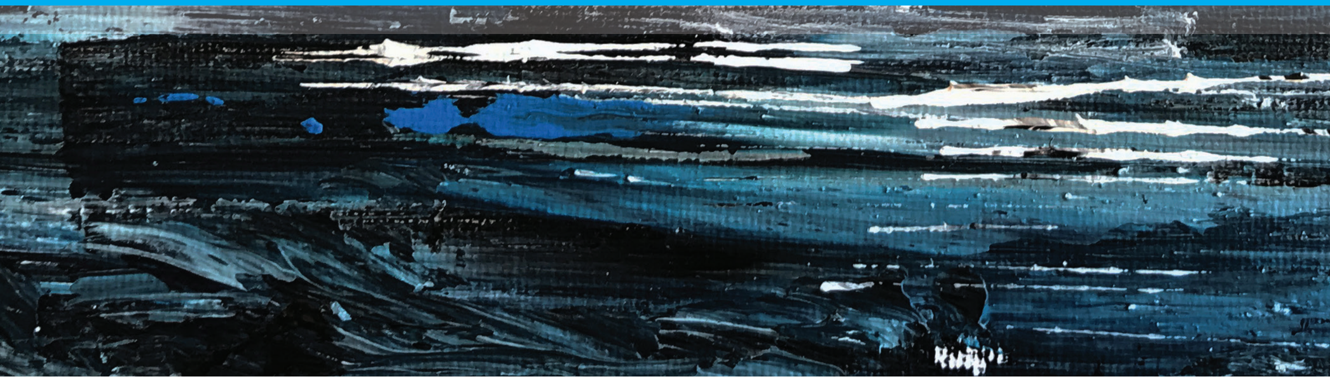
PLAXIS

PLAXIS MoDeTo Manual 2018



Edited by:

S. Panagoulias
PLAXIS bv, The Netherlands

R.B.J. Brinkgreve
Delft University of Technology & PLAXIS bv, The Netherlands

L. Zampich
PLAXIS bv, The Netherlands

With co-operation of:

H. J. Burd
B. W. Byrne
A. Chesaru
S. Hosseini
M. Lahoz
R. A. McAdam
N. A. Rooms

IMPORTANT WARNING AND DISCLAIMER

PLAXIS MoDeTo is a program for specific geotechnical/structural applications in which soil models are used to simulate the soil behaviour. The PLAXIS MoDeTo 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 MoDeTo 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 MoDeTo 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 MoDeTo, 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 MoDeTo calculations or documentation.

Trademark

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; E-mail: info@plaxis.nl; Internet site: www.plaxis.nl

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: 978-90-76016-25-2
© 2018 Plaxis bv

Printed in the Netherlands

TABLE OF CONTENTS

1 General Information 6
1.1 Preface 6
1.2 PLAXIS products and services 8
1.3 Short overview of features 10
1.4 Installation 11
1.5 Troubleshooting 12
1.6 Third Party Software Licenses/Notices 13

2 Reference manual 14
2.1 Introduction 14
2.1.1 The PISA project 14
2.1.2 The design methodology 14
2.1.3 Interoperability with PLAXIS 3D 15
2.1.4 The 1D FE model 16
2.1.5 The components of PLAXIS MoDeTo 17
2.1.6 Graphical user interface 18
2.1.7 Useful terminology 19
2.2 General information 20
2.2.1 Using PLAXIS MoDeTo with and without PLAXIS 3D 20
2.2.2 Program layout 20
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 23
2.2.7 Automatic saving 25
2.2.8 Help facilities 25
2.3 Soil mode 26
2.3.1 Soil mode layout 26
2.3.2 Material types 26
2.3.3 Creating soil layers 28
2.4 Calibration mode 29
2.4.1 Calibration mode layout 30
2.4.2 Geometry datasets (GeoDS) 31
2.4.3 Structural and material properties 38
2.4.4 Results inspection pane 38
2.4.5 Recommended workflow 47
2.5 Analysis mode 48
2.5.1 Depth variation functions 48
2.5.2 Monopile geometry 49
2.5.3 Structural properties 50
2.5.4 Workload (monopile head) 50
2.5.5 Thickness variation 51
2.5.6 Expert settings 53
2.5.7 Calculate 54
2.5.8 Results inspection pane 55
2.6 Results mode 59
2.6.1 Workload and load factor 59

2.6.2 Graph tab 60
2.6.3 Table tab 61
2.6.4 Accuracy metric η 62

3 Tutorial manual 64
3.1 Tutorial - numerical based design 64
3.1.1 Input 66
3.1.2 Verification of the calibration procedure 71
3.1.3 Final design 73
3.1.4 Verification of the final design 74

4 Scientific manual 76
4.1 Introduction 76
4.2 Material models 76
4.2.1 Clay: NGI-ADP material parameters 76
4.2.2 Sand: HSsmall material parameters 77
4.3 PLAXIS 3D models 79
4.3.1 Generating 3D models 79
4.3.2 Soft beam properties 82
4.3.3 Soil reaction curves 83
4.3.4 Results inspection pane 84
4.4 Optimisation module 85
4.5 Plain text file format rules 88
4.5.1 Format: depth variation functions 88
4.6 1D FE model 90
4.6.1 Formulation of the 1D FE model 90
4.6.2 Implementation aspects of the 1D FE model 95
4.7 Results mode 98
4.7.1 Realised H 98
4.7.2 Realised M 98
4.7.3 Load factor 99
4.7.4 Accuracy metric η 99
4.7.5 H-v and M-ψ plots 99
4.7.6 $v(z)$ and $\psi(z)$ plots 100
4.7.7 $s_u(z)$ and $\sigma'_{v0}(z)$ 101
4.7.8 $p(z)$ plot 101

5 References 102

Appendix A - Third party software licenses/notices 104

Appendix B - Warnings and errors 126

1 GENERAL INFORMATION

1.1 PREFACE

PLAXIS MoDeTo 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 MoDeTo 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 MoDeTo 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 MoDeTo 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 MoDeTo 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:

Delft University of Technology, Civil Engineering (NL)	Prof. Michael Hicks, Prof. Bert Sluys

Delft University of Technology, Mathematics & Informatics (NL)	Prof. Kees Vuik
Deltares (NL)	Mr. Mark Post, Dr. Cor Zwanenburg
BundesAnstalt für Wasserbau (DE)	Dr. Michael Heibaum, Mr. Oliver Stelzer
Technical University, Graz (AT)	Prof. Helmut Schweiger, Dr. Franz Tschuchnigg
Univ. of Grenoble, Laboratoire 3R (FR)	Prof. Cino Viggiani
University of Oxford (UK)	Prof. Harvey Burd, Prof. Byron Byrne, Prof. Ross McAdam
Chalmers University of Technology (S)	Prof. Minna Karstunen, Dr. Mats Olsson, Dr. Anders Kullingsjö
Massachusetts Institute of Technology (USA)	Prof. Andrew Whittle
University of California at Berkeley (USA)	Prof. Juan Pestana
Northwestern University (USA)	Prof. Richard Finno
Univ. of Illinois at Urbana-Champain (USA)	Prof. Youssef Hashash
California Department of Transportation (USA)	Dr. Anoosh Shamsabadi
Norwegian Univ. of Science and Tech (NO)	Prof. Steinar Nordal, Prof. Gustav Grimstad
Norwegian Geotechnical Institute (NO)	Dr. Lars Andresen, Prof. Hans Petter Jostad, Dr. Nallathamby Sivasithamparam
Technical University of Catalunya (ES)	Prof. Antonio Gens, Prof. Eduardo Alonso
National University of Singapore (SG)	Prof. Harry Tan
Sapienza University of Rome (IT)	Prof. Angelo Amorosi
Ain Shams University, Cairo (EG)	Prof. Yasser El-Mossallamy
Technical University, Dresden (DE)	Prof. Ivo Herle
Charles University, Prague (CZ)	Prof. David Mašin
University of Tampere (FI)	Prof. Tim Lansivaara
Ruhr University, Bochum (DE)	Prof. Tom Schanz †, Prof. Günther Meschke
National Technical University, Athens (GR)	Prof. George Gazetas, Dr. Nikos Gerolymos
University of Washington (USA)	Prof. Steven Kramer, Prof. Pedro Arduino
University of Liege (BE)	Prof. Christophe Geuzaine
INSA-Lyon (FR)	Prof. Yves Renard
University of British Columbia (CA)	Prof. Mahdi Taiebat
University of Bologna (IT)	Prof. Daniela Boldini

This support is gratefully acknowledged.

The editors

1.2 PLAXIS PRODUCTS AND SERVICES

Update 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 by means of the Plaxis bulletin and the website www.plaxis.nl.

PLAXIS 2D: A large range of geotechnical problems may be analysed using this high capacity version. It is possible to use extensive 2D finite element meshes. PLAXIS 2D is supplied as an extended package, including static elastoplastic deformation, advanced soil models, stability analysis, consolidation, updated mesh analysis (large deformations) and steady-state groundwater flow.

PLAXIS 3D: PLAXIS 3D is a geotechnical finite element program with a full 3D pre-processor that allows CAD objects to be imported and further processed within a geotechnical context. The program is supplied as an extended package, including static elastoplastic deformation, advanced soil models, stability analysis, consolidation, updated mesh analysis and steady-state groundwater flow.

PLAXIS MoDeTo: PLAXIS MoDeTo is a software package for the analysis and design of monopiles as foundation elements for offshore wind turbines, under lateral loading conditions. MoDeTo 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. In the latter case, soil reaction curves, used in the one-dimensional finite element model of MoDeTo, are derived and calibrated from the results of a series of 3D finite element calculations performed in PLAXIS 3D.

Dynamics: Dynamics is an add-on module to PLAXIS 2D and PLAXIS 3D. This module may be used to analyse vibrations in the soil and their influence on nearby structures as well as for geotechnical earthquake analysis. Excess pore pressures can be analysed. Liquefaction can be analysed by using one of the available liquefaction models (UBC3D-PLM or PM4Sand) The latter model (PM4Sand) is available as a user-defined soil model. (UBC3D-PLM is nowadays a standard model). Besides short-term (undrained) dynamic analysis, the PLAXIS 2D dynamic module includes dynamics with simultaneous consolidation of excess pore pressures. Dynamic calculations can also be executed taking large deformation effects (updated mesh) into account.

PlaxFlow: PlaxFlow is an add-on module to PLAXIS 2D and PLAXIS 3D. This module may be used for the analysis of fully coupled flow deformation analysis, steady-state and transient groundwater flow. The module incorporates sophisticated models for saturated / unsaturated groundwater flow, using the well-known "Van Genuchten" relations between pore pressure, saturation and permeability. It provides state-of-the-art facilities to incorporate time-dependent boundary conditions. The Barcelona Basic model for unsaturated soil behaviour is available as a user-defined soil model upon request.

Thermal: Thermal is an add-on module to PLAXIS 2D. This module may be used for the analysis of fully coupled thermo-hydro-mechanical analysis, steady-state and transient thermal flow.

PLAXIS VIP: PLAXIS VIP is an additional subscription system on top of the professional software licenses. PLAXIS VIP members benefit from the latest releases of their PLAXIS software maintenance and support from Plaxis technical experts. In addition, some features of PLAXIS programs are only available for PLAXIS VIP members. An overview of these features and more information about PLAXIS VIP are available at the website www.plaxis.nl.

User support: Priority technical support is provided by e-mail for members of PLAXIS VIP. A professional helpdesk is available for clients who wish to obtain prompt and extensive technical and scientific support.

PLAXIS Demo CD: An introductory version of PLAXIS software is available for interested persons who wish to learn about the program features and capabilities before ordering the software. The PLAXIS Demo CD is based on PLAXIS software but there is a limited number of material sets and calculation phases. In addition, it is not possible to copy or print. A Tutorial Manual with examples specifically generated for the PLAXIS Demo CD is included.

PLAXIS Academy: Under the PLAXIS Academy educational program several courses, trainings, seminars, webinars and workshops are organized through out the year at different locations all over the world.

PLAXIS Courses PLAXIS Courses consist of a mixture of lectures and hands-on computer analyses making the courses more than just a software training. The aim of both teaching and exercises is to concentrate on the use and background of advanced methods in geotechnical engineering. The principle of PLAXIS Courses is that they will give ongoing education and knowledge transfer for which the use of the PLAXIS software is only a tool to explain the topics learned.
There are two levels of courses, *Standard* and *Advanced*. The *Standard Courses* are meant for those who have little or no experience with PLAXIS and focuses on the engineering aspects of the use of finite element method. The *Advanced Courses* on the other hand assume a certain level of experience and therefore go deeper into the theoretical background.

PLAXIS Trainings The PLAXIS Trainings focus on developing knowledge on the use of PLAXIS Software. This is considered the main topic and any technical lecture is just to support and illustrate the use of the software. Under PLAXIS Trainings we distinguish between a PLAXIS Training (multi-day), a PLAXIS Workshop (1-day) or PLAXIS Seminar (without any hands-on training with the software).

PLAXIS Expert Services: PLAXIS Expert Services are professional services exclusively meant for users of PLAXIS software. The purpose of this service is to help our clients on any FE modelling related issue such that they can be assisted with in their simulation work and improve their own modelling capabilities. PLAXIS Expert Services provides high-level technical assistance with advanced finite element modelling issues, fit-for-purpose training courses which can be customized to your specific requirements, and personal mentoring with on-call simulation expertise.

PLAXIS Customisation: These professional services are aimed at specific user requirements in addition to the standard PLAXIS functionality. Customisation may not necessarily involve specific features in PLAXIS, it merely deals with functionality to interact with the software (using the PLAXIS Application Programming Interface) to make it more suitable or efficient for particular applications, or to integrate it with the user's work environment. Examples of possible customization projects are:

- Import of soil data from data bases and translation into PLAXIS model parameters
- Fast creation of PLAXIS models for standard applications (foundations, excavations, embankments, piles) based on templates
- Facilitating parameter variations for sensitivity analysis, probabilistic analysis, parameter optimisation or inverse analysis
- Implementation of specific constitutive models as user-defined soil model

PLAXIS Bulletin: An international bulletin, issued periodically, is provided to all registered PLAXIS users. This bulletin contains descriptions of practical projects in which PLAXIS has been used, backgrounds on the use of advanced soil models, information on new developments, hints for optimised usage of the program and a diary of activities. Ideas and experiences with the PLAXIS programs are highly appreciated.

Website: The Plaxis website www.plaxis.nl is the main source of information about the latest news, events, products and services. Besides this information on the site includes a support section and Knowledge Base, an extensive library with a collection of bulletins, publications, models and much more. Via the website it is also possible to purchase product and register for events. Visit the site on a regular basis to stay in contact with Plaxis.

For more information on products and user's services, contact:

Plaxis bv
P.O. Box 572
NL-2600 AN Delft
The Netherlands
E-mail: info@plaxis.nl
Internet: www.plaxis.nl

1.3 SHORT OVERVIEW OF FEATURES

PLAXIS MoDeTo 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 MoDeTo 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 MoDeTo is given below.

Input of soil stratigraphy: Based on a preliminary selection of either clay-type or sand-type soils, PLAXIS MoDeTo 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 VIP license.

Visualisation option: A 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 MoDeTo. 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 MoDeTo 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 MoDeTo 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 MoDeTo 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 build-in one-dimensional finite element model. PLAXIS MoDeTo 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 MoDeTo facilitates the presentation of various results in both graphical and tabulated format. Results can be copied to clipboard and printed.

1.4 INSTALLATION

- Insert the MoDeTo installation USB stick into the computer.
- Select Open folder to view files, or open the drive assigned to this USB device.
- Select the 'PlaxisMoDeToSetup.exe' application.
- A screen appears, which will guide the user through the rest of the installation process.
- When asked for user name and registration code please refer to the sticker on the back of this booklet. The registration data is spacing and case sensitive.
- With the CodeMeter dongle attached to the computer, the PLAXIS MoDeTo software is now ready for use.

Local dongle installation: PLAXIS MoDeTo continuously checks for the presence of the dongle that is included in the package. The dongle must be inserted in a USB port of the computer. Normally a device driver for the dongle is installed during the setup. The document 'CodeMeterInstallation.pdf' can be found on the *Knowledge base* on the PLAXIS website.

Network dongle installation: Alternatively it is possible to use a shared multiple license dongle over the network. The document 'CodeMeterInstallation.pdf' can be found on the *Knowledge base* on the PLAXIS website.

Program uninstall and install: Should you wish to uninstall or reinstall PLAXIS MoDeTo you can use the Windows *Add/Remove* programs utility from the *Control Panel*.

1.5 TROUBLESHOOTING

In exceptional cases the installation program fails to install the PLAXIS package. Some possible error messages during the execution of the program are:

- The program starts with a message and then closes immediately
- Codemeter problems with IP protocol
- When running PLAXIS MoDeTo for the first time warning about missing VCRUNTIME140.dll file pops up

The appropriate actions to be taken on the problems are described below:

Program starts with a message and then closes immediately: Make sure that the dongle is inserted in a USB port of the computer. In addition, make sure that the latest drivers are installed. These can be found on the PLAXIS website: www.plaxis.nl in the Downloads section. Download the drivers for the right system type (32-bit or 64-bit Operating System).

Codemeter problem with IP protocol: As Codemeter dongles require IP protocol and firewalls may prevent this, the firewall should explicitly allow the Codemeter dongles over the IP port. To allow this, one can allow both TCP and UDP protocol for port 22352.

A more detailed description is given in the document "CodeMeterInstallation.pdf" can be found on the *Knowledge base* on the PLAXIS website.

When running the program for the first time warning about missing VCRUNTIME140.dll file:

When running PLAXIS MoDeTo for the first time, there might appear a warning message about missing VCRUNTIME140.dll file. In this case, please install the Microsoft Visual C++ Redistributable for Visual Studio from the Microsoft website https://www.microsoft.com/en-us/download.

Other issues may be related to the following:

Antivirus - Whitelist recommendation: It is recommended to add *pythonw.exe* to the antivirus whitelist to prevent it from being blocked.

Problems with relaunching the program: If PLAXIS MoDeTo cannot be relaunched, please make sure to terminate the *PlaxisMoDeTo.exe*, to ensure a clean restart.

Disk storage: The PLAXIS 3D files generated via PLAXIS MoDeTo may have a size of several GB. Please ensure that your working station has enough storage space available.

1.6 THIRD PARTY SOFTWARE LICENSES/NOTICES

PLAXIS software makes use of and contains certain third party software (components). As a condition of the use of this third party software, we are obliged to also distribute the specific terms and conditions that apply to the use of it and that may differ from or are additional to PLAXIS' own conditions as they are contained in the PLAXIS User License Agreement. These terms and conditions of the third party software used, are deemed to form an integral part of the PLAXIS User License Agreement and thus of the right to use the PLAXIS software. The current applicable license terms of the third party software used can be found in Appendix A.

2 REFERENCE MANUAL

2.1 INTRODUCTION

PLAXIS MoDeTo (Monopile Design Tool) 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 recent conference publications (e.g. Byrne et al., 2018a, Byrne et al., 2017, Burd et al., 2017, Byrne et al., 2015a, Byrne et al., 2015b, Zdravkovic et al., 2015). 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., 2018, Byrne et al., 2018b).

PLAXIS MoDeTo provides a means of implementing the PISA design method in a daily engineering context, for the design of monopile foundations for offshore wind turbines (Panagoulias et al., 2018a, Panagoulias et al., 2018b), including large diameter monopiles.

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 MoDeTo 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. The numerical-based design approach is likely to be used for detailed design.

Note that PLAXIS MoDeTo 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 Section 2.3 and Section 4.2 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.

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 MoDeTo 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 MoDeTo 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 in Section 2.1.4. 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.

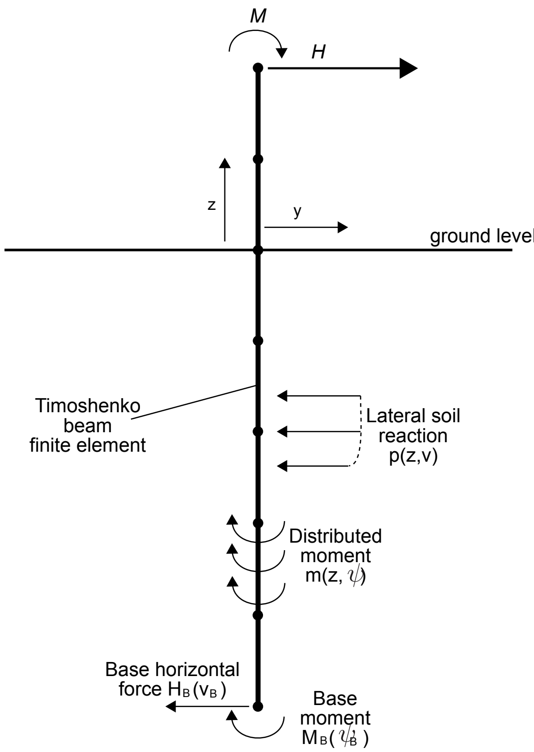


Figure 2.1 Components of the 1D FE model (based on Byrne et al., 2015b)

2.1.5 THE COMPONENTS OF PLAXIS MoDeTo

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).

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.

