient at rest (-) along depth (m).
$p(z)$	Lateral soil reaction (kN/m) along depth (m) and base horizontal force (kN) at the toe of the monopile.

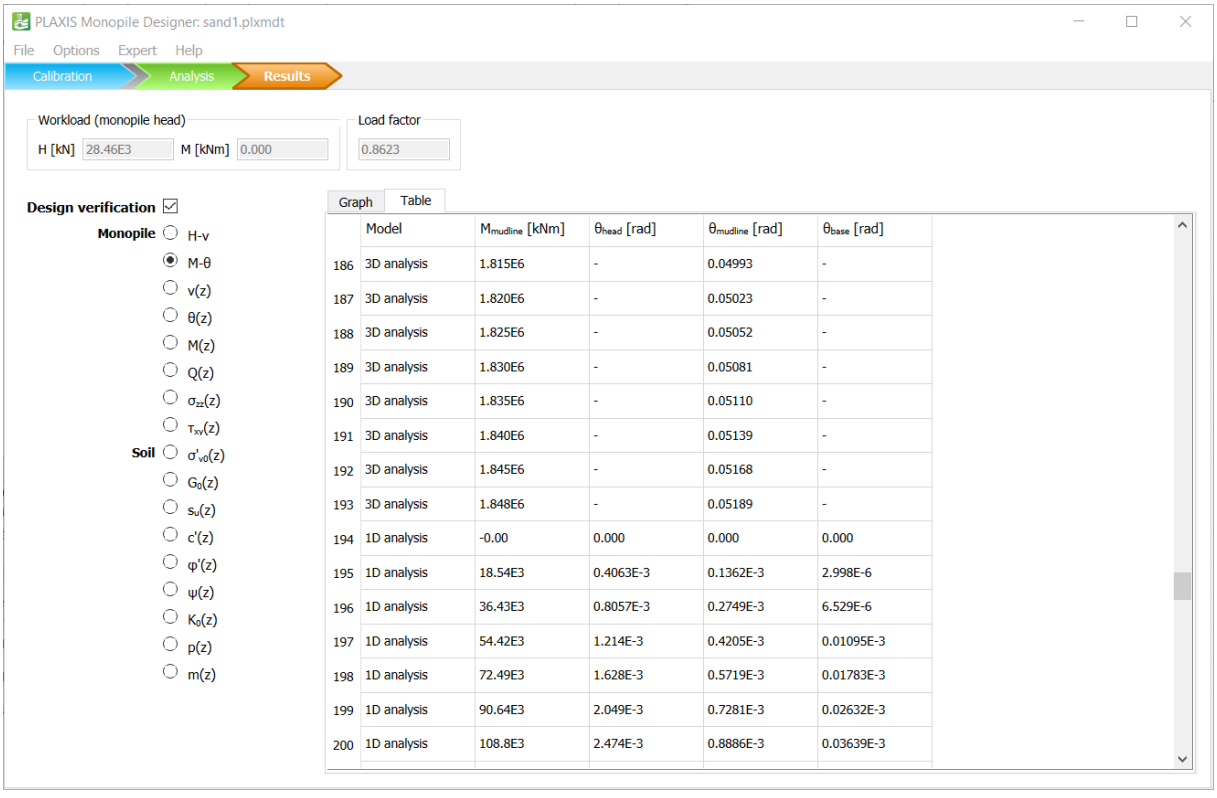
The symbol *(z)* indicates that the corresponding output quantity is plotted over the monopile's depth, where *max(|z|) = -L*.

Modal analysis results [technology preview]:

x-z modes	Modal amplitudes of the natural vibration modes in the x-z (Fore-Aft) direction
y-z modes	Modal amplitudes of the natural vibration modes in the y-z (Side-Side) direction
Frequencies	Natural frequencies and mass participations of the natural vibration modes

2.5.3 Table tab

Provides data displayed in the graph, including 3D data if selected and available (2.5.3 Table tab (p. 83)).



	Model	$M_{mudline}$ [kNm]	θ_{head} [rad]	$\theta_{mudline}$ [rad]	θ_{base} [rad]
186	3D analysis	1.815E6	-	0.04993	-
187	3D analysis	1.820E6	-	0.05023	-
188	3D analysis	1.825E6	-	0.05052	-
189	3D analysis	1.830E6	-	0.05081	-
190	3D analysis	1.835E6	-	0.05110	-
191	3D analysis	1.840E6	-	0.05139	-
192	3D analysis	1.845E6	-	0.05168	-
193	3D analysis	1.848E6	-	0.05189	-
194	1D analysis	-0.00	0.000	0.000	0.000
195	1D analysis	18.54E3	0.4063E-3	0.1362E-3	2.998E-6
196	1D analysis	36.43E3	0.8057E-3	0.2749E-3	6.529E-6
197	1D analysis	54.42E3	1.214E-3	0.4205E-3	0.01095E-3
198	1D analysis	72.49E3	1.628E-3	0.5719E-3	0.01783E-3
199	1D analysis	90.64E3	2.049E-3	0.7281E-3	0.02632E-3
200	1D analysis	108.8E3	2.474E-3	0.8886E-3	0.03639E-3

Figure 2–49: Example of table of results

2.5.4 Accuracy metrics η and ρ

The accuracy metrics η and ρ (Figure 2–50 (p. 84)) are indicators of the goodness of fit between the results of the 1D analysis and the selected 3D model. It is displayed only if a 3D model is selected and only for Lateral load-displacements (*H-v*) at mudline result type. The value are shown in percentage (%) and are visible at the top of the graph. The closer the η and ρ value is to unity, the closer the 1D analysis results are compared to the 3D model results. Desired η values are in the range of 90-100%. Desired ρ values are in the range of 90-110%.

> **Note:** The user should use the accuracy metric η and ρ as a match indicator of the calculation results between equivalent monopile geometries analysed under the same soil conditions. This can be used to check the soundness of the calibration procedure and the validity of the chosen final design.

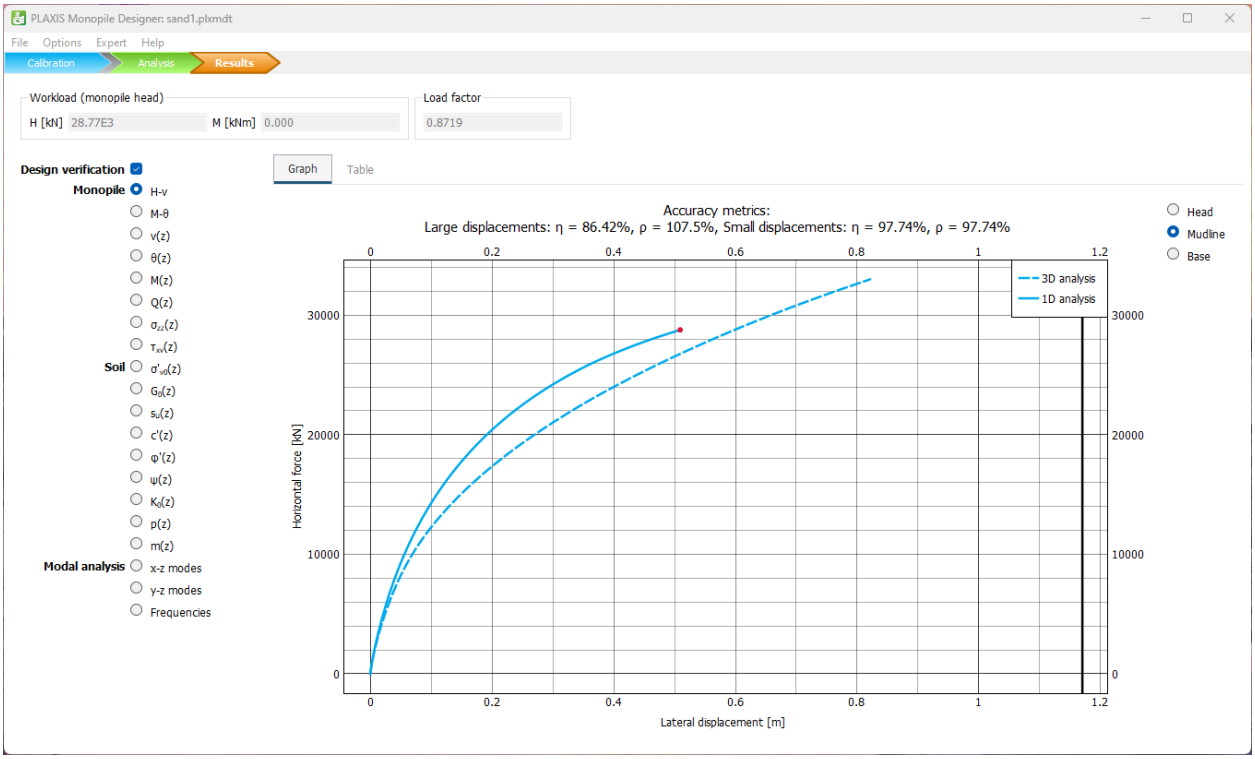


Figure 2–50: Example of accuracy metric η and ρ

2.6 Remote Scripting Interface

It is possible to run the 1D Analysis engine using Python scripting. This approach allows saving time when many calculations are needed. Most functions of the **Analysis** and **Results modes** are available through the remote scripting interface.

2.6.1 Anatomy of a command

The structure of the PLAXIS Monopile Designer remote scripting commands mirrors that of PLAXIS 2D and 3D. Each command has a name and a target. Some commands may also require a parameter.

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.`) or as a specific target command (after some object) in the scripting. In Python, each command corresponds to a method.

Python examples:

`g.Analysis.calculate()`	# calculate is the command name
`g.Analysis.SoilLayers[0].Bottom = -8.5`	#the equal sign stands for the special command 'set'
`g.getresults(g.ResultTypes.MonopileResponseDepthVariation, g.Models.Analysis1D)`	#getresults 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.Analysis.calculate()`	# the target is the analysis object
`g.Analysis.SoilLayers[0].Bottom = -8.5`	# the target is the first layer in the soil profile
`g.getresults(g.ResultTypes.MonopileResponseDepthVariation, g.Models.Analysis1D)`	# the target is the global environment object

Parameter

Parameters represent the extra information passed to a command when it is called. In programming terms, they are arguments to a method. 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.Analysis.calculate()`	#the calculate command is called without parameters
`g.Analysis.SoilLayers[0].Bottom = -8.5`	#the set command updates the value of the layer bottom to the parameter -8.5
`g.getresults(g.ResultTypes.MonopileResponseDepthVariation, g.Models.Analysis1D)`	# the getresults command uses the ResultTypes and Models objects as parameters

2.6.2 Creating scripts

Python scripts can be written in any plain text editor or integrated development environment (IDE). This documentation assumes that the SciTE editor will be used, which is included in the installation of PLAXIS 2D and 3D.

- In the SciTE editor create a new text file by going to **File** > **New** or by pressing **Ctrl+N**.
- First we must import the scripting library:

```
from plxmdt.easyimport new_server
```

- Connect to the PLAXIS Monopile Designer application:

```
s, g = new_server
```

- After that, you can type the code.
- Save this script on your hard drive. Make sure it ends with *.py* so it can be correctly identified as a Python script.

2.6.3 Running scripts

Scripts are run using the PLAXIS Python Internal environment. They can only be executed from the *Expert menu* in PLAXIS Monopile Designer. Either choose **Expert** > **Python** > **Run Python script** > **Open** or save it in the pytools folder and choose **Expert** > **Python** > **Run Python tool**.

> **Note:** The current version of the Remote Scripting Interface does not support file operations, such as importing new *Soil reaction curve* files. Import all necessary files before running your script.

3

Tutorial 1 - Numerical Based Design

This tutorial explains how to apply the Numerical-Based Design (NBD). The NBD is used for a detailed concept design or a final design of a set of monopile geometries.

A typical clay soil profile encountered in the North Sea is assumed, with the following depth variation profiles for the following characteristic:

- Submerged unit weight γ (Figure 3–1 (p. 87) (a)).
- Undrained shear strength s_u in triaxial compression (TXC) (Figure 3–1 (p. 87) (b)).
- Lateral earth pressure at rest K_0 in terms of effective stresses (Figure 3–1 (p. 87) (c)).
- Small strain shear modulus G_0 (Figure 3–1 (p. 87) (d)).

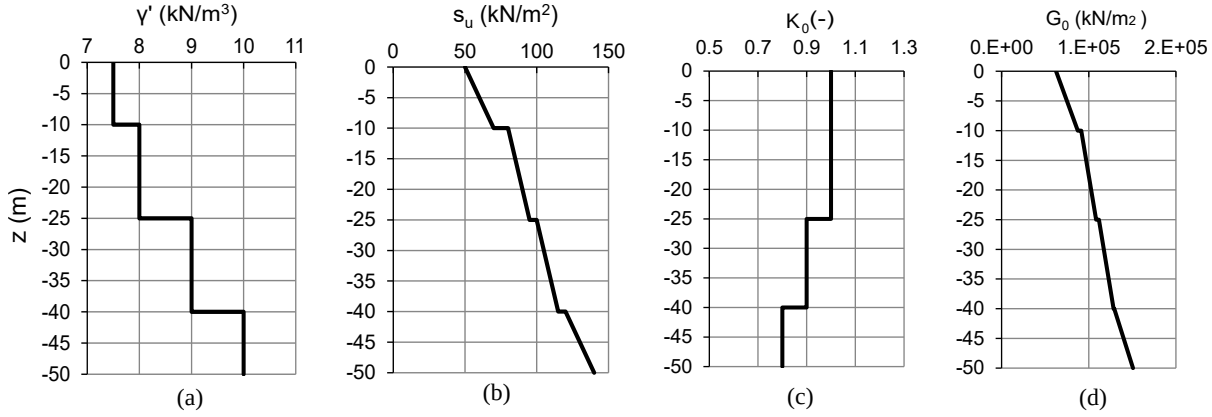


Figure 3–1: Typical soil profile North Sea

Tip: In PLAXIS Monopile Designer the submerged unit weight (γ') of the soil is used to generate an effective stress state without water in the PLAXIS 3D models.

A specific design space is also assumed. The design space consists of many monopile geometries (models) that define an envelope in which the optimum monopile design is expected to lie.

For this tutorial, the design space is defined by eight calibration models. Each model corresponds to a PLAXIS 3D project and is used for the calibration and parameterisation of the soil reaction curves. Figure 3–2 (p. 88) illustrates the adopted design space. The geometric dimensions of the assumed final design case are also presented in Figure 3–2 (p. 88).

The final design is done using a quick 1D design model and is considered to be the optimum design based on the examined soil profile, the assumed design space and the adopted design criteria.

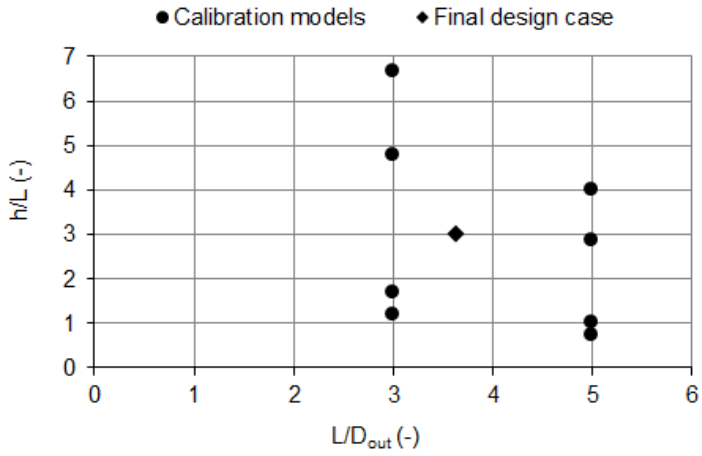


Figure 3–2: Adopted design space

The ultimate goal is to verify the results of the 1D model that represents the final design. This is done by comparing the results of the 1D model against an equivalent 3D model.

Objectives:

- Form the clay soil profile.
- Define the design space by specifying the eight different monopile GeoDS to be considered.
- Generate and calculate the 3D models which correspond to each GeoDS.
- Calibrate the 1D model, based on the data retrieved from all eight 3D model results, through parameterisation.
- Run a number of 1D analyses against some of the eight GeoDS to ensure that the 1D model is well calibrated.
- Run 1D analyses to determine the final monopile geometry, based on the required design criteria.
- Generate and calculate a new 3D design verification model with the presumed optimum monopile geometry.
- Compare the results between the 1D and 3D models with the optimum monopile geometry to verify the final design.

For simplicity, only one design criterion is used in this tutorial, being the displacement at mudline (or seabed surface) must be less than *0.1·D* when the design load *H* is applied at height *h* above mudline.

Also, for simplicity, no thickness variation is considered in this tutorial for the final design. The user might select to vary this parameter to achieve a further optimised final solution. The **Analysis mode**, in contrast to the **Calibration mode**, does allow for thickness variations.

3.1 Calibration mode - Input

- Start PLAXIS Monopile Designer.
- In the Quick select dialogue (Figure 3–3 (p. 90)) choose *Start a new project* and save the project with the name "Monopile Designer Tutorial" in the desired directory.

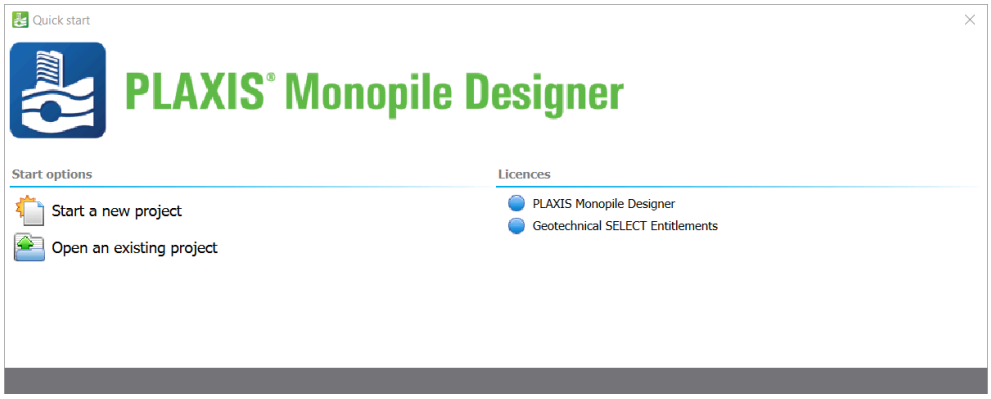


Figure 3–3: Quick select window

Creation of Soil profile and GeoDS

As initial step for the monopile definition, inside the Calibration matrix, create one soil profile and eight GeoDs as displayed in Figure.

> **Note:** To create Soil profiles press the **+** button in the columns of the Calibration matrix. To create GeoDS press the **+** button in the row of the Calibration matrix.

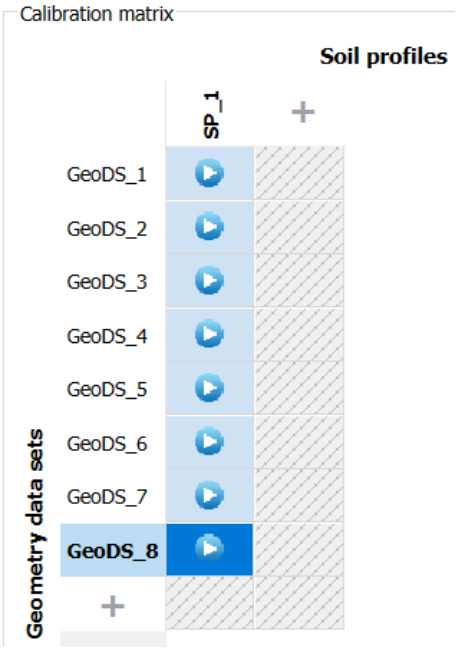


Figure 3–4: Creation of Soil profile and GeoDS

Definition of the soil stratigraphy

First, the soil is defined by following these steps:

1. Make sure that the program is in the **Soil profile** tabsheet.
2. Choose the option *Clay* (default) for material type, and generate the soil layers based on the assumed clay soil profile. Add the needed layers by pressing the *Add* button.

The layer data are provided in Table 3–1 (p. 91).

Table 3–1: Soil layer data

#	z_{top} [m]	z_{bottom} [m]	γ' [kN/m^3]	G_0 [kN/m^2]	$s_{u,top}$ [kN/m^2]	$s_{u,bottom}$ [kN/m^2]	K_0 [-]
1	0	-10	7.5	75.00E3	50	70	1.0
2	-10	-25	8.0	100.00E3	80	95	1.0
3	-25	-40	9.0	120.0E3	110	115	0.9
4	-40	-50	10.0	140.0E3	120	140	0.8

Tip: The input value of the small strain shear stiffness modulus G_0 corresponds to the mid-depth of each soil layer (Figure 3–1 (p. 87)). The values of G_0 at the top and bottom soil layer boundaries are calculated based on the G_0 over s_u (value at mid-depth of each layer) ratio. See 5.2.1 Clay: NGI-ADP material Parameters (p. 118) for more information on the parameters.

On Figure 3–5 (p. 92) all layers added are displayed.

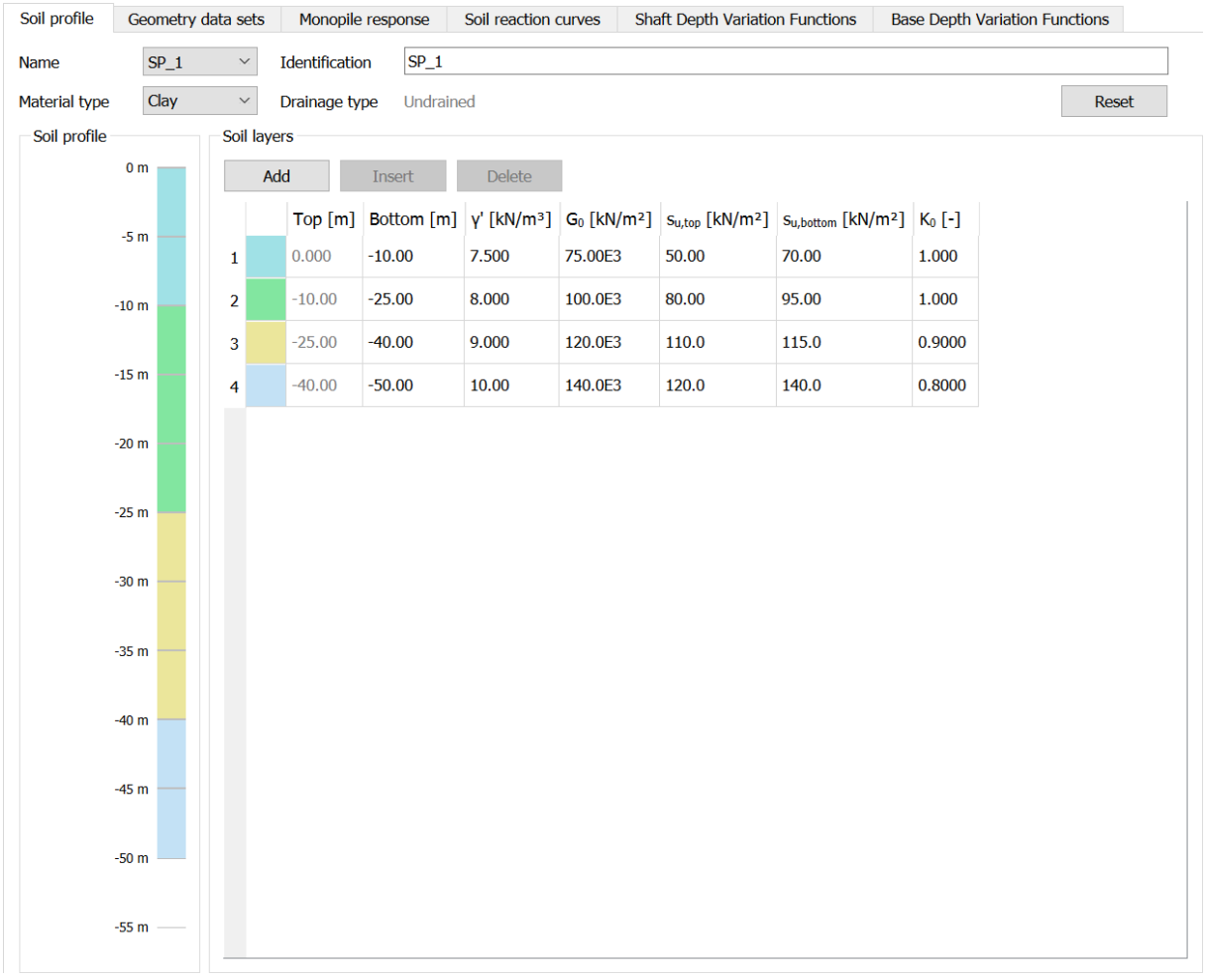


Figure 3–5: Layers generated in the Soil profile tabsheet

Definition of the geometry data sets

The next part is to define some GeoDS by following these steps:

1. Proceed to **Geometry data sets** tabsheet.
2. Add all the needed GeoDS using the *Add* button, one by one. To fill geometrical characteristics use the data presented in Table 3–2 (p. 92).

   To get a calibration with good quality, a lateral ground displacement of about *0.2·D* is needed. See 5.3.1 Generating 3D models (p. 120) for more information on how to estimate the needed value of v_g/D_{out}.

Table 3–2: Geometry data sets

#	*h* [m]	*L* [m]	D_{out} [m]	*t* [m]	v_g/D_{out} [-]
GeoDS_1	25.0	15.0	5.0	0.05	0.2
GeoDS_2	25.0	25.0	5.0	0.05	0.2
GeoDS_3	100.0	15.0	5.0	0.05	0.2
GeoDS_4	100.0	25.0	5.0	0.05	0.2
GeoDS_5	25.0	21.0	7.0	0.07	0.2

#	h [m]	L [m]	D_{out} [m]	t [m]	v_g/D_{out} [-]
GeoDS_6	25.0	35.0	7.0	0.07	0.2
GeoDS_7	100.0	21.0	7.0	0.07	0.2
GeoDS_8	100.0	35.0	7.0	0.07	0.2

The definition of the parameters in Table 3–2 (p. 92) can be found in 2.3.5 Geometry datasets tabsheet (p. 42).

> **Note:** This action automatically saves the project each time a new GeoDS is added to the list.

The added GeoDS are shown in Figure 3–6 (p. 93).