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

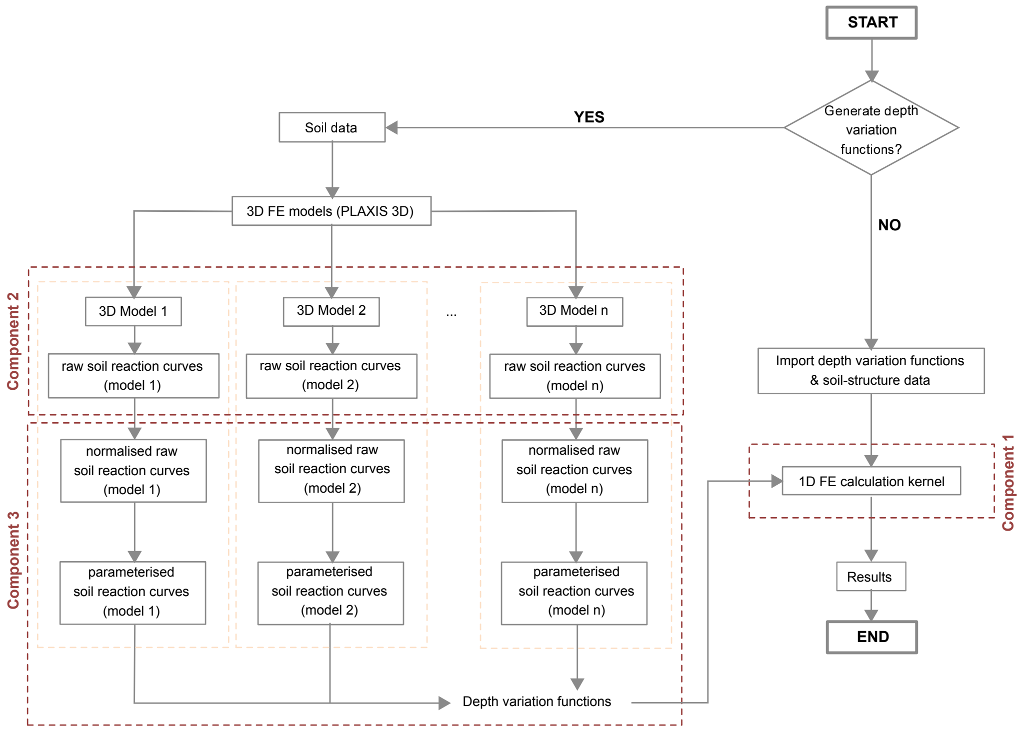


Figure 2.2 PLAXIS MoDeTo workflow

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 four operational modes, namely the Soil mode, 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 four modes of the tool are used sequentially. The Soil mode is used to define the site-specific soil layers and soil data. In the Calibration mode the various monopile geometric configurations are defined. The PLAXIS 3D models are generated and calculated based on the data coming from the Soil mode and the Calibration mode. The extraction and parameterisation of the soil reaction curves is part of the Calibration mode. 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.

Note that the present version of the tool supports homogeneous soil profiles. The user can define either a *Clay* or a *Sand* soil configuration in the Soil mode. Furthermore, the calibrated or user-defined soil reaction curves imported in the Analysis mode should represent homogeneous soil sites as well. Mixed soil layers are planned to be implemented in future versions of PLAXIS MoDeTo.

2.1.7 USEFUL TERMINOLOGY

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

Design space

The design space (or calibration space) 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.

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 ψ
- base horizontal force H_B - lateral displacement v_B
- base moment M_B - base rotation ψ_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). 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 MoDeTo WITH AND WITHOUT PLAXIS 3D

Functionality without PLAXIS 3D VIP

Without PLAXIS 3D VIP, 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 Section 2.4.2) is not available unless PLAXIS 3D VIP 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 VIP, please contact the PLAXIS Sales Department at sales@plaxis.com.

Functionality with PLAXIS 3D VIP

If PLAXIS 3D VIP 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 MoDeTo, the user has four modes to work with: Soil mode, Calibration mode, Analysis mode, and Results mode. Each mode appears as a coloured tabsheet in PLAXIS MoDeTo.

After starting the program, the user chooses whether to open an existing project or start a new one. A new project (Figure 2.3) must first be saved before being worked with.

The general layout of the program is shown in Figure 2.4.

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.

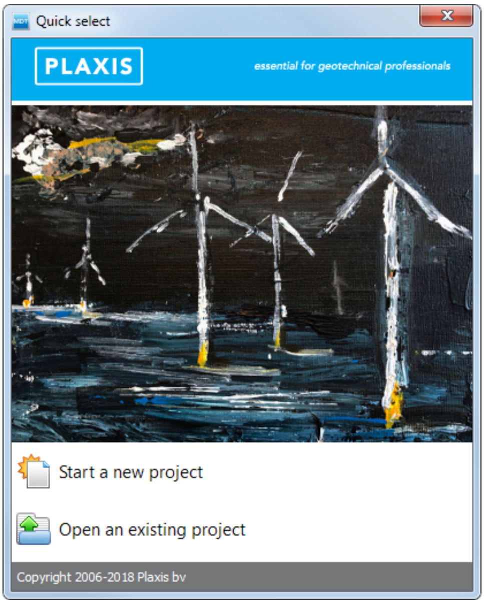


Figure 2.3 Start screen

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:

Soil	Optional mode allowing users with access to PLAXIS 3D to define the soil stratigraphy.
Calibration	Optional mode for the users with access to PLAXIS 3D, to generate and calculate 3D FE models, the soil reaction curves of which will be extracted and parameterised.
Analysis	To run the 1D FE Analysis.
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 *Soil* or *Calibration* modes, 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.

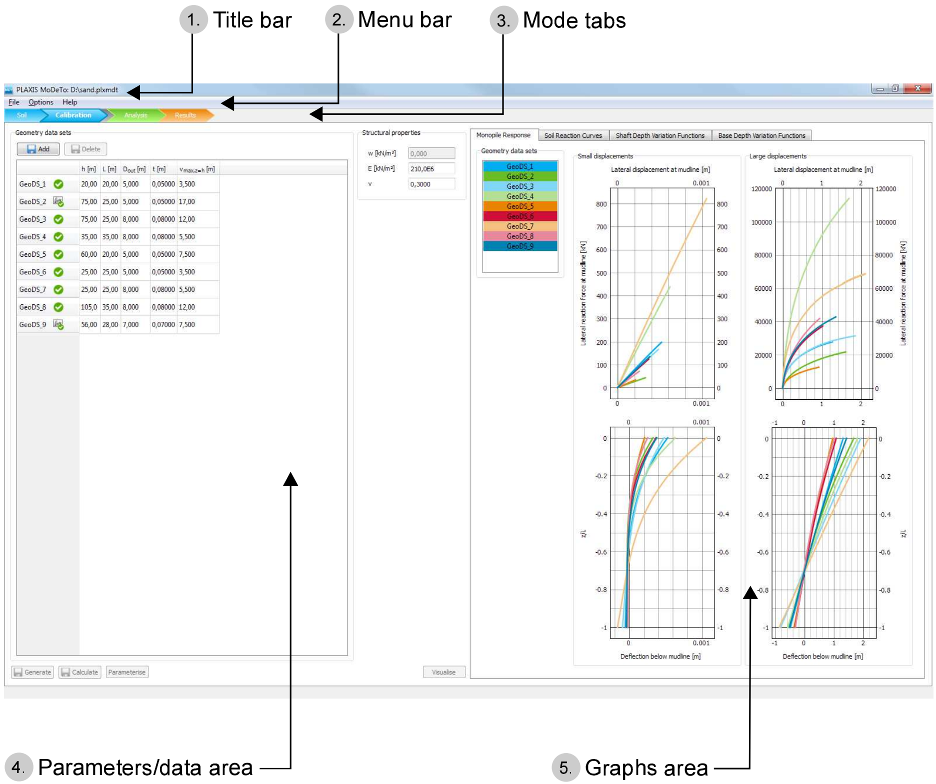


Figure 2.4 Layout of the program

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 *Soil* 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) 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 *Soil* mode, see Section 2.3.

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 mode information on the *Analysis* mode, see Section 2.5.

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 create 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.

Options 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.

Help menu

Manuals	To display the manuals.
Request support...	To send a request for support.
https://www.plaxis.com	To reach the PLAXIS website.
Disclaimer	To display the complete disclaimer text.
About	To display information about the program version and license.

2.2.6 UNITS AND SIGN CONVENTION

Standard units

PLAXIS MoDeTo 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 MoDeTo.

- Positive x-, y-, z-direction as displayed in Figure 2.5
- Positive moment: right-handed coordinate system
- Compressive stress: negative (solid mechanics convention)

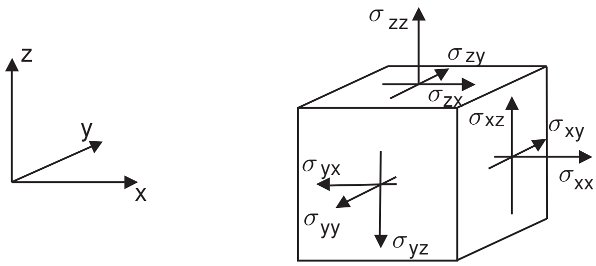


Figure 2.5 Coordinate system and the indication of positive stress components

Sign convention used in 1D model

The following applies to the 1D FE model.

- positive y-, z-direction as displayed in figure in Section 2.1.4 positive lateral load *p* and moment *m* as displayed in Figure 2.6

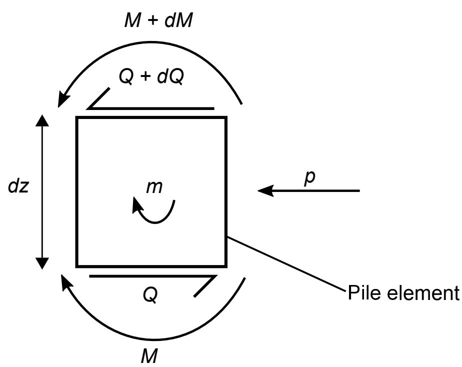


Figure 2.6 Sign convention in 1D FE model

With this sign convention the variables are related by:

$$p = -\frac{dQ}{dz} \qquad m - \frac{dM}{dz} = Q \tag{2.1}$$

where Q is the pile shear force and M is the 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 MoDeTo 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 MoDeTo provides extensive help facilities for the users. In the *Help* menu (Section 2.2.5), there is a link to all manuals in PDF form.

Manuals

To obtain a quick working knowledge of the main features of PLAXIS MoDeTo, it is suggested that users work through the example problem contained in the Tutorial Manual.

This 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 Manual. It also contains practical details on how to use PLAXIS MoDeTo for the design of monopiles according to the PISA method.

This 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.

Knowledge base

Additional information can be found in the PLAXIS Knowledge Base (https://www.plaxis.com/support/).

Customer support

Need Help? Tell us about your issue and find the best support option:
https://www.plaxis.com/contact/

2.3 SOIL MODE

The *Soil* mode is intended for users who want to follow the numerical-based design approach and use 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 *Soil* mode should be used before the *Calibration* and *Analysis* modes.

The *Soil* 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 project. Depending on the material type, 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 (Section 4.2). 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.

2.3.1 SOIL MODE LAYOUT

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

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

- default material type is *Clay*
- soil layers are deleted

2.3.2 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., 2018). The material behaviour (drainage type) is assumed to be undrained. To read more about the parameters of this material model, see Section 4.2.1. The following input parameters need to be defined per soil layer:

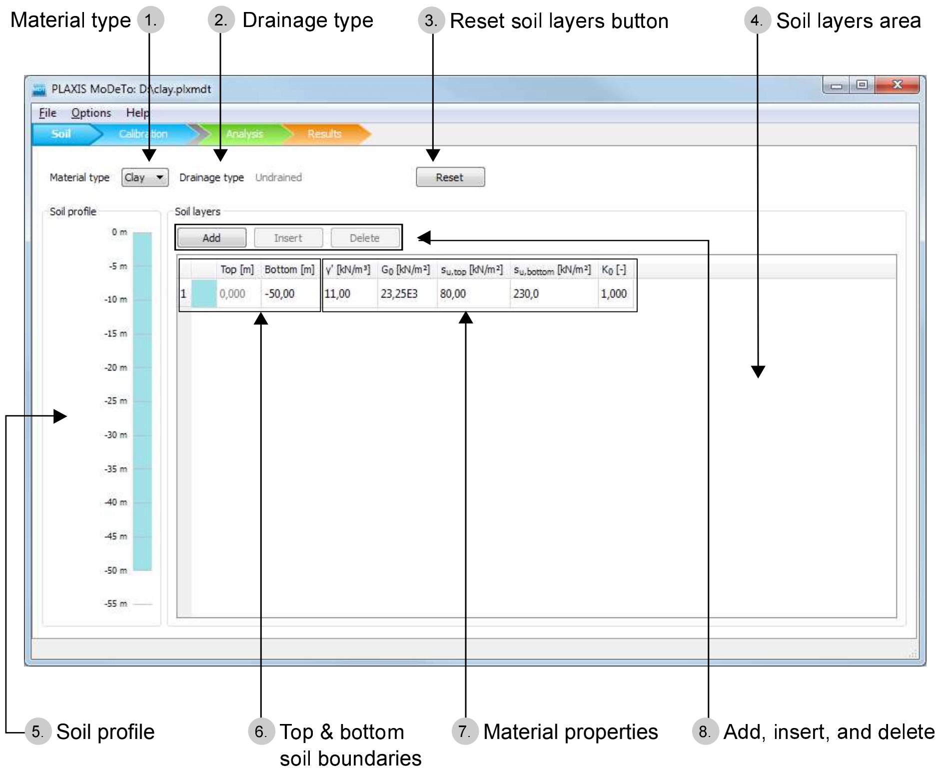


Figure 2.7 View of a project in the *Soil* mode

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	[-]

TIP: The user may change the constitutive model or use a user-defined soil constitutive model via PLAXIS 3D.

Sand

The sandy soil material type is formulated using the *HSsmall* model (for more information, see Brinkgreve et al., 2018). The material behaviour (drainage type) is assumed to be drained. To read more about the parameters of this material model, see Section 4.2.1. The following input parameters need to be defined per soil layer:

γ'	:	submerged unit weight	$[kN/m^3]$
G_0	:	small strain shear stiffness modulus in the middle of the soil layer	$[kN/m^2]$
φ'	:	effective angle of internal friction	[°]

ψ	:	angle of dilatancy	[°]
K_0	:	lateral earth pressure coefficient at rest	[-]

TIP: The user may change the constitutive model or use a user-defined soil constitutive model via PLAXIS 3D.

2.3.3 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 *Soil profile* (Figure 2.8), which is visible in the left panel of the *Soil* mode. It is a visual representation of the inserted soil layers and their top and bottom boundaries.

TIP: For easier reference and navigation, the selected soil layer is also highlighted in the *Soil profile* panel.

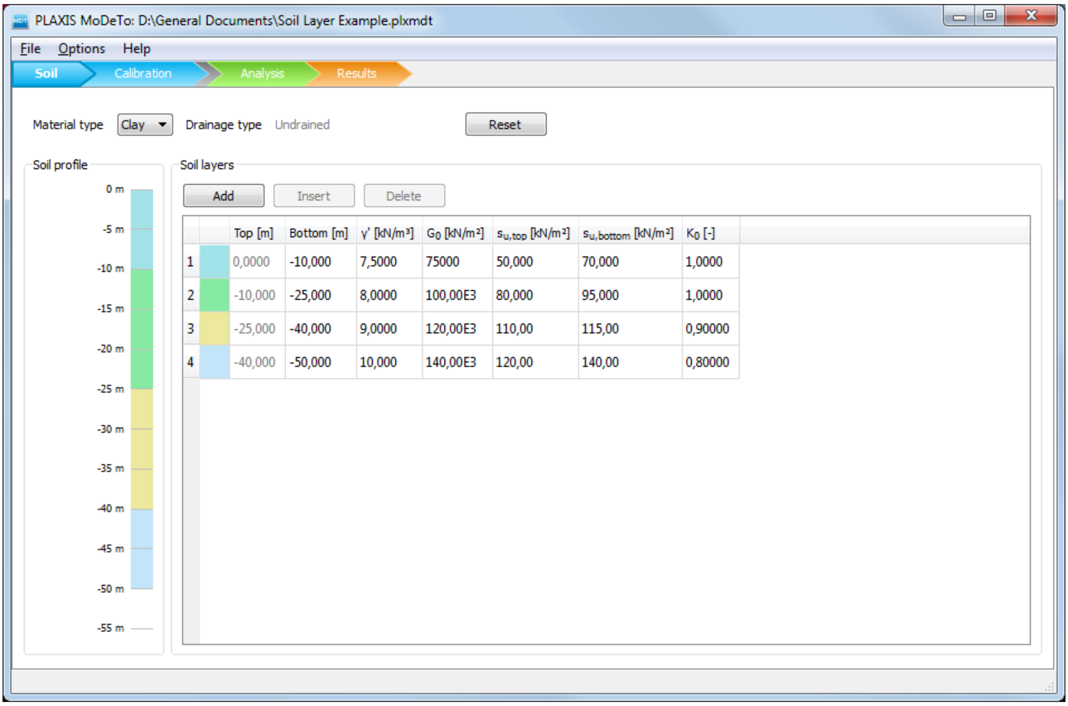


Figure 2.8 Soil profile in the *Soil* mode

2.4 CALIBRATION MODE

The *Calibration* mode is intended for users who want to follow the numerical-based design approach and to use in the numerical-based design. It 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), a maximum horizontal displacement needs to be specified which is applied at the height h ($v_{max,z=h}$).

Calibration procedure

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

1. Generating PLAXIS 3D models based on the soil profile in the *Soil* mode 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.

2.4.1 CALIBRATION MODE LAYOUT

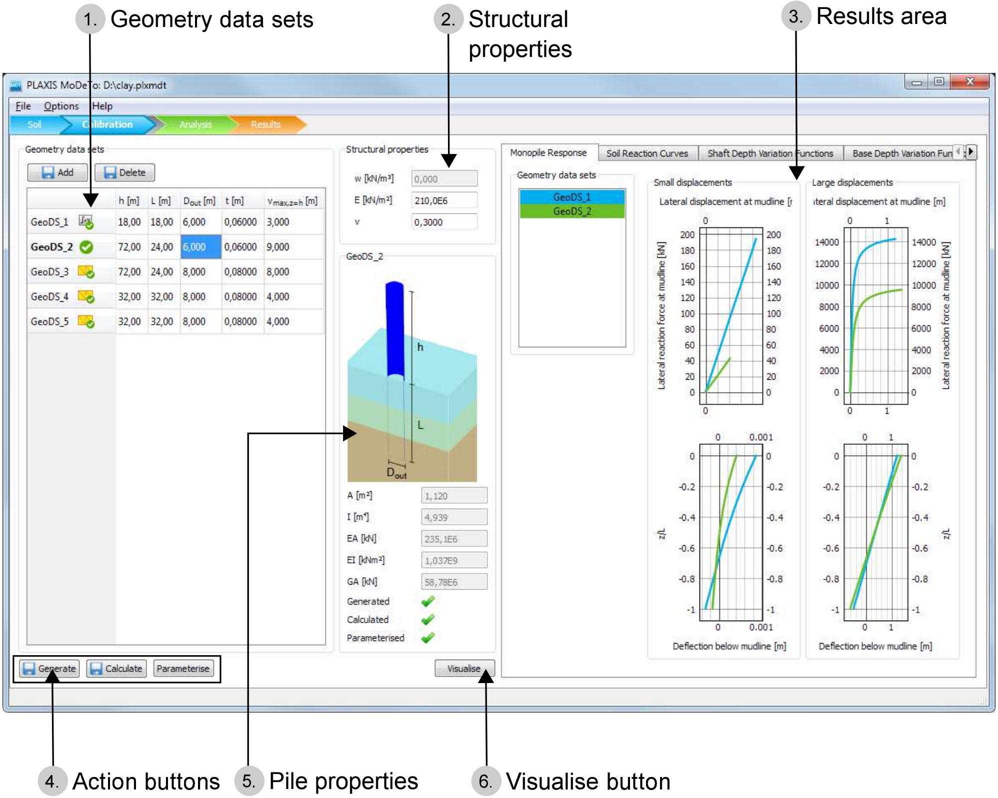


Figure 2.9 Layout of the *Calibration* mode

Geometry datasets

Tabular overview of the data sets. Used to add/delete sets via the corresponding buttons and edit data sets. Each data set is identified by the name GeoDS_#, where # is the number of the data set. 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).

Results area

View of results after calculation/parameterisation.

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.

Visualise button

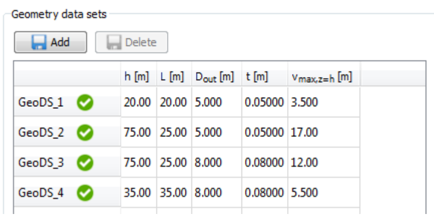
Visualisation of the 3D model that corresponds to the selected dataset in PLAXIS 3D, where the user is allowed to modify the models manually. For example, the user can change default material parameters. Note that this should be done with caution.

2.4.2 GEOMETRY DATASETS (GEODS)

A geometry data set (GeoDS) corresponds to a particular PLAXIS 3D model. The soil stratigraphy in the model comes from the *Soil* mode of PLAXIS MoDeTo. The geometrical characteristics of the monopile come from the input parameters of the *Calibration* mode for the specific data set GeoDS.

Add and delete GeoDS

The defined GeoDS are listed in a table, see Figure 2.10.



	h [m]	L [m]	D_{out} [m]	t [m]	$v_{max,z=h}$ [m]
GeoDS_1	20.00	20.00	5.000	0.05000	3.500
GeoDS_2	75.00	25.00	5.000	0.05000	17.00
GeoDS_3	75.00	25.00	8.000	0.08000	12.00
GeoDS_4	35.00	35.00	8.000	0.08000	5.500

Figure 2.10 Geometry data sets (GeoDS)

To add a new GeoDS, click the *Add* button. A new data set is added to the table, below the last data set.

NOTE: Creation of a new GeoDS copies the properties assigned to the last created data set, and the associated PLAXIS 3D project if it has been already generated, or generated and calculated. If a project was calculated, then the calculated project is copied, including the results.

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

TIP: The user can select a single GeoDS by clicking anywhere in the row. More than one GeoDS may be selected by using Shift+click (consecutive rows) or Ctrl+click (single rows).

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

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
$v_{max,z=h}$	Prescribed displacement applied to the top of the monopile, at height equal to h	m

The name of each geometry data set is assigned automatically and cannot be changed. The name is GeoDS_<n>, where n increases by one, adding to the highest existing number. Previously deleted numbers are not reused unless the deleted numbers were the highest.

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 Section B.1.1 for more information.
If there are no soil layers defined, the user is not able to fill in L.

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 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 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 *Visualise* button and check the error encountered during the generation or calculation of the model.

Generate

When the GeoDS have been added, you can generate the PLAXIS 3D models.

To generate a model, PLAXIS MoDeTo will:

- Generate the soil layers (as specified in the *Soil* mode of PLAXIS MoDeTo)
- Generate the soil materials and the model parameters based on the values specified in the *Soil* mode

  Note that all the needed material parameters are calculated based on predetermined relationships. See Section 4.2.1 and Section 4.2.2. 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 GeoDS, from the *Calibration* mode
- 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 *Calibration* mode
- 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 MoDeTo).
- 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 maybe used by the user to check additional results (in PLAXIS 3D Output) but they are not directly used by the PLAXIS MoDeTo workflow and calculations.

NOTE: Note that 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 MoDeTo. Note that any manual changes to the model will be copied to the next one added in the GeoDS menu of the tool (see Section 2.4.5).

A user may modify a 3D model that is automatically generated by PLAXIS MoDeTo 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 MoDeTo. Open the corresponding model in PLAXIS 3D.

This should preferably be done for the first GeoDS that is defined in PLAXIS MoDeTo since subsequent models are based on the previously generated (and modified) 3D model. In this way, subsequent models can be automatically generated by PLAXIS MoDeTo, 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).

Since the input value of the maximum lateral displacement v_{max} is specified at height h above mudline, the displacement at mudline is generally smaller than the input value of the displacement v_{max}. The user should realise this when specifying an appropriate input value for v_{max}.

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 Section B.1.2.

Regeneration:

The regeneration of models is different than the initial generation. See Section B.1.2 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 Section B.4.

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

Calculate

Model calculation includes the following:

- Calculation of all 4 phases (as described in Section Generate)
- 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 2.11. To stop all calculations, click the *Stop* button.

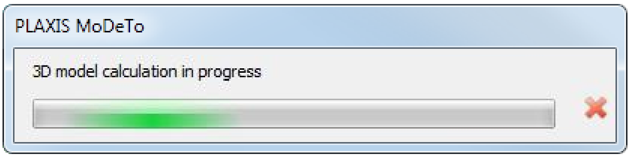


Figure 2.11 Model calculation progress dialog

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 Section B.1.3.

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 (Section 4.4) 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, (the 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 Section B.1.4.

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. 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 specified lateral displacement at the top of each monopile geometry should be such that the maximum pile deflection at mudline is about $0.2 \cdot D$ at the end of the large displacements calculation.

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

Visualise

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 2.12 appears (unless switched off by the user via the *'Do not show 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 MoDeTo

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

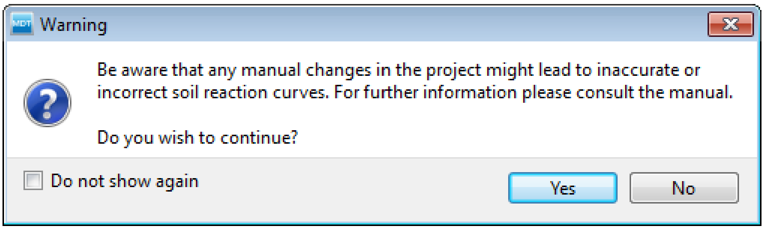


Figure 2.12 Warning about manual changes

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 MoDeTo).
- Modify the name of the project (it will not be recognised anymore by PLAXIS MoDeTo).
- 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 MoDeTo).
- 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 MoDeTo 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 MoDeTo 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.

2.4.3 STRUCTURAL AND MATERIAL PROPERTIES

The middle section of the *Calibration* mode view shows the structural and material properties and a schematic of the monopile. 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.

Structural properties

In the structural properties area, the Young's modulus E and Poisson's ratio ν 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 Section 2.4.

The structural properties are described in Table 2.2

Table 2.2 Structural properties

Property	Description	Unit	Default value
w	pile unit weight	kN/m^3	0.000 (fixed)
E	Young's modulus	kN/m^2	$210 \cdot 10^6$
ν	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.

Table 2.3 Equivalent pile properties

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

Also, see Section 2.4.

2.4.4 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 tabs include the *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 2.13.

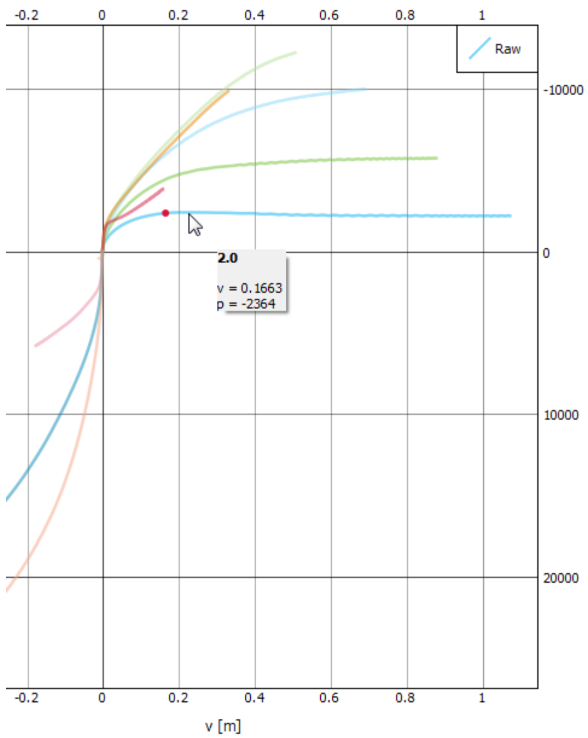


Figure 2.13 Hover over curve to show values

- 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 2.14). This allows users to adjust the appearance of the graph and to export it as an image or vector graphic.

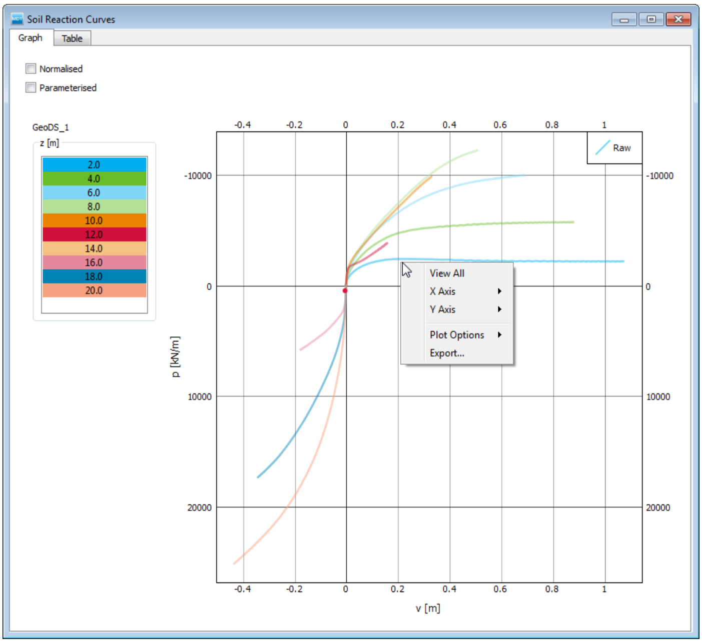


Figure 2.14 Context menu for curves

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 2.15.

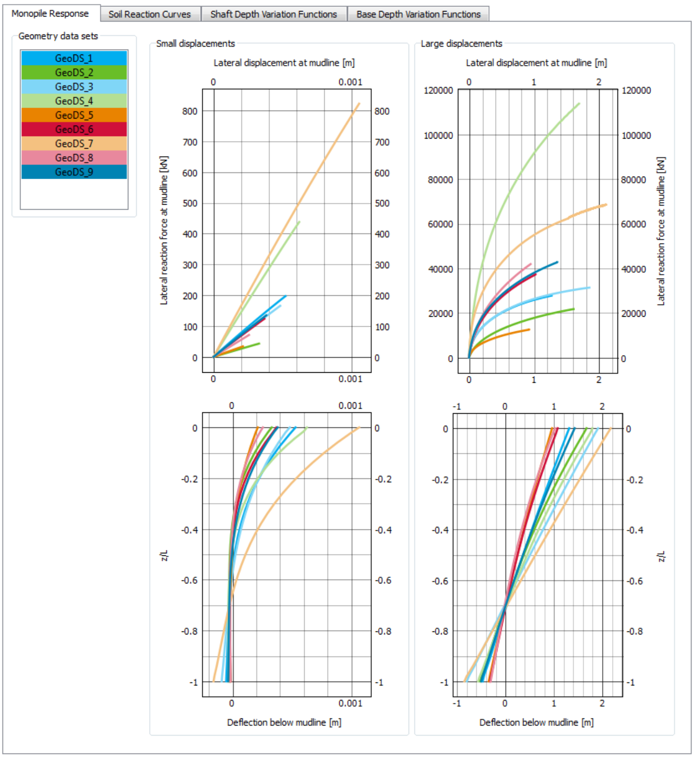


Figure 2.15 Monopile response tab

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 $D/10$.

NOTE: Charts are automatically updated after successful calculation of a selected model.

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 Section 2.4.4.

Soil reaction curves

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 (Figure 2.16). For more information on soil reaction curves, see Section 4.3.3.

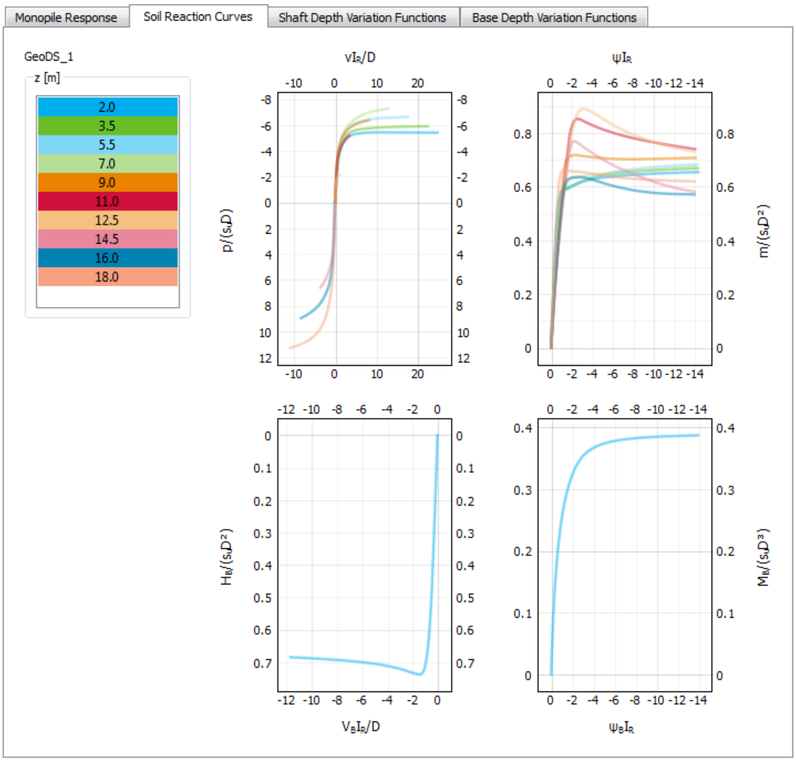


Figure 2.16 Soil reaction curves tab

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: $\{z_{target} = 0.1\,L, 0.2\,L, 0.3\,L, 0.4\,L, 0.5\,L, 0.6\,L, 0.7\,L, 0.8\,L, 0.9\,L, 1.0\,L\}$.
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 2.17). If the rounded z_{target} corresponds to the exact boundary of two slices, the bottom slice is selected.

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.

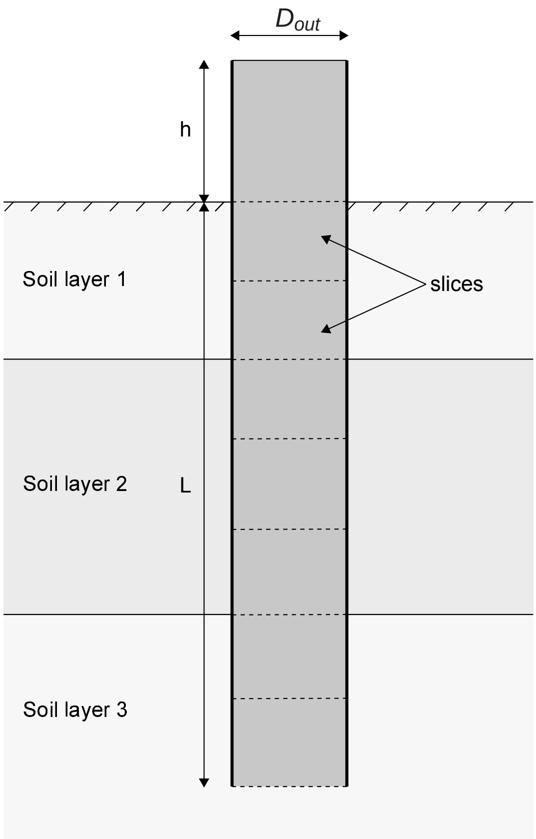


Figure 2.17 Slices of a monopile

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

The four types of soil reaction curves ($p - \nu$, $m - \psi$, $H_B - \nu_B$ and $M_B - \psi_B$) are displayed in four graphs.

- The top graphs show the $p - \nu$ and $m - \psi$ soil reaction curves along the monopile at the predefined depths mentioned above.
- The bottom graphs show the $H_B - \nu_B$ and $M_B - \psi_B$ 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 Section 2.2.6.

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 Section 2.4.4 for more information.

The enlarged graph allows choosing to show the normalised and/or parameterised 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.

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 2.4.4) 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. The vertical axis (z) is normalised. The normalisation context changes per graph. For further detail on the normalisation, see Table 4.2. In case that the parameter is constant over depth, then the outer diameter is used for the normalisation.

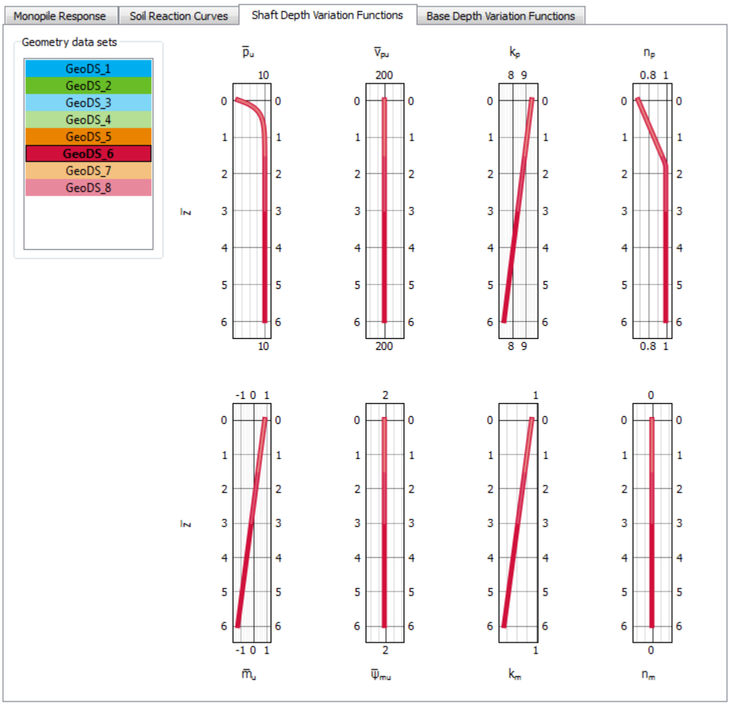


Figure 2.18 Shaft depth variation functions

Table 2.4 Shaft depth variables

Variable on x-axis	Definition
$\overline{\nu}_{pu}$	normalised ultimate lateral displacement
$\overline{p}_u$	normalised ultimate lateral soil reaction
k_p	normalised initial stiffness of the lateral soil reaction
n_p	normalised curvature of the lateral soil reaction
$\overline{\psi}_{mu}$	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 2.19) 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, and originate from equations used in Table 4.1. Note that the vertical axis (L/D_{out}) is normalised.

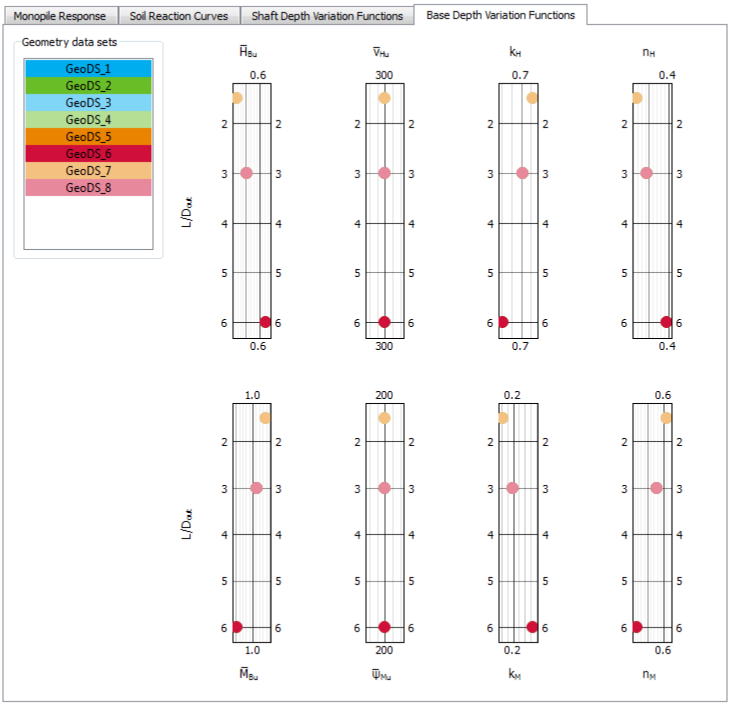


Figure 2.19 Base depth variation functions

Table 2.5 Base fitting parameters

Fitting parameter	Definition
$\overline{v}_{Hu}$	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
$\overline{\psi}_{Mu}$	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

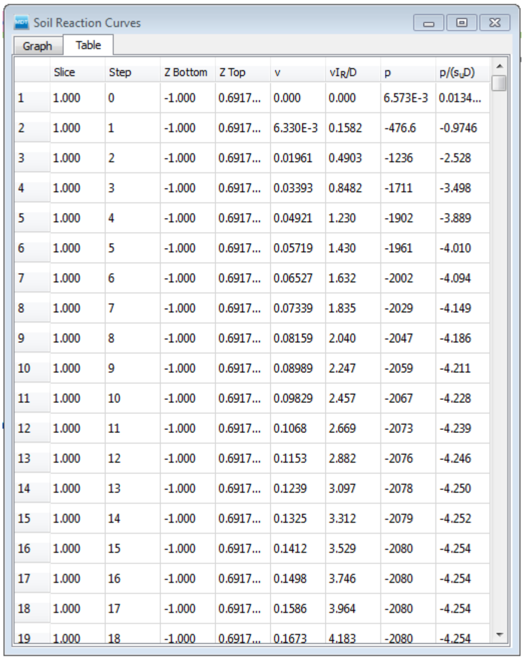
Graph and table windows

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	Step	Z Bottom	Z Top	v	vI_R/D	p	$p/(s_uD)$
1	1.000	0	-1.000	0.6917...	0.000	0.000	6.573E-3	0.0134...
2	1.000	1	-1.000	0.6917...	6.330E-3	0.1582	-476.6	-0.9746
3	1.000	2	-1.000	0.6917...	0.01961	0.4903	-1236	-2.528
4	1.000	3	-1.000	0.6917...	0.03393	0.8482	-1711	-3.498
5	1.000	4	-1.000	0.6917...	0.04921	1.230	-1902	-3.889
6	1.000	5	-1.000	0.6917...	0.05719	1.430	-1961	-4.010
7	1.000	6	-1.000	0.6917...	0.06527	1.632	-2002	-4.094
8	1.000	7	-1.000	0.6917...	0.07339	1.835	-2029	-4.149
9	1.000	8	-1.000	0.6917...	0.08159	2.040	-2047	-4.186
10	1.000	9	-1.000	0.6917...	0.08989	2.247	-2059	-4.211
11	1.000	10	-1.000	0.6917...	0.09829	2.457	-2067	-4.228
12	1.000	11	-1.000	0.6917...	0.1068	2.669	-2073	-4.239
13	1.000	12	-1.000	0.6917...	0.1153	2.882	-2076	-4.246
14	1.000	13	-1.000	0.6917...	0.1239	3.097	-2078	-4.250
15	1.000	14	-1.000	0.6917...	0.1325	3.312	-2079	-4.252
16	1.000	15	-1.000	0.6917...	0.1412	3.529	-2080	-4.254
17	1.000	16	-1.000	0.6917...	0.1498	3.746	-2080	-4.254
18	1.000	17	-1.000	0.6917...	0.1586	3.964	-2080	-4.254
19	1.000	18	-1.000	0.6917...	0.1673	4.183	-2080	-4.254

Figure 2.20 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/Save Selection*, see Figure 2.21.