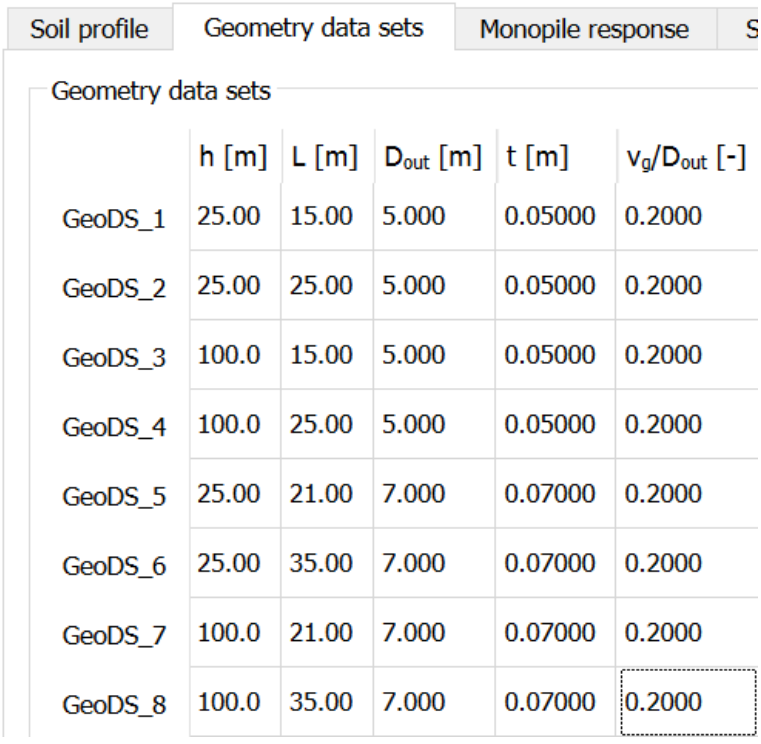


Figure 3–6: GeoDS parameters generated in the Geometry data sets tabsheet

3. The default values are used for the 2.4.2 Structural properties (p. 59), i.e. the Young's modulus E and the Poisson's ratio v of the plate material are $E = 210 \cdot 10^6$ kN/m^2 and $v = 0.3$.

4. In the Calibration Matrix select the first GeoDS and generate the PLAXIS 3D model by clicking the *Generate* button.

> **Note:** Note that this action automatically saves the project. The project is saved after each generation is completed.

> **Tip:** When the *Generate* button is pressed, PLAXIS Monopile Designer automatically verifies the value of 0.2 for all selected GeoDS models. If a value is specified outside the recommended range, a warning will be displayed suggesting appropriate values.

> **Tip:** If users want to perform changes in the default settings of the PLAXIS 3D models, then they are advised to:
>
> - Generate the first model (GeoDS_1),
> - Make the needed changes within PLAXIS 3D
> - Subsequently, add all the other GeoDS
>
> The addition of a GeoDS copies the last project, including all the user-modified parameters. To adjust the geometry of the newly added GeoDS, the *Generate* button should be used, which triggers regeneration of all selected PLAXIS 3D models, based on the input geometrical characteristics.
>
> Note that the regeneration process maintains all valid manual changes, as described in 2.3.5 Geometry datasets tabsheet (p. 42). However, be aware that any manual modifications apart from the suggested ones, might affect the calculation of the results and the validity of the parameterisation procedure.

5. Open the model in PLAXIS 3D by clicking the *View* button and ensure that the generated soil profile consists of four soil layers with the correct top and bottom boundaries.

   Additionally, the validity of the generated material properties may be checked against the formulations provided in 5.2.1 Clay: NGI-ADP material Parameters (p. 118).

   The geometry of the monopile and the parameters assigned to the plate material may be checked too for consistency.

   Check the structure and quality of the generated mesh in PLAXIS Monopile Designer Output by previewing Phase_1 ("Pile wished in place"). This can be done via the mesh quality metrics available under the *Mesh* menu item.

   Close PLAXIS Monopile Designer Output and Input after completing the checks suggested above. Note that there is no need to save the PLAXIS 3D project as no modifications were done.

6. Multi-select all the remaining GeoDS (i.e. GeoDS_2 to GeoDS_8), and generate the PLAXIS 3D models by clicking the *Generate* button.

7. Multi-select all eight GeoDS and press the *Calculate* button. The calculations are performed sequentially.

   > **Note:** Note that this action automatically saves the project after each calculation finishes.

Note: The calculations may take a long time (several hours) to finish.

8. The monopile response of all calculated GeoDS may be inspected in the right side panel, under the tab *Monopile Response*. Selecting a specific GeoDS in the menu highlights the corresponding curves in the graphs.

9. Focus on the "lateral reaction force against lateral displacement at mudline" graphs (two top graphs, see Figure 3–7 (p. 96)) for small and large displacements. Double-click the graph for "large displacements" (the right graph) and inspect the values more accurately in the pop-up window. Select the *Table* tab to extract (copy-paste) all data.

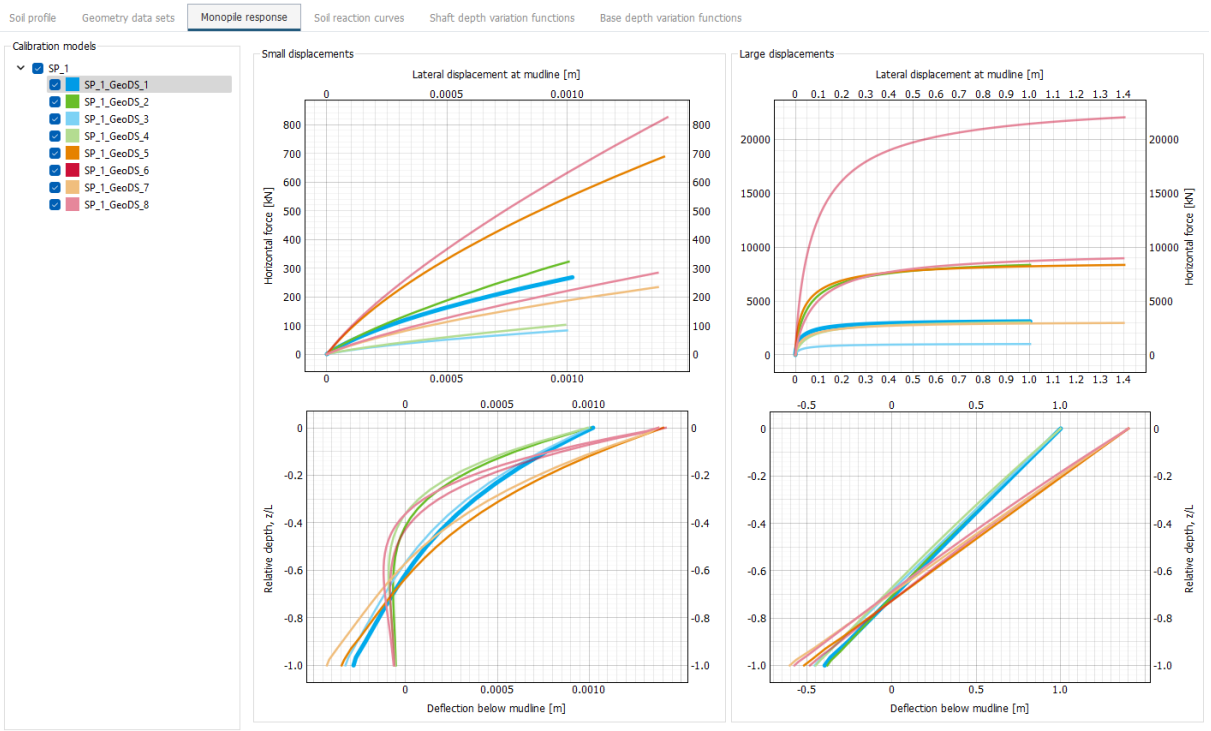


Figure 3–7: Monopile results tab - displacement

> **Tip:** Detailed 3D results can be inspected directly in PLAXIS Monopile Designer Output by selecting the calculated GeoDS and pressing the *View* button.

10. Focus on the "deflection below mudline" graphs (two bottom graphs, see Figure 3–7 (p. 96)) for small and large displacement to ensure that the lateral displacement at zero depth (at mudline, where z/L=0) is about D/10000 and D/5 respectively.

> **Note:** In this example the pile is pushed up to D/5 to ensure that the failure mechanism is fully developed.

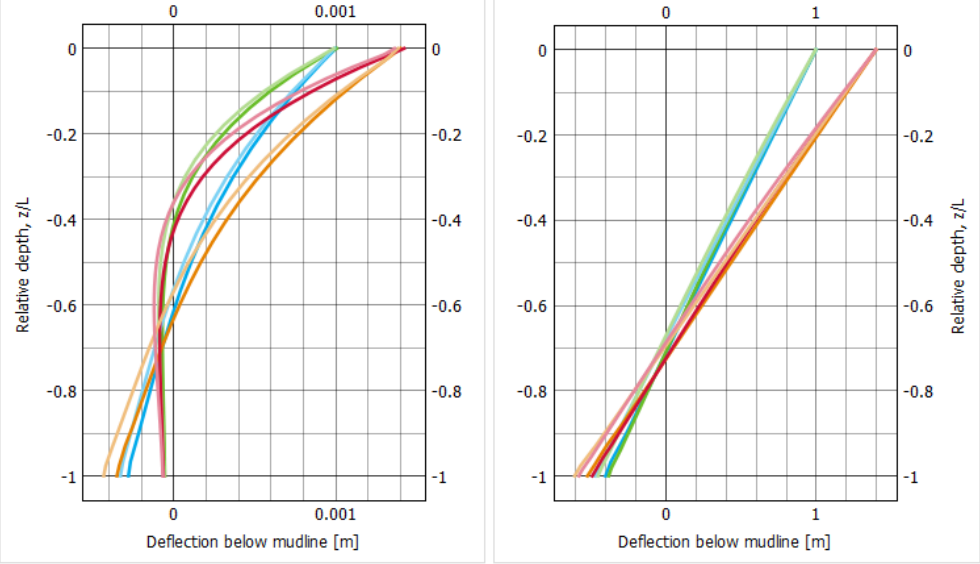


Figure 3–8: Monopile results tab - deflection

11. Select all eight GeoDS and press the *Parameterise* button. The soil reaction curves from all selected models are taken into account in the calibration process. This procedure results in the generation of the file `sp_1_calibrated.dvf` within the project's folder.

> **Note:** The project has to be saved manually.

12. Inspect the variation of the soil reaction fitting parameters along the depth as presented in the tabs *Shaft depth variation functions* and *Base depth variation functions*.

3.2 Verification of the calibration procedure

After all geometry sets have been generated, calculated and parameterised, the resulting depth variation functions can be analysed.

Analysis mode

For the analysis do the following:

1. Make sure that the program is in the **Analysis mode**.
2. In the *Depth variation functions* section, the file `sp_1_calibrated.dvf` that was created and saved during calibration is selected by default. Leave this selection as it is.
3. In the *Soil layers* section, enter a soil layer with *Bottom* at -50 m. Select the created Soil reaction curve as the default.
4. In the *Monopile geometry* section, enter the values of the geometric parameters (*Monopile head* = *h*, *L*, D_{out}) which correspond to GeoDS_1, see Table 3–2 (p. 92). Also you can inspect the table tab for data on the relation between load and displacement at the mudline.
5. Asign as a calculation type the *Workload* option. Subsequently, enter a value for the horizontal force at the top of the monopile (*H*). This value is equal to, or exceeds, the maximum reached lateral reaction force at mudline of the PLAXIS 3D model, which corresponds to the GeoDS_1 (see the *Monopile response* tab in the **Calibration mode**). For the 3D Model example, a value of 3200 kN should be used.
6. On the *Segments* tab add a pile segment by clicking the *Add* button. Enter the thickness value *t* that corresponds to GeoDS_1, see Table 3–2 (p. 92).
7. Start the *1D analysis* by clicking on the *Static* button.

> **Note:** The Static calculation takes only a few seconds.

> **Note:** In the *Expert settings* tab. Use the default values, see 2.4.8 Expert settings (p. 68) .

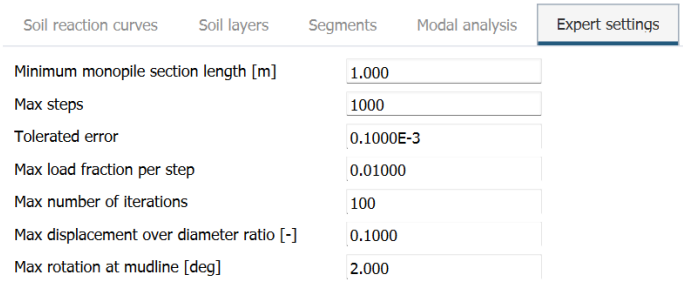


Figure 3–9: Expert settings - default values

8. In the 3D design verification section, *Generate* and *Calculate* the verification model to compare 1D and 3D results.

Results mode

To compare and verify the results, carry out the following steps:

1. Proceed to **Results mode**.
2. Select the Design verification checkbox.
3. Compare the results of the 1D and the selected 3D model by inspecting the *H-v* graph at mudline. See Figure 3–10 (p. 99).

   The high value (96.33%) of the accuracy metric (*η*) indicates that the calibration of the 1D model was done successfully via the parameterisation procedure.

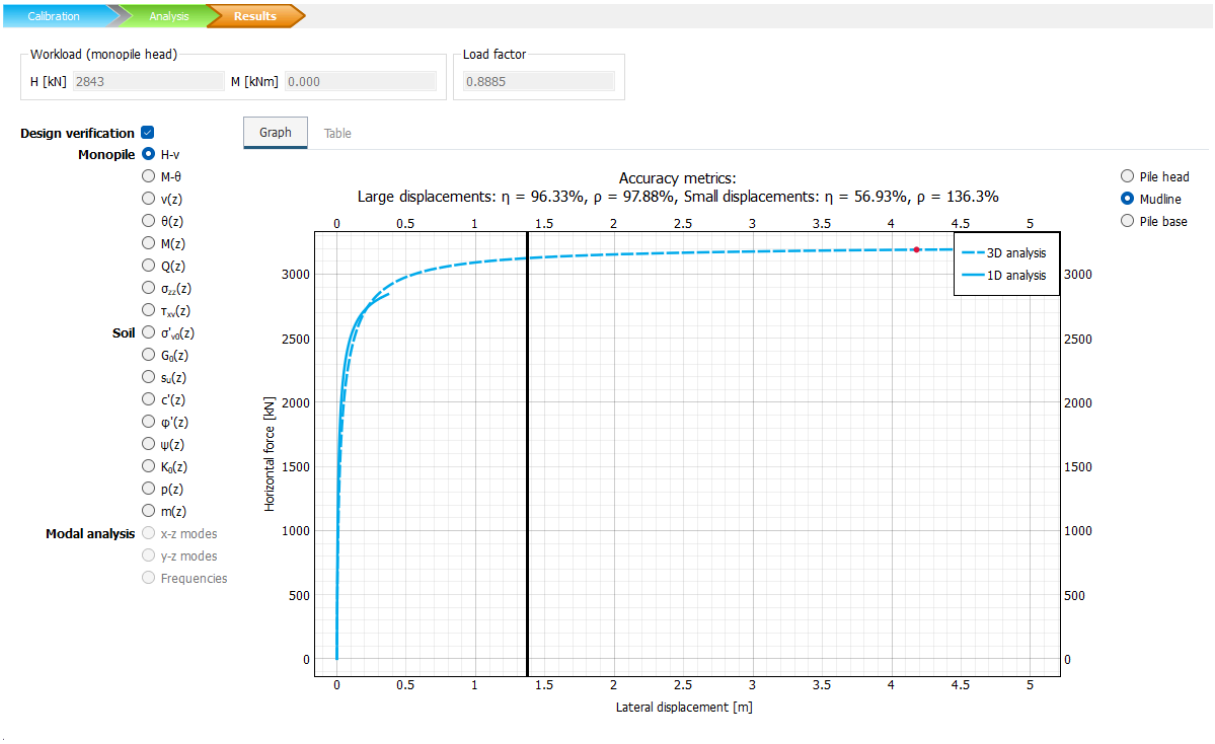


Figure 3–10: Comparison of 1D and 3D model results

> **Note:** By default for the 1D analysis, the target lateral displacement at ground level equals *0.1·D* (see *Expert settings* tab of the **Analysis mode**). The accuracy metric *η* is calculated based on that value. See 2.5.4 Accuracy metrics η and ρ (p. 84) for more information on how the accuracy metric *η* is computed.

> **Tip:** The user may also compare the response of the 1D and 3D models by inspecting the *M-θ*, *v(z)* and *θ(z)* graphs. Note that the *v(z)* and *θ(z)* plots are comparable only if the *maximum displacement over diameter ratio* parameter under the *Expert settings* tab (**Analysis mode**) is selected such that the achieved lateral displacement at mudline is approximately equal to the one of the 3D model. In this case, the applied workload *H* should be high enough in order to obtain the target displacement at ground level.

3.3 Final Design

In the present tutorial, a specific geometry is assumed to represent the final design, based on the defined design criteria. The user is advised to try different geometries as well in order to get familiar with the design tool.

> **Note:**
>
> Only geometrical configurations that fall within the assumed design space are recommended to be analysed.

Analysis mode

1. In the **Analysis mode** enter the geometric characteristics corresponding to the assumed final monopile design that meets the required design criteria. See Table 3–3 (p. 100).

   Table 3–3: Values of the final monopile design

#	h [m]	L [m]	D_{out} [m]	t [m]
1	60.0	20.0	6.0	0.05

   The assumed design load (*H*) for this tutorial is *3000* kN. This load corresponds to a bending moment at ground level (M_g) of $180.0 \cdot 10^3$ kNm , assuming that the load is applied at height (*h*) above mudline, equal to *60.0* m.

   > **Note:** The final design may consist of more than one pile segment with different thicknesses. This can be done by adding more monopile segment sections by clicking the *Add* button on the *Segments* tab in the **Analysis mode**.

2. Run the *1D analysis* by selecting the *Static* option.
3. Go to **Results mode** and inspect the *H-v* graph (at mudline) to ensure that the displacement is less than *0.1·D* when the design load *H* is applied (see Figure 3–11 (p. 101)).

   The design criterion is met, and the selected monopile design is adopted as the final one for the specified soil conditions and design space.

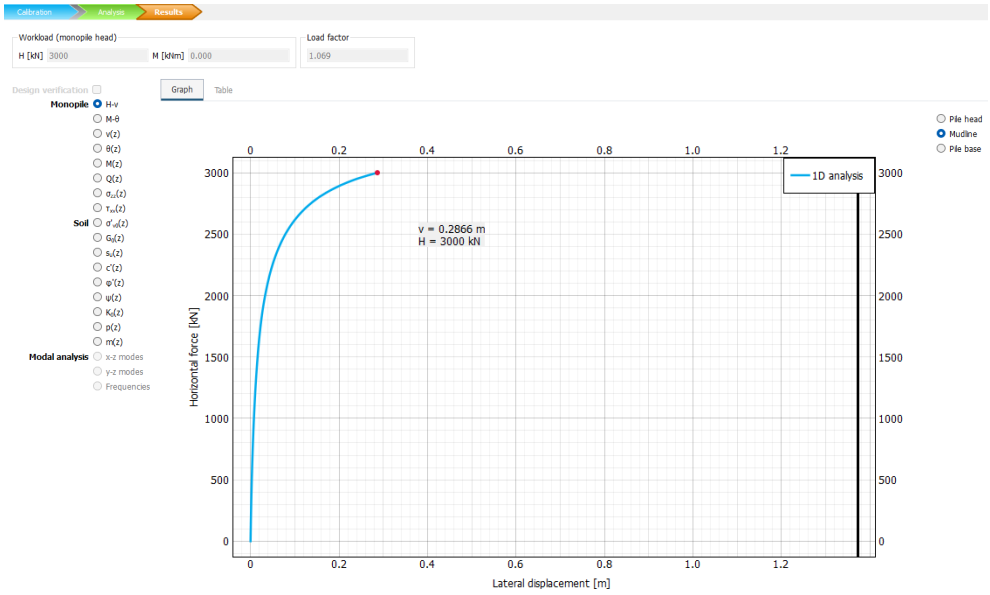


Figure 3–11: Check displacement at design load

3.4 Verification of the final design

Analysis mode

After determining the final monopile design, it is highly recommended that the results of the 1D analysis are validated against the results of a 3D FE model with the same geometrical and mechanical characteristics.

1. In the **Analysis mode**, *Generate* a 3D design verification model. This will automatically create a PLAXIS 3D model with the same monopile geometry, structural properties, thickness variation, workload, and soil layers as specified for the 1D analysis.
2. Calculate the 3D design verification model.

Results mode

In the **Results mode**, select the *Design verification* checkbox and compare the results between the 1D model and the PLAXIS 3D model. The user may also inspect all the graphs mentioned above. A very good match is achieved between the 1D and 3D results indicating a successfully validated design procedure. See Figure 3–12 (p. 102) for the comparison in the *H-v* graph at mudline.

Notice the high value of the accuracy metric (η) for large displacement (95.20%) , which indicates a good match. In the case of small displacement the value η obtained is 65.03%.

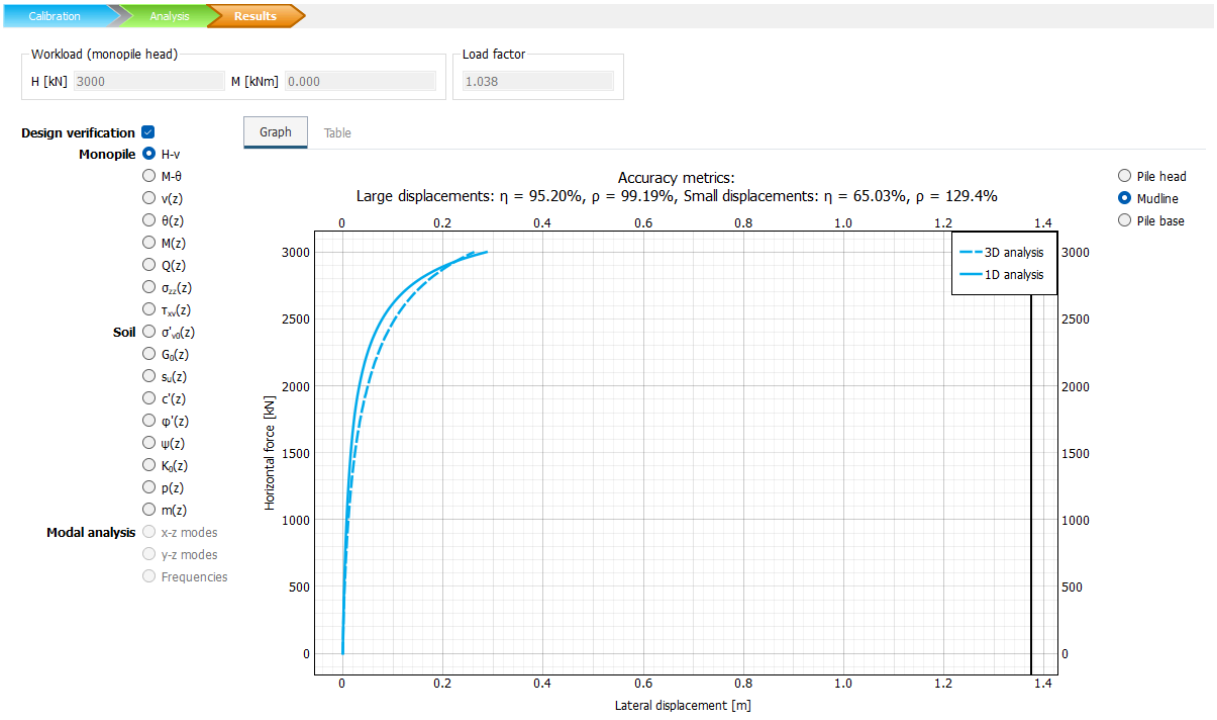


Figure 3–12: Compare 1D and 3D model results for final design

4

Tutorial 2 - Layered Soils

This tutorial is built in two parts:

- 4.1 Rule based-design (p. 105) uses the published depth variation functions for the Cowden till model and the general Dunkirk sand model .
- 4.2 Numerical-based design (p. 112) calibrates site-specific depth variation functions from 3D FEM analyses .

The soil profile, monopile geometry, and workload are the same in both parts. Thus, this tutorial will demonstrate the differences between rule-based design (RBD) and numerical-based design (NBD) in layered soil.

The location consists of two interbedded soil units, a stiff clay and a very dense sand (RD = 90%).

Table 4–1: Layered soil profile

Top(m)	Bottom (m)	Soil unit
0.0	-3.0	Clay
-3.0	-6.0	Sand
-6.0	-12.5	Clay
-12.5	-22.5	Sand
-22.5	-50.0	Clay

For simplicity, each of the soil units is modelled as a homogeneous single-layer profile, with the following depth variation profiles for the small strain shear modulus, G_0.

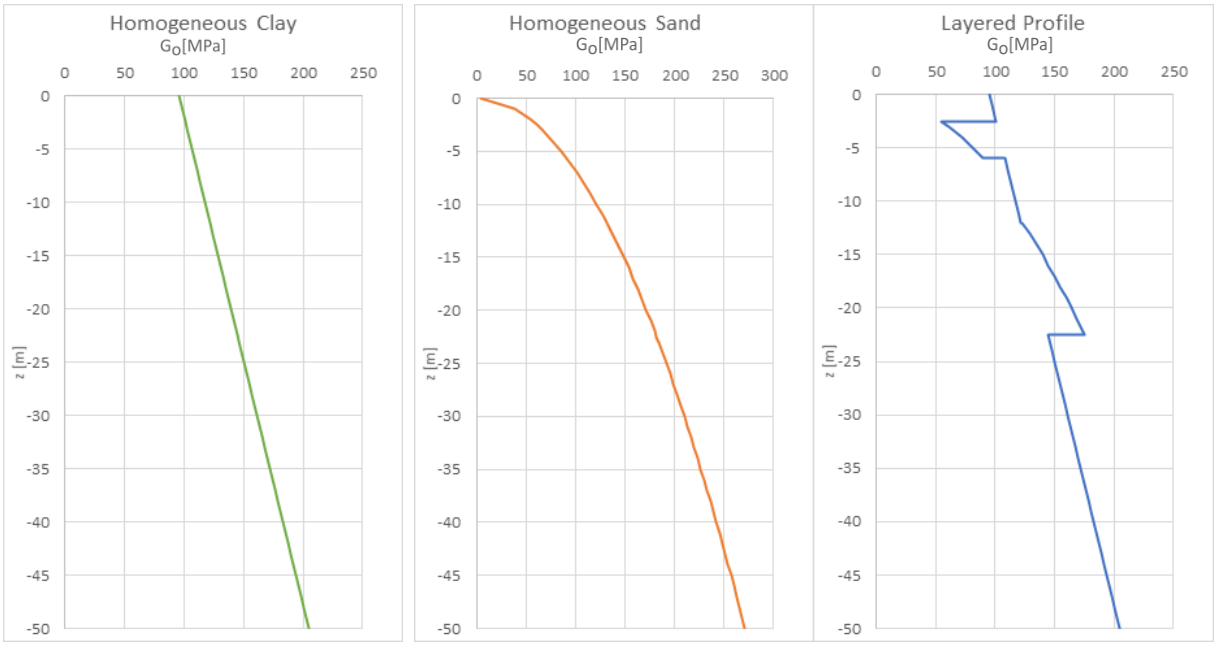


Figure 4–1: Homogeneous and layered shear stiffness profiles

4.1 Rule based-design

In the first part of the tutorial, the layered soil profile will be modelled using the published rule-based models.

- The stiff clay is modelled with the Cowden till model (Byrne et al., 2020).
- The dense sand is modelled with the general Dunkirk sand model, or GDSM (Burd et al., 2020a), particularised for a high value of the relative density, RD = 90%.

> **Note:** Cowden till and Dunkirk sand models are already included in the **Analysis Mode** of PLAXIS Monopile Designer. However, depth variation functions can be constructed manually for future models not incorporated yet in PLAXIS.