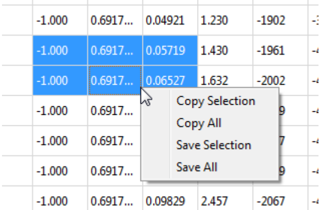


Figure 2.21 Context menu for the table of values

- Copy/Save entire table with right-click > *Copy All/Save All* or CTRL+C.
- To copy to the clipboard, use CTRL+C shortcut.

2.4.5 RECOMMENDED WORKFLOW

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

1. First add one GeoDS and generate it.
2. 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 Section 2.4.2 may be made.
3. Calculate it using the *Calculate* button in PLAXIS MoDeTo and inspect the results in

PLAXIS MoDeTo.

4. Visualise the calculated project in PLAXIS 3D to ensure that the calculation results are as expected.
5. Only after that add more data sets.
6. When adding a dataset, not only the values of the previous dataset are copied, but also the associated PLAXIS 3D project.
7. During the generation of an added dataset with different geometry configuration, PLAXIS MoDeTo 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 Section 2.4.2 have to be followed.

2.5 ANALYSIS MODE

The *Analysis* mode (Figure 2.22) 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 (Section 2.1).

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 MoDeTo, this information is included in the produced *dvf file*. If the rule-based design is adopted, then this information should be specified by the user 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 MoDeTo.

2.5.1 DEPTH VARIATION FUNCTIONS

The depth variation functions define the soil reaction curves that are used in the 1D calculation. 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).

NOTE: Note that the tool remembers the previously selected option (calibrated/custom) and does not reset to the default option (calibrated). The selected option is maintained in memory temporarily, i.e. as long as the project is running, or permanently if the project is manually saved by the user.

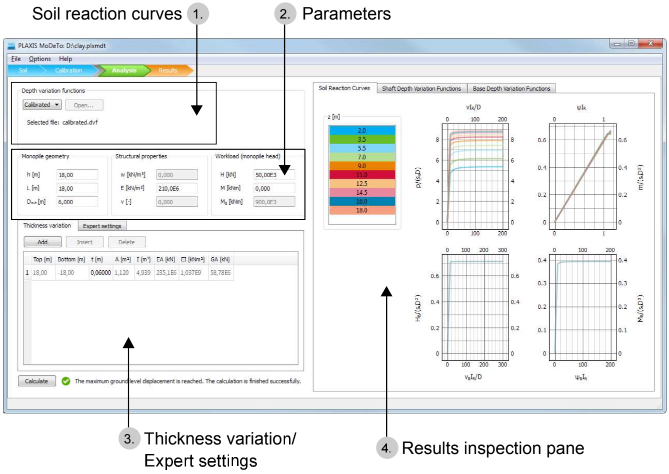


Figure 2.22 Analysis mode layout

TIP: The user may select any other user-defined `dvf` file as long as it complies with the required format.

The two methods for specifying depth variation functions are:

Calibrated This option is only available if the *Calibration* mode has been used. Then a generated file has been created and saved in the project folder's location. This is the default option if available (Figure 2.23). Note that each time the *Analysis* mode is (re)accessed, the *.*dvf* file is always (re)imported to make sure that the *.*dvf* is updated.

Custom User-defined depth variation functions can be imported by the user from the drop-down menu. This is the default option if *Calibrated* is not available. Once a file has been imported, it is displayed as *Selected file:* *.*dvf*

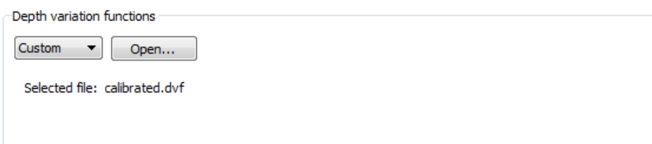


Figure 2.23 Calibrated option for depth variation functions

2.5.2 MONOPILE GEOMETRY

One *Monopile geometry* is used for each calculation. The available geometry parameters are the same as in the *Calibration mode*.

The monopile geometry properties are shown in Table 2.6.

Table 2.6 Monopile geometry parameters

Parameter	Description	Unit
h	height above mudline of the application of the resultant horizontal load	m
L	monopile embedded depth	m
D_{out}	monopile outer diameter	m

2.5.3 STRUCTURAL PROPERTIES

In the *Structural properties* (Figure 2.24) panel it is possible to add/change values of the Young's modulus E. The user cannot edit values of w and ν. These are automatically set to zero by the software.

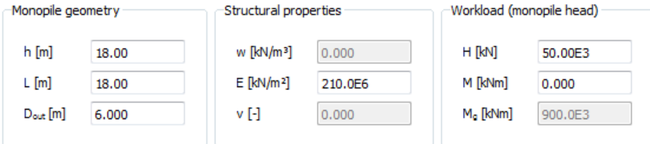


Figure 2.24 Parameters for calculation in *Analysis mode* which includes *Monopile geometry*, *Structural properties* and *Workload (monopile head)*

The *Structural properties* are shown in Table 2.7.

Table 2.7 Structural properties

Property	Description	Unit
γ	pile distributed weight	kN/m
E	Young's modulus	kN/m^2
ν	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.

2.5.4 WORKLOAD (MONOPILE HEAD)

A horizontal load H and a bending moment M may be applied to the monopile head, at height h above the ground level. 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 gives the minimum, maximum and default values for the input horizontal force H and the moment M.

Table 2.8 Minimum, maximum and default values for horizontal force *H* and moment *M*

Parameter	Description	Unit
H	Horizontal force at monopile head	kN
M	Moment at monopile head	kMm

Figure 2.25 Parameters and units

2.5.5 THICKNESS VARIATION

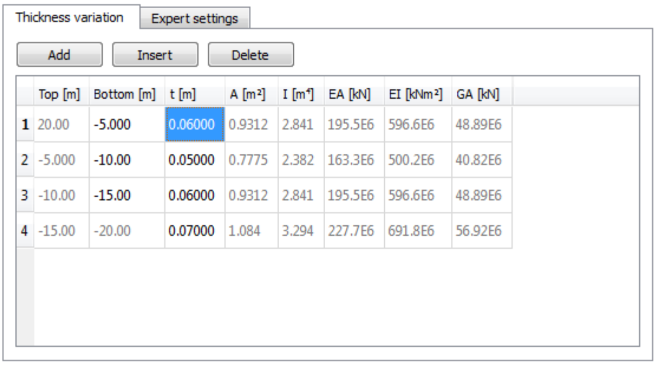


Figure 2.26 Thickness variation tabsheet

In the *Thickness variation* tabsheet, which is shown in Figure 2.26, it is possible to insert and edit pile segments. These are displayed in Figure 2.27.

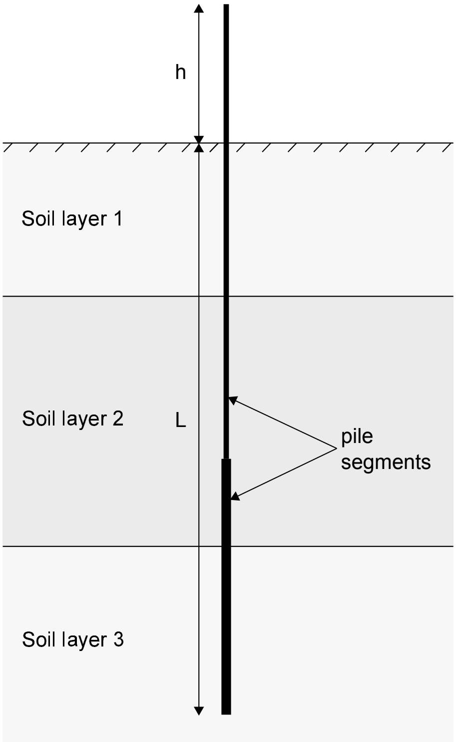


Figure 2.27 Schematic representation of the pile with two different segments (thickness variation) used in 1D model

The user can create pile segments using buttons above the table:

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.

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 newly inserted segment is placed right above a selected segment.
6. A segment's bottom boundary cannot be lower than the next segment's bottom boundary.
7. 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.

2.5.6 EXPERT SETTINGS

The *Expert settings* tabsheet (Figure 2.28) has parameters as described below.

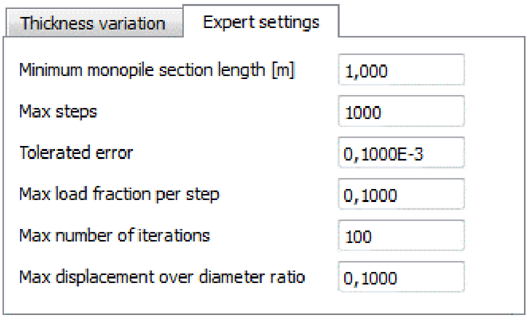


Figure 2.28 Expert settings tabsheet

Table 2.9 gives the minimum, maximum and default values for the expert settings parameters.

Table 2.9 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	0.1000
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: $N = L\, /$ $<$ Minimum monopile section length $>$.

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 (Section B.2).

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.

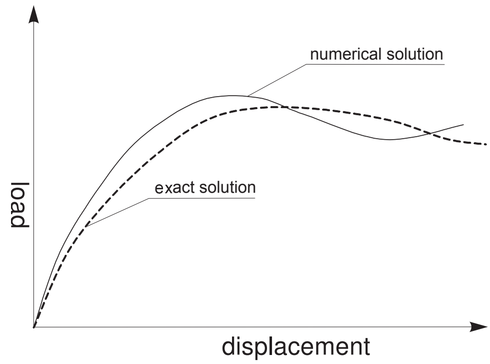


Figure 2.29 Computed solution versus exact solution

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.

2.5.7 CALCULATE

The user can click on the *Calculate* button to start the calculation process after specifying the monopile geometry, loading, and importing the depth variation functions. For more information on importing depth variation functions refer to Section 2.5.1.

Calculation 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.

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 Section B.2 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.

2.5.8 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 2.30) 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.

TIP: Charts are updated after a successful calculation.

The four types of soil reaction curves ($p - \nu$, $m - \psi$, $H_B - \nu_B$ and $M_B - \psi_B$) are displayed in four graphs.

- The top graphs show the $p - \nu$ and $m - \psi$ soil reaction curves along the monopile at the predefined depths
- The bottom graphs show the $H_B - \nu_B$ and $M_B - \psi_B$ soil reaction curves at the base of the monopile (at depth L)

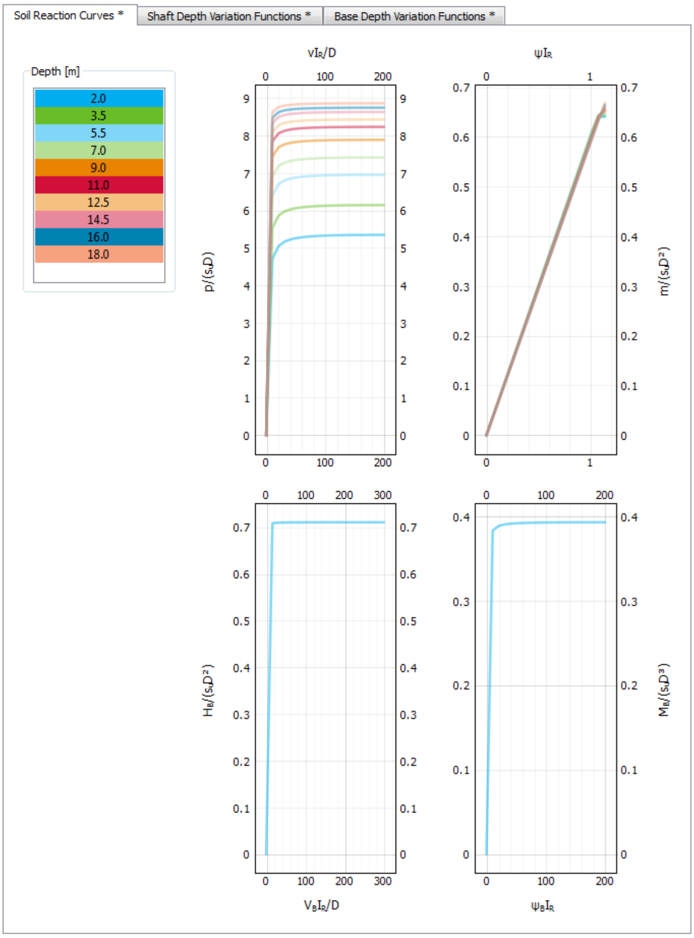


Figure 2.30 Soil reaction curves tabsheet

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. Refer to Section 2.4.4 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.

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 2.4. Note that the vertical (z) axis is not normalised, in contrast to the *Calibration* mode.

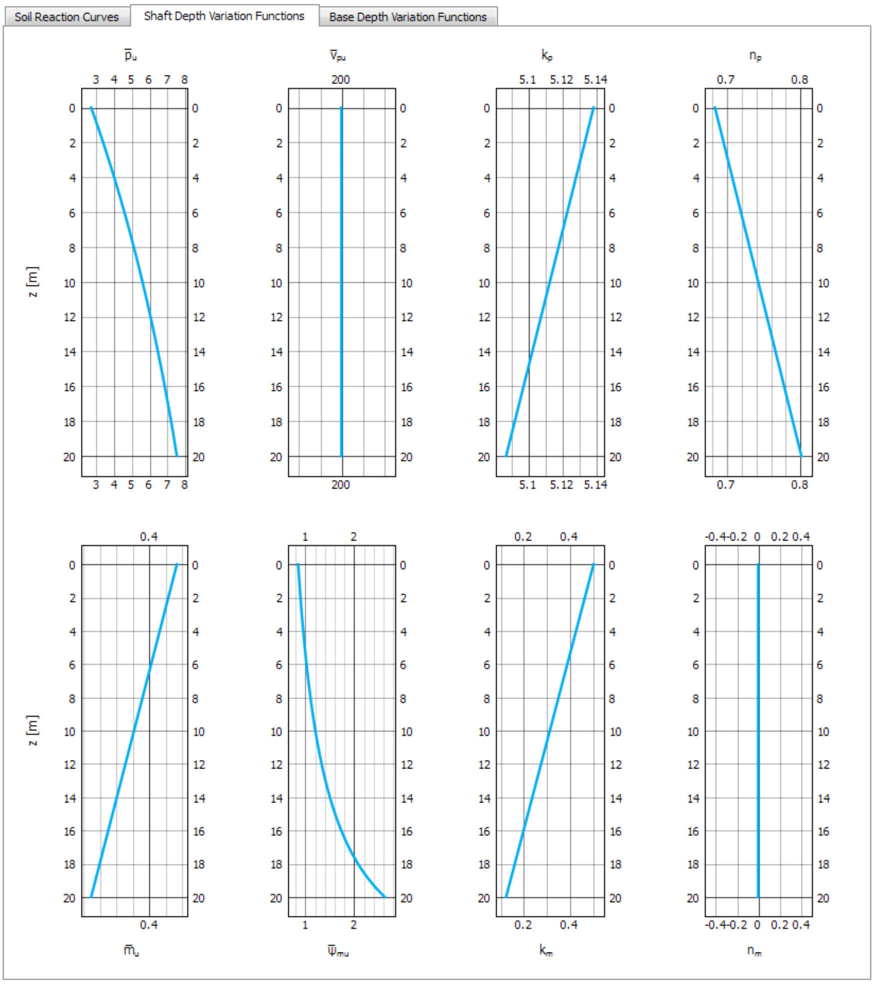


Figure 2.31 Shaft depth variation tab

For more information on the shaft fitting parameters, see Table 2.4.

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 2.5. 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.

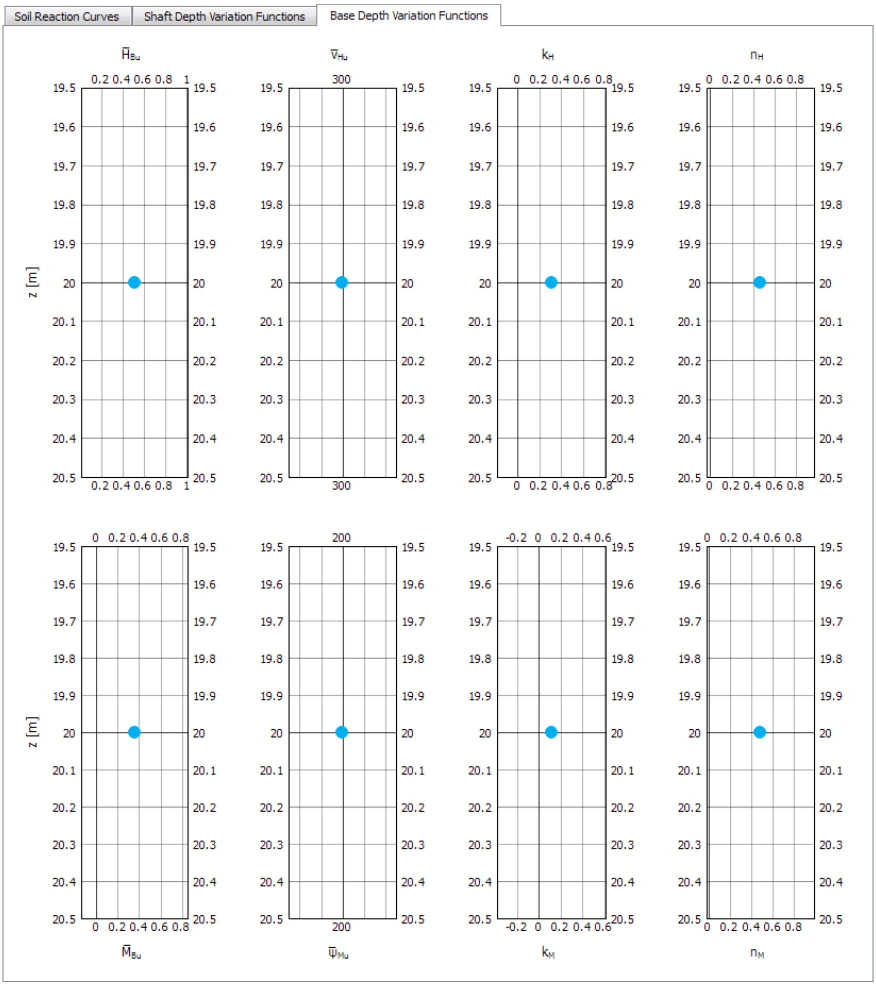


Figure 2.32 Base depth variation tab

For more information on the base fitting parameters, see Table 2.5.

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 2.4.4 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.

2.6 RESULTS MODE

This mode presents the results of the 1D calculation run in the *Analysis* mode. Additional results for one of the calculated 3D models in the *Calibration* mode may be presented as well for comparison between the 3D model and 1D model results. To display the results from the 3D models the exact corresponding 3D model needs to be selected from the drop-down list. 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 models presented in the combo box. Thus if new models are generated and calculated in the *Calibration* mode, the 1D calculation has to be repeated to update the data in this mode.

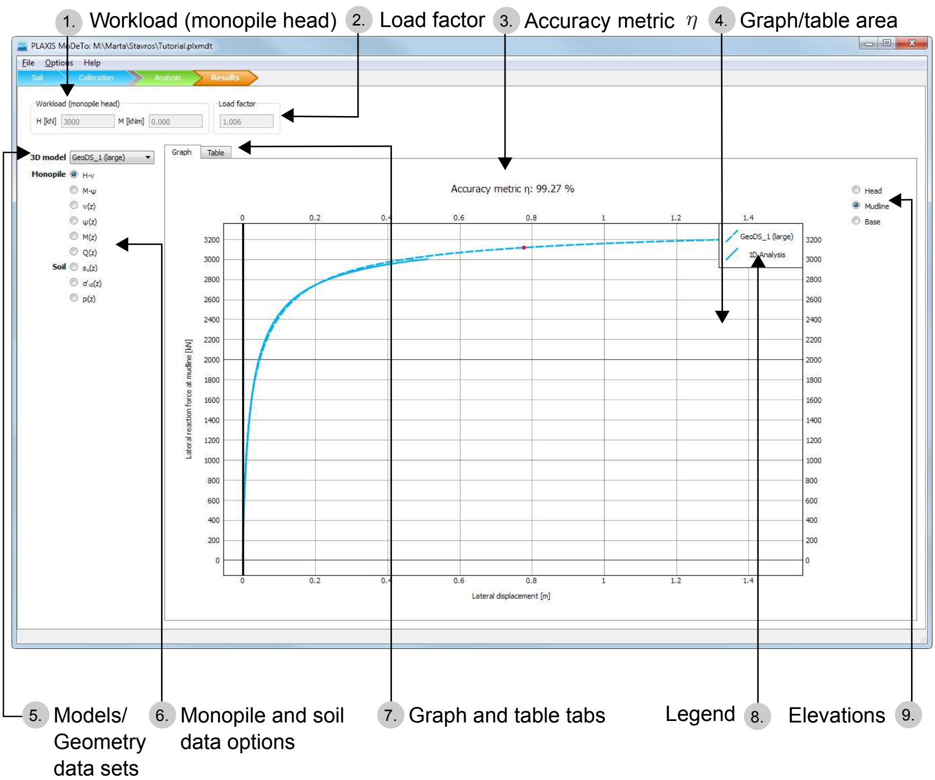


Figure 2.33 View of the *Results* mode

2.6.1 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 $\geq$ 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.

2.6.2 GRAPH TAB

The graph tab shows the graphical results of the 1D model. It can also show the results from one of the 3D models in the *Calibration* mode to enable a comparison between the 1D and 3D models, either for small deformations or large deformations.

The user can choose from various GeoDS present in the project using the 3D model selector.

NOTE: To update the 3D results after any changes done in the *Calibration* mode, the user needs to rerun the 1D analysis in the *Analysis* mode.

NOTE: No significant meaning can be derived from comparing different geometries. It's possible to go back to the *Calibration* mode and create a 3D model with the final geometry for validation purposes.

Examples:

GeoDS_x (small)	results from the small displacements 3D calculation
GeoDS_x (large)	results from the large displacements 3D calculation

NOTE: The 3D data are only available at mudline for the $H - v$ and $M - \psi$ plots.

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 labeled by the legend.

For the first two plots ($H - v$ and $M - \psi$), 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.

Monopile results:

H-v	Lateral reaction force at mudline (kN) versus lateral displacement (m) at the mudline, base or head

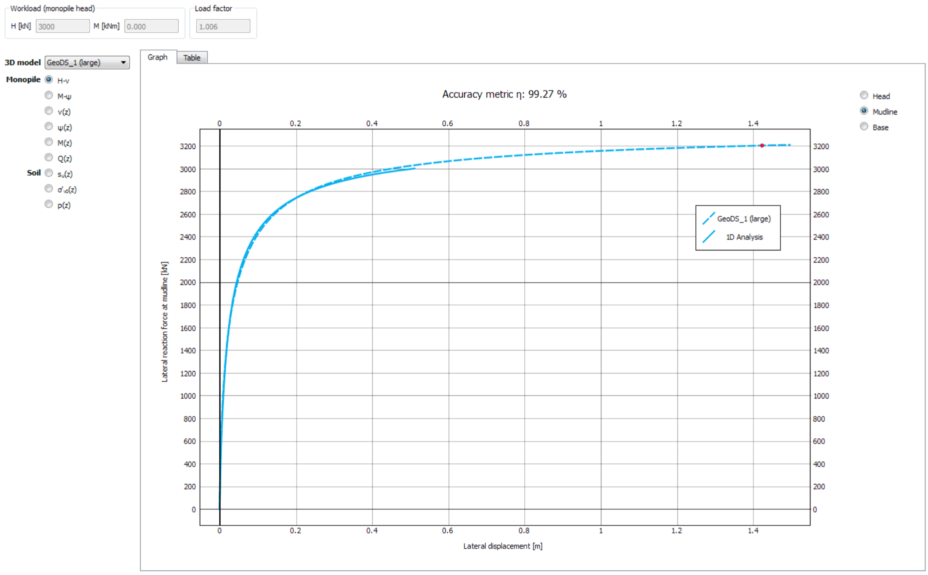


Figure 2.34 Example of H-v at mudline curve

M-ψ	Bending moment (kNm) at mudline versus monopile cross section rotation (rad) at the mudline, base or head
$v(z)$	Monopile deflection (m) over depth (m)
$\psi(z)$	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:

$s_u(z)$	Undrained shear strength (kN/m^2) over depth (m).
$\sigma'_{v0}(z)$	Initial vertical effective stress (kN/m^2) 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 $max(|z|) = -L$.

NOTE: In the case that a sandy material type has been selected in the *Soil* mode specified via the **.dvf* file (*Analysis* mode), the option to display the undrained shear strength (s_u) along depth (z) is inactive.

2.6.3 TABLE TAB

Provides data displayed in the graph, including 3D data if selected and available (Figure 2.6.3).

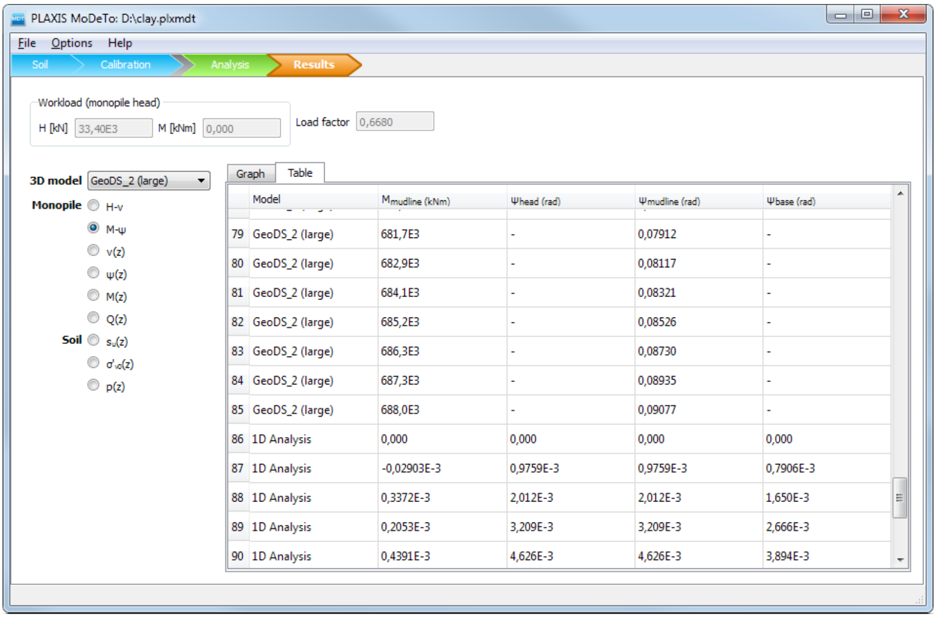


Figure 2.35 Example of table of results

2.6.4 ACCURACY METRIC η

The accuracy metric η (Figure 2.6.4) is an indicator 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 is shown in percentage (%) and is visible at the top of the graph. The closer the η 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%

NOTE: The user should use the accuracy metric η 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.

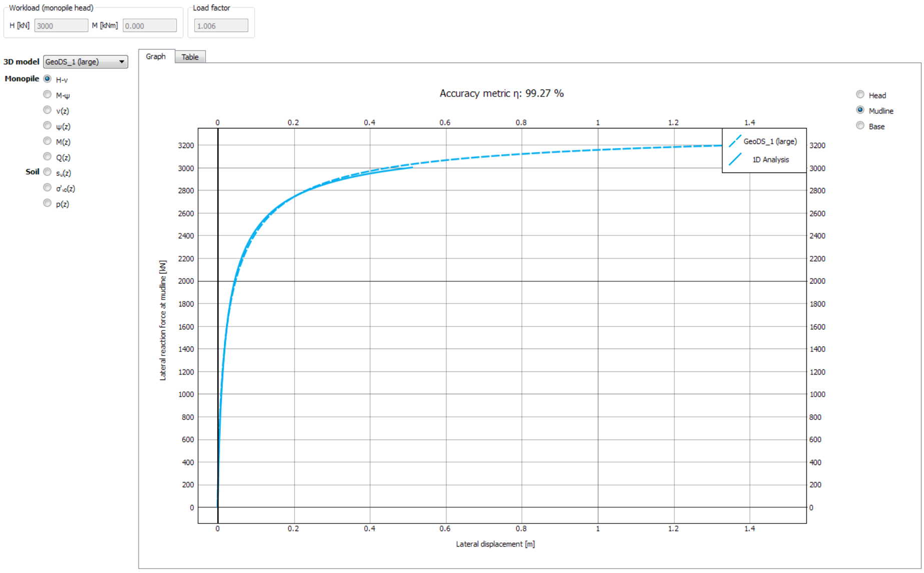


Figure 2.36 Example of accuracy metric η

3 TUTORIAL MANUAL

3.1 TUTORIAL - 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 submerged unit weight γ (Figure 3.1 (a)), the undrained shear strength s_u in triaxial compression (TXC) (Figure 3.1 (b)), the lateral earth pressure at rest K_0 in terms of effective stresses (Figure 3.1 (c)), and the small strain shear modulus G_0 (Figure 3.1 (d)).

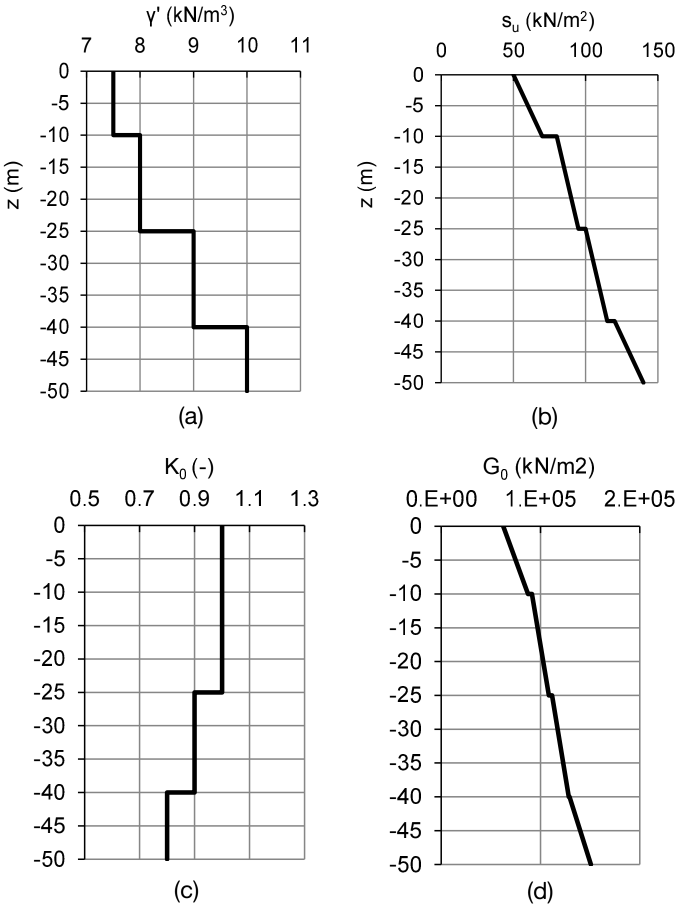


Figure 3.1 Submerged unit weight γ (a), undrained shear strength s_u in triaxial compression (TXC) (b), lateral earth pressure at rest K_0 in terms of effective stresses (c), small strain shear modulus G_0 (d).

TIP: In PLAXIS MoDeTo 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 illustrates the adopted design space. The geometric dimensions of the assumed final design case are also presented in Figure 3.2. 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.

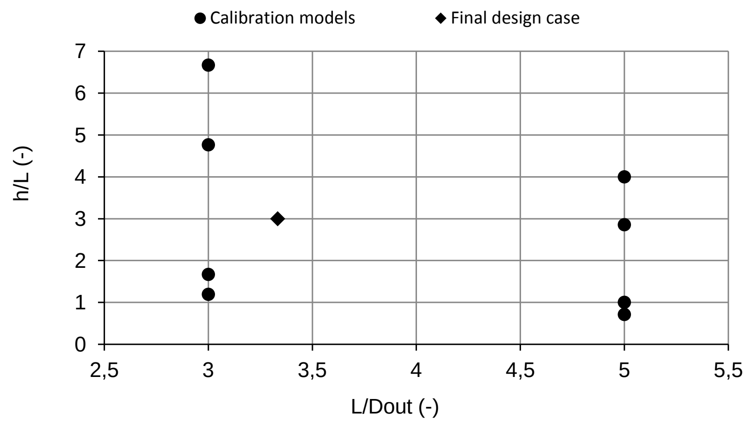


Figure 3.2 Adopted design space

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 GeoDS 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 $0.1 \cdot D$ 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 1D model, in contrast to the 3D model, does allow for thickness variations.

3.1.1 INPUT

General settings

- Start PLAXIS MoDeTo via the executable `PlaxisMoDeTo.exe`.
- In the *Quick select* dialogue (Figure 3.3) choose *Start a new project* and save the project with the name "MoDeTo Tutorial" in the desired directory.

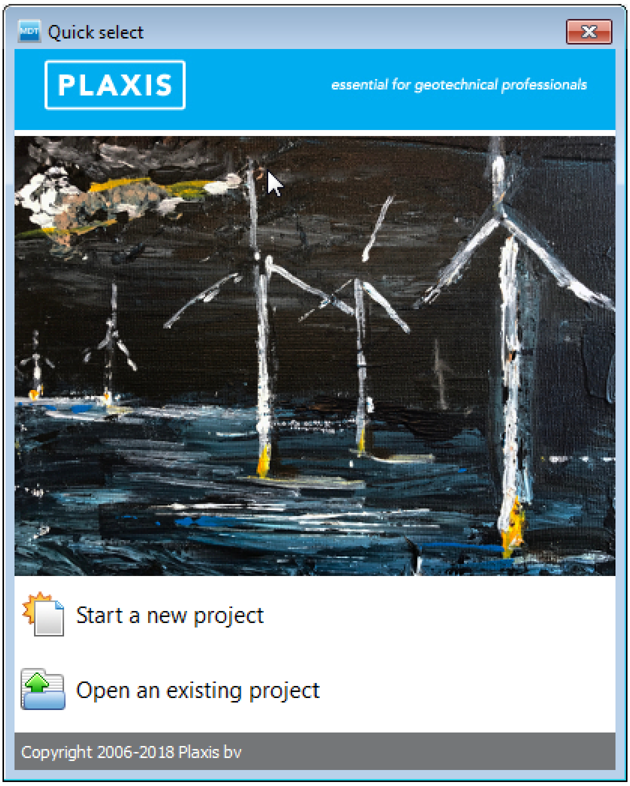


Figure 3.3 Quick select window

Soil mode - definition of the soil stratigraphy

First, the soil is defined by following these steps:

1. Make sure that the program is in the *Soil* mode.
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.

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

The result after all layers have been added is shown in Figure 3.4.

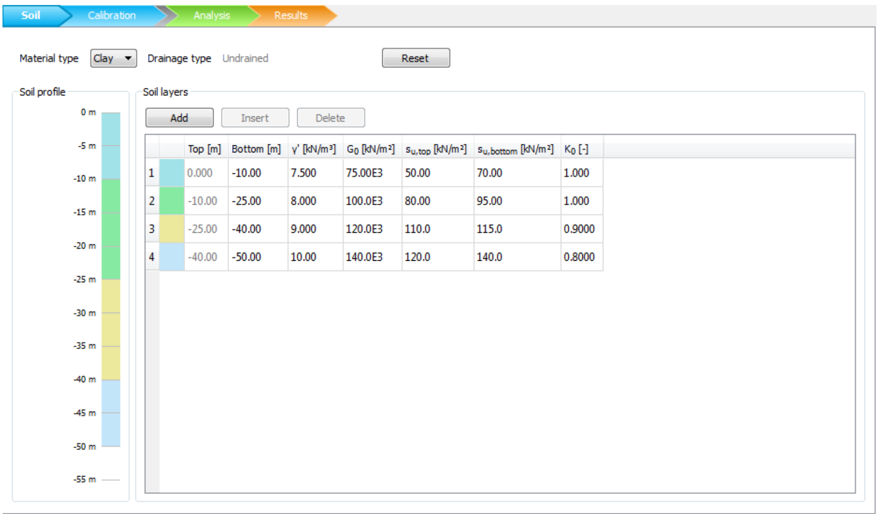


Figure 3.4 Layers generated in *Soil* mode

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). 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 Section 4.2.1 for more information on the parameters.

Calibration mode - definition of the geometry data sets

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

1. Proceed to *Calibration* mode.
2. Add all the needed GeoDS using the *Add* button, one by one. In the geometrical characteristics fill in the data presented in Table 3.2.

   To get a calibration with good quality, a lateral ground displacement of about 0.2·D is needed. See Section 4.3.1 for more information on how to estimate the needed value of $v_{max,z=h}$.

TIP: When the *Generate* button is pressed, PLAXIS MoDeTo automatically verifies the value of $v_{max,z=h}$ 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 first generate the first model (GeoDS_1), then make the needed changes within PLAXIS 3D and afterwards 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 Section 2.4.2. 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.

Table 3.2 Geometry data sets

#	h (m)	L (m)	D_{out} (m)	t (m)	$v_{max,z=h}$ (m)
1	25.0	15.0	5.0	0.05	5.0
2	25.0	25.0	5.0	0.05	3.0
3	100.0	15.0	5.0	0.05	15.0
4	100.0	25.0	5.0	0.05	9.0
5	25.0	21.0	7.0	0.07	5.0
6	25.0	35.0	7.0	0.07	4.0
7	100.0	21.0	7.0	0.07	15.0
8	100.0	35.0	7.0	0.07	10.0

The definition of the parameters in Table 3.2 can be found in Section 2.4.2.

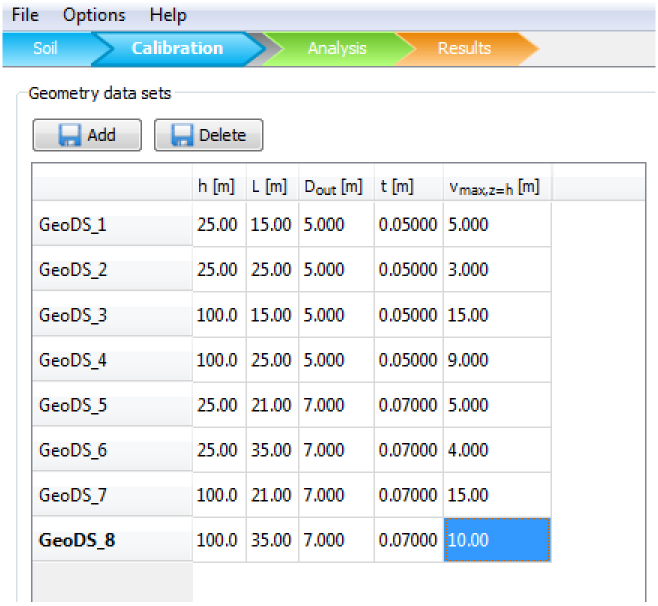
NOTE: This action automatically saves the project each time a new GeoDS is added to the list.

The result after adding all GeoDS is shown in Figure 3.5.

3. The default values are used for the *Structural properties*, i.e. the Young's modulus E and the Poisson's ratio ν of the plate material are $E = 210 \cdot 10^6$ kN/m^2 and $\nu = 0.3$.
4. 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.

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



	h [m]	L [m]	D_{out} [m]	t [m]	$v_{max,z=h}$ [m]
GeoDS_1	25.00	15.00	5.000	0.05000	5.000
GeoDS_2	25.00	25.00	5.000	0.05000	3.000
GeoDS_3	100.0	15.00	5.000	0.05000	15.00
GeoDS_4	100.0	25.00	5.000	0.05000	9.000
GeoDS_5	25.00	21.00	7.000	0.07000	5.000
GeoDS_6	25.00	35.00	7.000	0.07000	4.000
GeoDS_7	100.0	21.00	7.000	0.07000	15.00
GeoDS_8	100.0	35.00	7.000	0.07000	10.00

Figure 3.5 GeoDS generated in *Calibration* mode

Additionally, the validity of the generated material properties may be checked against the formulations provided in Section 4.2.1.

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 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 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 9) 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.

TIP: Detailed 3D results can be inspected directly in PLAXIS Output by selecting the calculated GeoDS and pressing the *View* button

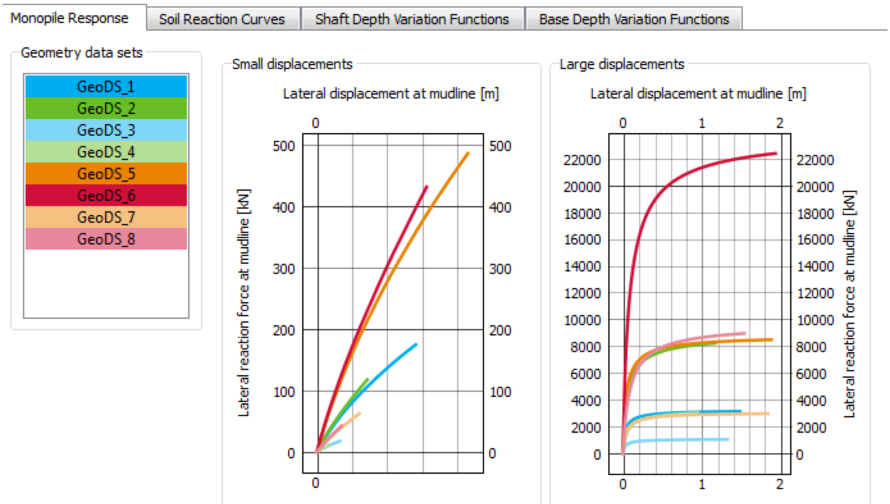


Figure 3.6 Monopile results tab - displacement

10. Focus on the "deflection below mudline" graphs (two bottom graphs, see Figure 10) 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.

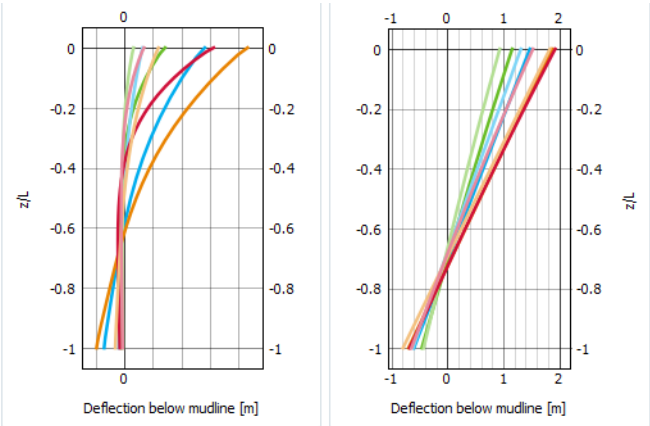


Figure 3.7 Monopile results tab - deflection

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 `calibrated.dvf` within the project its 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*.