4.1.1 Analysis - Layered profile

To proceed with the analysis do the following:

1. Start PLAXIS Monopile Designer.
2. In the Quick select dialogue (Figure 3–3 (p. 90)) choose *Start a new project* and save it as 'RBD Layered.plxmdt'.
3. Advance to the **Analysis mode**.
4. If constructed manually, in the *Soil reaction curves* tab, select *Import*... and navigate, to the respective clay and sand RD 90 dvf files (e.g . 'rule-based Clay.dvf' and 'rule-based Sand RD90.dvf'.)

> **Tip:**
> - It is possible to multi-select files to import manually constructed soil reaction curves.
> - If desired the already included Cowden and Dunkirk soil reaction curves can be used directly on the Soil layers tab of the **Analysis mode**.

5. In the *Soil layers* tab, assign each dvf file to its corresponding layers (see Figure 4–2 (p. 106)), according to Table 4–2 (p. 106).

Table 4–2: DVF files for layers

Top [m]	Bottom [m]	Soil unit	DVF file
0.0	-3.0	Clay	rule-based Clay.dvf
-3.0	-6.0	Sand	rule-based Sand RD90.dvf
-6.0	-12.5	Clay	rule-based Clay.dvf
-12.5	-22.5	Sand	rule-based Sand RD90.dvf
-22.5	-50.0	Clay	rule-based Clay.dvf

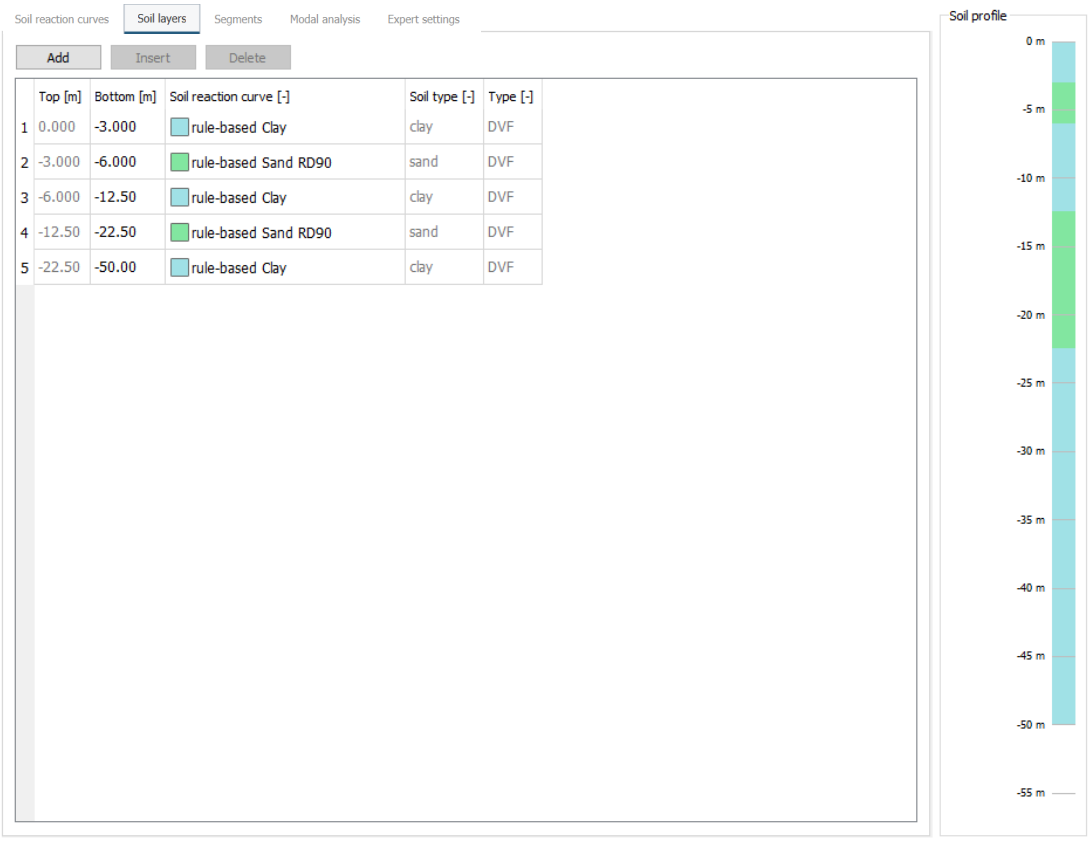


Figure 4–2: Soil layers for rule-based design (based on imported dvf files)

On the other hand:

If the included PLAXIS rule-based models are used (i.e. models not manually created), it is necessary to insert on the interface the material properties for each layer. To do this **Click** on the soil reaction curve assigned for each layer; the *Material parameters* window will be displayed as shown in Figure 4–3 (p. 107).

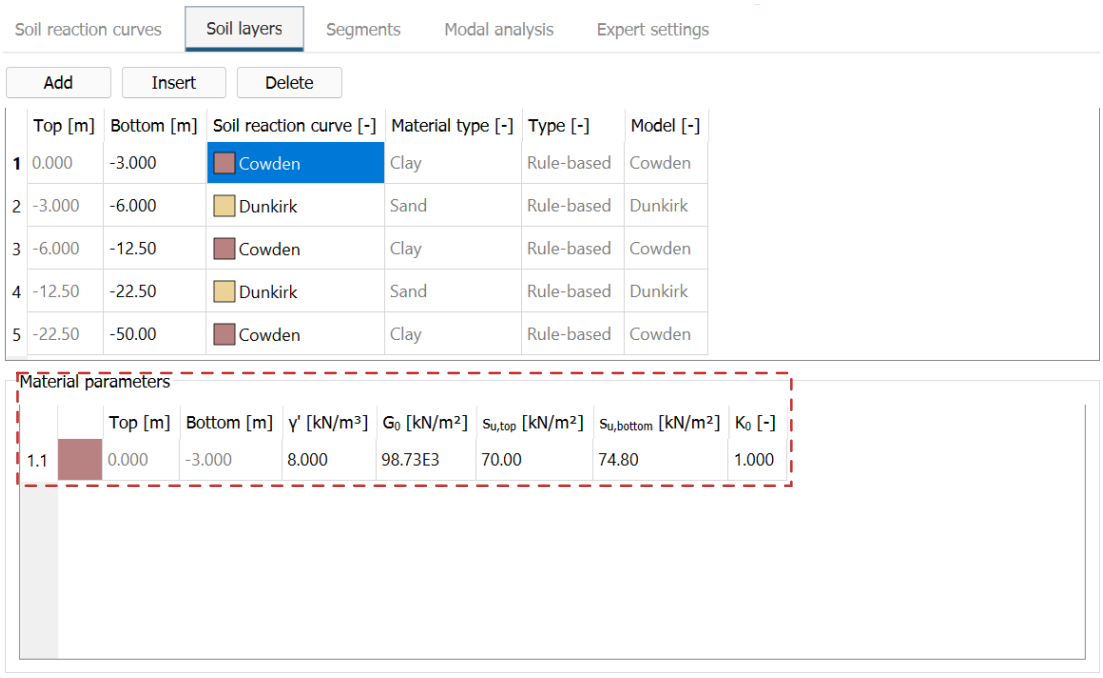


Figure 4–3: Material Parameters window for soil reaction curves included in PLAXIS

For the current example, the material parameters for the layers of Cowden clay and Dunkirk sand are displayed on Table 4–3 (p. 107). Notice that the values of G_0, $s_{u,top}$, $s_{u,bottom}$ vary with depth.

Table 4–3: Material parameters of Cowden clay and Dunkirk sand

Cowden clay						
Top	**Bottom**	**γ_{sub} [kN/m^3]**	**G_0 (kN/m^2)**	**$s_{u,top}$ (kN/m^2)**	**$s_{u,bottom}$ (kN/m^2)**	**K_0**
0	-3	8	98727	70	74.8	1
-6	-12.5	8	115640	79.6	90	1
-22.5	-50	8	174550	106	150	1

Dunkirk sand									
Top	**Bottom**	**γ_{sub} [kN/m^3]**	**G_0 (kN/m^2)**	**c' (kN/m^2)**	**m [-]**	**φ [deg]**	**ψ [deg]**	**K_0**	**Relative density (%)**
-3	-6	10	75777	0.1	0.5	39	9.0	1	90
-12.5	-22.5	10	151370	0.1	0.5	39	9.0	1	90

6. In the *Monopile geometry* and *Structural properties* area, define the geometry and material parameters of the final design. Assumed:

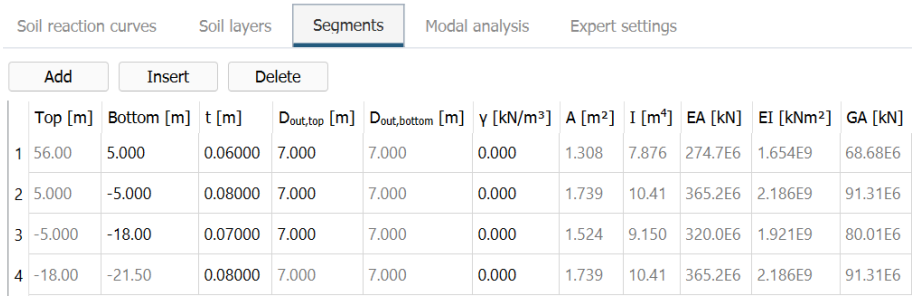
Table 4–4: Geometry and structural properties

	Monopile head [m]	L [m]	D_{out} [m]	γ [kN/m^3]	E [kN/m^2]	v [-]
Final Design	56.0	21.5	7.0	0.0	210.0E6	0.0

7. In the *Segments* section, input the thickness segments of the final design. A simple pattern is assumed:

Table 4–5: Segments - Thickness variation

Top [m]	Bottom [m]	t [m]
56.0	5.0	0.06
5.0	-5.0	0.08
-5.0	-18.0	0.07
-18.0	-21.5	0.08



	Top [m]	Bottom [m]	t [m]	$D_{out,top}$ [m]	$D_{out,bottom}$ [m]	γ [kN/m³]	A [m²]	I [m⁴]	EA [kN]	EI [kNm²]	GA [kN]
1	56.00	5.000	0.06000	7.000	7.000	0.000	1.308	7.876	274.7E6	1.654E9	68.68E6
2	5.000	-5.000	0.08000	7.000	7.000	0.000	1.739	10.41	365.2E6	2.186E9	91.31E6
3	-5.000	-18.00	0.07000	7.000	7.000	0.000	1.524	9.150	320.0E6	1.921E9	80.01E6
4	-18.00	-21.50	0.08000	7.000	7.000	0.000	1.739	10.41	365.2E6	2.186E9	91.31E6

Figure 4–4: Thickness variation of segments (t)

> **Note:**
>
> For this example, it is assumed that the monopile diameter D_{out} is constant for all segments. Only the first segment can be modified seenotes of 2.4.6 Pile segments and segment properties (p. 63).

8. In *Calculation type > Workload*, input the resultant horizontal load and moment. In this example, a single load case is considered corresponding to a horizontal load in the head of the monopile H = 10 MN.

9. Under *1D analysis*, select *Static*.

10. Advance to the **Results mode** to inspect the output of the analysis.

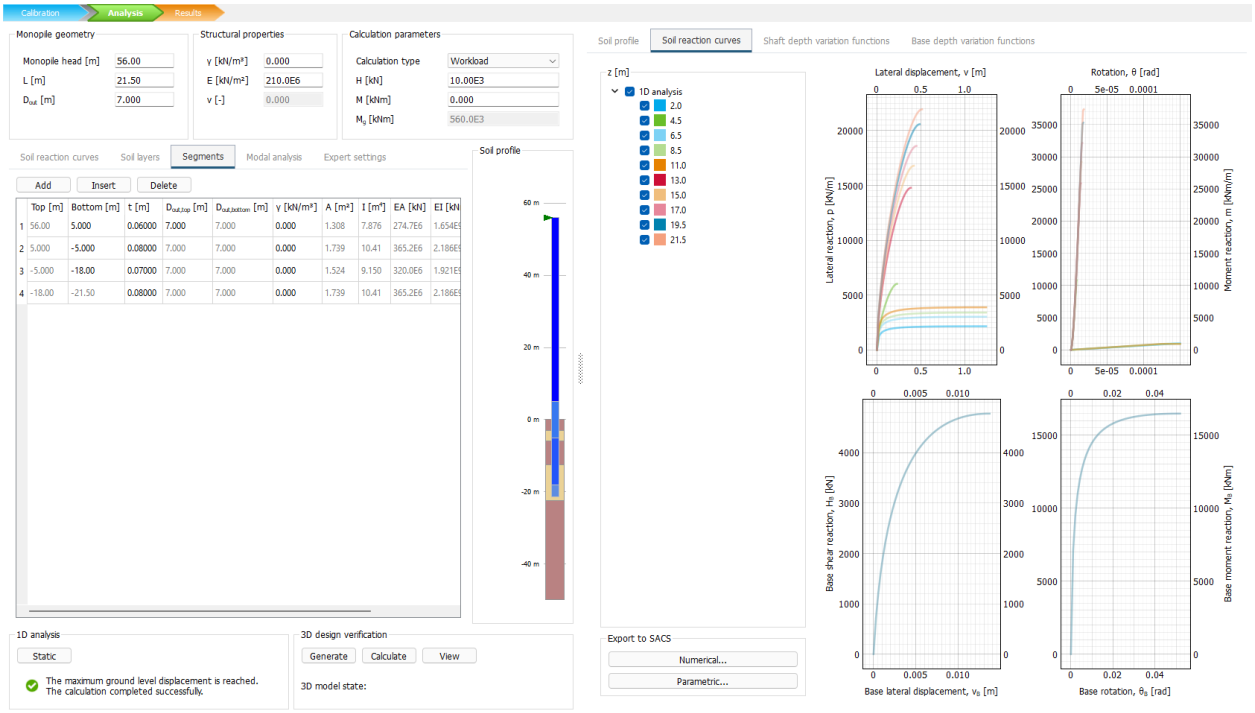


Figure 4–5: Soil reaction curves from rule-based analysis

4.1.2 Design verification

1. Return to the **Analysis mode**.
2. Run the *3D Design verification*:
   a. Click on *Generate* and wait until the PLAXIS 3D model is completed.
   b. Click on *Calculate*.

> **Note:** A warning message about the mesh quality because of the distance between the toe of the monopile and the closest soil boundary is small might appear. For this case, this message is ignored and the calculation the generation of the PLAXIS 3D Model is allowed.

3. In the **Results mode**, select the *Design verification* checkbox to compare the results between the rule-based 1D analysis and the 3D FEM model.

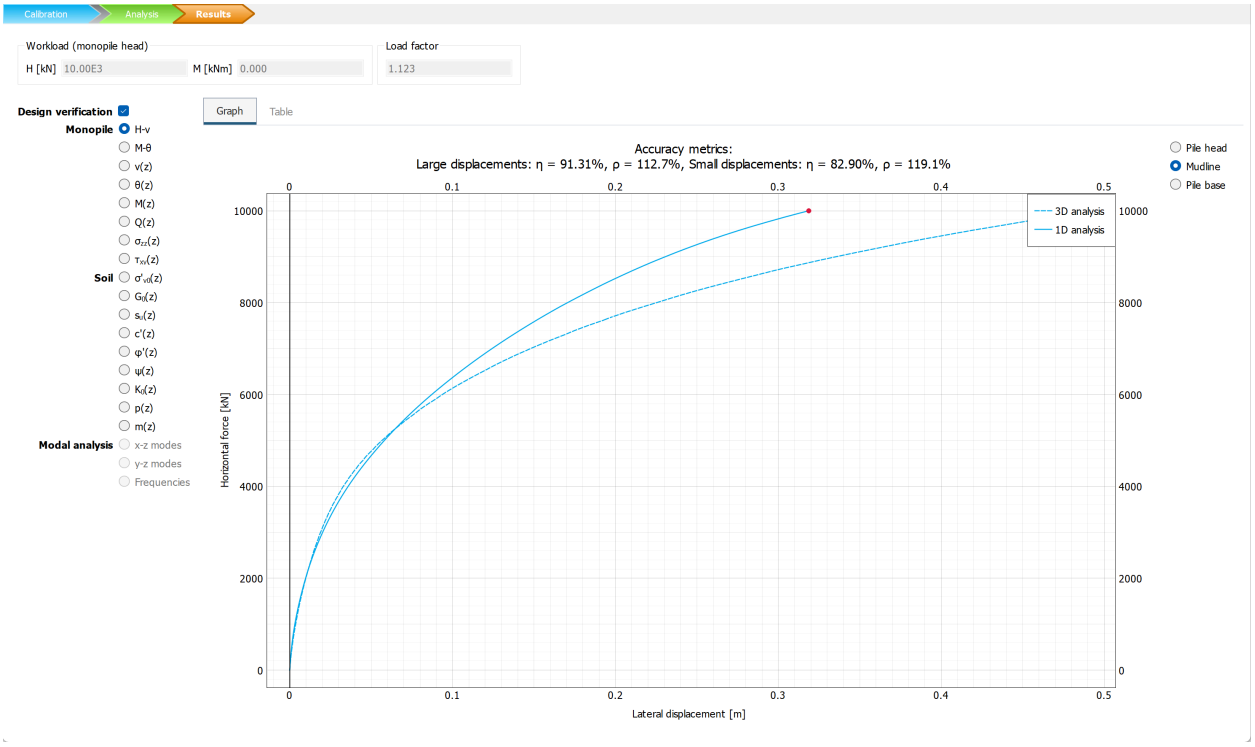


Figure 4–6: Comparison between 1D rule-based analysis and 3D model

The value of eta is acceptable (η = 91.31%); the value of rho is slightly beyond desirable (ρ = 112.7%). This indicates the need to use the results of the rule-based method with caution.

4.2 Numerical-based design

In the numerical-based design, the depth variation coefficients are obtained from the calibration of the homogeneous soil units in 3D FEM.

A narrower design space than that of RBD is assumed. The design space consists of many monopile geometries (models) that define an envelope in which the optimum monopile design is expected to lie.

Table 4–6: Design space for numerical-based design

	h [m]	L [m]	D_{out} [m]	t [m]
GeoDS_1	20.0	20.0	5.0	0.050
GeoDS_2	75.0	25.0	5.0	0.050
GeoDS_3	75.0	25.0	8.0	0.080
GeoDS_4	35.0	35.0	8.0	0.080
GeoDS_5	60.0	20.0	5.0	0.050
GeoDS_6	25.0	25.0	5.0	0.050
GeoDS_7	25.0	25.0	8.0	0.080
GeoDS_8	105.0	35.0	8.0	0.080

The same design space will be used in the calibration of the clay and the sand units. The value of the target displacement, v_g/D_{out} will be set to 0.15 for all clay and sand models.

4.2.1 Calibration – Layered soil profile

1. From the *File* menu, create a *New project*. Save it as 'NBD Layered.plxmdt'.
2. Start in the **Calibration mode**. Make sure that Soil type is set to Clay.
3. In the 2.3.3 Calibration Matrix (p. 31) add two soil profiles and eight datasets.
4. Go to the **Soil profile** tabsheet.
5. Select a material for each soil profile from the *Material type* drop down list. Assign *Clay* to the first soil profile and *Sand* for the second. Adding an *Identification* is optional.
6. *Add* one Layer to each soil profile.
7. For each soil profile, input the material parameters for each created layer (see Table 4–7 (p. 112) and Table 4–8 (p. 113)).

Table 4–7: Material parameter for the Clay Soil Profile

Layer	Top [m]	Bottom [m]	γ' [kN/m^3]	G_0 [kN/m^2]	$s_{u,top}$ [kN/m^2]	$s_{u,bottom}$ [kN/m^2]	K_0 [-]
1	0.00	-50.0	8.0	150.0E3	70.0	150.0	1.0

Table 4–8: Material parameter for the Sand Soil Profile

Layer	Top [m]	Bottom [m]	γ' [kN/ m^3]	G_0 [kN/ m^2]	c' [kN/ m^2]	power (m)[-]	φ' [deg]	ψ [deg]	K_0 [-]
1	0.00	-50.00	10.0	191.6E3	0.1	0.5	39.0	9.0	1.0

8. Advance to the *Calibration data sets*.
9. Input the properties for the eight GeoDS according to Table 4–6 (p. 112). Set all values of v_g/D_{out} = 0.15.
10. Multi-select all eight GeoDS and press *Generate*. This operation will take a few minutes.
11. Multi-select all eight GeoDS and press *Calculate*. This operation may take a long time.
12. Inspect the results of the calibration in the *Results inspection pane*.

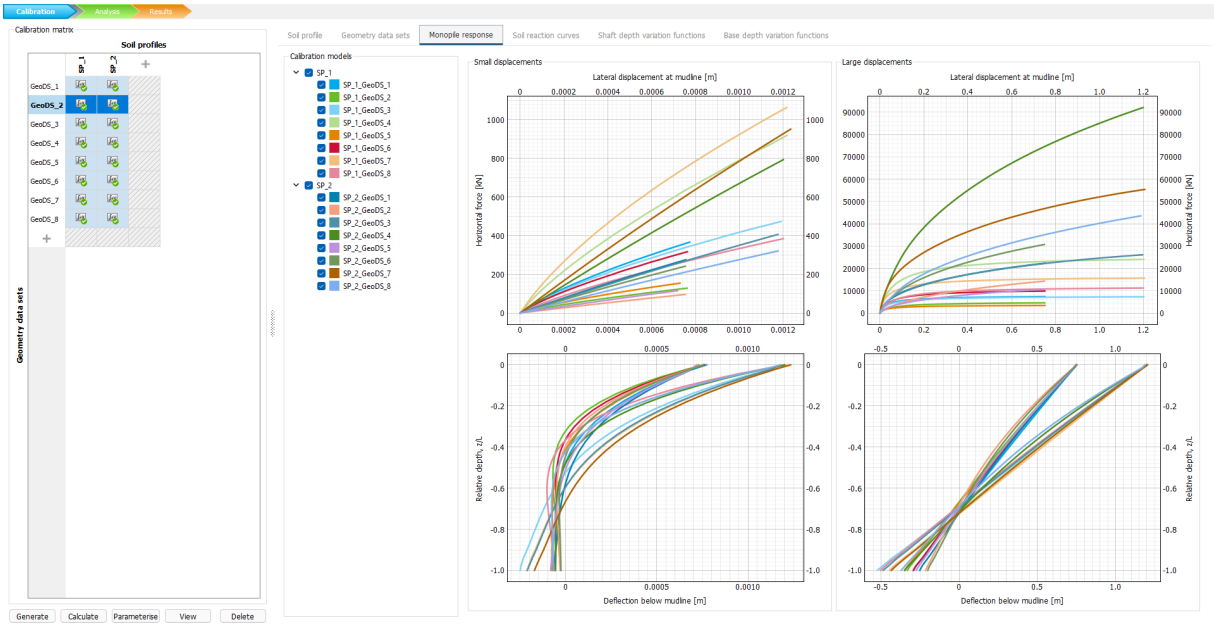


Figure 4–7: Numerical-based calibration for Clay(Sp_1) and Sand(SP_2) units

> **Note:**
>
> - Calibration results in the form of curves can be turn on or turn off in the *Calibration models* sub-window by clicking on the corresponding Soil Profile-GeoDS combination (see Figure 67).

13. Multi-select all 8 GeoDS and press *Parameterise*. This will automatically generate the files 'sp_1_calibrated.dvf' (for the clay unit) and sp_2_calibrated.dvf (for the sand unit) on the project folder and import it into the **Analysis mode**.

4.2.2 | Analysis – Layered profile

1. Advance to the **Analysis mode**.
2. In the *Soil reaction curves* tab, select *Import...* and navigate to the 'sp_1_calibrated.dvf' (clay unit) and 'sp_2_calibrated.dvf' (sand unit) files.

   > **Tip:** It is possible to multi-select when importing.

3. In the *Soil layers* tab, assign each dvf file to its corresponding layers, according to Table 4–9 (p. 114).

   Table 4–9: Soil layers and dvf files

Top [m]	Bottom [m]	Soil unit	DVF file
0.0	-3.0	Clay	sp_1_calibrated.dvf
-3.0	-6.0	Sand	sp_2_calibrated.dvf
-6.0	-12.5	Clay	sp_1_calibrated.dvf
-12.5	-22.5	Sand	sp_2_calibrated.dvf
-22.5	-50.0	Clay	sp_1_calibrated.dvf

4. In *Monopile geometry* and *Structural properties*, define the geometry and material parameters of the final design. Assumed values are provided in Table 4–4 (p. 109).
5. In the *Segments* section, input the thickness segments of the final design. A simple pattern is assumed (Table 4–5 (p. 109)).

   > **Note:**
   > For this example, it is assumed that the monopile diameter D_{out} is constant for all segments.

6. In **Calculation parameters** > **Workload**, input the resultant horizontal load and moment. In this example, a single load case is considered corresponding to a horizontal load H = 10 MN applied at a Monopile head = 56 m.
7. To run the *1D analysis*, select the *Static* option.
8. Advance to the **Results mode** to inspect the output of the analysis.

4.2.3 Design verification

1. Return to the **Analysis mode**.
2. Under 3D *Design verification*, generate and calculate the design verification model.

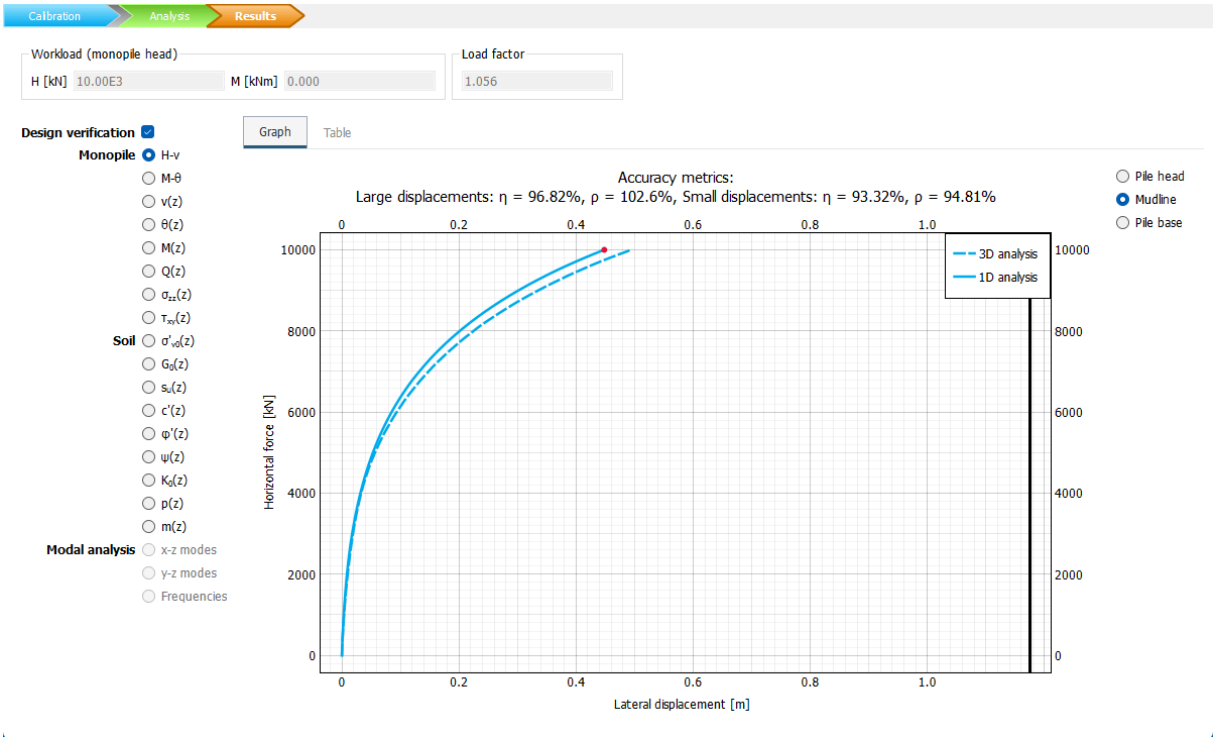


Figure 4–8: Comparison between 1D numerical-based analysis and 3D model

3. In the **Results mode**, select the *Design verification* checkbox to compare the results between the rule-based 1D analysis and the 3D FEM model.

The value of eta is much better than before (η = 96.82%), while the value of rho is now excellent (ρ = 102.6%). These metrics indicate that a better approximation is achieved by the numerical-based method.

4.3 Comparison between rule-based and numerical-based design

To provide a clear comparison between the two methods, the predicted monopile behaviour curves are exported and plotted in a spreadsheet together with the predictions of the 3D FEM:

1. On the 'NBD Layered.plxmdt' project, make sure you are in the **Results mode** and the *Design verification* checkbox is selected.
2. Select the *H-v* plot and open the *Table* tab.
3. Right-click on the results table and select *Copy All*.
4. Paste the results table on a spreadsheet. Rename the *1D analysis* cells to 'NBD' .
5. From the *File* menu, open 'RBD Layered.plxmdt' and advance to the **Results mode**.

6. Select the *H-v* plot and open the *Table* tab.
7. Right-click on the results table and select *Copy All*.

> **Note:** The design verification model does not depend on the input depth variation functions, but only on the material parameters of each layer. Thus, design verification results from 'RBD Layered.plxmdt' and 'NBD Layered.plxmdt' should match.

> **Tip:** If the *Design verification* checkbox is unselected, *Table* will only display the results of the 1D analysis.

8. Paste the results table to the spreadsheet. Rename the *1D analysis* cells to 'RBD'.
9. Plot all 3 series (*RBD*, *NBD*, and *3D analysis*) on the same X-Y graph.

The differences between the predictions of the three methods are now apparent. A similar comparison can be plotted for the M-θ curves.

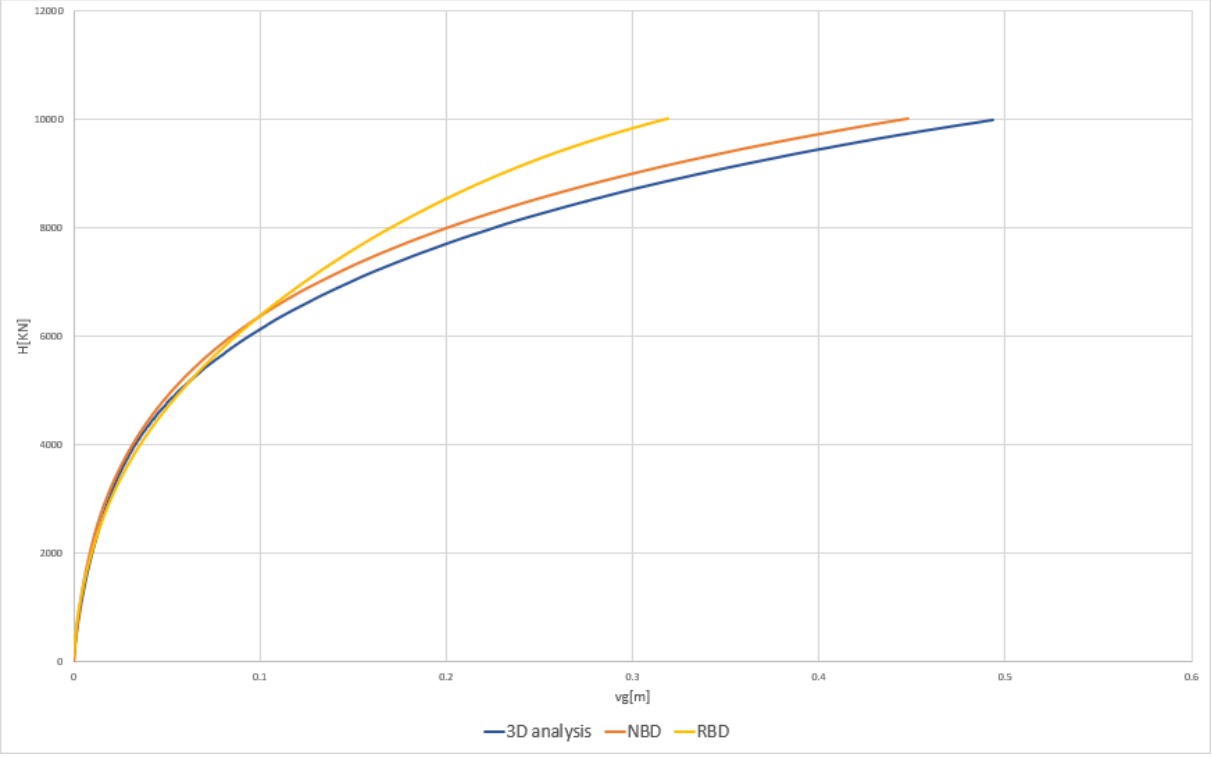


Figure 4–9: Monopile behaviour predictions from rule-based, numerical-based, and 3D analysis

5

Scientific Manual

5.1 Introduction

This part of the manual describes the technical basis of PLAXIS Monopile Designer. The material models that are used, the algorithm and modules, as well as assumptions and limitations, are explained here.

> **Note:**
>
> More references to publications on the use of PLAXIS Monopile Designer,PLAXIS and related background information can be found on the Research archive of the TUDelft webpage.

5.2 Material Models

The following predefined soil types may be used in PLAXIS Monopile Designer:

- Clay: modelled via the NGI-ADP constitutive soil model.
- Sand: modelled via the Hardening Soil small-strain (HSsmall) constitutive soil model.

The calculation of the material model parameters is explained in this chapter. For further detail on the material models, please see Brinkgreve et al. (2021).

5.2.1 Clay: NGI-ADP material Parameters

The NGI-ADP model (Andresen & Jostad, 1999) may be used for capacity, deformation and soil-structure interaction analyses involving undrained loading of clay. The basis of the material model is:

- Input parameters for (undrained) shear strength for three different stress paths/ states (Active, Direct Simple Shear, Passive).
- A yield criterion based on a translated approximated Tresca criterion.
- Elliptical interpolation functions for plastic failure strains and shear strengths in arbitrary stress paths.
- Isotropic elasticity, given by the unloading/reloading shear modulus, G_{ur} .

The NGI-ADP model parameters are determined based on the user-defined values as follows:

- Drainage type: Undrained C[1]
- $\gamma_{unsat} = \gamma'$ (input[2])) (kN/m^3)
- $\gamma_{\text{sat}} = \gamma_{\text{unsat}}$ (kN/m^3)
- $e_{\text{init}} = 0.500$ (default)
- $G_{ur} = G_0$, input at mid-depth of the layer (kN/m^2)
- $s^A_{u,\text{ref}} = s_{u,top}$ (kN/m^2)
- $G_{ur}/s^A_u = G_0/\left((s_{u,\text{bottom}} + s_{u,top})/2\right)$

  Note that, s^A_u is the value of the active undrained shear strength at the mid-layer level. [3]
- $\gamma^C_f = 60 \cdot 100/\left(G_{ur}/s^A_u\right) \quad (\%)$
- $0.5\,\% \leq \gamma^C_f \leq 75\,\%$
- $\gamma^E_f = 2.0 \cdot \gamma^C_f \quad (\%)$
- $1.0\,\% \leq \gamma^E_f \leq 150\,\%$
- $\gamma^{DSS}_f = 1.5 \cdot \gamma^C_f \quad (\%)$
- $0.75\,\% \leq \gamma^{DSS}_f \leq 100\,\%$
- $s^{C,TX}_u/s^A_u = 0.99 \quad (-)$
- $z_{\text{ref}} = z_{\text{top}} \quad (m)$ (negative value)
- $s^A_{u,\text{inc}} = (s_{u,\text{bottom}} - s_{u,\text{top}})/(z_{\text{top}} - z_{\text{bottom}})$ (kN/m^2/m)
- $s^P_u/s^A_u = 0.5 \quad (-)$
- $\tau_0/s^A_u = -0.5 \cdot (1 - K_0) \cdot \sigma'_{v0}/s^A_u$

---

[1] For hydrostatic cases, the situations different from the Head are considered. If the phreatic level in hydrostatic conditions is equal to Head, no extra water level will be generated. For more information on head and water conditions, see Section Defining water conditions in the PLAXIS 3D Reference Manual.

[2] By using γ' (Input value) as γ_{sat} and γ_{unsat} effective stresses are calculated without the need to calculate the water pressures. The phreatic level is set at the bottom of the model and that is why γ' is used as γ_{sat} and γ_{unsat}. (kN/m^3)

[3] A minimum value of G_{ur}/s^A_u equal to 10 is adopted for robustness.

where σ'_{v0} = initial vertical effective stress at the mid-layer level (kN/m^2). Note that compression is negative.

> **Note:** $0.0 \leq \tau_0/s_u^A \leq 0.95$ (overconsolidated clays should be modelled with care)

- $s_u^{DSS}/s_u^A = \left(1 + s_u^P/s_u^A\right)/2 \quad (-)$
- $\nu' = 0.495 \quad (-)$
- $R_{\text{inter}} = 1.0 \quad (-)$
- $K_0 = \text{input} \quad (-)$

To allow tension cut-off, the Mohr-Coulomb (MC) model is assigned to the interfaces instead of the NGI-ADP. The stiffness MC properties match the derived NGI-ADP parameters presented above. The stiffness properties of the MC model, in terms of s_u and $s_{u,inc}$, are adjusted to 65% of the strength of the adjacent soil material (Palix et al., 2011).

5.2.2 Sand: HSsmall material Parameters

The Hardening Soil model with small-strain stiffness (Brinkgreve et al., 2021) implemented in PLAXIS is based on the Hardening Soil model and uses almost entirely the same parameters. In fact, only two additional parameters are needed to describe the variation of stiffness with strain:

- The initial or very small-strain shear modulus G_0.
- The shear strain level $\gamma_{0.7}$ at which the secant shear modulus G_s is reduced to about 70% of G_0.

The HSsmall model parameters are determined based on the user-defined values as follows, based on Brinkgreve et al. (2010).

- Drainage type: Drained
- $\gamma_{\text{sat}} = \gamma' \quad$ (input[4], [5])
- $\gamma_{\text{unsat}} = \gamma_{\text{sat}} \quad$ (kN/m^3)
- $e_{\text{init}} = 0.500 \quad (-)$
- $G_0 = \text{input at mid-depth of the layer} \quad$ (kN/m^2)
- $\sigma'_3 = K_0 \cdot \sigma'_1 \quad \text{where} \quad \sigma'_1 = \sigma'_{v0} \quad \text{at the mid-layer level} \quad$ (kN/m^2)
- $G_0^{\text{ref}} = G_0/\left[\left(c'_{\text{ref}} \cdot \cos\varphi' - \sigma'_3 \cdot \sin\varphi'\right)/\left(c'_{\text{ref}} \cdot \cos\varphi' + p_{\text{ref}} \cdot \sin\varphi'\right)\right]^m \quad$ (kN/m^2)
- $RD = 100 \cdot \left(G_0^{\text{ref}} - 60000\right)/68000 \quad (\%)$
- $E_{50}^{ref} = 60000 \cdot RD/100 \quad$ (kN/m^2)
- $E_{oed}^{ref} = E_{50}^{ref} \quad$ (kN/m^2)
- $E_{ur}^{ref} = 3 \cdot E_{50}^{ref} \quad$ (kN/m^2)
- $m = 0.5 \quad (-)$
- $c'_{ref} = 0.1 \quad$ (kN/m^2)
- $\varphi' = \text{input} \quad (^\circ)$

[4] By using γ′ (input value) as γ_{sat} and γ_{unsat} effective stresses are calculated without the need to calculate the water pressures.

[5] Note that γ′ is used as both γ_{sat} and γ_{unsat} because the phreatic level is set at the bottom of the model.

- $\psi = \text{input} \quad (^\circ)$
- $\gamma_{0.7} = (2 - RD/100) \cdot 1E-4 \quad (-)$
- $\nu'_{ur} = 0.2 \quad (-)$
- $p_{\text{ref}} = 100.0 \quad \text{(kN/m}^2\text{)}$
- $K_0^{NC} = 1 - \sin\varphi' \quad (-)$
- $c'_{inc} = 0.0 \quad \text{(kN/m}^2\text{)}$
- $z_{\text{ref}} = 0.0 \quad \text{(m)}$
- $R_f = 1 - RD/800 \quad (-)$
- Tensile strength = 0.0 (kN/m^2)
- $R_{\text{inter}} = 1.0 \quad (-)$
- $K_0 = \text{input} \quad (-)$

The allowed minimum and maximum Relative Density (RD), RD_{min} and RD_{max}, are not defined for G_0^{ref}, i.e. 100% may be exceeded, but a lower and an upper bound of 10% and 100% is applied to the calculation of E_{50}^{ref} and $\gamma_{0.7}$ and R_f.

A separate interface material (HSsmall) is generated with the same properties but dilatancy ψ equal to 0.0 and friction angle φ equal to 29.0 deg (Jardine et al., 2005). This also allows the user to modify the properties of the interfaces separately if needed.

5.3 PLAXIS 3D Models

5.3.1 Generating 3D models

The generation of 3D PLAXIS models is based on the model assumptions that are listed below.

Model geometry

- Only half of a symmetric model of the monopile is modelled via the **Tunnel designer**. This offers controllable geometry (re)generation, based on user-defined parameters. The vertical plane at y = 0 is the plane of symmetry.
- The model contour is based on input parameters:
  - The bottom depth of the last soil layer is as specified in the **Soil mode**.
  - The total model length in the x-direction is equal to 12 monopile outer diameters (D_{out}) The distance from the centre of the pile to the right and left model boundaries in the x-direction is $6 \cdot D_{out}$.
  - Total model length in the y-direction equal to 4 monopile outer diameters D_{out} The distance from the plane of symmetry (front model boundary) to the rear model boundary in the y-direction is $4 \cdot D_{out}$.
- Borehole at (0,0) from z_{top} = 0.0 to a user-defined depth.

> **Note:** The water table is placed at the bottom of the model.

- Fully saturated soil conditions for offshore applications and effective stress approach.
- The user defines the number of soil layers in **Soil mode**.

- The basic soil parameter can be directly entered by the user (**Soil mode**), whereas secondary parameters are automatically defined based on correlations.
- The user may change constitutive models and soil materials parameters in PLAXIS 3D .

> **Note:** If the user changes the constitutive model or the material parameters in PLAXIS 3D, the parameters defined in the **Soil mode** of PLAXIS Monopile Designer should match the updated values used in PLAXIS 3D. This is because the values that are used during parameterisation come from the **Soil mode** and not directly from the PLAXIS 3D models.

- The soil layer thickness cannot be less than 0.5 m. This is to prevent bad quality meshes that lead to long calculation time and possibly inaccurate results.

Monopiles

- The embedded part of the monopile is divided into slices of approximately 1.0 m depth during its generation via the **Tunnel designer**. The slicing takes place within each soil layer, assuring that no monopile slice intersects boundaries between soil layers. Note that if a soil layer is less than 1.0 m deep, then only one monopile slice is generated within that soil layer.
- The number of monopile slices per soil layer is determined based on the following two hypotheses:
  - The target thickness of a monopile slice is 1.0 m (fixed value).
  - To determine the number of monopile slices per soil layer, the (user-defined) depth of that soil layer is rounded up or down to the closest integer.

> **Tip:** If the depth of the soil layer is 1.3 m, 1 monopile slice is created with a thickness of 1.3 m. On the other hand, if the layer depth is 1.75 m, 2 monopile slices are created with a thickness of 0.875 m each.

- Linear-elastic isotropic plate elements (shells) are used to model the monopile structure.
- The input properties of the plate elements (shells): are Young's modulus E, Poisson's ratio v and wall thickness t.
- The top of the monopile is closed with a plate with rigid body properties, to apply a prescribed displacement.
- The bottom of the monopile remains open.
- The monopile is weightless, i.e. the pile unit weight w is set to zero. The weight is not taken into account because lateral loading and not vertical loading is considered.
- The monopile is 'wished-in-place' (i.e. no installation effects is considered) and then loaded laterally.
- A prescribed displacement is applied to the top surface in the horizontal x-direction thereby introducing a lateral force and bending moment at the ground level (the latter is valid if the prescribed displacement is applied at a particular height h above seabed).

> **Note:**
> - The applied prescribed displacement is only valid for Calibration models (**Calibration mode**). In contrast, verification models are force controlled (**Analysis mode**).
> - The prescribed displacement at the top of the monopile is firstly approximated as an "a priori" value so as to be slightly higher than the target displacement, v_g/D_{out}. For clays this value depends on S_u (undrained shear strength), while for sands depends on ϕ' (internal friction angle) and p' (mean effective stress). Please note that the calculation will stop whenever the v_g/D_{out} is reached.

- Interfaces are used at the outer surface of the monopile to model the soil-structure interaction.
- Another (horizontal) interface is used at the monopile bottom to retrieve soil reactions at the base.
- Drainage type of interface elements is always set to *drained*, to prevent suction from developing at the active soil side of the monopile. In this case a gap is formed between the monopile and the soil. Note that this requires the generation of an extra (drained) material set for the interfaces.
- Based on the user-defined input values per PLAXIS 3D model and the structural parameters specified, the following are calculated:
  - $D_{in} = D_{out} - 2 \cdot t$
  - cross section area: $A = \varpi \, (D_{out}^{\,2} - D_{in}^{\,2}) / 4$
  - moment of inertia: $I = \varpi \, (D_{out}^{\,4} - D_{in}^{\,4}) / 64$
  - axial stiffness: EA
  - flexural rigidity: EI
  - shear stiffness: $GA = 0.5 \cdot k \, EA / (1+\nu)$

  A constant value of k = 0.5 (independent of Poisson's ratio effects) is assumed to calculate the shear stiffness GA.

> **Note:** The values of the parameters above, which are presented on the GUI once a GeoDS is selected, are indicative based on the input diameter and thickness. The plate elements in PLAXIS 3D do not directly take these quantities as input. For more information on the definition of the plate elements the reader may refer to the PLAXIS 3D Reference Manual and Scientific Manual.

- Two 'soft' beams (5.3.2 Soft beam properties (p. 124)) are placed on both the front and back edges (sides) of the monopile for post-processing purposes.

Mesh

- The embedded part of the monopile cross-section is divided into 9 arcs of 20 degrees each to force a structured mesh at the circumference.
- A (refinement) zone around the monopile is generated to have structured mesh extended:
  - $0.20 \cdot D_{out}$ at the monopile's circumference.
  - $0.15 \cdot D_{out}$ below the monopile toe.

- The considered default mesh settings are:
  - pile above ground level: coarseness factor = 1.0
  - embedded pile: coarseness factor = 0.5
  - bottom of the pile: coarseness factor = 0.07
  - surrounding soil: coarseness factor = 1.0
  - beams: coarseness factor = 1.0
  - coarse mesh:
    - mesh command used in PLAXIS 3D 2026.1 : `_mesh 0.075 256 True 2.2 0.0175 1.0`

Calculation phases

- The considered calculation phases are:
  - Initial phase: K0-procedure
  - **Phase 1:** Monopile installation (wished-in-place), plastic calculation
  - **Phase 2 (small displacements):** Applying prescribed lateral displacements equal to $v_g/D_{out}/1000$, plastic calculation
  - **Phase 3 (Large displacements):** Applying prescribed lateral displacements (input value, v_g/D_{out}), plastic calculation
- The considered default numerical settings are:
  - **Phase 2:**
    - Solver = Pardiso
    - Max load fraction per step = 0.02
    - Tolerated error = 0.001
  - **Phase 3:**
    - Solver = Pardiso
    - Max load fraction per step = 0.5
    - Tolerated error for Sand = 0.01
    - Tolerated error for Clay = 0.001
    - Max unloading steps = 50
    - Max steps = 10000
    - Max iterations = 90
    - Desired min iterations = 4
    - Desired max iterations = 30

> **Tip:** For more details, see the PLAXIS 3D Reference Manual

Suggested Values of v_g/D_{out}

A common failure condition used in design is v_g/D_{out} = 0.10 (i.e. the monopile is considered to fail when the horizontal displacement at the mudline reaches 10% of the outer diameter). However, at this value of displacement the failure mechanism is commonly not completely developed. Calibration requires the full development of plasticity in most of the soil reaction curves, for which we generally need to allow a higher relative displacement than the failure criterion. At the same time, too high values of v_g/D_{out} will result in long calculations that do not contribute additional information about the shape of the soil reaction curves

Suggested values for the relative target displacement at the mudline are:

- Minimum: v_g/D_{out} = 0.15
- Maximum: v_g/D_{out} = 0.30

> **Note:** The DVF parameters resulting from Calibration will vary slightly depending on the reached displacement.

> **Note:**
>
> - Previous versions of PLAXIS Monopile Designer used the prescribed displacement at the monopile head, $v_{max,z=h}$, as input a parameter instead of the relative target displacement at the mudline, v_g/D_{out}. The suggested values of $v_{max,z=h}$ were estimated based on the assumption of a linearly deformed pile with a rotation point a depth approximately equal to 2/3·L. This neglected the bending of the pile.
> - When opening a project created with an older version, the input value of $v_{max,z=h}$ will be automatically converted to v_g/D_{out} using the reached displacement at the mudline, v_g, and the outer diameter, D_{out} (See Figure 5–1 (p. 124)) .

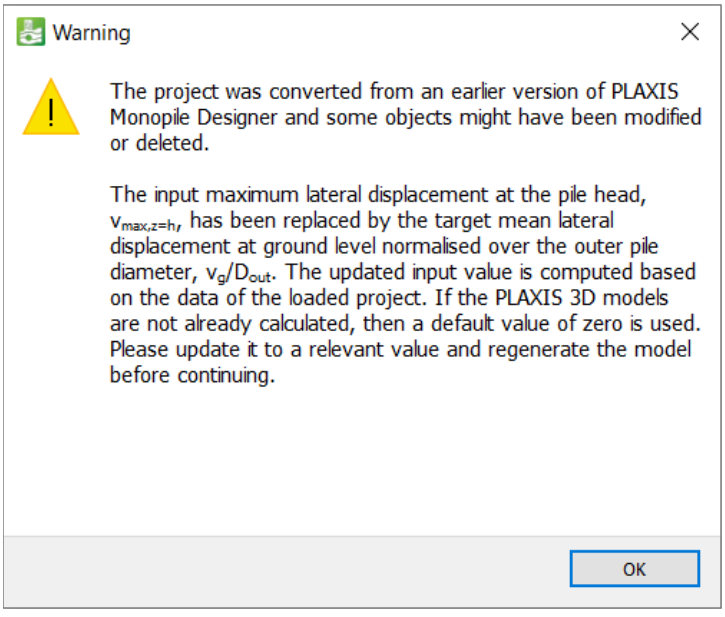


Figure 5–1: Warning - update projects for the use of v_g/D_{out}

5.3.2 Soft beam properties

Two soft beams are attached to the monopile (at the front and back of the tube), to be used for post-processing purposes. In the present version of PLAXIS Monopile Designer the post-processing merely concerns a manual comparison between the deflected shape of the beams against the deflection of the monopile in the **Calibration mode** of the tool. The stiffness of the soft beam is taken equal to $E_{steel}/1000$ (= $210 \cdot 10^3$ kPa) to avoid any influence on the results.

5.3.3 Soil reaction curves

There are four types of soil reaction curves:

- Distributed lateral load vs. lateral displacement: (p, v).
- Distributed moment vs. pile rotation: (m, θ).
- Base horizontal force vs. lateral base displacement: (H_B, v_B).
- Base moment vs. base rotation: (M_B, θ_B).

After every successful PLAXIS 3D calculation, data is obtained from the nodal reaction forces of the interface elements. This information is turned into resultant forces, moments, average displacements and rotations along the shaft (with corresponding depth) and at the base (with corresponding depth), to obtain the four types of raw soil reaction curves. The following quantities are calculated:

- Monopile slices with thickness of about 1.0 m are considered. Note that the actual slice thickness depends on the soil layering too (see 5.3.1 Generating 3D models (p. 120)).
- Force values are multiplied by 2 since only one symmetric half is modelled.
- For every interface element, it is determined to which monopile slice it belongs.
- The *slice* forces are the summation of the forces from all the interfaces belonging to this slice: $p_{slice} = \Sigma(Fx_{interface})$.
- The *base (toe)* forces are the summation of the forces from the bottom interface: $p_{bottom} = \Sigma(Fx_{interface,\ bottom})$.
- The *slice* displacement is the average value of the lateral displacement of the slice nodes $v_{slice} = \underline{u}_{\mathrm{x,\ interface}}$.
- The *base* displacements: same as above, but considering only the bottom interface nodes at the pile circumference.
- The slice moment is calculated as a summation of the vertical node forces F_z times the lateral distance from the axis of symmetry d_x $m_{slice} = \Sigma\ (F_z \cdot d_x)$.
- The base (toe) moment: same as above, considering the bottom interface.
- The slice rotation θ_{slice} is calculated using a least-squares linear fit to the vertical displacement of the nodes on the cross-section.
- The base rotation: same as above, but considering only the bottom interface nodes at the pile circumference.
- The following quantities are calculated at mudline (ground level):
  - Lateral displacement at the front and the back of the pile (based on single nodes), to check pile ovalisation.
  - Rotation as described above (least-squares linear fit).
  - Horizontal force H as the result of integration from all horizontal soil reactions along the monopile, including the base. This corresponds to the structural (monopile) shear force at the mudline.
  - Moment M as the result of the integration of all soil reaction contributions along the monopile, including the base, considering both vertical and lateral arms. This corresponds to the structural (monopile) bending moment at the mudline.

The normalised soil reaction curves are generated by the *Optimisation Module*, which takes as input the data derived from the post-processor and the soil-structure data provided in PLAXIS Monopile Designer (**Soil** and **Calibration modes** respectively). The normalisation process is based on local stiffness and soil strength parameters. It is triggered by the *Parameterise* button.