3.1.2 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 `calibrated.dvf` that was created and saved during calibration is selected by default. Leave this selection as it is.
3. In the *Monopile geometry* section, enter the values of the geometric parameters (h, L, D_{out}) which correspond to GeoDS_1, see Figure 3.5.
4. In the *Workload (monopile head)* section, enter a value for the horizontal force which 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 present example, a value of 3200 kN should be used.
5. Add a pile segment by clicking the *Add* button on the *Thickness variation* tab. Enter the thickness value t that corresponds to GeoDS_1, see Figure 3.5.
6. Click on the *Calculate* button to start the 1D analysis.

NOTE: The 1D calculation takes only a few seconds.

NOTE: Next to the *Thickness variation* tab, there is an *Expert settings* tab. Use the default values, see Figure 3.1.2.

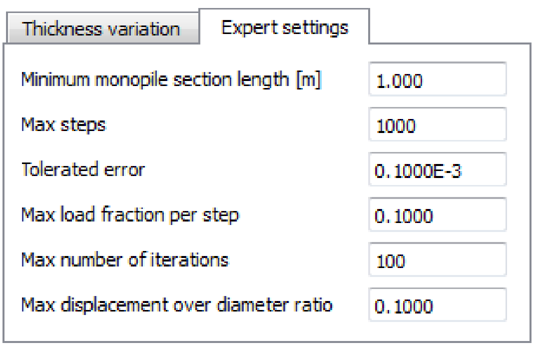


Figure 3.8 Expert settings - default values

Results mode

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

1. Proceed to *Results* mode.

2. Select GeoDS_1 (large) from the options included in the 3D model combobox.
3. Compare the results of the 1D and the selected 3D model by inspecting the $H - v$ graph at mudline. See Figure 3.

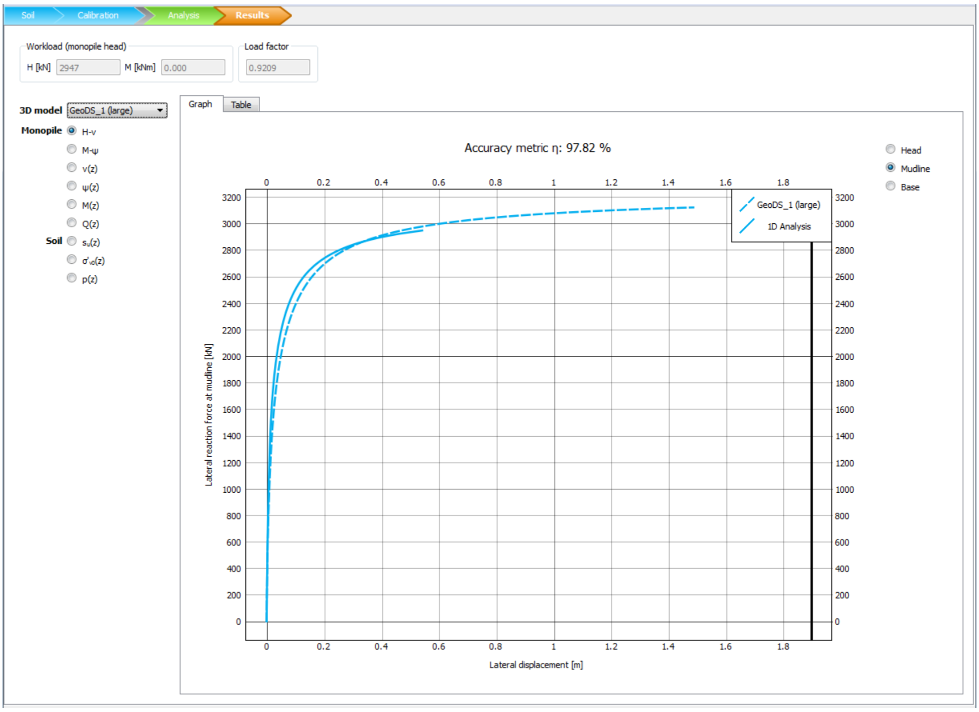


Figure 3.9 Comparison of 1D and 3D model results

The high value (97.82%) of the accuracy metric (η) indicates that the calibration of the 1D model was done successfully via the parameterisation procedure.

NOTE: By default for the 1D analysis, the target lateral displacement at ground level equals $0.1 \cdot D$ (see *Expert settings* tab of the *Analysis* mode). The accuracy metric η is calculated based on that value. See Section 4.7.4 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 $M - \psi$, $v(z)$ and $\psi(z)$ graphs.
Note that the $v(z)$ and $\psi(z)$ 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.

3.1.3 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 that only geometrical configurations that fall within the assumed design space are recommended to be analysed.

Analysis mode

1. In the *Analysis* mode enter the following geometrical characteristics, which correspond to the assumed final monopile design that meets the required design criteria. See Table 3.3.

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

TIP: The final design may consist of more than one pile segment with different thicknesses. This can be done by adding more thickness sections by clicking the *Add* button on the *Thickness variation* tab of the *Analysis* mode.

2. In the *Results* mode, inspect the $H - v$ graph (at mudline) to ensure that the displacement is less than $0.1{\cdot}D$ when the design load H is applied, see Figure 3.10

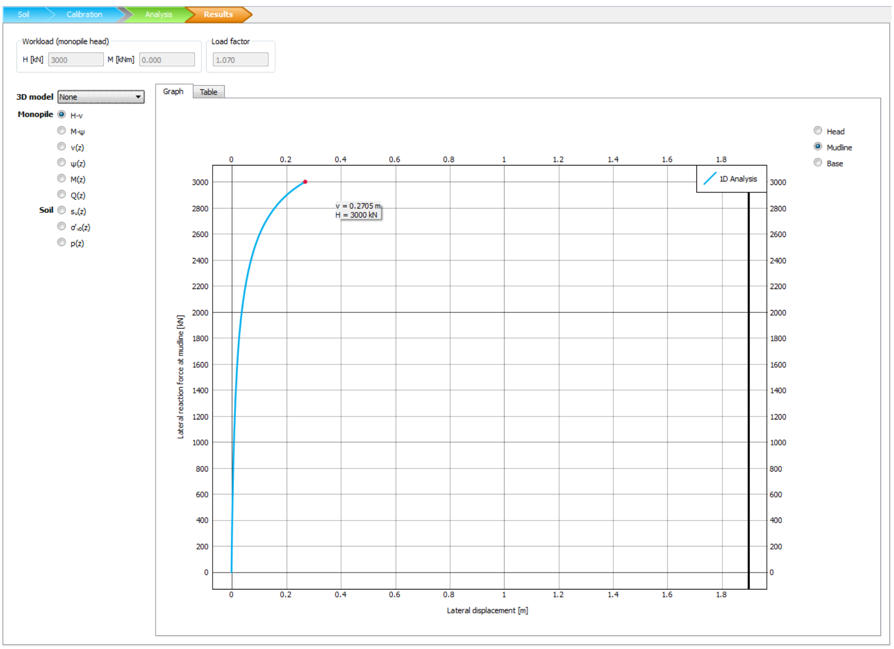


Figure 3.10 Check displacement at design load

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

3.1.4 VERIFICATION OF THE FINAL DESIGN

Calibration mode

1. 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. In the *Calibration* mode, add a ninth geometry data set with the geometrical characteristics given in Table 3.3. The value of the parameter $v_{max,z=h}$ should be defined such that a lateral displacement at mudline of about $0.1 \cdot D$ is achieved (based on the specified design criteria). A value of 7.0 m satisfies this condition for the present case study.
2. Generate and calculate the PLAXIS 3D model.

> **TIP:** In case that more than one pile segment is employed in the final design, the user needs to generate and then access the model in PLAXIS 3D to manually introduce the needed additional plate materials.
> The properties may be determined based on the parameters displayed within the *Thickness variation* tab of the *Analysis* mode, namely: A, I, EA, EI, GA.
> After creating the additional plate materials in PLAXIS 3D, it is necessary to assign them to the corresponding plate elements, based on the thickness of each monopile segment. The 3D project needs to be saved and then calculated via PLAXIS MoDeTo.

Results mode

1. In the *Results* mode, select the final design (GeoDS_9 (large) in the 3D model combobox) 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 3.11 for the comparison in the $H - v$ graph at mudline.

   Notice the high value (96.82%) of the accuracy metric (η), which indicates a good match.

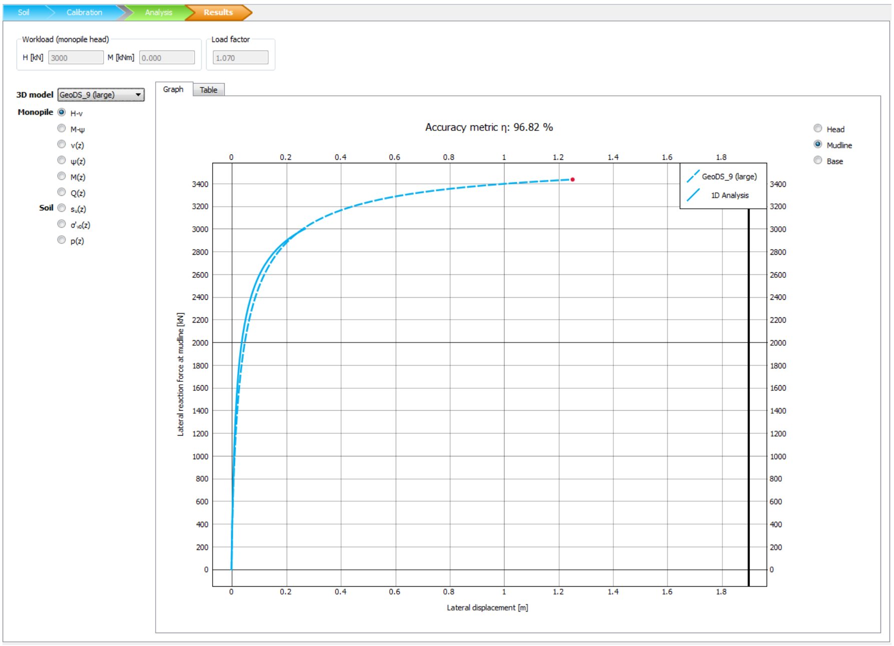


Figure 3.11 Compare 1D and 3D model results for final design

4 SCIENTIFIC MANUAL

4.1 INTRODUCTION

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

4.2 MATERIAL MODELS

The following predefined soil types may be used in PLAXIS MoDeTo:

- 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., (2017).

4.2.1 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[†]
- $\gamma_{unsat} = \gamma'$ (input)[‡] (kN/m^3)
- $\gamma_{sat} = \gamma_{unsat}$ (kN/m^3)
- e_{init} = 0.500 (default)
- $G_{ur} = G_0$, input at mid-depth of the layer (kN/m^2)
- $s^A_{u\,ref} = s_{u,top}$ (kN/m^2)
- $G_{ur}/s^A_u = G_0/((s_{u,bottom} + s_{u,top})/2)$
  Note that, s^A_u is the value of the active undrained shear strength at the mid-layer

† 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.

‡ 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}.

level[§]

- $\gamma_f^C = 60 \cdot 100/G_{ur}/s_u^A$ (%)
- $0.5\% \leq \gamma_f^C \leq 75\%$
- $\gamma_f^E = 2.0 \cdot \gamma_f^C$ (%)
- $1.0\% \leq \gamma_f^E \leq 150\%$
- $\gamma_f^{DSS} = 1.5 \cdot \gamma_f^C$ (%)
- $0.75\% \leq \gamma_f^{DSS} \leq 100\%$
- $s_u^{C,TX}/s_u^A = 0.99$ (-)
- $z_{ref} = z_{top}$ (m) (negative value)
- $s_{u\,inc}^A = (s_{u,bottom} - s_{u,top})/(z_{top} - z_{bottom})$ (kN/m^2/m)
- $s_u^P/s_u^A = 0.5$ (-)
- $\tau_0/s_u^A = -0.5 \cdot (1 - K_0) \cdot \sigma'_{v0}/s_u^A$
  where $\sigma'_{\nu 0}$ = initial vertical effective stress at the mid-layer level (kN/m^2). Note that compression is negative.
- $0.0 \leq \tau_0/s_u^A \leq 0.95$ (-)
- $s_u^{DSS}/s_u^A = (1 + s_u^P/s_u^A)/2$ (-)
- $\nu' = 0.495$ (-)
- $R_{inter} = 1.0$ (-)
- K_0 = input (-)

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., 2017).

4.2.2 SAND: HSsmall MATERIAL PARAMETERS

The Hardening Soil model with small-strain stiffness (Brinkgreve et al., 2017) 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

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

- $\gamma_{sat} = \gamma'$ (input)[¶] [‖] (kN/m^3)
- $\gamma_{unsat} = \gamma_{sat}$ (kN/m^3)
- e_{init} = 0.500 (-)
- G_0 = input at mid-depth of the layer (kN/m^2)
- $\sigma'_3 = K_0 \cdot \sigma'_1$
  where $\sigma'_1 = \sigma'_{\nu 0}$ at the mid-layer level (kN/m^2)
- $G_0^{ref} = G_0/[(c'_{ref} \cdot \cos\varphi - \sigma'_3 \cdot \sin\varphi')/(c'_{ref} \cdot \cos\varphi + p_{ref} \cdot \sin\varphi')]^m$ (kN/m^2)
- RD = $100 \cdot (G_0^{ref} - 60000)/68000$ (%)
- $E_{50}^{ref} = 60000 \cdot RD/100$ (kN/m^2)
- $E_{oed}^{ref} = E_{50}^{ref}$ (kN/m^2)
- $E_{ur}^{ref} = 3 \cdot E_{50}^{ref}$ (kN/m^2)
- m = 0.5 (-)
- c'_{ref} = 0.1 (kN/m^2)
- φ' = input (°)
- ψ = input (°)
- $\gamma_{0.7} = (2 - RD/100) \cdot 1E\text{-}4$ (-)
- ν'_{ur} = 0.2 (-)
- p_{ref} = 100.0 (kN/m^2)
- $K_0^{NC} = 1 - \sin\varphi'$ (-)
- c'_{inc} = 0.0 (kN/m^2/m)
- z_{ref} = 0.0 (m)
- $R_f = 1 - RD/800$ (-)
- tensile strength = 0.0 (kN/m^2)
- R_{inter} = 1.0 (-)
- K_0 = input (-)

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.

[¶] By using γ' (input value) as γ_{sat} and γ_{unsat} effective stresses are calculated without the need to calculate the water pressures.
[‖] Note that γ' is used as both γ_{sat} and γ_{unsat} because the phreatic level is set at the bottom of the model.

4.3 PLAXIS 3D MODELS

4.3.1 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 MoDeTo 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 ν and wall thickness t.
- The top of the monopile is closed with a plate of the same 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).
- 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:
  - inner diameter: $D_{in} = D_{out} - 2 \cdot t$
  - cross section area: $A = \pi(D_{out}^2 - D_{in}^2)/4$
  - moment of inertia: $I = \pi(D_{out}^4 - D_{in}^4)/64$
  - axial stiffness: EA
  - flexural rigidity: EI
  - shear stiffness: $GA = 0.5 \cdot kEA/(1 + \nu)$

  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 and Scientific manuals.

- Two 'soft' beams (Section 4.3.2) 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 segments of 20° 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 2017.01 or older versions: _mesh 0.075 256 True
    - mesh command used in PLAXIS 3D 2018: _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: Applying prescribed lateral displacements equal to $v_{max,z=h}/1000$, plastic calculation
  - Phase 3: Applying prescribed lateral displacements (input value, $v_{max,z=h}$), 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_{max}

Under the assumption of having the monopile rotation point in the ground at a depth approximately equal to $2 \cdot L/3$, the following formulas are used to derive the suggested values of the parameter $v_{max,z=h}$ for clay and sand based on the Similar Triangles Theorem.

- min target displacement at the ground level is around $0.20 \cdot D$:
  $v_{max,z=h} = (h + 2 \cdot L/3) \cdot 0.30 \cdot D/L$
- max target displacement at ground level is around $0.30 \cdot D$:
  $v_{max,z=h} = (h + 2 \cdot L/3) \cdot 0.45 \cdot D/L$

NOTE: The estimated values are based on the assumption of a linearly deformed pile. In the 3D FE model, the pile will bend. This implies that the displacement at mudline for a given $v_{max,z=h}$ will be smaller than the one estimated based on the Similar Triangles Theorem. The difference is larger for a higher h.

4.3.2 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 MoDeTo 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.

4.3.3 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, ψ)
- Base horizontal force vs. lateral base displacement: (H_B, v_B)
- Base moment vs. base rotation: (M_B, ψ_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 Section 4.3.1)
- 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: $p_{slice} = \sum (Fx_{interface})$
- The *base (toe)* forces are the summation of the forces from the bottom interface: $p_{bottom} = \sum (Fx_{interface,bottom})$
- The slice displacement is the average value of the lateral displacement of the slice nodes $v_{slice} = \overline{u}_{x,interface}$
- 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 $m_{slice} = \sum (F_z \cdot d_x)$
- The base (toe) moment: same as above, considering the bottom interface
- The slice rotation ψ_{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 MoDeTo (*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 4.1:

Table 4.1 Normalisation formulae for *Sand* and *Clay* (Burd et al., 2018, Byrne et al., 2018b)

Component	Clay normalisation	Sand normalisation
Distributed load, $\overline{p}$	$\overline{p} = p/(s_u/D)$	$\overline{p} = p/(\sigma'_{\nu 0} D)$
Lateral displacement, $\overline{\nu}$	$\overline{\nu} = vI_R/D$	$\overline{\nu} = (vI_R/D) \cdot \sqrt{(}p_a/\sigma'_{\nu 0}) =$ $\nu G_0/(\sigma'_{\nu 0} D)$
Distributed moment, $\overline{m}$	$\overline{m} = m/(s_u D^2)$	$\overline{m} = m/(pD)$
Pile cross section rotation, $\overline{\psi}$	$\overline{\psi} = \psi I_R$	$\overline{\psi} = \psi I_s \cdot \sqrt{(}p_a/\sigma'_{\nu 0}) = \psi G_0/\sigma'_{\nu 0}$
Base horizontal force, $\overline{H}_B$	$\overline{H}_B = H_B/(s_u D^2)$	$\overline{H}_B = H_B/(\sigma'_{\nu 0} D^2)$
Base moment, $\overline{M}_B$	$\overline{M}_B = M_B/(s_u D^3)$	$\overline{M}_B = M_B/(\sigma'_{\nu 0} D^3)$

The normalised curves depend on the undrained shear strength, s_u, or the initial vertical effective stress, $\sigma'_{\nu 0}$, 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: $I_S = (G_0/p_a) \cdot \sqrt{p_a/\sigma'_{\nu 0}}$.

4.3.4 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 *Parameterised* checkboxes (default): the data of the raw soil reaction curves (extracted from the 3D FE models) are plotted for the following predefined depths:
  $0.1{\cdot}L, 0.2{\cdot}L, 0.3{\cdot}L, 0.4{\cdot}L, 0.5{\cdot}L, 0.6{\cdot}L, 0.7{\cdot}L, 0.8{\cdot}L, 0.9{\cdot}L, L$ (base)
- Checked *Normalised* and unchecked *Parameterised* checkboxes: if the normalised raw data are selected to be plotted, then the data is derived from the raw soil reaction curves and normalisation formulas presented in Table 4.1
- Checked *Normalised* and checked *Parameterised* checkboxes: the parameterised

soil reaction curves obtained from the parameterisation procedure are plotted

- Unchecked *Normalised* and checked *Parameterised* checkboxes: the parameterised soil reaction curves obtained from the parameterisation procedure are denormalised and plotted

Depth variation functions: shaft and base

The plots are derived based on the `.dvf` files and the corresponding functions (Table 4.2).

4.4 OPTIMISATION 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 4.1) 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 Eq. (4.1):

$$-n \cdot \left(\frac{\overline{y}}{\overline{y}_u} - \frac{\overline{x}}{\overline{x}_u}\right)^2 + (1 - n) \cdot \left(\frac{\overline{y}}{\overline{y}_u} - \frac{\overline{x} \cdot k}{\overline{y}_u}\right) \cdot \left(\frac{\overline{y}}{\overline{y}_u} - 1\right) = 0 \tag{4.1}$$

where $\overline{x}$ refers to a normalised displacement (or rotation) variable and $\overline{y}$ signifies the corresponding normalised soil reaction component. The conic function is calibrated by the specification of four parameters (k, n, $\overline{x}_u$, $\overline{y}_u$), each of which has a straightforward interpretation. The parameter k specifies the initial slope; $\overline{y}_u$ is the ultimate value of the normalised soil reaction and $\overline{x}_u$ is the normalized displacement (or rotation) at which this ultimate value of soil reaction is reached. The parameter n ($0 < n < 1$) determines the shape of the curve. This particular function was selected during the PISA project to represent the soil reaction curves in the 1D design model (Burd et al., 2018, Byrne et al., 2018b). 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, $\overline{x}_u$, $\overline{y}_u$) 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 $\overline{y}$ are:

$$\overline{y} = \overline{y}_u \cdot \frac{2c}{-b + \sqrt{b^2 - 4ac}} \qquad \text{for} \qquad \overline{x} \leqslant \overline{x}_u \tag{4.2}$$

$$\overline{y} = \overline{y}_u \qquad \text{for} \qquad \overline{x} > \overline{x}_u \tag{4.3}$$

where:

$$a = 1 - 2 \cdot n$$
$$b = 2 \cdot n \cdot \frac{\overline{x}}{\overline{x}_u} - (1 - n) \cdot (1 + \frac{\overline{x} \cdot k}{\overline{y}_u})$$
$$c = \frac{\overline{x} \cdot k}{\overline{y}_u} \cdot (1 - n) - n \cdot \frac{\overline{x}^2}{\overline{x}_u{}^2} \tag{4.4}$$

The shape of the conic function is strongly conditioned by the value of n, as illustrated in Figure 4.1. For $n = 0$ and $n = 1$, bi-linear forms are obtained. For intermediate values of n the function is curved.

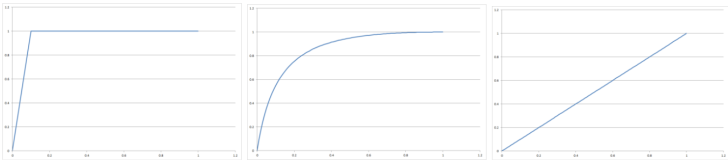

n = 0 (k = 10)   n = 0.5 (k = 10)   n = 1

Figure 4.1 Curves for different values of the n-parameter

The parameters corresponding to each soil reaction curve are:

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

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., 2018b. 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/rotation $(\overline{v}_{pu}, \overline{\psi}_{mu}, \overline{v}_{Hu}, \overline{\psi}_{mu})$ to match the form of the numerical data.
2. Determine values of the ultimate values $(\overline{p}_u, \overline{m}_u, \overline{H}_u, \overline{M}_u)$.
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 4.2.

Table 4.2: 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	$c4 + c5 \cdot (z/D_{out})$	$s4$
	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, $\overline{\psi}_{mu}$	$c9$	$s7$
	Initial stiffness, k_m	$c10 + c11 \cdot (z/D_{out})$	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	$c16 + c17 \cdot (L/D_{out})$	$s14+s15 \cdot (L/D_{out})$
	Curvature, n_H	$c18 + c19 \cdot (L/D_{out})$	$s16+s17 \cdot (L/D_{out})$
	Ultimate reaction, $\overline{H}_{Bu}$	$c20 + c21 \cdot (L/D_{out})$	$s18+s19 \cdot (L/D_{out})$
Base moment, $\overline{M}_B$	Ultimate rotation, $\overline{\psi}_{Mu}$	$c22$	$s20$
	Initial stiffness, k_M	$c23 + c24 \cdot (L/D_{out})$	$s21$
	Curvature, n_M	$c25 + c26 \cdot (L/D_{out})$	$s22$
	Ultimate reaction, $\overline{M}_{Bu}$	$c27 + c28 \cdot (L/D_{out})$	$s23+s24 \cdot (L/D_{out})$

Figure 4.2 presents the adopted workflow.

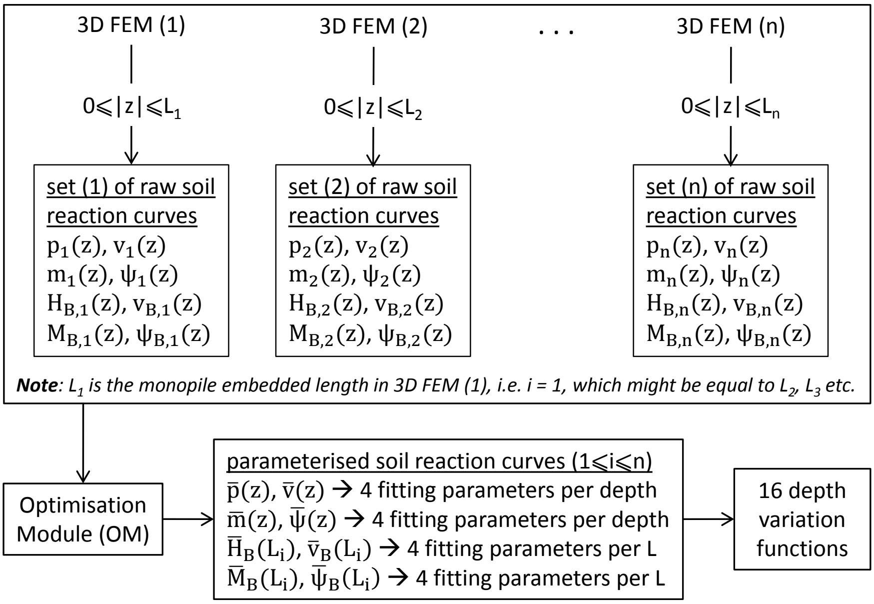


Figure 4.2 Calibration workflow

4.5 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 Section 2.2.6.
- 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. $\sim$ 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 files should comply with the rules presented above.

4.5.1 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

Note that the strings used in the file are case insensitive.

In total 28 parameters are needed to define the depth variation functions for Sand, and 24 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 MODETO DEPTH VARIATION FUNCTIONS
# Version number
1
# Parameterisation function type
conic
# Material type
clay
# Drainage type
undrained
# Number of soil layers
2
# SoilLayer ztop(m) zbottom(m) G0top(kN/m2) G0bottom(kN/m2) sutop(kN/m2) subottom(kN/m2) gammasubmerged(kN/m3)
  1          0.00     -20.00     25.0E3         30.0E3        15.00         22.00           16.50
  2        -20.00    -100.00     30.0E3        120.0E3        25.00         45.00           17.50
# GeoDS
# Number of GeoDS
4
# L(m)   h(m)   t(m)   Dout(m)   E(kN/m2)
60.00   50.00   0.010   10.00   210.0E+06
30.00   50.00   0.010   6.000   210.0E+06
50.00   45.00   0.010   6.000   210.0E+06
40.00   40.00   0.010   6.000   210.0E+06
# Max displacement reached at ground level (m)
3.005
# Max rotation reached at ground level (rad)
0.212
# Fitting parameters
c1
c2
c3
...
c28
```

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

```
# Depth variation functions flag
PLAXIS MODETO DEPTH VARIATION FUNCTIONS
# Version number
1
# Parameterisation function type
```

```
conic
# Material type
sand
# Drainage type
drained
# number of soil layers
2
# SoilLayer ztop(m) zbottom(m) G0top(kN/m2) G0bottom(kN/m2) ceff(kN/m2) phieff(deg) psi(deg) gammasubmerged(kN/m3)
  1          0.00     -20.00       25.0E3      30.0E3          0.00        0.00        0.00       6.50
  2        -20.00    -100.00       30.0E3      90.0E3          0.00        0.00        0.00       7.50
# number of GeoDS
4
# L(m)    h(m)    t(m)   Dout(m)   E(kN/m2)
35.00   50.00   0.010  10.00     210E6
30.00   50.00   0.010   6.000    210E6
75.00   45.00   0.010   6.000    210E6
60.00   40.00   0.010   6.000    210E6
# Max displacement reached at ground level
2.012
# Max rotation reached at ground level (rad)
0.546
# 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).

4.6 1D FE MODEL

4.6.1 FORMULATION OF THE 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., 2018, and Byrne et al., 2018b. 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 4.3. 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.

Kinematics

Adopted kinematics are presented in Figure 4.4. Equation 4.6.1 describes the axial and transverse displacements in a pile in a case where the neutral axis coincides with the centroid of the pile.

$$w(y,z) = y\psi(z) \qquad\qquad v(y,z) = v_0(z) + f(y) \tag{4.5}$$

where ψ is the clockwise rotation of the beam cross-section (assumed to remain plane), v_0 is the lateral displacement of the pile centroid, and $f(y)$ is a function to represent the

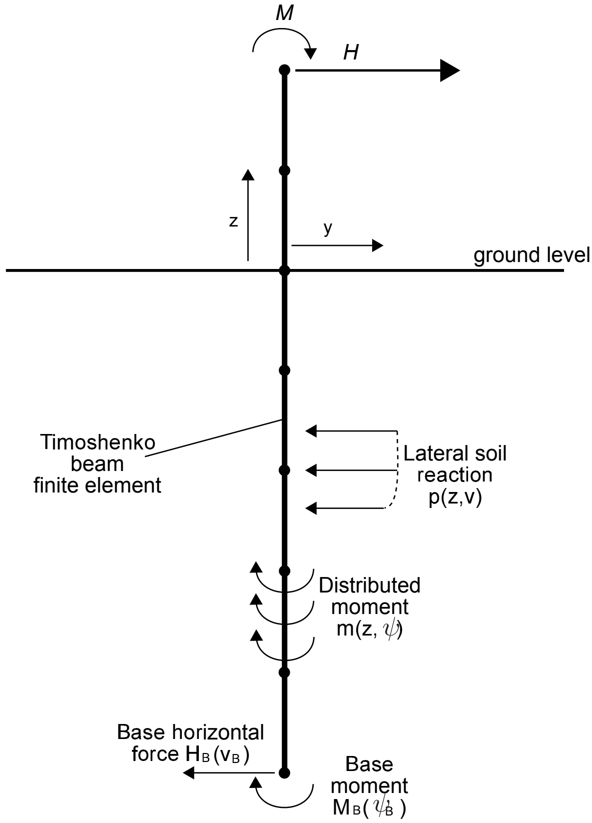


Figure 4.3 1D structural model of monopile foundation for a wind turbine (redrawn from Byrne et al., 2018b)

coupling between the axial and transverse strains, which are defined by:

$$\varepsilon_{zz} = \frac{\partial w}{\partial z} = y\frac{d\psi}{dz} \qquad \varepsilon_{yy} = \frac{\partial f}{\partial z} \qquad \gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} = \theta + \psi \qquad \left(where \quad \theta = \frac{\partial v}{\partial z}\right) \tag{4.6}$$

Bending moment and shear force

The axial stress in the pile is $\sigma_{zz} = E\varepsilon_{zz}$. The bending moment is:

$$M = \int \sigma_{zz} y dA = E\left[\int y^2 dA\right]\frac{d\psi}{dz} = EI\frac{d\psi}{dz} \tag{4.7}$$

A positive bending moment causes tension on the y-positive side of the pile.

The shear force is:

$$V = \int G\gamma_{yz} dA \quad = \quad GA\kappa(\theta + \psi) \tag{4.8}$$

where G is the shear modulus, A is the cross-sectional area of the pile and κ is a shear factor.

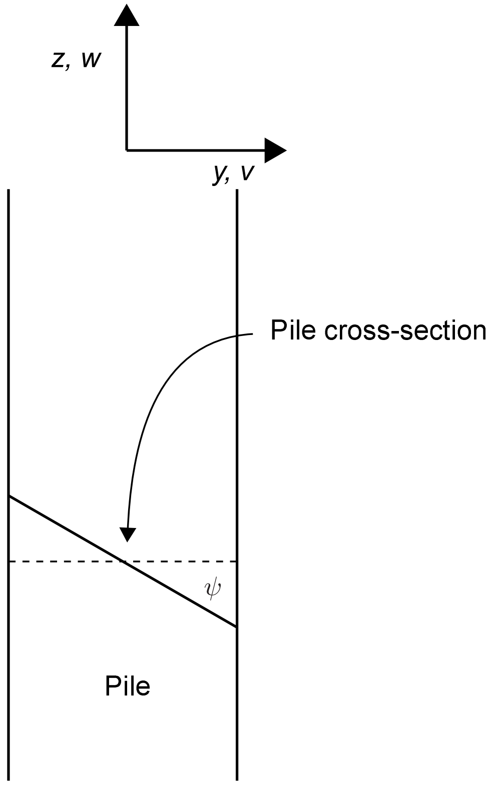


Figure 4.4 Monopile foundation for a wind turbine support structure (redrawn from Byrne et al., 2018b)

Virtual Work

At equilibrium the total virtual work is zero. The external virtual work is:

$$\delta W_E = -H_T \delta v_T - M_T \delta \psi_T \tag{4.9}$$

where H_T and M_T are the horizontal force and moment applied at the top of the pile (as in Figure 4.4) and v_T and ψ_T are the lateral displacement and cross-section rotation at the top of the pile. The internal virtual work is:

$$\delta W_I = \int_{pile} \left(M \frac{d\delta\psi}{dz} + V(\delta\theta + \delta\phi) + p(z, v)\delta v + m(z, \psi)\delta\psi \right) dz + H_B \delta v_B + M_B \delta \psi_B \tag{4.10}$$

This may be expressed as:

$$\delta W_I = \int_{pile} \left(\frac{d\delta\psi}{dz} EI \frac{d\psi}{dz} + (\delta\theta + \delta\phi) GA\kappa(\theta + \psi) + \delta v p(z, v) + \delta\psi m(z, \psi) \right) dz + \delta v_B H_B + \delta \psi_B M_B \tag{4.11}$$

Finite element discretisation

The pile is discretised into 2-noded finite elements as shown in Figure 4.5. The lateral displacement within each element is determined using Equation 4.12:

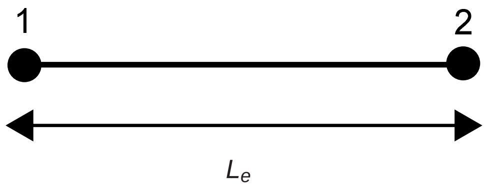


Figure 4.5 2-noded beam element

$$v = N_1^h V_1 + N_2^h \Theta_1 + N_3^h V_2 + N_4^h \Theta_2 \tag{4.12}$$

where N_i^h are set of Hermitian shape functions given by:

$$N_1^h = 1 - 3\alpha^2 + 2\alpha^3 \tag{4.13}$$

$$N_2^h = \alpha L_e(1 - 2\alpha + \alpha^2) \tag{4.14}$$

$$N_3^h = 3\alpha^2 - 2\alpha^3 \tag{4.15}$$

$$N_4^h = \alpha L_e(-\alpha + \alpha^2) \tag{4.16}$$

where α $(0 < \alpha < 1)$ is a parametric variable defined by the interpolation:

$$z = N_1^l Z_1 + N_2^l Z_2 \tag{4.17}$$

and N_i^l are the Langrangian interpolation functions:

$$N_1 = 1 - \alpha \qquad N_2 = \alpha \tag{4.18}$$

According to Astley (1992), within each element the shear strain is constant, γ_0 and therefore $\theta = \gamma_0 - \psi$. This gives:

$$v = N_1 V_1 - N_2 \Psi_1 + (N_2 + N_4)\gamma_0 + N_3 V_2 - N_4 \Psi_2 \tag{4.19}$$

The first and second derivatives are defined as shown in Section 4.6.1.

Finite element equations for the pile

Bending terms:

The bending moment in the pile is $M = EI\dfrac{d\psi}{dz}$. For the current beam element formulation:

$$\frac{d\psi}{dz} = -\frac{d\theta}{dz} + \frac{d\gamma_0}{dz} = -\frac{d\theta}{dz} = -\frac{d^2 v}{dz^2} \tag{4.20}$$

This may be expressed in matrix form:

$$\frac{d\psi}{dz} = \underline{B_B V} \tag{4.21}$$

where:

$$\underline{B}_B = \left[\frac{d^2(N_1^h)}{dz^2} \quad -\frac{d^2(N_2^h)}{dz^2} \quad \frac{d^2(N_2^h + N_4^h)}{dz^2} \quad \frac{d^2(N_3^h)}{dz^2} \quad -\frac{d^2(N_4^h)}{dz^2} \right] \tag{4.22}$$

$$\underline{V}^T = \left[V_1 \ \Psi_1 \ \gamma_0 \ V_2 \ \Psi_2 \right] \tag{4.23}$$

The terms in B_B are:

$$\underline{B}_B = \frac{1}{L_e^2} \left[6 - 12\alpha \ \ L_e(6\alpha - 4) \ \ L_e(6 - 12\alpha) \ \ 12\alpha - 6 \ \ L_e(6\alpha - 2) \right] \tag{4.24}$$

The element force vector f_B and the element stiffness matrix k_B are:

$$\underline{f}_B = \int_{element} \underline{B}_B^T M dz \qquad \underline{k}_B = \int_{element} \underline{B}_B^T EI \underline{B}_B dz \tag{4.25}$$

Shear terms:

The shear force within the pile element is:

$$\gamma_0 = \underline{B_S V} \tag{4.26}$$

where:

$$\underline{B}_S = \left[0 \ 0 \ 1 \ 0 \ 0 \right] \tag{4.27}$$

The corresponding element force and stiffness matrices are:

$$\underline{f}_S = \int_{element} \underline{B}_S^T V dz \qquad \underline{k}_S = \int_{element} \underline{B}_S^T \kappa GA \underline{B}_S dz \tag{4.28}$$

Finite element equations for the soil

The lateral displacement v within the element is:

$$v = \underline{B}_V V \quad where \quad \underline{B}_V = \left[N_1^h \ \ -N_2^h \ \ (N_2^h + N_4^h) \ \ N_3^h \ \ -N_4^h \right] \tag{4.29}$$

The element force and stiffness matrices are:

$$\underline{f}_{DP} = \int_{element} \underline{B}_V^T p dz \qquad \underline{k}_{DP} = \int_{element} \underline{B}_V^T \left(\frac{dp}{dv} \right) \underline{B}_V dz \tag{4.30}$$

The cross-section pile rotation ψ is:

$$\psi = \underline{B}_\psi V \quad where \quad \underline{B}_\psi = \left[\frac{dN_1^h}{dz} \quad -\frac{dN_2^h}{dz} \quad \left(\frac{dN_2^h}{dz} + \frac{dN_4^h}{dz} + 1 \right) \quad \frac{dN_3^h}{dz} \quad -\frac{dN_4^h}{dz} \right] \tag{4.31}$$

The element force and stiffness matrices are:

$$\underline{f}_\psi = \int_{element} \underline{B}_\psi^T m dz \qquad \underline{k}_\psi = \int_{element} \underline{B}_v^T \left(\frac{dm}{d\psi}\right) \underline{B}\psi dz \tag{4.32}$$

Hermitian shape functions

$$N_1^h = 1 - 3\alpha^2 + 2\alpha^3 \tag{4.33}$$

$$N_2^h = \alpha L_e(1 - 2\alpha + \alpha^2) \tag{4.34}$$

$$N_3^h = 3\alpha^2 - 2\alpha^3 \tag{4.35}$$

$$N_4^h = \alpha L_e(-\alpha + \alpha^2) \tag{4.36}$$

The first derivatives are:

$$\frac{dN_1^h}{d\alpha} = -6\alpha + 6\alpha^2 \qquad \frac{dN_1^h}{dx} = \frac{1}{L_e}(-6\alpha + 6\alpha^2) \tag{4.37}$$

$$\frac{dN_2^h}{d\alpha} = L_e(1 - 4\alpha + 3\alpha^2) \qquad \frac{dN_2^h}{dx} = 1 - 4\alpha + 3\alpha^2 \tag{4.38}$$

$$\frac{dN_3^h}{d\alpha} = 6\alpha - 6\alpha^2 \qquad \frac{dN_3^h}{dx} = \frac{1}{L_e}(6\alpha - 6\alpha^2) \tag{4.39}$$

$$\frac{dN_4^h}{d\alpha} = L_e(-2\alpha + 3\alpha^2) \qquad \frac{dN_4^h}{dx} = -2\alpha + 3\alpha^2 \tag{4.40}$$

The second derivatives are:

$$\frac{d^2N_1^h}{d\alpha^2} = 12\alpha - 6 \qquad \frac{d^2N_1^h}{dx^2} = \frac{1}{L_e^2}(12\alpha - 6) \tag{4.41}$$

$$\frac{d^2N_2^h}{d\alpha^2} = L_e(6\alpha - 4) \qquad \frac{d^2N_2^h}{dx^2} = \frac{1}{L_e}(6\alpha - 4) \tag{4.42}$$

$$\frac{d^2N_3^h}{d\alpha^2} = 6 - 12\alpha \qquad \frac{d^2N_3^h}{dx^2} = \frac{1}{L_e^2}(6 - 12\alpha) \tag{4.43}$$

$$\frac{d^2N_4^h}{d\alpha^2} = L_e(6\alpha - 2) \qquad \frac{d^2N_4^h}{dx^2} = \frac{1}{L_e}(6\alpha - 2) \tag{4.44}$$

4.6.2 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, Crisfield, Remmers & Verhoosel (2012).