The normalisation formulae for Sand and Clay are presented in Table 5–1 (p. 126):

Table 5–1: Normalisation formulae for Sand and Clay (Burd et al., 2020a, Byrne et al., 2020)

Component	Clay normalisation	Sand normalisation
Distributed load, $\bar{p}$	$\bar{p} = p/\left(s_u D\right)$	$\bar{p} = p/\left(\sigma'_{v0} D\right)$
Lateral displacement, $\bar{\nu}$	$\bar{\nu} = \nu I_R / D$	$\bar{\nu} = \left(\nu I_R / D\right) \cdot \sqrt{\left(p_a / \sigma'_{v0}\right)} = \nu G_0 / \left(\sigma'_{v0} D\right)$
Distributed moment, $\overline{m}$	$\overline{m} = m/\left(s_u D^2\right)$	$\overline{m} = m/(pD)$
Pile cross section rotation, θ	$\bar{\theta} = \theta I_R$	$\bar{\theta} = \theta I_s \cdot \sqrt{\left(p_a / \sigma'_{v0}\right)} = \theta G_0 / \sigma' \nu_0$
Base horizontal force, $\overline{H}_B$	$\overline{H}_B = H_B/\left(s_u D^2\right)$	$\overline{H}_B = H_B/\left(\sigma'_{v0} D^2\right)$
Base moment, $\overline{M}_B$	$\overline{M}_B = M_B/\left(s_u D^3\right)$	$\overline{M}_B = M_B/\left(\sigma'_{v0} D^3\right)$

The normalised curves depend on the undrained shear strength, s_u, or the initial vertical effective stress, σ'_{v0}, and therefore are depth-dependent (z); p_a is the atmospheric pressure. The parameter I_R is the rigidity index, defined as $I_R = G_0 / s_u$, where G_0 is the small-strain shear modulus and s_u the undrained shear strength in triaxial compression. The parameter I_s is a stiffness coefficient defined as: $I_s = \left(G_0/p_a\right)\sqrt{p_a/\sigma'_{v0}}$.

5.3.4 | Results inspection pane

Monopile response of the 3D models

- The lateral reaction force at ground level is plotted against the lateral displacements at ground level. The latter is a mean value of the displacements at the front and the back of the monopile.
- The pile deflection profile below mudline is plotted based on the average lateral displacements of the front and back of the monopile for each monopile segment.

Soil reaction curves

The following combinations may be plotted:

- Unchecked *Normalised* and unchecked *Numerical* checkboxes (default): the *Parametric* soil reaction curves obtained from the depth variation functions are denormalised from the formulae presented in Table 5–1 (p. 126) and plotted for the following predefined depths: *0.1·L, 0.2·L, 0.3·L, 0.4·L, 0.5·L, 0.6·L, 0.7·L, 0.8·L, 0.9·L, L* (base).
- Checked *Normalised* and unchecked *Numerical* checkboxes: the Parametric soil reaction curves obtained from the depth variation functions are plotted in non-dimensional space.
- Unchecked *Normalised* and checked *Numerical* checkboxes: the data of the raw soil reaction curves (extracted from the 3D design verification model) are plotted together with the Parametric soil reaction curves obtained from the depth variation functions and denormalised from the formulae presented in Table 5–1 (p. 126).
- Checked *Normalised* and checked *Numerical* checkboxes: the data of the soil reaction curves (extracted from the 3D design verification model) are normalised from the formulae presented in Table 5–1 (p. 126) and plotted together with the Parametric soil reaction curves obtained from the depth variation functions, in non-dimensional space.

> **Tip:** Although the graph only displays the curves at 10 predefined depths, soil reaction curves are in general computed at finer intervals. The full set of results are available in the *Table* tab.

Depth variation functions: shaft and base

The plots are derived based on the .dvf files and the corresponding functions (Table 5–2 (p. 129)).

5.4 Optimization Module

At a first stage, the raw soil reaction curves (obtained from the 3D finite element calibration models) are normalised (using the forms in Table 5–1 (p. 126)) and pre-processed to obtain purely monotonic curves. These normalised soil reaction curve data are then represented with the 4-parameter conic function shown in Eqn. 5–1 (p. 127) :

$$-n \cdot \left(\frac{\bar{y}}{\bar{y}_u} - \frac{\bar{x}}{\bar{x}_u}\right)^2 + \left(1-n\right) \cdot \left(\frac{\bar{y}}{\bar{y}_u} - \frac{\bar{x} \cdot k}{\bar{y}_u}\right) \cdot \left(\frac{\bar{y}}{\bar{y}_u} - 1\right) = 0 \tag{5–1}$$

where

$\bar{x}$	=	Normalised displacement (or rotation) variable
$\bar{y}$	=	The corresponding normalised soil reaction component.

The conic function is calibrated by the specification of four parameters (k, n, $\bar{x}_u$, $\bar{y}_u$), each of which has a straightforward interpretation.

k	=	Initial slope
$\bar{y}_u$	=	Ultimate value of the normalised soil reaction
$\bar{x}_u$	=	The normalized displacement (or rotation) at which this ultimate value of soil reaction is reached
n	=	Curvature parameter (0<n<1)

This particular function was selected during the PISA project to represent the soil reaction curves in the 1D design model (Burd et al., 2020a; Byrne et al., 2020). To an extent, however, the choice of function is arbitrary and other possibilities exist for the choice of functional form of the soil reaction curves.

The four parameters (k, n, $\bar{x}_u$, $\bar{y}_u$) for each of the soil reaction components, and the way in which the parameters vary with depth, are determined from the normalised raw soil reaction curves by the optimisation module. This optimisation process incorporates data from the results of all user-selected PLAXIS 3D models in the **Calibration mode**.

For numerical implementation purposes, the positive roots of $\bar{y}$ are:

$$\bar{y} = \bar{y}_u \cdot \frac{2c}{-b + \sqrt{b^2 - 4ac}} \text{for} \bar{x} \leq \bar{x}_u \tag{5–2}$$

$$\bar{y} = \bar{y}_u \text{for} \bar{x} > \bar{x}_u \tag{5–3}$$

where:

$$a = 1 - 2 \cdot n$$
$$b = 2 \cdot n \cdot \frac{\overline{x}}{\overline{x}_u} - \left(1 - n\right) \cdot \left(1 + \frac{\overline{x} \cdot k}{\overline{y}_u}\right) \quad (5\text{–}4)$$
$$c = \frac{\overline{x} \cdot k}{\overline{y}_u} \cdot \left(1 - n\right) - n \cdot \frac{\overline{x}^2}{\overline{x}_u^2}$$

The shape of the conic function is strongly conditioned by the value of *n*, as illustrated in Figure 5–2 (p. 128). For *n = 0* and *n = 1*, bi-linear forms are obtained. For intermediate values of *n* the function is curved.

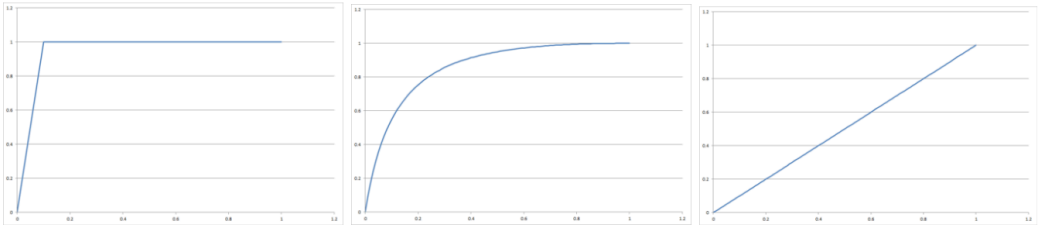


Figure 5–2: Curves for different values of the n-parameter

The parameters corresponding to each soil reaction curve are:

- Distributed lateral load vs. lateral displacement: $(\bar{p}, \bar{v}) \rightarrow (\bar{p}_u, \bar{v}_{pu}, k_p, n_p)$.
- Distributed moment vs. pile rotation: $\left(\overline{m}, \bar{\theta}\right) \rightarrow \left(\overline{m}_u, \bar{\theta}_{mu}, k_m, n_m\right)$.
- Base horizontal force vs. lateral base displacement: $\left(\overline{H}_B, \bar{v}_B\right) \rightarrow \left(\overline{H}_{Bu}, \bar{v}_{Hu}, k_H, n_H\right)$.
- Base moment vs. base rotation: $\left(\overline{M}_B, \bar{\theta}_B\right) \rightarrow \left(\overline{M}_{Bu}, \bar{\theta}_{Mu}, k_M, n_M\right)$.

The parameters needed to calibrate the soil reaction curves are determined by obtaining a best-fit (based on least-squares) with the raw soil reaction curves. This process, conducted by the optimisation module, is described in detail in Byrne et al., 2018. The procedure is summarised as follows.

Initially, values of the calibration parameters are determined for the distributed load and distributed moment at depths where data are available, for all of the piles in the calibration set. Calibration parameters are also determined for the base horizontal force and moment. The general approach that is employed by the optimisation module to determine these parameters is summarised below.

1. Determine appropriate values of ultimate displacement $\left(\bar{v}_{pu}, \bar{\theta}_{mu}, \bar{v}_{Hu}, \bar{\theta}_{mu}\right)$ to match the form of the numerical data.
2. Determine values of the ultimate values $\left(\bar{p}_u, \overline{m}_u, \overline{H}_u, \overline{M}_u\right)$.
3. Find values of initial stiffness (k_p, k_m, k_H, k_M) to provide a match with the initial portions of the raw soil reaction curves.
4. Determine the curvature parameters (n_p, n_m, n_H, n_M) to provide a fit with the data. At this stage the curvature parameter for the distributed moment is set to zero (to produce a bi-linear form).

Once the calibration parameters have been determined for the calibration set piles on a point-wise basis, a further model is developed (referred to as 'depth variation function') which represent the variation with depth of the calibration parameters. The form of the depth variation functions is specified in Table 5–2 (p. 129).

Table 5–2: Depth variation functions

Soil reaction component	Fitting parameter	Clay depth variation functions	Sand depth variation functions
Distributed lateral load, $\overline{p}$	Ultimate strain, $\overline{v}_{pu}$	$c1$	$s1$
	Initial stiffness, k_p	$c2 + c3 \cdot (z/D_{out})$	$s2 + s3 \cdot (z/D_{out})$
	Curvature, n_p	$c4 + c5 \cdot (z/D_{\text{out}})$	$s4$
	Ultimate reaction $\overline{p}_u$	$c6 + c7 \cdot e^{c8 \cdot z/D_{out}}$	$s5 + s6 \cdot (z/L)$
Distributed moment, $\overline{m}$	Ultimate rotation, $\bar{\theta}_{mu}$	$c9$	$s7$
	Initial stiffness, k_m	$c10 + c11 \cdot (z/D_{\text{out}})$	s8
	Curvature, n_m	$c12$	$s9$
	Ultimate moment, $\overline{m}_u$	$c13 + c14 \cdot (z/D_{out})$	$s10 + s11 \cdot (z/L)$
Base horizontal force, $\overline{H}_B$	Ultimate strain, $\overline{V}_{Hu}$	$c15$	$s12 + s13 \cdot (L/D_{out})$
	Initial stiffness, k_H	$c16 + c17 \cdot (L/D_{\text{out}})$	$s14 + s15 \cdot (L/D_{\text{out}})$
	Curvature, n_H	$c18 + c19 \cdot (L/D_{\text{out}})$	$s16 + s17 \cdot (L/D_{\text{out}})$
	Ultimate reaction, $\overline{H}_{Bu}$	$c20 + c21 \cdot (L/D_{\text{out}})$	$s18 + s19 \cdot (L/D_{\text{out}})$
Base moment, $\overline{M}_B$	Ultimate rotation, $\bar{\theta}_{Mu}$	c22	$s20$
	Initial stiffness, k_M	$c23 + c24 \cdot (L/D_{\text{out}})$	$s21$
	Curvature, n_M	$c25 + c26 \cdot (L/D_{\text{out}})$	$s22$
	Ultimate reaction, $\overline{M}_{Bu}$	$c27 + c28 \cdot (L/D_{\text{out}})$	$s23 + s24 \cdot (L/D_{out})$

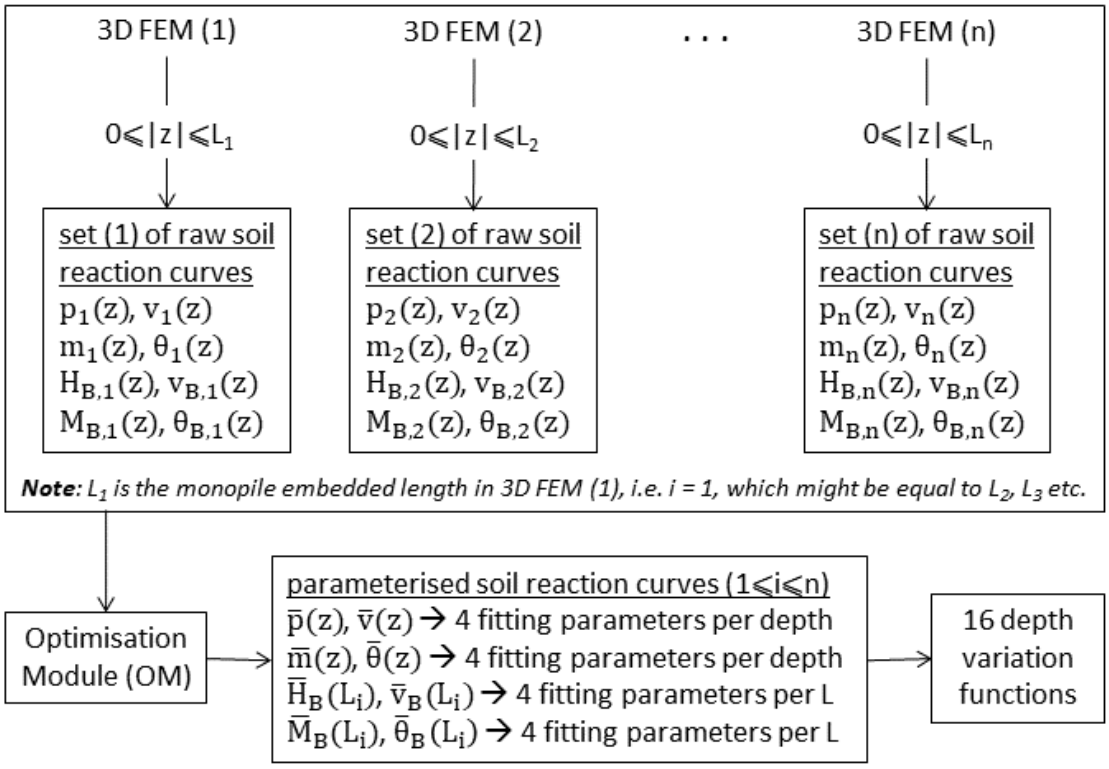


Figure 5–3: Calibration workflow

5.5 Rule based models

The PISA rule-based method defines parametric, normalised depth variation functions (DVF) for several generic soil types. Thus, it is possible to obtain semi-particularised soil reaction curves by de-normalising the rule-based DVF according to the site-specific soil parameters. This approach necessarily provides a looser approximation to the actual response of the soil, since it relies on a generic fitting of DVF instead of numerically calibrating them from the actual soil units present on site. Conversely, it enables a faster time to results at the early stages of a project, when site-specific information required for 3D FEM analyses may not yet be fully available.

The following rule-based design models have been published to date:

- Cowden till model (Byrne et al., 2020) for stiff clays.
- Bothkennar clay model (Burd et al., 2020b) for soft clays.
- General Dunkirk Sand Model (Burd et al., 2020a) for medium to dense sands (45% ≤ RD ≤ 90%).

Rule-based models can be used in PLAXIS Monopile Designer by creating and importing *.dvf* files with coefficient values corresponding to those in the published models. To create these *.dvf* files please refer to 5.5.3.3 Format: Depth variation functions (p. 140).

> **Note:** PISA rule-based models such as Cowden till, Bothkennar clay model and Dunkirk sand model are built into the **Analysis mode** (see 2.4.5 Soil Layers (p. 62)), therefore they can be used without the need for input files.

Conventional p-y curves, codified in industry standards such as API RP 2A-WSD (API, 2014), can also be considered as a form of rule-based models. These can also be used in PLAXIS Monopile Designer, importing a different file format (*.spy* for sand and *.cpy* for clay) for input.

5.5.1 Plain text file format rules

The general rules for all plain text file formats are:

- No particular units are needed for the data, assuming that a consistent set of units is used throughout the tool. Information regarding units can be found in 2.2.6 Units and sign convention (p. 24)
- Lines starting with # are regarded as comments (and skipped).
- Leading spaces and tabs are ignored, i.e. a line starting with " #" is still regarded as a comment.
- Tabular data columns are separated by single tabs (there is no intentional visual alignment of numbers).
- Floating point numbers are written in full-accuracy scientific notation floating point (i.e. ~ 16 digits such as 4.659996895060823E-19). The readers must not rely on this. They must check tab separators rather than field length.
- The files are written in ASCII and must not contain any Unicode characters, nor single-byte characters outside the allowed range (Char(9), Char(10), Char(13), Char(32)-Char(126)).

> **Note:**
>
> - User-defined *dvf*, *cpy* and *spy* files should comply with the rules presented above.
> - To insure version compatibility of PLAXIS Monopile Designer with *dvf*, *cpy* and *spy* files, it is necessary that users adapt the corresponding *#Version number* (e.g see sample in 5.5.3.3 Format: Depth variation functions (p. 140)) according Table 5–3 (p. 131)

Table 5–3: Version compatibility for soil reaction curves files

Version compatibility			
PLAXIS Monopile Designer	**dvf**	**cpy**	**spy**
PLAXIS MoDeTo V1	1	-	-
PLAXIS Monopile Designer V20	2	1	1
PLAXIS Monopile Designer V21	2	1	1
PLAXIS Monopile Designer V22	3	2	2

5.5.2 Conventional p-y curves

Although not the primary usage of PLAXIS Monopile Designer, conventional horizontal soil reaction curves ('p-y' curves) can be specified for method comparison. Conventional p-y curves for both clay and sand are defined in accordance with the American Petroleum Institute Recommended Practice 2A-WSD, API RP 2A-WSD (API , 2014). Since the PISA method only accounts for horizontal loading, conventional vertical soil reaction curves ('t-z' and 'Q-z' curves) are not supported.

Note: Rotational soil reaction curves are not defined in conventional methods.

5.5.2.1 Soft and stiff clay

In accordance with the API Recommended Practice 2A-WSD (API, 2014)

$$p_{uc} = c_1 s_u + \gamma' z + J s_u z / D_{out} \quad \text{for} \quad z < z_R \tag{5–5}$$

$$p_{uc} = c_2 s_u \quad \text{for} \quad z \geq z_R \tag{5–6}$$

where

p_{uc} = ultimate resistance of a clayey soil (kPa).

D_{out} = pile diameter (m).

s_u = undrained shear strength (undisturbed clay) (kPa).

γ' = submerged soil unit weight (kN/m^3).

c_1, c_2 = dimensionless coefficients that, in the absence of more definitive criteria, take the values suggested in (API, 2014):

c_1 = 3 (suggested value).

c_2 = 9 (suggested value; general variation: 8.0 - 12.0).

J = dimensionless empirical constant (general variation: 0.25 - 0.5).

z = depth below mudline (m).

z_R = distance from the mudline to the bottom of the reduced resistance zone (m).

- In general $z_R \geq 2.5D$
- For a condition of constant strength with depth: $z_R = 6D/(\gamma' D/s_u + J)$
- If strength varies with depth, z_R is taken as the point of first intersection between the values predicted by Eq.[7] and Eq. [8].

Note: Note that the value of s_u to compute the default value of z_R is taken as the value at mid-depth based on the specified $s_{u,\,\text{top}}$ and $s_{u,\,\text{bottom}}$: $s_{u,mid} = s_{u,top} + (z_{mid} - z_{top})^*(s_{u,bottom} - s_{u,top})/(z_{bottom} - z_{top})$.

Lateral soil resistance-deflection relationships for piles in soft clay are generally non-linear. The p-y curves for the short-term static load case may be generated from the following table.

p/p_{uc}	y/y_c
0	0
0.23	0.1
0.33	0.3
0.5	1
0.72	3
1	8
1	∞

p = lateral resistance (kPa)

y = lateral deflection (m)

y_c = 2.5 ε_c D_{out} (m)

ε_c = strain at one-half the maximum stress during a laboratory during a laboratory drained compression of undisturbed soil samples. The default value of ε_c is determined based on the input s_u values, as given in the table below

s_u (kPa)	ε_c
<48	0.020
48-96	0.010
96-192	0.005
>192	0.004

5.5.2.2 Sand

According to the API Recommended Practice 2A-WSD (API, 2014), at a given depth the equation giving the smallest value of p_u should be used as the ultimate bearing capacity:

$$p_{us,s} = (s_1 z + s_2 D_{out})\gamma' z \quad (5\text{–}7)$$

$$p_{us,d} = s_3 z D_{out}\gamma' z \quad (5\text{–}8)$$

Thus, the bearing capacity of sand is obtained as:

$$p_{us,d} = \min\{\, p_{us,s}(z), p_{us,d}(z)\,\} \quad (5\text{–}9)$$

where

$p_{us,d}$ = ultimate resistance (kN/m).

γ' = submerged soil unit weight (kN/m^3)

z = depth

D_{out} = pile diameter

s_1, s_2, s_3 = dimensionless coefficients, obtained as function of the angle of internal friction of sand, φ', from the following equations:

$$s_1 = \tan\beta\{K_p \tan(\alpha) + K_0 \quad [\tan(\varphi\prime)\sin(\beta)(1/\cos(\alpha)+1) - \tan(\alpha)]\} \tag{5–10}$$

$$s_2 = K_p - K_A \tag{5–11}$$

where :

$$K_0 = 0.4$$

$$K_p = \left[\tan\left(45\deg + \tfrac{\varphi\prime}{2}\right)\right]^2$$

$$K_A = \left[\tan\left(45\deg - \tfrac{\varphi\prime}{2}\right)\right]^2$$

$$\alpha = \tfrac{\varphi\prime}{2}$$

$$\beta = 45\deg + \tfrac{\varphi\prime}{2}$$

$$s_3 = (K_p)^2 (K_p + K_0 \tan(\varphi)) - K_A \tag{5–12}$$

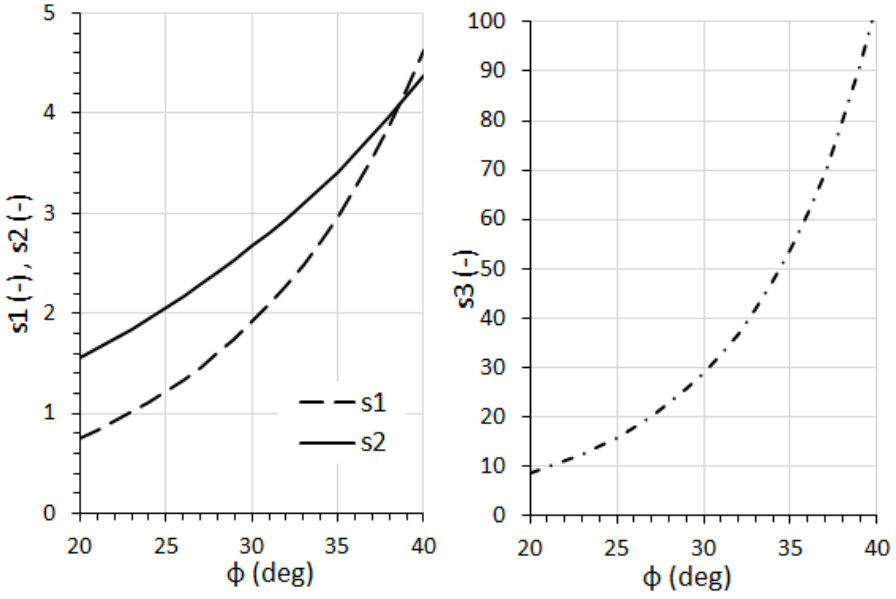


Figure 5–4: Dependency of coefficients s_1, s_2, s_3 with the internal angle of friction

The lateral soil resistance-deflection (p-y) relationships for sand are also non-linear, reading:

$$p = Ap_{us} \tanh[kzy/(Ap_{us})] \tag{5–13}$$

where

p = lateral resistance (kPa).

y = lateral deflection (m).

A = factor to account for cyclic or static loading conditions, evaluated as:

- cyclic loading: A=0.9
- static loading: A=max{(3.0-0.8 z/D); 0.9}

k = initial modulus of subgrade reaction (kN/m^3) -- determined as a function of φ' by the following equation for sand below the water table (adapted from Thieken et al.,2015), where φ' in deg and k in kN/m^3 .

$$k = 8.085*\left(\varphi^{2.45}\right) - 26090 \quad \text{(5–14)}$$

Note: Eqn. 5–14 (p. 135) is valid for 29 deg < φ'< 45 deg. Outside of this range, a warning will be issued.

5.5.2.3 | Format: Coventional p-y curves

The inputs for p-y curves are stored following a simple format in a text file readable as ASCII table by the tool. The name can be user-defined, but the extension of the file should always be "*.cpy*" for clay and "*.spy*" for sand.

This file starts with a recognisable flag (see sample files further below) and contains:

- Version number.

  Note: Before filling this data see Table 5–3 (p. 131) to ensure version compatibility of the file with your current PLAXIS Monopile Designer instance.

- Soil material parameters, including soil type. These are the same parameters as in the *dvf* files.
- Default secondary parameters or input values.
- Default *p-y* curves or input table.

Note: Although some of the material parameters are not necessary to obtain the p-y curves according to the conventional approach (API, 2014), they are included for the purpose of generating the corresponding 3D design verification model.

This is a sample file for conventional *p-y* curves for Clay:

```
# p-y curves flag
PLAXIS MONOPILE DESIGNER PY CURVES
# Version number
2
# Material type
clay
# ztop(m)    zbottom (m) gammasubmerged(kN/m3)    G0top(kN/m2) G0bottom(kN/m2)     sutop(kN/m2)     subottom(kN/m2) K0
0.00         -50.00      9.0                     70.0E3       145.0E3             75.00            140.00          1.0
# Default secondary parameters (yes/no)
yes
# Additional input parameters (used if default is set to no)
c1
c2
J
zR
epsC
yc
# Default p-y curves (yes/no)
yes
# p/puc - y/yc table (used if default p-y curves is set to no)
0.00    0.0
0.23    0.1
0.33    0.3
0.50    1.0
0.72    3.0
1.00    8.0
1.00    20.0
```

This is a sample file for conventional *p-y* curves for Sand:

```
# p-y curves flag
PLAXIS MONOPILE DESIGNER PY CURVES
# Version number
2
# Material type
sand
# ztop(m)    zbottom (m) gammasubmerged(kN/m3)    G0top(kN/m2) G0bottom(kN/m2)     ceff(kN/m2) phieff(deg) psi(deg) K0
0.00         -50.00      10.50                   50.0E3       180.0E3             1.0         35.00       5.000    1.0
# Default secondary parameters (yes/no)
yes
# Additional input parameters (used if default is set to no)
s1
s2
s3
A
k
# Default p-y curves (yes/no)
yes
# p/puc - y/yc table (used if default p-y curves is set to no)
0.00    0.0
```

5.5.3 Depth Variation Functions

The PISA rule-based models are defined in a common calibration space, so that the only factor that changes between models is soil type. This rule-based calibration space is more extensive than those that would be used in a typical numerical-based calibration, as it aims to cover a wide variety of geometries and design cases, namely:

- $2 \leq L/D_{out} \leq 6$
- $5 \leq h/D_{out} \leq 15$
- $60 \leq D_{out}/t \leq 111$

Care should be taken when applying the PISA rule-based models outside of these ranges.

Table 5–4: Calibration space for PISA rule-based depth variation functions (Burd et al., 2020a, Byrne et al., 2020)

GeoDS_#	L [m]	h [m]	t [m]	D [m]	E [kN/m^2]	h/D [-]	L/D [-]	D/t [-]
GeoDS_1	20	50	0.091	10	2.00E +08	5	2	110
GeoDS_2	20	150	0.091	10	2.00E +08	15	2	110
GeoDS_3	20	50	0.125	10	2.00E +08	5	2	80
GeoDS_4	60	50	0.091	10	2.00E +08	5	6	110
GeoDS_5	60	150	0.091	10	2.00E +08	15	6	110
GeoDS_6	10	25	0.045	5	2.00E +08	5	2	111
GeoDS_7	10	25	0.083	5	2.00E +08	5	2	60
GeoDS_8	30	25	0.045	5	2.00E +08	5	6	111
GeoDS_9	30	75	0.045	5	2.00E +08	15	6	111
GeoDS_10	15	37.5	0.068	7.5	2.00E +08	5	2	110
GeoDS_11	45	37.5	0.068	7.5	2.00E +08	5	6	110

Tip: For numerical-based design, a narrower calibration space may be defined by the project design envelope.

5.5.3.1 Cowden till and Bothkennar clay

The two PISA rule-based design models for clayey soils provide tabulated values for each of the 28 depth variation function coefficients (see Table 5–5 (p. 138)) for two distinct clay types. The Cowden till model (Byrne et al., 2020) describes a stiff, overconsolidated glacial clay till with a typical North Sea strength and stiffness profile, while the Bothkennar clay model (Burd et al., 2020b) defines a well-known soft silty estuarine clay.

Table 5–5: DVF coefficients for Cowden till and Bothkennar clay models

Component	Parameter	Symbol	DVF Coefficient	Cowden till (Byrne et al., 2020)	Bothkennar clay (Burd et al., 2020b)
Distributed lateral load, p-v	Ultimate displacement	v_{pu}	c1	241.4	173.8
	Initial stiffness	k_{p1}	c2	10.6	12.05
		k_{p2}	c3	-1.65	-1.547
	Curvature	n_{p1}	c4	0.939	0.7204
		n_{p2}	c5	-0.03345	-0.00268
	Ultimate reaction	p_{u1}	c6	10.7	7.743
		p_{u2}	c7	-7.101	-3.945
		p_{u3}	c8	-0.3085	-0.8456
Distributed moment, m-θ	Ultimate rotation	θ_{mu}	c9	0.2042	0.2863
	Initial stiffness	k_{m1}	c10	1.42	1.698
		k_{m2}	c11	-0.09643	-0.1576
	Curvature	n_m	c12	0	0
	Ultimate moment	m_{u1}	c13	0.2899	0.4862
		m_{u2}	c14	-0.04775	-0.05674
Base shear, H_B-v_B	Ultimate displacement	v_{Hu1}	c15	235.7	291.5
	Initial stiffness	k_{H1}	c16	2.717	3.008
		k_{H2}	c17	-0.3575	-0.2701
	Curvature	n_{H1}	c18	0.8793	0.3113
		n_{H2}	c19	-0.0315	0.04263
	Ultimate reaction	H_{Bu1}	c20	0.4038	0.5279
		H_{Bu2}	c21	0.04812	0.06864

Component	Parameter	Symbol	DVF Coefficient	Cowden till (Byrne et al., 2020)	Bothkennar clay (Burd et al., 2020b)
Base moment, M_B-θ_B	Ultimate rotation	θ_{Mu}	c22	173.1	187
	Initial stiffness	k_{M1}	c23	0.2146	0.3409
		k_{M2}	c24	-0.00213	-0.01995
	Curvature	n_{M1}	c25	1.079	0.699
		n_{M2}	c26	-0.1087	-0.1155
	Ultimate reaction	M_{Bu1}	c27	0.8192	0.8756
		M_{Bu2}	c28	-0.08588	-0.09195

5.5.3.2 General Dunkirk Sand Model (GDSM)

The General Dunkirk Sand Model (GDSM) provides tabulated values for each of the 24 depth variation function coefficients defined in Table 5–4 (p. 137), accounting for linear dependencies on the relative density (RD). The model was initially calibrated from the homogeneous dense sand (RD = 75%) at the Dunkirk test site and generalised to arbitrary values of the relative density in the range 45% ≤ RD ≤ 90%.

Table 5–6: DVF coefficients as a function of Relative Density for General Dunkirk Sand Model; 45% ≤ RD ≤ 90%

Component	Parameter	Symbol	DVF Coefficient	GDSM (Burd et al., 2020a)
Distributed lateral load, p-y	Ultimate displacement	v_{pu}	s1	146.1-92.11·RD
	Initial stiffness	k_{p1}	s2	8.731-0.6982*RD
		k_{p2}	s3	-0.9178
	Curvature	n_p	s4	0.917+0.06193*RD
	Ultimate reaction	p_{u1}	s5	0.3667+25.89*RD
		p_{u2}	s6	0.3375-8.900*RD
Distributed moment, m-θ	Ultimate rotation	θ_{mu}	s7	0.01532
	Initial stiffness	k_m	s8	17
	Curvature	n_m	s9	0
	Ultimate moment	m_{u1}	s10	0.2605
		m_{u2}	s11	-0.1989+0.2019*RD
Base shear, H_B-v_B	Ultimate displacement	v_{Hu1}	s12	0.5150+2.883*RD
		v_{Hu2}	s13	0.1695-0.7018*RD
	Initial stiffness	k_{H1}	s14	6.505-2.985*RD

Component	Parameter	Symbol	DVF Coefficient	GDSM (Burd et al., 2020a)
		k_{H2}	s15	-0.007969-0.4299*RD
	Curvature	n_{H1}	s16	0.09978+0.7974*RD
		n_{H2}	s17	0.004994-0.07005*RD
	Ultimate reaction	H_{Bu1}	s18	0.09952+0.7996*RD
		H_{Bu2}	s19	0.03988-0.1606*RD
Base moment, M_B-θ_B	Ultimate rotation	θ_{Mu}	s20	44.89
	Initial stiffness	k_M	s21	0.3515
	Curvature	n_M	s22	0.300+0.4986*RD
	Ultimate reaction	M_{Bu1}	s23	0.09981+0.3710*RD
		M_{Bu2}	s24	0.01998-0.09041*RD

Note: In the PISA models, the distributed rotational soil reaction curves are assumed linear elastic perfectly plastic, with coefficient of curvature n_m = 0. Thus, for any given depth the ultimate rotation, θ_{mu}(z), is uniquely defined from the ultimate moment, m_u(z), and the initial stiffness, k_m(z), and coefficient s7 is unnecessary. In PLAXIS Monopile Designer, the value of coefficient s7 is only used for display purposes. The value of s7 in Table 5–6 (p. 139) corresponds to the ultimate rotation at depth z = 0 and is obtained as s7 = s10/s8.

5.5.3.3 | Format: Depth variation functions

The depth variation functions are stored following a simple format in a text file readable as ASCII table by the tool. The name can be user-defined, but the extension of the file should always be ".dvf". The file produced from the *Optimisation Module* in the **Calibration mode**, if the numerical-based design is followed, is named as "calibrated.dvf". User-defined files of the same format can be created and imported in the tool via the **Analysis mode**. This file starts with a recognisable flag (see sample files further below) and contains:

- Version number.
- Parameterisation function type, e.g. conic.
- Soil material parameters, including soil profile (Sand/Clay) and drainage type (drained/ undrained) per soil layer.
- The used geometry data sets during the calibration: *L, h, t,* D_{out} *, E.*
- The maximum reached displacement and rotation at ground level during the calibration.
- The fitting parameters.

In total 24 parameters are needed to define the depth variation functions for Sand, and 28 parameters for Clay respectively. If the numerical-based design is followed, the values of the needed parameters are defined by the *Optimisation Module*. If the rule-based design is followed the user needs to specify the values.

This is a sample file for depth variations functions for Clay:

```
# Depth variation functions flag
PLAXIS MONOPILE DESIGNER DEPTH VARIATION FUNCTIONS
# Version number
3
# Parameterisation function type
conic
# Material type
clay
# Drainage type
undrained
# Number of soil layers
4
#SoilLayer   ztop(m) zbottom(m)   gammasubmerged(kN/m3)   G0(kN/m2)   sutop(kN/m2)   subottom(kN/m2)   K0
1             0.0     -10.0           7.5                  75000.0     50.0           70.0              1.0
2           -10.0     -25.0           8.0                  100000.0    80.0           95.0              1.0
3           -25.0     -40.0           9.0                  120000.0    100.0          115.0             0.9
4           -40.0     -50.0           10.0                 140000.0    120.0          140.0             0.8
# Number of Geometry data sets
8
#h(m)        L(m)    Dout(m)   t(m)     E(kN/m2)
25.0         15.0     5.0      0.05     210000000.0
25.0         25.0     5.0      0.05     210000000.0
100.0        15.0     5.0      0.05     210000000.0
100.0        25.0     5.0      0.05     210000000.0
25.0         21.0     7.0      0.07     210000000.0
25.0         35.0     7.0      0.07     210000000.0
100.0        21.0     7.0      0.07     210000000.0
100.0        35.0     7.0      0.07     210000000.0

# Max displacement reached at ground level (m)
1.371147902105263
# Max rotation reached at ground level (rad)
0.0984185195521575
# Fitting parameters
c1
c2
c3
...
c28
```

This is a sample file for depth variations functions for Sand:

```
# Depth variation functions flag
PLAXIS MONOPILE DESIGNER DEPTH VARIATION FUNCTIONS
# Version number
3
# Parameterisation function type
conic
# Material type
sand
# Drainage type
drained
# Number of soil layers
1
#SoilLayer ztop(m) zbottom(m)   gammasubmerged(kN/m3)    G0(kN/m2)    ceff(kN/m2)    phieff(deg)   psi(deg)    K0
1           0.0      -50.0           10.0                191600.0       0.1          39.0          9.0         1.0
# Number of Geometry data sets
8
#h(m)     L(m)     Dout(m)  t(m)    E(kN/m2)
20.0      20.0     5.0         0.05    210000000.0
75.0      25.0     5.0         0.05    210000000.0
75.0      25.0     8.0         0.08    210000000.0
35.0      35.0     8.0         0.08    210000000.0
60.0      20.0     5.0         0.05    210000000.0
25.0      25.0     5.0         0.05    210000000.0
25.0      25.0     8.0         0.08    210000000.0
105.0     35.0     8.0         0.08    210000000.0

# Max displacement reached at ground level (m)
1.5609173689473692
# Max rotation reached at ground level (rad)
0.09742615046144398
# Fitting parameters
s1
s2
s3
...
s24
```

The 16 fitting parameters used in the conic function are derived from the values of the 28 (or 24) parameters presented above. They are used by the 1D FE model to conduct the 1D analysis and additionally to plot the depth variation functions in the **Calibration** and **Analysis modes** (Results inspection pane).

5.6 1D FE Model

5.6.1 Formulation of the model 1D FE Model

The 1D finite element model employed to represent the monopile, and the soil-structure interaction behaviour, is based on the use of Timoshenko beam elements combined with conforming finite elements for the soil. The numerical approach is described in Burd et al. (2020a), and Byrne et al. (2020). The implementation details provided below are based on a set of notes developed by Prof. H.J. Burd.

A 1D model of the monopile is shown in Figure 5–5 (p. 142). Plane sections of the cross-section stay plane, although cross-sections orthogonal to the centroidal axis may not remain orthogonal according to Timoshenko beam theory used to model the monopile.

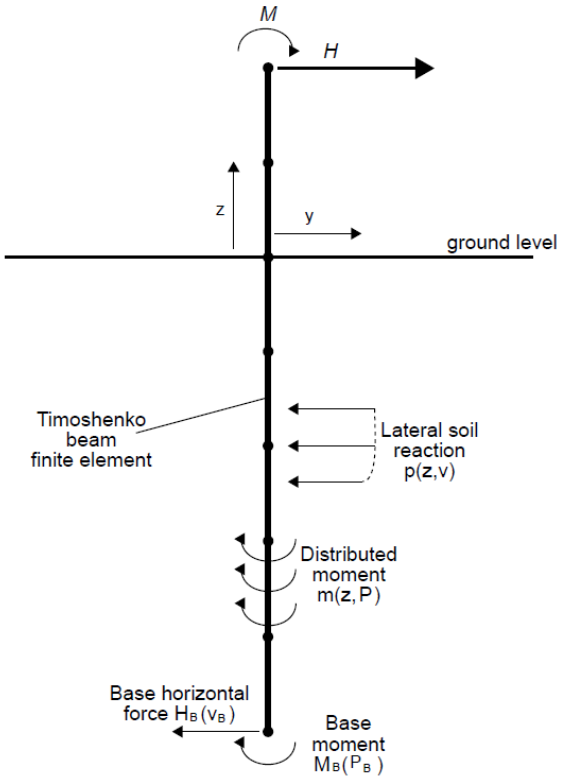


Figure 5–5: 1D structural model of monopile foundation for a wind turbine (redrawn from Byrne et al., 2020)

Kinematics

Adopted kinematics are presented in Figure 5–6 (p. 143). Eqn. 5–15 (p. 142) describes the axial and transverse displacements in a pile in a case where the neutral axis coincides with the centroid of the pile.

$$w(y,z) = y\theta(z) \qquad v(y,z) = v_0(z) + f(y) \tag{5–15}$$

θ = the clockwise rotation of the beam cross-section (assumed to remain plane)

v_o = lateral displacement of the pile centroid

$f(y)$ = function to represent the coupling between the axial and transverse strains, which are defined by:

$$\varepsilon_{zz} = \frac{\partial w}{\partial z} = y\frac{d\theta}{dz}\varepsilon_{yy} = \frac{\partial f}{\partial z}\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} = \psi + \theta\left(\text{where}\psi = \frac{\partial \nu}{\partial z}\right) \tag{5–16}$$

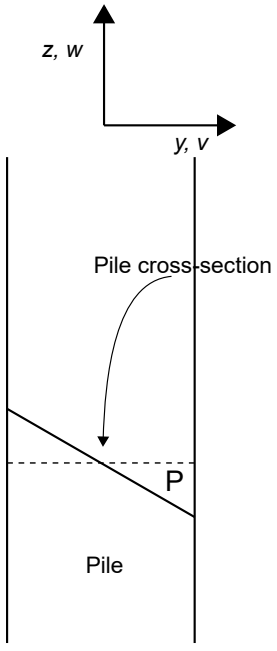


Figure 5–6: Monopile foundation for a wind turbine support structure (redrawn from Byrne et al., 2018b)

Bending moment and shear force

The axial stress in the pile is $\sigma_{zz} = E\varepsilon_{zz}$. The bending moment is:

$$M = \int \sigma_{zz} y dA = E\left[\int y^2 dA\right]\frac{d\theta}{dz} = EI\frac{d\theta}{dz} \tag{5–17}$$

> **Note:** A positive bending moment causes tension on the y-positive side of the pile.

The shear force is:

$$Q = \int G\gamma_{yz} dA = GA\kappa\left(\theta + \psi\right) \tag{5–18}$$

G = The shear modulus

A = the cross-sectional area of the pile

κ = shear factor

Virtual work

At equilibrium the total virtual work is zero. The external virtual work is:

$$\delta W_E = -H_T \delta v_T - M_T \delta \theta_T \tag{5–19}$$

H_T and M_T = horizontal force and moment applied at the top of the pile (as in Figure 5–6 (p. 143))

v_T and θ_T = the lateral displacement and cross-section rotation at the top of the pile.

The internal virtual work is:

$$\delta W_I = \int_{pile} \left(M \frac{d\delta\theta}{dz} + V\left(\delta\psi + \delta\theta\right) + p\left(z, v\right)\delta v + m\left(z, \theta\right)\delta\theta \right) dz + H_B \delta v_B + M_B \delta \theta_B \tag{5–20}$$

This may be expressed:

$$\delta W_I = \int_{\text{pile}} \left(\frac{d\delta\theta}{dz} EI \frac{d\theta}{dz} + \left(\delta\psi + \delta\theta\right) GA\kappa\left(\psi + \theta\right) + \delta v p\left(z, v\right) + \delta\theta m\left(z, \theta\right) \right) dz + \delta v_B H_B + \delta\theta_B M_B \tag{5–21}$$

Finite element discretisation

The pile is discretised into 2-noded finite elements as shown in Figure 5–7 (p. 144). The lateral displacement within each element is determined using Eqn. 5–22 (p. 144) :

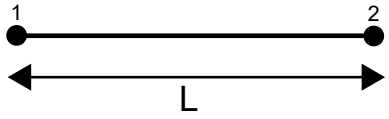


Figure 5–7: 2-noded beam element

$$v = N_1^h V_1 + N_2^h \Theta_1 + N_3^h V_2 + N_4^h \Theta_2 \tag{5–22}$$

where N_i^h are set of Hermitian shape functions given by:

$$N_1^h = 1 - 3\alpha^2 + 2\alpha^3 \tag{5–23}$$

$$N_2^h = \alpha L_e \left(1 - 2\alpha + \alpha^2\right) \tag{5–24}$$

$$N_3^h = 3\alpha^2 - 2\alpha^3 \tag{5–25}$$

$$N_4^h = \alpha L_e \left(-\alpha + \alpha^2\right) \tag{5–26}$$

where $\alpha\,(0 < \alpha < 1)$ is a parametric variable defined by the interpolation:

$$z = N_1' Z_1 + N_2' Z_2 \tag{5–27}$$

and N_i^l are the Langrangian interpolation functions:

$$N_1 = 1 - \alpha \qquad N_2 = \alpha \tag{5–28}$$

According to Astley (1992) , within each element the shear strain is constant, γ_0 and therefore $\psi = \gamma_0 - \theta$. This gives:

$$v = N_1 V_1 - N_2 \theta_1 + (N_2 + N_4)\gamma_0 + N_3 V_2 - N_4 \theta_2 \tag{5–29}$$

The first and second derivatives are defined as shown in 5.6.1 Formulation of the model 1D FE Model (p. 142).

Finite element equations for the pile

1. **Bending terms:**

The bending moment in the pile is $M = EI\frac{d\theta}{dz}$. For the current beam element formulation:

$$\frac{d\theta}{dz} = -\frac{d\psi}{dz} + \frac{d\gamma_0}{dz} = -\frac{d\psi}{dz} = -\frac{d^2v}{dz^2} \tag{5–30}$$

This may be expressed in matrix form:

$$\frac{d\theta}{dz} = \underline{B}_B \underline{V} \tag{5–31}$$

where:

$$\underline{B}_B = \left[\frac{d^2(N_1^h)}{dz^2} \quad -\frac{d^2(N_2^h)}{dz^2} \quad \frac{d^2(N_2^h+N_4^h)}{dz^2} \quad \frac{d^2(N_3^h)}{dz^2} \quad -\frac{d^2(N_4^h)}{dz^2}\right] \tag{5–32}$$

$$\underline{V}^T = [V_1 \quad \theta_1 \quad \gamma_0 \quad V_2 \quad \theta_2] \tag{5–33}$$

The terms B_B are:

$$\underline{B}_B = \frac{1}{L_e^2}[6-12\alpha \quad L_e(6\alpha-4) \quad L_e(6-12\alpha) \quad 12\alpha-6 \quad L_e(6\alpha-2)] \tag{5–34}$$

The element force vector f_B and the element stiffness matrix k_B are:

$$\underline{f}_B = \int_{\text{element}} \underline{B}_B^T M dz \qquad \underline{k}_B = \int_{\text{element}} \underline{B}_B^T EI \underline{B}_B dz \tag{5–35}$$

2. Shear terms:

The shear force within the pile element is:

$$\gamma_0 = \underline{B}_S \underline{V} \tag{5–36}$$

where:

$$\underline{B}_S = [0 \quad 0 \quad 1 \quad 0 \quad 0] \tag{5–37}$$

The corresponding element force and stiffness matrices are:

$$\underline{f}_S = \int_{\text{element}} \underline{B}_S^T V dz \qquad \underline{k}_S = \int_{\text{element}} \underline{B}_S^T \kappa G A \underline{B}_S dz \tag{5–38}$$

Finite element equations for the soil

The lateral displacement v within the element is:

$$v = \underline{B}_V V \text{where} \underline{B}_V = \begin{bmatrix} N_1^h - N_2^h & \left(N_2^h + N_4^h\right) & N_3^h - N_4^h \end{bmatrix} \tag{5–39}$$

The element force and stiffness matrices are:

$$\underline{f}_{DP} = \int_{\text{element}} \underline{B}_V^T p dz \quad \underline{k}_{DP} = \int_{\text{element}} \underline{B}_V^T \left(\frac{dp}{dv}\right) \underline{B}_V dz \tag{5–40}$$

The cross-section pile rotation θ is:

$$\theta = \underline{B}_\theta V \text{where} \underline{B}_\theta = \begin{bmatrix} \frac{dN_1^h}{dz} - \frac{dN_2^h}{dz} & \left(\frac{dN_2^h}{dz} + \frac{dN_4^h}{dz} + 1\right) & \frac{dN_3^h}{dz} - \frac{dN_4^h}{dz} \end{bmatrix} \tag{5–41}$$

The element force and stiffness matrices are:

$$\underline{f}_\theta = \int_{\text{element}} B_\theta^T m dz \underline{k}_\theta = \int_{\text{element}} B_V^T \left(\frac{dm}{d\theta}\right) \underline{B} \theta dz \tag{5–42}$$

Hermitian shape functions

$$N_1^h = 1 - 3\alpha^2 + 2\alpha^3 \tag{5–43}$$

$$N_2^h = \alpha L_e \left(1 - 2\alpha + \alpha^2\right) \tag{5–44}$$

$$N_3^h = 3\alpha^2 - 2\alpha^3 \tag{5–45}$$

$$N_4^h = \alpha L_e \left(-\alpha + \alpha^2\right) \tag{5–46}$$

The first derivatives are:

$$\frac{dN_1^h}{d\alpha} = -6\alpha + 6\alpha^2 \qquad \frac{dN_1^h}{dx} = \frac{1}{L_e}\left(-6\alpha + 6\alpha^2\right) \tag{5–47}$$

$$\frac{dN_2^h}{d\alpha} = L_e \left(1 - 4\alpha + 3\alpha^2\right) \frac{dN_2^h}{dx} = 1 - 4\alpha + 3\alpha^2 \tag{5–48}$$

$$\frac{dN_3^h}{d\alpha} = 6\alpha - 6\alpha^2 \frac{dN_3^h}{dx} = \frac{1}{L_e}\left(6\alpha - 6\alpha^2\right) \tag{5–49}$$

$$\frac{dN_4^h}{d\alpha} = L_e \left(-2\alpha + 3\alpha^2\right) \frac{dN_4^h}{dx} = -2\alpha + 3\alpha^2 \tag{5–50}$$

The second derivatives are:

$$\frac{d^2N_1^h}{d\alpha^2} = 12\alpha - 6 \frac{d^2N_1^h}{dx^2} = \frac{1}{L_e^2}\left(12\alpha - 6\right) \tag{5–51}$$

$$\frac{d^2N_2^h}{d\alpha^2} = L_e \left(6\alpha - 4\right) \frac{d^2N_2^h}{dx^2} = \frac{1}{L_e}\left(6\alpha - 4\right) \tag{5–52}$$

$$\frac{d^2N_3^h}{d\alpha^2} = 6 - 12\alpha \frac{d^2N_3^h}{dx^2} = \frac{1}{L_e^2}\left(6 - 12\alpha\right) \tag{5–53}$$

$$\frac{d^2N_4^h}{d\alpha^2} = L_e \left(6\alpha - 2\right) \frac{d^2N_4^h}{dx^2} = \frac{1}{L_e}\left(6\alpha - 2\right) \tag{5–54}$$

5.6.2 | Implementation aspects of the 1D FE model

Mesh

Pile height and embedded pile length mesh with the same element size. The maximum number of elements in each part of the pile (height and embedded length) is set to 100. Therefore, the finest mesh does not exceed a total number of 200 elements.

It should also be noted that the finite element equations of both pile and soil reaction are assembled along the same 2-noded elements of the mesh. In other words, there is no distinction between pile elements and soil elements.

Solution control and arc-length method

In the 1D model, the pile structure is modelled by linear elastic beam elements whereas a series of non-linear curves model soil reactions. This combination results in a non-linear problem which needs to be solved in a series of steps. An iteration process is performed in each step, to reduce the equilibrium error to a relatively small number. The pile-soil system might also fail at a certain level of external loads. Here, the failure is reached when the soil-reaction springs reach their ultimate capacity. Therefore, the solution method should be able to trace the post-failure response of such a system. In the current implementation of the 1D model, arc-length control is used as the solution method. The main idea of the arc-length method is that the load increment $\Delta\lambda$ is considered as an additional unknown. Among the various types of the method, Riks' formulation (Riks, 1979) is implemented. The main equations are presented. For more details on the topic, see Borst et al. (2012).

$$\Delta \underline{u}^{i+1} = \Delta \underline{u}^{i} + d\underline{u}^{i+1} \tag{5–55}$$

$$d\underline{u}^{i+1} = d\underline{u}_{/}^{i+1} + \Delta\lambda d\underline{u}_{//}^{i+1} \tag{5–56}$$

$$d\underline{u}_{|}^{i+1} = \underline{K}_{\text{elastic}}^{-1}\underline{r} \tag{5–57}$$

$$\underline{r} = \underline{F}_{ext}^{i} - \underline{F}_{\text{int}}^{i} \tag{5–58}$$

$$d\underline{u}_{//}^{i+1} = \underline{K}^{-1}\widehat{\underline{F}} \tag{5–59}$$

$$\Delta\lambda^{i+1} = \frac{\left[\Delta \underline{u}^{1}\right]^{T} d\underline{u}_{/}^{i+1}}{\left[\Delta \underline{u}^{1}\right]^{T} d\underline{u}_{//}^{i+1}} \tag{5–60}$$

$\underline{F}_{ext}^{i}$ = external applied load at iteration i

$\underline{F}_{\text{int}}^{i}$ = internal forces at iteration i

$\underline{K}_{\text{elastic}}$ = elastic stiffness matrix

$\Delta \underline{u}^{i}$ = cumulative vector of unknowns (displacements and rotation) at iteration i

$\Delta\lambda$ = load factor

$\widehat{\underline{F}}$ = unit external load

In the 1D model, the finite element equations of the pile-soil system need to be solved for different variables, namely displacements and rotations. These displacements and rotations are used to compute internal forces (shear and bending moments) which should be in equilibrium with external forces. The presence of different units for forces and moments requires an accurate convergence checking. For this purpose, an energy norm is used to check for equilibrium:

$$\text{Error} = \frac{\left(\underline{F}_{ext}^{i} - \underline{F}_{\text{int}}^{i}\right) \cdot \Delta \underline{u}^{i}}{\underline{F}_{ext}\underline{K}_{elastic}^{-1}\underline{F}_{ext}} \quad < \text{tolerance} \tag{5–61}$$

Error checking using energy norms requires tighter tolerances compared to other methods such as residual or displacement based methods. In the 1D FE calculation, a default value of 0.0001 (10^{-4})) is selected.

Automatic stepping procedure

In the 1D model, an automatic stepping scheme is adopted. The very first step size is set to the user-defined *Max load fraction* per step parameter. At the end of each step, the size of the next step is predicted. This new step size is a function of the total number of iterations required for the convergence of the current step.

If a step takes maximum 6 iterations to converge, then the next step size is twice the current one (up-scaling). On the other hand, if a step does not converge within maximum 15 iterations, then the current step size is reduced by a factor of 0.5 (down-scaling). For a certain step with a successive scale-down process, scaling-down is stopped if the total number of iterations exceeds the user-defined *Max number of iterations*.

Numerical integration

All the integral equations are evaluated using 4 Gaussian integration points. Since all the shape functions and their derivatives are functions of the parametric coordinate α, the following transformation is performed:

$$\alpha = 0.5 \cdot (1 + \xi) \tag{5–62}$$

which transforms the standard Gaussian coordinates $\xi \epsilon$ [-1, 1] to the parametric coordinates $\alpha \epsilon$ [0, 1].

Precalculation checks

The following precalculation checks are performed by the 1D FE model before the analysis is performed:

For the distributed lateral load:

1. If $k_p < k_{p,\min} \rightarrow k_p = c3 \cdot (z/D) \cdot k_{p,\min} / (k_{p,\min} - c2)$, otherwise $k_p = c2 + c3 \cdot (z/D)$
2. If $n_p < 0 \rightarrow n_p = 0$
3. If $n_p > 1 \rightarrow n_p = 1$
4. If $\overline{v}_{pu} < \bar{p}_u / k_p \rightarrow \bar{v}_{pu} = \bar{p}_u / k_p$
5. If $\bar{p}_u \leq 0$ or $k_p = 0 \rightarrow$ the output of the conic function is set to zero: $\bar{p} = 0$ and $d\bar{p}/d\bar{v}_p = 0$

Note that $k_{p,min}$ is a minimum value of the initial stiffness parameter , determined during the parameterisation. This is to prevent negative values of stiffness close to the ground level.

For the distributed moment:

1. $If\ \bar{\theta}_{mu} < \overline{m}_u / k_m \rightarrow \bar{\theta}_{mu} = \overline{m}_u / k_m$
2. If $\overline{m}_u \leq 0$ or $k_m \leq 0 \rightarrow$ the output of the conic function is set to zero: $\overline{m} = 0$ and $d\overline{m}/d\bar{\theta}_m = 0$

For the base horizontal force:

1. If $n_H < 0 \rightarrow n_H = 0$
2. If $n_H > 1 \rightarrow n_H = 1$
3. If $\overline{V}_{Hu} < \overline{H}_{Bu} / k_H \rightarrow \overline{V}_{Hu} = \overline{H}_{Bu} / k_H$
4. If $\overline{H}_{Bu} \leq 0$ or $k_H \leq 0 \rightarrow$ the output of the conic function is set to zero: $\overline{H}_B = 0$ and $d\overline{H}_B / d\bar{v}_H = 0$

For the base moment:

1. If $n_M < 0 \rightarrow n_M = 0$

2. If $n_M > 1 \rightarrow n_M = 1$
3. If $\bar{\theta}_{Mu} < \overline{M}_{Bu}/k_M \rightarrow \bar{\theta}_{Mu} = \overline{M}_{Bu}/k_M$
4. If $\overline{M}_{Bu} \leq 0$ or $k_M \leq 0 \rightarrow$ the output of the conic function is set to zero: $\overline{M}_B = 0$ and $d\overline{M}_B/d\bar{\theta}_M = 0$

Post-processing

The 1D model calculates the primary variables, i.e. displacements and rotations on the mesh nodes. The shear forces and bending moments of each finite element are computed using the interpolated primary variables on the Gauss points. These forces and moments are then extrapolated to the nodes. Finally, the values on the common nodes of the neighbouring elements are averaged to ensure only one value per node.

5.7 Results Mode

5.7.1 Realised *H*

The realised horizontal force *H* applied to the monopile head is defined as a minimum of H_{input} and the H_{input} multiplied by the *Load factor*:

$$H = \min\left(H_{\text{input}}; \text{Load factor} \cdot H_{\text{input}}\right) \tag{5–63}$$

> **Note:** The mentioned H_{input} is referred to as *H* in the user interface (**Analysis** and **Results modes**).

The *H-v* plot is based on the realised load *H*.

5.7.2 Realised *M*

The realised moment *M* applied to the monopile head is defined as a minimum of M_{input} and the M_{input} multiplied by the *Load factor*:

$$M = \min\left(M_{\text{input}}; \text{Load factor} \cdot M_{\text{input}}\right) \tag{5–64}$$

> **Note:** The mentioned M_{input} is referred to as *M* in the **Analysis** and **Results modes**.

The *M-θ* plot is based on the realised moment *M*.

5.7.3 Load factor

$$\text{Load factor} = \frac{H_{\max}}{H_{\text{input}}} = \frac{M_{\max}}{M_{\text{input}}} \tag{5–65}$$

> **Note:** In the UI, *H* and *M* are indicated as H_{input} and M_{input}

The *Load factor* is calculated based on the input horizontal load H_{input} and moment M_{input}. The input horizontal load and moment are multiplied by *3.0* to derive the values of H_{max} and M_{max} respectively. The horizontal load and moment are applied incrementally. The increments *dH* and *dM* are determined per step until the maximum applied load H_{max} or moment M_{max} are reached.

If H_{max} and/or M_{max} at the end of the calculation is less than H_{input} and/or M_{input} then the load factor will be less than *1.0* indicating that under the selected analysis settings the specified input load and/or moment cannot be fully applied.

A load factor of *3.0* means that the input load and/or moment is at least *3.0* times less than the maximum load and/or moment that can be applied to the pile, under the selected analysis settings.

5.7.4 Accuracy metrics η and ρ

An accuracy metrics, η and ρ, are computed to quantify the quality of the match between the 3D finite element calibration analyses and the 1D model. The formulation and application of the accuracy metric is described in Byrne et al. (2020).

The accuracy metrics η and ρ is displayed at the top of the graph of a selected 3D model of only the lateral load-displacements (H-v) at mudline.

The accuracy metric η is calculated as follows:

$$\eta = \frac{(A_{\text{ref}} - A_{\text{dif}})}{A_{\text{ref}}} \leq 1.0 \tag{5–66}$$

where A_{ref} is the area under the reference curve up to a specific lateral displacements threshold, i.e. the curve that corresponds to the (selected) 3D results, and A_{dif} is the area in between the curve of the 1D results and the curve of the 3D results, up to the same lateral displacements threshold (Figure 5–8 (p. 151)a)). A perfect match means the accuracy metric equals 1.0. The lateral displacements threshold is defined as the lower value of the maximum displacement reached by the 1D model and the selected 3D model.

> **Note:** Low values of the accuracy metric η (close to zero) should also be interpreted as a bad match. In all cases, the user is advised to visually inspect the compared curves and not simply rely on the computed metric.

The second accuracy metric ρ is calculated as follows:

$$\rho = \frac{H_{1D}}{H_{3D}} \tag{5–67}$$

where H_{1D} and H_{3D} are, respectively, the horizontal forces predicted by the 1D model and the 3D model for a constant value of the displacement at mudline.

A perfect match means the accuracy metric ρ equals 1.0. Values larger than 1.0 imply an over-prediction of ultimate capacity by the 1D model, while values smaller than 1.0 imply an under-prediction (Figure 5–8 (p. 151)b).

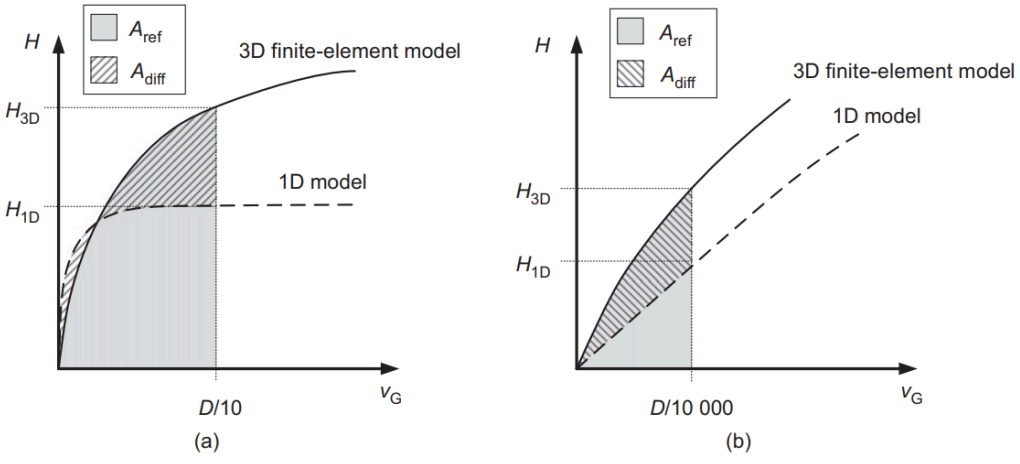


Figure 5–8: Graphic representation of accuracy metrics η and ρ for large (a) and small displacements (b) (After Burd et al., 2020a)

5.7.5 *H-v* and *M-θ*

The values of *H* and *M* that are plotted on the charts and provided in the tables are selected as follows:

- For the 1D analysis results the values of *H* and *M* are always retrieved from the values of *H* and *M* at the head (H_{head}, M_{head}) from the 1D model results.
- The radio buttons in the chart influence only the displayed lateral displacements and do not have any control over the displayed forces and moments:
  - If head is selected → v_{head} and θ_{head} are plotted
  - If head is selected → v_{head} and θ_{head} are plotted
  - If mudline is selected → $v_{mudline}$ and $\theta_{mudline}$ are plotted
  - If base is selected → v_{base} and θ_{base} are plotted
- The table for the *H-v* graph contains all the information, no matter the selected radio button. The following columns are provided:
  - Model: 1D Analysis, GeoDS_#
  - $H_{mudline}$ [kN]
  - v_{head} [m]
  - $v_{mudline}$ [m]
  - v_{base} [m]
- The table for the *M-θ* graph contains all the information, no matter the selected radio button. The following columns are provided:
  - Model: 1D Analysis, GeoDS_#
  - $M_{mudline}$ [kNm]
  - θ_{head} [rad]
  - $\theta_{mudline}$ [rad]
  - θ_{base} [rad]
- The available data from a selected 3D model are plotted only in the case that the mudline radio button is selected.

5.7.6 *v*(*z*) and *θ*(*z*) plots

The data from the last calculation step of the 1D analysis are plotted. For the plots of the data retrieved from 3D models, average values are used per monopile slice for the deflection and the cross section rotation. Note that the lateral load applied to the 1D model should be adjusted properly in order to obtain a legitimate comparison between the 1D and the corresponding 3D model.

5.7.7 Soil profile plots

The data plot on the $\sigma'_{v0}(z)$, $G_0(z)$, $s_u(z)$, $c'(z)$, $\varphi'(z)$, $\varphi(z)$, and $K_0(z)$ graphs are retrieved from the imported *dvf* file in the **Analysis mode**.

5.7.8 *p(z)* plot

The retrieving of the data from the 1D FE model output results is done as follows:

- Shaft:

- Continuous line
- Only for the last calculation step, all depths from zero to *L* are used, and the corresponding lateral soil reaction is plotted

- Base:
  - Single point
  - Only for the last calculation step, the base horizontal force data at depth *L* are plotted

5.7.9 SACS pile models

Exporting to SACS/OpenWindPower automatically creates a Pile Input file in the native format of SACS and OpenWindPower (Bentley Systems, 2021), which contains information on the monopile geometry, soil parameters, and Numerical or Parametric soil reaction curves. The different exported variables, their units, and transformations are listed in this section. For each table, the 'Transformation' column contains the SACS variable expressed in terms of PLAXIS variables.

Note: Transformations include changes of units. All files are exported using the 'MN - Metric units with forces in kiloNewtons' unit setting.

Tip: Exported Pile Input files can be inspected and modified using SACS Datagen (Bentley Systems, 2021) or any plain text editor.

5.7.9.1 Pile geometry

By default, it is assumed the pile-structure joint is located at the mudline. Thus, only the geometry below the mudline is exported. Sections, or parts of sections, above $z = 0$ are ignored.

Table 5–7: Mapping of pile geometry parameters

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
Load eccentricity (*h*)	[m]	Pile head height	[m]	Not modelled. It is assumed the pile-structure joint is located at the mudline.
Embedded length (*L*)	[m]	Pile segment length	[m]	• For Calibration models (GeoDS), a single pile segment is generated, ranging from $z = 0$ to $z = -L$ • For 3D verification models, one segment is generated per each thickness section between z = 0 and z = -L, where Segment length = | Bottom – Top |

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
Outside diameter (D_{out})	[m]	Outside diameter	[cm]	D_{out} * 100
Thickness (t)	[m]	Wall thickness	[cm]	t * 100
Unit weight ($\gamma = 0$)	[kN/m^3]	-	-	Only used in 2.4.7 Modal Analysis (p. 65)
Elasticity modulus (E)	[kN/m^2]	Elasticity modulus (E)	[MN/cm^2]	E * 10 $^{-7}$
Poisson's ratio (v)	-	-	-	Not used
-	[kN/m^2]	Shear modulus (G)	[MN/cm^2]	E / [2 * (1 + v)] * 10 $^{-7}$
-	[kN/m^2]	Yield stress (SY)	[kN/cm^2]	36.0 (Assumed)
Cross-section area (A)	[m^2]	Available end bearing area	[m^2]	A (Assuming unplugged behaviour)

5.7.9.2 | Soil reaction curves

Since the PISA method only accounts for horizontal loading, only horizontal and rotational soil reaction curves are exported from PLAXIS Monopile Designer Vertical soil reaction curves are internally generated in SACS/OpenWindPower from the soil parameter data in PLAXIS Monopile Designer.

Horizontal and rotational curves

For each monopile model, either the Numerical or the Parametric soil reaction curves can be exported.

- In **Calibration mode**:
  - *Numerical*: exports the raw soil reaction curves from 3D FEM. Requires the GeoDS to be *Calculated*.
  - *Parametric*: exports the parameterised soil reaction curves, obtained from the depth variation functions and denormalised from the formulae presented in Table 5–1 (p. 126): Normalisation formulae for Sand and Clay (Burd et al., 2020a, Byrne al., 2020). Requires the 1D analysis to be *Calculated*.
- In **Analysis mode**:
  - *Numerical*: exports the raw soil reaction curves from the 3D design verification model. Requires the 3D verification model to be *Calculated*.
  - o Parametric: exports the parameterised soil reaction curves, obtained from the depth variation functions and denormalised from the formulae presented in Table 5–1 (p. 126): Normalisation formulae for Sand and Clay (Burd et al., 2020a, Byrne al., 2020). Requires the 1D analysis to be *Calculated*.

Before exporting, soil reaction curves are simplified to a maximum of 30 data points, while preserving their general shape.

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
p	[kN/m]	P	[kN/cm]	p / 100
v	[m]	Y	[cm]	v * 100
m	[kNm/m]	m	[kNm/m]	m / 100
θ	[rad]	T	[rad]	θ
H_B	[kN]	BH	[kN]	H_B
v_B	[m]	V	[cm]	v_B * 100
M_B	[kNm]	BM	[kNm]	M_B
θ_B	[rad]	BT	[rad]	θ_B

Note: For versions previous to SACS 2023.2 the variable m in PLAXIS is denoted as M in SACS with corresponding units of [kN.m/cm].

Vertical curves

Conventional vertical soil reaction curves ('t-z' and 'Q-z' curves) are automatically generated in SACS/OpenWindPower from a series of basic input parameters, following the API RP 2A-WSD (API, 2014). These are calculated from the Soil parameters in PLAXIS Monopile Designer via the following relations:

Sand parameters:

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
Slice top	[m]	Top of stratum	[m]	| Top |
Slice bottom	[m]	Bottom of stratum	[m]	| Bottom |
Earth gravity (g)	[m/s^2]	-	-	Not used
-	-	Location	-	B - Below water table
Submerged unit weight (γ')	[kN/m^3]	Submerged density	[t/m^3]	γ' / g
Shear stiffness (G_0)	[kN/m^2]	-	-	Not used
Effective internal friction angle (φ')	[deg]	Friction angle between pile and soil (delta)	[deg]	φ' * 0.9 (Liu et al., 2019)

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
		Soil type	[-]	• $\varphi' < 22.5$: SILT • $2.5 \leq \varphi' < 27.5$: SNSL • $27.5 \leq \varphi' < 32.5$: SLSN • $32.5 \leq \varphi' < 37.5$: SAND • $37.5 \leq \varphi'$: GRAV
Dilation angle (psi)	[deg]	-	-	Not used
Coefficient of lateral earth pressure at rest (K_0)	-	Coefficient of lateral earth pressure [at failure] (K)	-	0.8 (Recommended value for unplugged piles)
-	-	Limiting end bearing cap.	[kN/ cm^2]	Default: From soil type
-	-	Overburden pressure	[kg/ cm^2]	Default: Internal calculation
-	-	Limiting skin friction	-	Default: From soil type

Clay parameters:

Table 5–8: PLAXIS-SACS mapping of soil variables (clay)

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
Slice top	[m]	Top of stratum	[m]	| Top |
Slice bottom	[m]	Bottom of stratum	[m]	| Bottom |
Earth gravity (g)	[m/s^2]	-	-	Not used
Submerged unit weight (γ')	[kN/m^3]	Submerged density	[t/m^3]	γ' / g
Shear stiffness (G_0)	[kN/m^2]	-	-	Not used
Undrained shear strength ($s_{u,top}$, $s_{u,bottom}$)	[kN/m^2]	Undrained shear strength	[kN/ cm^2]	($s_{u,top}$ + $s_{u,bottom}$) / 2 * 10^{-4}
Coefficient of lateral earth pressure at rest (K_0)	-	-	-	Not used
-	-	Soil type	-	CLAY

Variable in PLAXIS	Units	Variable in SACS	Units	Transformation
-	-	Overburden pressure	[kg/ cm^2]	Default: Internal calculation

6

References

1. API RP 2A-WSD 22nd Ed (2014). API Recommended Practice 2A-WSD. Planning, Designing, and Constructing Fixed Offshore Platforms-Working Stress Design. 22nd Edition. American Petroleum Institute Publishing Services. Washington, DC.
2. Andresen, L., Jostad, H.P. (1999). Application of an anisotropic hardening model for undrained response of saturated clay. Proc. NUMOG VII, 581–585.
3. Astley, R. (1992). Finite elements in solids and structures. Chapman and Hall Publishers.
4. Bentley Systems (2020). SACS Pile-Soil Interaction Manual. SACS CONNECT Edition Version 15.1. Bentley Systems, New Orleans, LA.
5. Borst, R.D., Crisfield, M.A., Remmers, J.J.C., Verhoosel, C.V. (2012). Non-Linear Finite Element Analysis of Solids and Structures. Wiley Online Library, USA.
6. Brinkgreve, R.B.J., Lisi, D., Lahoz, M., Panagoulias, S. (2020). Validation and application of a new software tool implementing the PISA design methodology. In J. Mar. Sci. Eng. 2020, 8(6), 457.
7. Brinkgreve, R.B.J., Engin, E., Engin, H.K. (2010). Validation of empirical formulas to derive model parameters for sands. 7th NUMGE, Trondheim, Norway.
8. Burd, H.J., Taborda, D.M.G., Zdravkovic, L., Abadie, C.N., Byrne, B.W., Houlsby, G.T., Gavin, K., Igoe, D., Jardine, R.J., Martin, C.M., McAdam, R.A., Pedro, A.M.G., Potts, D.M. (2020a). PISA Design Model for Monopiles for Offshore Wind Turbines: Application to a Marine Sand. In Géotechnique 2020 70:11, 1048-1066.
9. Burd, H.J., Abadie, C.N., Byrne, B.W., Houlsby, G.T., Martin, C.T., McAdam, R.A., Jardine, R.J., Pedro, A.M.G., Potts, D.M., Taborda, D.M.G., Zdravkovic, L., Pacheco Andrade, M. (2020b). Application of the PISA design model to monopiles embedded in layered soils. In Géotechnique 2020 70:11, 1067-1082.
10. Burd, H.J., Byrne, B.W., McAdam, R.A., Houlsby, G.T., Martin, C.M., Beuckelaers, W.J.A.P., Zdravkovic, L., Taborda, D.M.G., Potts, D.M., Jardine, R.J., Gavin, K., Doherty, P., Igoe, D., Skov Gretlund, J., Pacheco Andrade, M., Muir Wood, A. (2017). Design aspects for monopile

foundations. In proceedings of TC209 Workshop - Foundation Design of Offshore Wind Structures, At Seoul, Korea, 35–44.

11. Byrne, B.W., Houlsby, G.T., Burd, H.J., Gavin, K., Igoe, D., Jardine, R.J., Martin, C.M., McAdam, R.A., Potts, D.M., Taborda, D.M.G., Zdravkovic, L. (2020). PISA design model for monopiles for offshore wind turbines: Application to a stiff glacial clay till. Géotechnique 2020 70:11, 1030-1047
12. Byrne, B.W. (2020). Editorial: geotechnical design for offshore wind turbine monopiles. In Géotechnique 2020 70:11, 943-944.
13. Byrne, B.W., Burd, H.J., Kenneth, G., Houlsby, G.T., Jardine, R.J., McAdam, R.A., Martin, C.M., Potts, D.M., Taborda, D.M.G., Zdravkovic, L. (2018). PISA: Recent Developments in Offshore Wind Turbine Monopile Design. In Vietnam Symposium on Advances in Offshore Engineering.
14. Byrne, B.W., McAdam, R.A., Burd, H.J., Houlsby, G.T., Martin, C.M., Beuckelaer, W.J.A.P., Taborda, L.Z.D.M.G., Jardine, R.J., Liu, E.U.T., Abadias, D., Gavin, K., Igoe, D., Doherty, P., Skov Gretlund, J., Pacheco Andrade, M., Muir Wood, A., Schroeder, F.C., Turner, S., Plummer, M.A.L. (2017). PISA: new design methods for offshore wind turbine monopiles. In proceedings of the Offshore Site Investigation and Geotechnics International Conference (OSIG), 142–161.
15. Byrne, B.W., McAdam, R.A., Burd, H.J., Houlsby, G.T., Martin, C.M., Gavin, K., Doherty, P., Igoe, D., Zdravkovic, L., Tabordo, D.M.G., Potts, D.M., Jardine, R.J., Sideri, M., Schroeder, F.C., Muir Wood, A., Kellehave, D., Skov Gretlund, J. (2015a). Field testing of large diameter piles under lateral loading for offshore wind applications. Third International Symposium on Frontiers in Offshore Geotechnics (ISFOG 2015).
16. Byrne, B.W., McAdam, R.A., Burd, H.J., Houlsby, G.T., Martin, C.M., Zdravkovic, L., Taborda, D.M.G., Potts, D.M., Jardine, R.J., Sideri, M., Schroeder, F.C., Gavin, K., Doherty, P., Igoe, D., Muir Wood, A., Kellahave, D., Skov Gretlund, J. (2015b). New design methods for large diameter piles under lateral loading for offshore wind applications. In proceedings of Third International Symposium on Frontiers in Offshore Geotechics, volume 1, 705–7010.
17. Kaltekis, K., Panagoulias, S., van Dijk, B.F.J., Brinkgreve, R.B.J., Ramos da Silva, M. (2019). Comparative concept design study of laterally loaded monopiles. In: Journal of Physics: Conference Series. 2019; Vol. 1222, No. 1.
18. Jardine, R., Chow, F., Overy, R. (2005). ICP design methods for driven piles in sands and clays. Thomas Telford, London.
19. Liu, J., Guo, Z., Han, B. (2019). Load transfer of offshore open-ended pipe piles considering the effect of soil plugging. In J. Mar. Sci. Eng. 2019, 7, 313.
20. Palix, E., Willems, T., Kay, S. (2011). Caisson capacity in clay: VHM resistance envelope - part 1: 3D FEM numerical study. Frontiers in Offshore Geotechnics (ISFOG), 753–758.
21. Panagoulias, S., Nernheim, A., Lisi, D., Lahoz, M., Brinkgreve, R.B.J. (2020). Concept design study of laterally loaded monopiles in sand. In Proceedings of the 4th International Symposium on Frontiers in Offshore Geotechnics (ISFOG 2021), Austin, TX.
22. Panagoulias, S., Hosseini, S., Brinkgreve, R.B.J., Burd, H.J. (2019). Design of laterally-loaded monopiles in layered soils. 2nd International Conference on Natural Hazards & Infrastructure (ICONHIC), Chania, Greece.
23. Panagoulias, S., Brinkgreve, R.B., Minga, E., Burd, H.J., McAdam, R.A. (2018a). Application of the PISA framework to the design of offshore wind turbine monopile foundations. WindEurope 2018 Conference at the Global Wind Summit, Hamburg, Germany.
24. Panagoulias, S., Hosseini, S., Brinkgreve, R. (2018b). An innovative design methodology for offshore wind monopile foundations. 26th European Young Geotechnical Engineers Conference, Graz, Austria.
25. Riks, E. (1979). An incremental approach to the solution of snapping and buckling problems. International Journal of Solids and Structures, 524–551.
26. Seequent (2026). PLAXIS 3D - Materials Models Manual. PLAXIS 2026.1. Bentley Systems, The Netherlands.

27. Thieken K., Achmus, M., Lemke, K. (2015). A new static p-y approach for piles with arbitrary dimensions in sand. In geotechnik 2015 38:4, 267-288.

28. Winkler, E. (1867). Die Lehre von Elasticität und Festigkeit. H. Dominicus, Prague.

29. Zdravkovic, L., Taborda, D., Potts, D., Jardine, R., Sideri, M., Schroeder, F.C., Byrne, B.W., Burd, R.A.M.H.J., Houlsby, G.T., Martin, C.M., Gavin, K., Doherty, P., Igoe, D., Muir Wood, A., Kellahave, D., Skov Gretlund, J. (2015). Numerical modelling of large diameter piles under lateral loading for offshore wind applications. In proceedings of Third International Symposium on Frontiers in Offshore Geotechnics (ISFOG 2015), 1, 759–764.

7

Appendices

A

Warnings and errors

A.1 Calibration Mode - Warnings and errors

In the **Calibration mode**, when the Generate, Calculate and Parameterise actions are performed, error or warning messages could be displayed from the tool. The details are explained in the following sections.

A.1.1 Generate

Table A–1: Checks and feedback for model generation

Condition	Severity	Message
Monopile embedded length *L* plus a distance of 0.15 times the pile diameter (D_{out}) equals or exceeds the maximum available length based on the specified bottom depth of the last soil layer in the **Soil profile** tabsheet.	Error	In the following models, the selected embedded monopile length *L* is very close to or meets the bottom soil boundary. Please enter a proper value to continue
A check is performed for the thickness of the layers. Zero thickness is not allowed	Error	Invalid values for the depth parameters in the **Calibration mode**. Please correct them to continue

Condition	Severity	Message
A check is performed to prohibit the following: • L = 0.0 m • h = 0.0 m • D_{out} = 0.0 m • t = 0.0 m • E = 0.0 kN/m^2	Error	Invalid values for the parameters in the **Calibration mode**. Please correct them to continue.
No soil layers are present in the **Soil profile** tabsheet.	Error	There are no soil layers present in the **Calibration mode**. Please define at least one soil layer to continue
The thickness of a single soil layer should not be less than 0.5 m. This is to prevent ending up with bad mesh qualities and many mesh elements	Error	The thickness of a single soil layer cannot be less than 0.5 m. Please use greater thickness to continue
The model is already successfully generated and the input parameters are altered (or not)	Error	The following models will be regenerated. Any manual modifications to these models may be completely or partially lost
The model is already successfully generated and the input parameters are altered (or not)	Warning	The following models will be regenerated. Any manual modifications to these models may be completely or partially lost
Monopile embedded length L exceeds the recommended max length based on the specified bottom depth of the last soil layer in the **Soil profile** tabsheet.	Warning	The recommended maximum pile length is 70% of the soil depth, specified in the **Calibration mode**. In the following models, this value is exceeded. Results may be incorrect
Relative target displacement at the mudline v_g/D_{out} equals zero	Warning	The value assigned to the parameter v_g/D_{out} is zero. Assign a higher value for valid calculation results
A high value is assigned to the relative target displacement at the mudline v_g/D_{out}, which exceeds the recommended value. Refer to Suggested Values of vg/Dout (p. 123) for more information.	Warning	A value lower than 0.3 (GeoDS_1), 0.3 (GeoDS_2), etc. is suggested to be used for the parameter v_g/D_{out}. The currently assigned value may result in excessive lateral displacement at ground level. This may lead to a long computation time

Condition	Severity	Message
A low value is assigned to the prescribed displacement, probably not high enough to result in the required displacement at ground level	Warning	A value higher than 0.15(GeoDS_1), (GeoDS_2), etc. is suggested to be used for the parameter v_g/D_{out}. The currently assigned value may result in insufficient lateral displacement at ground level. See 5.3 PLAXIS 3D Models (p. 120) for more information.
The distance above the toe of the pile (depth equal to L) and the closest layer boundary is less than 0.25 m or the distance below the toe of the pile (depth equal to L), and the closest layer boundary is less than $0.2 \cdot D_{out}$	Warning	The distance between the toe of the monopile and the closest soil boundary is small. This might lead to bad mesh quality and inaccuracy of results

Table A–2: Checks and feedback for model generation for Sand

Condition	Severity	Message
$G_{ur}/s_u^A > 2000$	Error	The ratio of the shear modulus over the average active shear strength is too high, exceeding 2000. Please decrease the value of the shear modulus or increase the value of the shear strength
$G_{ur}/s_u^A < 25$	Error	The ratio of the shear modulus over the average active shear strength is less than 25. Please increase the value of the shear modulus or decrease the value of the shear strength
$s_{u,top}$ = 0 or $s_{u,bottom}$ =0	Error	The shear strength is zero

Table A–3: Checks and feedback for model generation for Sand

Condition	Severity	Message
$\varphi' = 0$	Error	The friction angle is zero
$\psi > \varphi'$	Error	The dilatancy angle exceeds the friction angle
$G_0^{ref} < 66800\ kN/m^2$	Warning	The value of the small strain shear modulus is too low. Please consider inspecting the material parameters after generating the PLAXIS 3D model

Condition	Severity	Message
$G_0^{ref} > 128000\ kN/m^2$	Warning	The value of the small strain shear modulus is too high. Please consider inspecting the material parameters after generating the PLAXIS 3D model
$1 < \frac{G_0^{ref}}{G_{ur}^{ref}} < 20$ $where:$ $G_{ur}^{ref} = E_{ur}^{ref}/(2 \cdot (1 + \upsilon_{ur}))$	Error	The value of the small strain shear modulus is outside the allowable range. Please use a different value
$G_0^{ref} or\ \phi\prime\ too\ low\ kN/m^2$	Error	The value of the small strain shear modulus or of the friction angle is too low. Please use higher values
$G_0^{ref} or\ \phi\prime\ too\ high\ kN/m^2$	Error	The value of the small strain shear modulus or of the friction angle is too high. Please use lower values

A.1.2 | Regenerate

Any changes since the model was last generated/calculated/parameterised are detected. The purpose of this approach is to maintain most changes that the user might have done manually through PLAXIS 3D.

The model modifications during regeneration, following the user actions (per mode), are presented in Table B.4.

Table A–4: Model regeneration

Mode	User action	PLAXIS 3D regeneration procedure
Soil	Change material type or soil parameters	All existing soil and interface materials are deleted and regenerated based on the new soil material type and the associated material parameters. The new materials are assigned to the already existing soil layers and interfaces
Soil	Add/delete/insert soil layer or modify the top/bottom layer boundaries	All the existing soil layers and the accompanying soil and interface materials are deleted. New layers and materials are generated based on the user input
Calibration	Change structure parameters (*t*, v_{max} , *E*, *v*)	The plate material assigned to the structure is modified, and the prescribed displacement applied to the top of the pile is readjusted
Calibration	Change geometry parameters (*L*, D_{out} , *h*)	The structure based on the modified parameter(s) is regenerated, without entirely deleting and recreating it. Note that a change in D_{out} also affects the size of the soil contour

A.1.3 | Calculate

Table A–5: Checks and feedback for model calculation

Condition	Severity	Message
Model is not successfully generated	Error	The following models are not successfully generated and can therefore not be calculated
Model is successfully generated or already calculated and input parameters are altered	Error	The input parameters of the following models are modified. The models should be regenerated before calculation
Model is already successfully calculated	Warning	The following models have already been calculated. Recalculating them is not necessary unless manual modifications to these models have been performed

A.1.4 | Parameterise

Before parameterisation begins, PLAXIS Monopile Designer might give the following error and warning if the corresponding condition is met:

Table A–6: Checks and feedback before model parameterisation

Condition	Severity	Message
Model is not successfully calculated	Error	The following models have not been calculated successfully and can therefore not be included in the parameterisation
Model is successfully calculated and input parameters are altered	Error	The input parameters of the following models have been modified since the models were calculated. The models should be regenerated and recalculated before trying to parameterise

During parameterisation, the Optimisation Module may give the following errors or warnings:

Table A–7: Checks and feedback during model parameterisation

Severity	Message
Error	Unrecognised soil type. The depth variation functions cannot be derived
Error	None of the data sets has advanced enough to be used for the definition of ultimate load values. The depth variation functions cannot be derived

Severity	Message
Warning	The 1D calculation kernel did not run successfully in the background under the applied default settings. The parameterised soil reaction curves and the depth variation functions cannot be displayed. After the parameterisation is performed, the produced calibrated.dvf file can be used in the **Analysis mode** to carry out the 1D numerical analysis. Note that the default expert settings might need to be changed based on the error messages that will be displayed

A.1.5 | Additional checks and feedback for PLAXIS 3D models

Table A–8: Checks and feedback for model calculation

Condition	Severity	Message
The displacement at the top of the monopile is fully reached but not yet at the mud-level	Warning	The calculation finished successfully. The prescribed input displacement is not fully applied but the target displacement at the reference level is reached.

A.2 | Analysis Mode - Warnings and errors

Table A–9: Analysis mode errors, warnings and success messages

Severity	Message
Error	Unknown error
Error	Unknown parameterisation function type
Error	Unknown soil material type
Error	The dvf file cannot be opened
Error	The input file cannot be opened
Error	The minimum element size should be less than the pile height
Error	The minimum element size should be less than the pile length
Error	The pile base defined in the thickness layers table should match the pile length
Error	Please provide input and output paths as command line arguments
Error	Invalid number of soil layers in the dvf file
Error	Invalid number of GeoDS in the dvf file
Error	Mismatch between the pile layers definition and specified pile length/height

Severity	Message
Error	The embedded part of the pile is outside the soil layers
Error	The pile base is outside the soil layers
Error	There is no thickness assigned to the pile
Error	The pile length is zero. Please use a higher value
Error	The pile outer diameter is zero. Please use a higher value
Error	The pile thickness is zero. Please use a higher value
Error	The workload is not defined correctly. Please use a value greater than zero
Error	The Young's modulus is not defined correctly. Please use a value greater than zero
Warning	Not enough load steps to reach the user specified load
Warning	Not enough load steps for the calculation of the load factor
Warning	The mudline lateral displacement exceeds the maximum value obtained by calibration
Warning	The mudline rotation exceeds the maximum value obtained by calibration
Success	The maximum ground level displacement is reached. The calculation is finished successfully
Success	The maximum load factor is reached. The calculation is finished successfully

B

Scripting Reference

B.1 Commands Reference

1. **Server object commands:**
   - **new(location):** Creates a new project at the specified location.

     ```
     s.new(r'D:\example\New.plxmdt')
     ```

   - **open(location):** Loads an existing project.

     ```
     s.open(r'D:\example\Existing.plxmdt')
     ```

2. **Global object commands:**
   - ***append(**kwargs):*** Adds a new item at the bottom of a list.

     ```
     g.Analysis.SoilReactionCurves.append(location=r'D:\example
     \Clay.dvf')
     ```

     ```
     g.Analysis.SoilLayers.append(Bottom=-60.0,
      SoilReactionCurveFilename='Sand.dvf')
     ```

     ```
     g.Analysis.Segments.append(t=0.05)
     ```

- ***calculate( ):*** Runs the 1D analysis. Returns the analysis feedback. [Deprecated]

```
g.Analysis.calculate()
```

- ***static_analysis( ):*** Runs the static analysis. Returns the analysis feedback messages.

```
g.Analysis.static_analysis()
```

- ***modal_analysis():*** Runs the modal analysis. Returns the modal analysis feedback messages.

```
g.Analysis.modal_analysis()
```

- ***del( ):*** Deletes an item in a list.

```
del g.Analysis.SoilLayers[5]
```

```
del g.Analysis.Segments[-1]
```

- ***insert(position, **kwargs):*** Inserts a new layer or segment at the specified position in the list.

```
g.Analysis.SoilLayers.insert(2, Bottom=-60.0,
 SoilReactionCurveFilename='Sand.dvf')
```

```
g.Analysis.Segments.insert(1, Bottom=10, t=0.05)
```

- ***getresults(ResultType, Model):*** Returns an object with the calculation results for the selected ResultType and Model.

```
g.getresults(g.ResultTypes.MonopileResponseDepthVariation,
 g.Models.Analysis1D)
```

- ***save( ):*** Saves the project to the current location (if not specified) or to the specified location and name.

```
g.save()
```

```
g.save(r'D:\example\SaveAs.plxmdt')
```

B.2 | Object Reference

The global object (denoted g in this Manual) is used to view and change data in the project. It contains the following objects:

1. ***Calibration*** : Settings of the **Calibration mode**, including Structural properties for the monopile.
    - ***Calibration.E:*** The Young's modulus of the monopile material.
    - ***Calibration.nu:*** The Poisson's ratio of the monopile material.
    - ***Calibration.w:*** The pile unit weight.
2. ***Calibration.GeometryDataSets*** : List containing all the geometry data sets for the **Calibration mode**.
    - ***Calibration.GeometryDataSets*** : A specific geometry data set in the calibration mode.
        - ***Calibration.GeometryDataSets[i].Identification*** : The user defined identification of the geometry data set.
        - ***Calibration.GeometryDataSets[i].Name*** : The unique identification of the geometry data set [Read-only].
        - ***Calibration.GeometryDataSets[i].Dout*** : The outer diameter of the monopile.
        - ***Calibration.GeometryDataSets[i].L*** : The embedded length of the monopile.
        - ***Calibration.GeometryDataSets[i].h*** : Height of application of the resultant horizontal load.
        - ***Calibration.GeometryDataSets[i].t*** : The monopile thickness.
        - ***Calibration.GeometryDataSets[i].vgDoutRatio*** : The target displacements over diameter ratio at mudline.
3. ***Calibration.SoilProfiles*** : List containing all the soil profiles for the **Calibration mode**.
    - ◦ ***Calibration.SoilProfile*** : A soil profile in the **Calibration mode**.
        - ***Calibration.SoilProfiles[i].Identification*** : The user defined identification of the soil profile.
        - ***Calibration.SoilProfiles[i].ID*** : The unique identification of the soil profile.
        - ***Calibration.SoilProfiles[i].SoilLayers[j]*** : A specific soil layer for a soil profile.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].Bottom*** : The soil layer bottom elevation.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].Top*** : The soil layer top elevation.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].G0*** : The small strain shear stiffness modulus in the middle of the soil layer.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].GammaSub*** : The soil layer submerged unit weight.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].K0*** :The soil layer lateral earth pressure coefficient at rest.
            - ***Calibration.SoilProfiles[i].SoilLayers[j].Phi*** : The soil layer effective angle of internal friction (only for sand).
            - ***Calibration.SoilProfiles[i].SoilLayers[j].Psi*** : The soil layer angle of dilatancy (only for sand).
            - ***Calibration.SoilProfiles[i].SoilLayers[j].SuBottom*** : The undrained shear strength at the bottom of the soil layer (only for clay).
            - ***Calibration.SoilProfiles[i].SoilLayers[j].SuTop*** : The undrained shear strength at the top of the soil layer (only for clay.)
4. ***Calibration.Matrix*** : Matrix containing all the calibration matrix cells.

- ***Calibration.Matrix.parameterise([Calibration.Matrix[i][j], ...])*** : Runs the parameterisation with all the PLAXIS 3D models that are passed on the list.
- ***Calibration.Matrix.parameterise([(i, j), ...])*** : Runs the parameterisation with all the PLAXIS 3D models corresponding to the tuples (row, column) passed on the list.
- ***Calibration.Matrix[i][j].Row*** : The column of the cell (0 based index).
- ***Calibration.Matrix[i][j].Column*** : The row of the cell (0 based index).
- ***Calibration.Matrix[i][j].Changed*** : Boolean indicator that is True when there are changes on the conditions used when the PLAXIS 3D model was generated.
- ***Calibration.Matrix[i][j].Optimized*** : Boolean indicator that is True when there are changes on the conditions used when the PLAXIS 3D model was generated.
- ***Calibration.Matrix[i][j].State*** : The state of the PLAXIS 3D model.
- ***Calibration.Matrix[i][j].generate()*** : Generates the PLAXIS 3D model.
- ***Calibration.Matrix[i][j].calculate()*** : Calculates the PLAXIS 3D model.
- ***Calibration.Matrix[i][j].view()*** : Open the PLAXIS 3D model for visualization on PLAXIS 3D.
- ***Calibration.Matrix.Rows*** : List of lists of matrix cells grouped per row.
    - ***Calibration.Matrix.Rows[i].Cells*** : The list of cells of the i^{th} row (0 based index).
        - ***Calibration.Matrix.Rows[i].Identification*** : The user defined identification of the i^{th} row (0 based index).
        - ***Calibration.Matrix.Rows[i].Name*** : The unique identification of the of the i^{th} row (0 based index) [Read-only].
        - ***Calibration.Matrix.Rows[i].Cells*** : The list of cells of the i^{th} row (0 based index).
        - ***Calibration.Matrix.Rows[i].Cells[j].Row*** : same as *Calibration.Matrix[i][j].Row*.
        - ***Calibration.Matrix.Rows[i].Cells[j].Column.*** : same as *Calibration.Matrix[i][j].Column*.
        - ***Calibration.Matrix.Rows[i].Cells[j].Changed*** : same as *Calibration.Matrix[i][j].Changed*.
        - ***Calibration.Matrix.Rows[i].Cells[j].Optimized*** : same as *Calibration.Matrix[i][j].Optimized*.
        - ***Calibration.Matrix.Rows[i].Cells[j].State*** : same as *Calibration.Matrix[i][j].State*.
        - ***Calibration.Matrix.Rows[i].Cells[j].generate()*** : same as *Calibration.Matrix[i][j].generate()*.
        - ***Calibration.Matrix.Rows[i].Cells[j].calculate()*** : same as *Calibration.Matrix[i][j].calculate()*.
        - ***Calibration.Matrix.Rows[i].Cells[j].view()*** : same as *Calibration.Matrix[i][j].view()*.
- ***Calibration.Matrix.Columns*** : List of lists of matrix cells grouped per column.
    - ***Calibration.Matrix.Columns[i].Cells*** : The list of cells of the i^{th} column (0 based index).
        - ***Calibration.Matrix.Columns[i].Identification*** : The user defined identification of the i^{th} column (0 based index).
        - ***Calibration.Matrix.Columns[i].Name*** : The unique identification of the of the i^{th} column (0 based index) [Read-only].
        - ***Calibration.Matrix.Columns[i].Cells:*** The list of cells of the i^{th} column (0 based index).
        - ***Calibration.Matrix.Columns[i].Cells[j].Row*** : same as *Calibration.Matrix[i][j].Row*.
        - ***Calibration.Matrix.Columns[i].Cells[j].Column.*** : same as *Calibration.Matrix[i][j].Column*.

        - ***Calibration.Matrix.Columns[i].Cells[j].Changed*** : same as *Calibration.Matrix[i][j].Changed*.
        - ***Calibration.Matrix.Columns[i].Cells[j].Optimized*** : same as *Calibration.Matrix[i][j].Optimized*.
        - ***Calibration.Matrix.Columns[i].Cells[j].State*** : same as *Calibration.Matrix[i][j].State*.
        - ***Calibration.Matrix.Columns[i].Cells[j].generate()*** : same as *Calibration.Matrix[i][j].generate()*.
        - ***Calibration.Matrix.Columns[i].Cells[j].calculate()*** : same as *Calibration.Matrix[i][j].calculate()*.
        - ***Calibration.Matrix.Columns[i].Cells[j].view()*** : same as *Calibration.Matrix[i][j].view()*.
5. ***Analysis*** : Settings of the **Analysis mode**, including Monopile geometry, Structural properties and Calculation Type (workload or target displacement).
    - ***Analysis.CalculationType = "workload" or 1***
        - ***Analysis.H*** : Horizontal force to be applied at the Monopile head.
        - ***Analysis.M*** : Moment to be applied the monopile head.
        - ***Analysis.Mg*** : Moment at ground level [Read-only].
    - ***Analysis.CalculationType = "targetdisplacement" or 2***
        - ***Analysis. VgDoutRatio*** : target displacement at ground level.
    - ***Analysis.Dout*** : Outer diameter of the monopile.
    - ***Analysis.E*** : Young's modulus of the monopile material.
    - ***Analysis.H*** : Horizontal force to be applied in the monopile.
    - ***Analysis.L*** : Embedded length of the monopile.
    - ***Analysis.LumpedMass*** : Lumped mass used when running modal analysis.
    - ***Analysis.LumpedMassHorizontalOffset*** : Lumped mass horizontal offset (x-axss) for the modal analysis.
    - ***Analysis. LumpedMassOutOfPlaneOffset*** : Lumped mass out-of-plane offset(y-axis) for the modal analysis.
    - ***Analysis.LumpedMassVerticalOffset*** : Lumped mass vertical offset (z-axis) for the modal analysis.
    - ***Analysis.MaxDisplacementDiameterRatio*** : Stop criteria for diverging iterations (displacement refers to the lateral displacement of the monopile at mudline).
    - ***Analysis.MaxLoadFractionPerStep*** : Controls the size of the load step applied during the calculation.
    - ***Analysis.MaxNumberOfIterations*** : Maximum allowable number of iterations within any individual calculation step.
    - ***Analysis. MaxRotationAtMudline*** : Stop criterion for the rotation of the monopile at ground level.
    - ***Analysis.MaxSteps*** : The maximum number of calculation steps (load steps) that are performed during the 1D calculation.
    - ***Analysis.MaximumCutOffFrequency*** : Maximum cut-off frequency used when running modal analysis.
    - ***Analysis.MaximumNumberOfModes*** : Maximum number of modes to search for when running modal analysis.
    - ***Analysis.MinMonopileSectionLength*** : Minimum monopile section length used to discretise the monopile into beam elements.
    - ***Analysis.MonopileHead*** : Height of the monopile head.
    - ***Analysis.ToleratedError*** : The error that will be tolerated within each step of the calculation.
    - ***Analysis.TowerTop*** : Height of the top of the tower.

- ***Analysis.w*** : Unit weight of the material.

6. ***Analysis.SoilLayers*** : List containing all soil layers defined on the analysis Soil profile.
   - ***Analysis.SoilLayer*** : A specific soil layer.
     - ***Analysis.SoilLayers[i].Top*** : Elevation of the layer's upper boundary [Read-only].
     - ***Analysis.SoilLayers[i].Bottom*** : Elevation of the layer's lower boundary.
     - ***Analysis.SoilLayers[i].SoilReactionCurveFilename*** : Name of the soil reaction curve file assigned to the soil layer. The file must be present in the project.
     - ***Analysis.SoilLayers[i].MaterialParametersLayers[j]*** : Values of the material parameters for the soil layer.
       - *GammaSub*: Submerged unit weight.
       - *G0*: Small strain shear stiffness modulus in the middle of the soil layer.
       - *SuTop*: Undrained shear strength at the top of the layer (Clay only).
       - *SuBottom*: Undrained shear strength at the bottom of the layer (Clay only).
       - *Phi*: Effective angle of internal friction (Sand only).
       - *Psi*: Angle of dilatancy (Sand only).
       - *K0*: Lateral earth pressure coefficient at rest.
       - *RelativeDensity*: Relative density (General Dunkirk Sand only).

7. ***Analysis.Segments*** : List containing all pile segments defining Thickness variation.
   - ***Analysis.Segment*** : A specific pile segment.
     - ***Analysis.Segments[i].Top*** : Elevation of the segment's upper boundary [Read-only].
     - ***Analysis.Segments[i].Bottom*** : Elevation of the segment's lower boundary. The Bottom of the last segment is read-only and equal to the embedded length (L).
     - ***Analysis.Segments[i].DoutBottom*** : The outer diameter at the bottom of the segment.
     - ***Analysis.Segments[i].DoutTop*** : The outer diameter at the top of the segment.
     - ***Analysis.Segments[i].t*** : Segment's wall thickness.
     - ***Analysis.Segments[i].w*** : The unit weight of the material.
     - ***Analysis.Segments[i].A*** : Segment's cross-section area [Read-only].
     - ***Analysis.Segments[i].I*** : Segment's moment of inertia [Read-only].
     - ***Analysis.Segments[i].EA*** : Segment's axial stiffness [Read-only].
     - ***Analysis.Segments[i].EI*** : Segment's bending stiffness [Read-only].
     - ***Analysis.Segments[i].GA*** : Segment's shear stiffness [Read-only].

8. ***Analysis.TowerSegments*** : List containing all tower segments[Only used for modal analysis].
   - ***Analysis.Segment*** : A specific tower segment.
     - Same properties as *Analysis.Segment*

9. ***ResultTypes*** : Results from the analysis, available in the **Analysis** and **Results modes**.
   - ***ResultTypes.MonopileResponseCurves*** : Values of the H-v and M-q global response curves, per calculation Step.
     - *Step*
     - *BaseDisplacement*
     - *BaseRotation*
     - *HeadDisplacement*
     - *HeadRotation*
     - *MudlineDisplacement*
     - *MudlineForce*

- *MudlineMoment*
- *MudlineRotation*

- ***ResultTypes.MonopileResponseDepthVariation*** : Values of structural stresses, deformations, and forces along the monopile, per elevation (z) and calculation Step.
  - *z*
  - *Step*
  - *AxialStress*
  - *ShearForce*
  - *BendingMoment*
  - *LateralDisplacement*
  - *PileCrossSectionRotation*
  - *TangentialStress*
- ***ResultTypes.SoilProfile*** : Discretised soil profile and variation of soil material parameters with elevation (z).
  - *z*
  - *Dilatancy Angle Variation*
  - *EffectiveCohesionVariation*
  - *FrictionAngleVariation*
  - *InitialVerticalEffectiveStress*
  - *LateralEarthPressureCoefficientVariation*
  - *SmallStrainStiffnessVariation*
  - *UndrainedShearStrength*
- ***ResultTypes.ShaftSoilReactionCurves*** : Non-normalised values of the soil reaction curves along the shaft of the pile (p-v, m-θ), per elevation (z) and calculation Step.
  - *z*
  - *Step*
  - *DistributedLateralLoad*
  - *DistributedMoment*
  - *LateralDisplacement*
  - *PileCrossSectionRotation*
- ***ResultTypes.BaseSoilReactionCurves*** : Non-normalised values of the soil reaction curves at the pile base (H-v_B, M-θ_B), per calculation Step.
  - *z*
  - *Step*
  - *BaseMoment*
  - *BaseRotation*
  - *BaseShearLoad*
  - *BaseLateralDisplacement*
- ***ResultTypes.NormalisedShaftSoilReactionCurves*** : Normalised values of the soil reaction curves along the shaft of the pile ($\bar{p} - \bar{v}$, $\overline{m} - \bar{\theta}$), per elevation (z) and calculation Step.
  - *z*
  - *Step*
  - *DistributedLateralLoad*
  - *DistributedMoment*
  - *LateralDisplacement*

    - *PileCrossSectionRotation*

- ***ResultTypes.NormalisedBaseSoilReactionCurves*** : Normalised values of the soil reaction curves along the shaft of the pile ($\overline{H} - \bar{v}_B,\ \overline{M} - \bar{\theta}_B$), per elevation (z) and calculation Step.
    - *z*
    - *Step*
    - *BaseMoment*
    - *Base Rotation*
    - *BaseShearLoad*
    - *BaseLateralDisplacement*
- ***ResultTypes.ShaftDepthVariationFunctions*** : Values of the DVF parameters determining the soil reaction curves along the shaft of the pile, per elevation (z)
    - *z*
    - *km*
    - *kp*
    - *nm*
    - *np*
    - *mu*
    - *pu*
    - *thetamu*
    - *vpu*
- ***ResultTypes.BaseDepthVariationFunctions*** : Values of the DVF parameters determining the soil reaction curves at the pile base.
    - *z*
    - *kM*
    - *kH*
    - *nM*
    - *HBu*
    - *MBu*
    - *nH*
    - *thetaMu*
    - *VHu*
- ***ResultTypes.Workload*** : Values of the load applied on the 1D analysis.
    - *LoadFactor*
    - *h*
    - *ReachedH*
    - *ReachedM*
    - *ReachedStep*
- ***ResultTypes.AccuracySmallDisplacement*** : Values of the accuracy metrics for the 1D - 3D model verification, in the small displacements condition.
    - *eta*
    - *rho*
- ***ResultTypes.AccuracyLargeDisplacement*** : Values of the accuracy metrics for the 1D - 3D model verification, in the large displacements condition.
    - *eta*
    - *rho*
- ***ResultTypes.Frequencies*** : Values of frequencies calculated by modal analysis.

    - *Direction*
    - *MassParticipation*
    - *ModalAmplitude*
    - *Frequency*
    - *z*
- ***ResultTypes.ForeAftModes*** : Values of modal analysis fore-aft modes.
    - *MassParticipation*
    - *ModalAmplitude*
    - *Frequency*
    - *z*
- ***ResultTypes.SideSideModes*** : Values of modal analysis fore-aft modes.
    - *MassParticipation*
    - *ModalAmplitude*
    - *Frequency*
    - *z*

10. **Models**: The two models available in the **Analysis mode**, for which results can be calculated.
    - ***Models.Analysis1D*** : The 1D finite element model (if calculated).
    - ***Models.DesignVerification*** : The 3D design verification model (if generated and calculated).