The incremental generated displacement (a vector of unknowns) $\Delta\mathbf{u}^{i+1}$ and incremental load factor $\Delta\lambda^{i+1}$ are calculated using the following equations:

$$\Delta\underline{u}^{i+1} = \Delta\underline{u}^{i} + d\underline{u}^{i+1} \tag{4.45}$$

$$d\underline{u}^{i+1} = d\underline{u}_{I}^{i+1} + \Delta\lambda d\underline{u}_{II}^{i+1} \tag{4.46}$$

$$d\underline{u}_{I}^{i+1} = \underline{K}_{\mathrm{elastic}}^{-1}\underline{r} \tag{4.47}$$

$$\underline{r} = \underline{F}_{ext}^{i} - \underline{F}_{int}^{i} \tag{4.48}$$

$$d\underline{u}_{II}^{i+1} = \underline{K}^{-1}\hat{\underline{F}} \tag{4.49}$$

$$\Delta\lambda^{i+1} = \frac{[\Delta\underline{u}^{1}]^{T} d\underline{u}_{I}^{i+1}}{[\Delta\underline{u}^{1}]^{T} d\underline{u}_{II}^{i+1}} \tag{4.50}$$

Where:

$\underline{F}_{ext}^{i}$: external applied load at iteration i (4.51)

$\underline{F}_{int}^{i}$: internal forces at iteration i (4.52)

$\underline{K}_{\mathrm{elastic}}$: elastic stiffness matrix (4.53)

$\Delta\underline{u}^{i}$: cumulative vector of unknowns (displacements and rotation) at iteration i (4.54)

$\Delta\lambda$: load factor (4.55)

$\hat{\underline{F}}$: unit external load (4.56)

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:

$$Error = \frac{(\underline{F}_{ext}^{i} - \underline{F}_{int}^{i}) \cdot \Delta\underline{u}^{i}}{\underline{F}_{ext}\underline{K}_{elastic}^{-1}\underline{F}_{ext}} < tolerance \tag{4.57}$$

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:

$$\alpha = 0.5 \cdot (1 + \xi) \tag{4.58}$$

which transforms the standard Gaussian coordinates $\xi \in [-1, 1]$ to the parametric coordinates $\alpha \in [0, 1]$.

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 $\overline{v}_{pu} < \overline{p}_u/k_p \rightarrow \overline{v}_{pu} = \overline{p}_u/k_p$
5. If $\overline{p}_u \leq 0$ or $k_p = 0 \rightarrow$ the output of the conic function is set to zero: $\overline{p} = 0$ and $d\overline{p}/d\overline{v}_p = 0$

Note that $k_{p,min}$ is a minimum value of the initial stiffness parameter k_p, determined during the parameterisation. This is to prevent negative values of stiffness close to the ground level.

For the distributed moment:

1. if $\overline{\psi}_{mu} < \overline{m}_u/k_m \rightarrow \overline{\psi}_{mu} = \overline{m}_u/k_m$
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 $d\overline{m}/d\overline{\psi}_m = 0$

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 $d\overline{H}_B/d\overline{v}_H = 0$

For the base moment:

1. If $n_M < 0 \rightarrow n_M = 0$
2. If $n_M > 1 \rightarrow n_M = 1$
3. If $\overline{\psi}_{Mu} < \overline{M}_{Bu}/k_M \rightarrow \overline{\psi}_{Mu} = \overline{M}_{Bu}/k_M$
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 $d\overline{M}_B/d\overline{\psi}_M = 0$

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.

4.7 RESULTS MODE

4.7.1 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*:

$$H = \min\left(H_{input}; \textit{Load factor} \cdot H_{input}\right) \tag{4.59}$$

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.

4.7.2 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*:

$$M = \min\left(M_{input}; \textit{Load factor} \cdot M_{input}\right) \tag{4.60}$$

NOTE: The mentioned M_{input} is referred to as M in the *Analysis and Results modes*.

The M-ψ plot is based on the realised moment M.

4.7.3 LOAD FACTOR

$$Load\ factor = \frac{H_{max}}{H_{input}} = \frac{M_{max}}{M_{input}} \tag{4.61}$$

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.

4.7.4 ACCURACY METRIC η

An accuracy metric, η, is 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., 2018b.

The accuracy metric η 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 is calculated as follows:

$$\eta = \frac{(A_{ref} - A_{dif})}{A_{ref}} \leq 1.0 \tag{4.62}$$

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 4.6). 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 that a low value 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.

4.7.5 *H-v* AND *M-ψ* PLOTS

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.

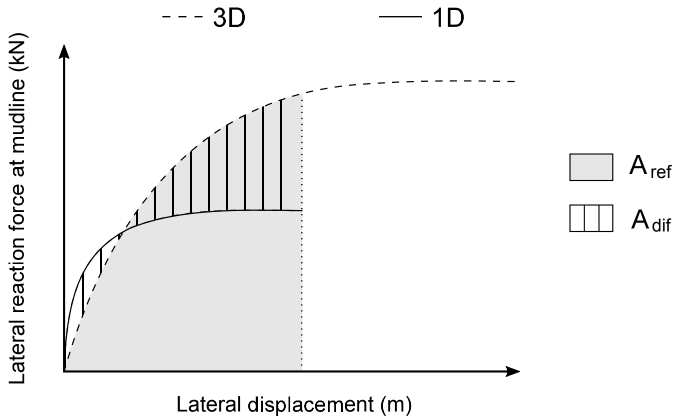


Figure 4.6 Graphic representation of the quantities used to calculate the accuracy metric η

- 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 $\rightarrow$ v_{head} and ψ_{head} are plotted
  - If mudline is selected $\rightarrow$ $v_{mudline}$ and $\psi_{mudline}$ are plotted
  - If base is selected $\rightarrow$ v_{base} and ψ_{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_x
  - $H_{mudline}$ [kN]
  - v_{head} [m]
  - $v_{mudline}$ [m]
  - v_{base} [m]
- The table for the M-ψ graph contains all the information, no matter the selected radio button. The following columns are provided:
  - Model: 1D Analysis, GeoDS_x
  - $_{mudline}$ [kNm]
  - ψ_{head} [rad]
  - $\psi_{mudline}$ [rad]
  - ψ_{base} [rad]
- The available data from a selected 3D model are plotted only in the case that the mudline radio button is selected.

4.7.6 $v(z)$ AND $\psi(z)$ 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.

4.7.7 $s_u(z)$ AND $\sigma'_{v0}(z)$

The data plot on these graphs are retrieved from the imported dvf file in the *Analysis* mode.

4.7.8 $p(z)$ 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 REFERENCES

[1] Andresen, L., Jostad, H.P. (1999). Application of an anisotropic hardening model for undrained response of saturated clay. Proc. NUMOG VII, 581–585.

[2] Astley, R. (1992). Finite elements in solids and structures. Chapman and Hall Publishers.

[3] 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.

[4] Brinkgreve, R.B.J., Engin, E., Engin, H.K. (2010). Validation of empirical formulas to derive model parameters for sands. 7th NUMGE, Trondheim, Norway.

[5] Brinkgreve, R.B.J., Kumarswamy, S., Swolfs, W. (2018). PLAXIS 3D Material Model Manual. PLAXIS B.V., The Netherlands.

[6] Burd, H.J., Byrne, B.W., McAdam, R.A., Houlsby, G.T., Martin, C.M., Beuckelaers, W.J.A.P., Zdravković, 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.

[7] Burd, H.J., Taborda, D.M.G., Zdravković, 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. (2018). PISA Design Model for Monopiles for Offshore Wind Turbines: Application to a Marine Sand (submitted).

[8] 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., Zdravković, L. (2018a). PISA: Recent Developments in Offshore Wind Turbine Monopile Design. In Vietnam Symposium on Advances in Offshore Engineering.

[9] 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., Zdravković, L. (2018b). PISA design model for monopiles for offshore wind turbines: Application to a stiff glacial clay till (submitted).

[10] 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.

[11] Byrne, B.W., McAdam, R.A., Burd, H.J., Houlsby, G.T., Martin, C.M., Gavin, K., Doherty, P., Igoe, D., Zdravković, 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).

[12] Byrne, B.W., McAdam, R.A., Burd, H.J., Houlsby, G.T., Martin, C.M., Zdravković, 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.

[13] Jardine, R., Chow, F., Overy, R. (2005). ICP design methods for driven piles in sands and clays. Thomas Telford, London.

[14] 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.

[15] 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.

[16] Panagoulias, S., Hosseini, S., Brinkgreve, R. (2018b). An innovative design methodology for offshore wind monopile foundations. 26th European Young Geotechnical Engineers Conference, Graz, Austria.

[17] Riks, E. (1979). An incremental approach to the solution of snapping and buckling problems. International Journal of Solids and Structures, 524–551.

[18] Winkler, E. (1867). Die lehre von elasticitat und festigkeit. H. Dominicus, Prague.

[19] Zdravković, 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.

APPENDIX A - THIRD PARTY SOFTWARE LICENSES/NOTICES

comtypes

The MIT License:

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

EasyGUI

EasyGUI License:

About...: EasyGUI is a module for very simple, very easy GUI programming in Python. Author..: Stephen Raymond Ferg HomePage: http://easygui.sourceforge.net

==
Executive summary
==

This license is what is generally known as the "modified BSD license" (aka "revised BSD", "new BSD", 3-clause BSD").

This license is GPL-compatible.

This license is less restrictive than GPL. It allows software released under the license to be incorporated into proprietary products. Works based on the software may be released under a proprietary license or as closed-source software.

See: `http://www.opensource.org/licenses/bsd-license.php`
`http://en.wikipedia.org/wiki/License_compatibility`
`http://www.gnu.org/licenses/license-list.html#GPLCompatibleLicenses`
`http://en.wikipedia.org/wiki/BSD_licenses#3-clause_license_.28.22New_BSD_License.22.29`

==
LICENSE
==

EasyGui version0.96 EasyGui version0.95 Copyright (c) 2010, Stephen Raymond Ferg
All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

3. The name of the author may not be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Flask

Flask License:

Copyright ©2010 by the Pallets team.

Some rights reserved.

Redistribution and use in source and binary forms of the software as well as documentation, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of the copyright holder nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE AND DOCUMENTATION IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS ŞAS ISŤ AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS

SOFTWARE AND DOCUMENTATION, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Flask-restful

License:

Copyright (c) 2013, Twilio, Inc. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. - Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. - Neither the name of the Twilio, Inc. nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

IPython

IPython License:

Copyright (c) 2011, IPython Development Team

All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

Neither the name of the IPython Development Team nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES,

INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

IPython documentation, examples and other materials are licensed under the terms of the Attribution 4.0 International (CC BY 4.0) license, as follows:

Creative Commons Attribution 4.0 International Public License

By exercising the Licensed Rights (defined below), You accept and agree to be bound by the terms and conditions of this Creative Commons Attribution 4.0 International Public License ("Public License"). To the extent this Public License may be interpreted as a contract, You are granted the Licensed Rights in consideration of Your acceptance of these terms and conditions, and the Licensor grants You such rights in consideration of benefits the Licensor receives from making the Licensed Material available under these terms and conditions.

Section 1 – Definitions.

a. Adapted Material means material subject to Copyright and Similar Rights that is derived from or based upon the Licensed Material and in which the Licensed Material is translated, altered, arranged, transformed, or otherwise modified in a manner requiring permission under the Copyright and Similar Rights held by the Licensor. For purposes of this Public License, where the Licensed Material is a musical work, performance, or sound recording, Adapted Material is always produced where the Licensed Material is synched in timed relation with a moving image.

b. Adapter's License means the license You apply to Your Copyright and Similar Rights in Your contributions to Adapted Material in accordance with the terms and conditions of this Public License.

c. Copyright and Similar Rights means copyright and/or similar rights closely related to copyright including, without limitation, performance, broadcast, sound recording, and Sui Generis Database Rights, without regard to how the rights are labeled or categorized. For purposes of this Public License, the rights specified in Section 2(b)(1)-(2) are not Copyright and Similar Rights.

d. Effective Technological Measures means those measures that, in the absence of proper authority, may not be circumvented under laws fulfilling obligations under Article 11 of the WIPO Copyright Treaty adopted on December 20, 1996, and/or similar international agreements.

e. Exceptions and Limitations means fair use, fair dealing, and/or any other exception or limitation to Copyright and Similar Rights that applies to Your use of the Licensed Material.

f. Licensed Material means the artistic or literary work, database, or other material to which the Licensor applied this Public License.

g. Licensed Rights means the rights granted to You subject to the terms and conditions of this Public License, which are limited to all Copyright and Similar Rights that apply to Your use of the Licensed Material and that the Licensor has authority to license.

h. Licensor means the individual(s) or entity(ies) granting rights under this Public License.

i. Share means to provide material to the public by any means or process that requires permission under the Licensed Rights, such as reproduction, public display, public performance, distribution, dissemination, communication, or importation, and to make material available to the public including in ways that members of the public may access the material from a place and at a time individually chosen by them.

j. Sui Generis Database Rights means rights other than copyright resulting from Directive 96/9/EC of the European Parliament and of the Council of 11 March 1996 on the legal protection of databases, as amended and/or succeeded, as well as other essentially equivalent rights anywhere in the world.

k. You means the individual or entity exercising the Licensed Rights under this Public License. Your has a corresponding meaning.

Section 2 – Scope.

a. License grant.

1. Subject to the terms and conditions of this Public License, the Licensor hereby grants You a worldwide, royalty-free, non-sublicensable, non-exclusive, irrevocable license to exercise the Licensed Rights in the Licensed Material to:

a. reproduce and Share the Licensed Material, in whole or in part; and

b. produce, reproduce, and Share Adapted Material.

2. Exceptions and Limitations. For the avoidance of doubt, where Exceptions and Limitations apply to Your use, this Public License does not apply, and You do not need to comply with its terms and conditions.

3. Term. The term of this Public License is specified in Section 6(a).

4. Media and formats; technical modifications allowed. The Licensor authorizes You to exercise the Licensed Rights in all media and formats whether now known or hereafter created, and to make technical modifications necessary to do so. The Licensor waives and/or agrees not to assert any right or authority to forbid You from making technical modifications necessary to exercise the Licensed Rights, including technical modifications necessary to circumvent Effective Technological Measures. For purposes of this Public License, simply making modifications authorized by this Section 2(a) (4) never produces Adapted Material.

5. Downstream recipients.

a. Offer from the Licensor – Licensed Material. Every recipient of the Licensed Material automatically receives an offer from the Licensor to exercise the Licensed Rights under the terms and conditions of this Public License.

b. No downstream restrictions. You may not offer or impose any additional or different terms or conditions on, or apply any Effective Technological Measures to, the Licensed Material if doing so restricts exercise of the Licensed Rights by any recipient of the Licensed Material.

6. No endorsement. Nothing in this Public License constitutes or may be construed as permission to assert or imply that You are, or that Your use of the Licensed Material is, connected with, or sponsored, endorsed, or granted official status by, the Licensor or others designated to receive attribution as provided in Section 3(a)(1)(A)(i).

b. Other rights.

1. Moral rights, such as the right of integrity, are not licensed under this Public License, nor are publicity, privacy, and/or other similar personality rights; however, to the extent possible, the Licensor waives and/or agrees not to assert any such rights held by the Licensor to the limited extent necessary to allow You to exercise the Licensed Rights, but not otherwise.

2. Patent and trademark rights are not licensed under this Public License.

3. To the extent possible, the Licensor waives any right to collect royalties from You for the exercise of the Licensed Rights, whether directly or through a collecting society under any voluntary or waivable statutory or compulsory licensing scheme. In all other cases the Licensor expressly reserves any right to collect such royalties.

Section 3 – License Conditions.

Your exercise of the Licensed Rights is expressly made subject to the following conditions.

a. Attribution.

1. If You Share the Licensed Material (including in modified form), You must:

a. retain the following if it is supplied by the Licensor with the Licensed Material:

i. identification of the creator(s) of the Licensed Material and any others designated to receive attribution, in any reasonable manner requested by the Licensor (including by pseudonym if designated);

ii. a copyright notice;

iii. a notice that refers to this Public License;

iv. a notice that refers to the disclaimer of warranties;

v. a URI or hyperlink to the Licensed Material to the extent reasonably practicable;

b. indicate if You modified the Licensed Material and retain an indication of any previous modifications; and

c. indicate the Licensed Material is licensed under this Public License, and include the text of, or the URI or hyperlink to, this Public License.

2. You may satisfy the conditions in Section 3(a)(1) in any reasonable manner based on the medium, means, and context in which You Share the Licensed Material. For example, it may be reasonable to satisfy the conditions by providing a URI or hyperlink to a resource that includes the required information.

3. If requested by the Licensor, You must remove any of the information required by Section 3(a)(1)(A) to the extent reasonably practicable.

4. If You Share Adapted Material You produce, the Adapter's License You apply must not prevent recipients of the Adapted Material from complying with this Public License.

Section 4 – Sui Generis Database Rights.

Where the Licensed Rights include Sui Generis Database Rights that apply to Your use of the Licensed Material:

a. for the avoidance of doubt, Section 2(a)(1) grants You the right to extract, reuse, reproduce, and Share all or a substantial portion of the contents of the database;

b. if You include all or a substantial portion of the database contents in a database in which You have Sui Generis Database Rights, then the database in which You have Sui Generis Database Rights (but not its individual contents) is Adapted Material; and

c. You must comply with the conditions in Section 3(a) if You Share all or a substantial portion of the contents of the database.

For the avoidance of doubt, this Section 4 supplements and does not replace Your obligations under this Public License where the Licensed Rights include other Copyright and Similar Rights.

Section 5 – Disclaimer of Warranties and Limitation of Liability.

a. UNLESS OTHERWISE SEPARATELY UNDERTAKEN BY THE LICENSOR, TO THE EXTENT POSSIBLE, THE LICENSOR OFFERS THE LICENSED MATERIAL AS-IS AND AS-AVAILABLE, AND MAKES NO REPRESENTATIONS OR WARRANTIES OF ANY KIND CONCERNING THE LICENSED MATERIAL, WHETHER EXPRESS, IMPLIED, STATUTORY, OR OTHER. THIS INCLUDES, WITHOUT LIMITATION, WARRANTIES OF TITLE, MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, NON-INFRINGEMENT, ABSENCE OF LATENT OR OTHER DEFECTS, ACCURACY, OR THE PRESENCE OR ABSENCE OF ERRORS, WHETHER OR NOT KNOWN OR DISCOVERABLE. WHERE DISCLAIMERS OF WARRANTIES ARE NOT ALLOWED IN FULL OR IN PART, THIS DISCLAIMER MAY NOT APPLY TO YOU.

b. TO THE EXTENT POSSIBLE, IN NO EVENT WILL THE LICENSOR BE LIABLE TO YOU ON ANY LEGAL THEORY (INCLUDING, WITHOUT LIMITATION, NEGLIGENCE) OR OTHERWISE FOR ANY DIRECT, SPECIAL, INDIRECT, INCIDENTAL, CONSEQUENTIAL, PUNITIVE, EXEMPLARY, OR OTHER LOSSES, COSTS, EXPENSES, OR DAMAGES ARISING OUT OF THIS PUBLIC LICENSE OR USE OF THE LICENSED MATERIAL, EVEN IF THE LICENSOR HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH LOSSES, COSTS, EXPENSES, OR DAMAGES. WHERE A LIMITATION OF LIABILITY IS NOT ALLOWED IN FULL OR IN PART, THIS LIMITATION MAY NOT APPLY TO YOU.

c. The disclaimer of warranties and limitation of liability provided above shall be interpreted in a manner that, to the extent possible, most closely approximates an absolute disclaimer and waiver of all liability.

Section 6 – Term and Termination.

a. This Public License applies for the term of the Copyright and Similar Rights licensed here. However, if You fail to comply with this Public License, then Your rights under this Public License terminate automatically.

b. Where Your right to use the Licensed Material has terminated under Section 6(a), it reinstates:

1. automatically as of the date the violation is cured, provided it is cured within 30 days of Your discovery of the violation; or

2. upon express reinstatement by the Licensor.

For the avoidance of doubt, this Section 6(b) does not affect any right the Licensor may have to seek remedies for Your violations of this Public License.

c. For the avoidance of doubt, the Licensor may also offer the Licensed Material under separate terms or conditions or stop distributing the Licensed Material at any time; however, doing so will not terminate this Public License.

d. Sections 1, 5, 6, 7, and 8 survive termination of this Public License.

Section 7 – Other Terms and Conditions.

a. The Licensor shall not be bound by any additional or different terms or conditions communicated by You unless expressly agreed.

b. Any arrangements, understandings, or agreements regarding the Licensed Material not stated herein are separate from and independent of the terms and conditions of this Public License.

Section 8 – Interpretation.

a. For the avoidance of doubt, this Public License does not, and shall not be interpreted to, reduce, limit, restrict, or impose conditions on any use of the Licensed Material that could lawfully be made without permission under this Public License.

b. To the extent possible, if any provision of this Public License is deemed unenforceable, it shall be automatically reformed to the minimum extent necessary to make it enforceable. If the provision cannot be reformed, it shall be severed from this Public License without affecting the enforceability of the remaining terms and conditions.

c. No term or condition of this Public License will be waived and no failure to comply consented to unless expressly agreed to by the Licensor.

d. Nothing in this Public License constitutes or may be interpreted as a limitation upon, or waiver of, any privileges and immunities that apply to the Licensor or You, including from the legal processes of any jurisdiction or authority.

matplotlib

License:

Matplotlib only uses BSD compatible code, and its license is based on the PSF license. See the Open Source Initiative licenses page for details on individual licenses. Non-BSD compatible licenses (e.g., LGPL) are acceptable in matplotlib toolkits. For a discussion of the motivations behind the licencing choice, see Licenses. Copyright Policy

John Hunter began matplotlib around 2003. Since shortly before his passing in 2012, Michael Droettboom has been the lead maintainer of matplotlib, but, as has always been the case, matplotlib is the work of many.

Prior to July of 2013, and the 1.3.0 release, the copyright of the source code was held by John Hunter. As of July 2013, and the 1.3.0 release, matplotlib has moved to a shared copyright model.

matplotlib uses a shared copyright model. Each contributor maintains copyright over their contributions to matplotlib. But, it is important to note that these contributions are typically only changes to the repositories. Thus, the matplotlib source code, in its entirety, is not the copyright of any single person or institution. Instead, it is the collective copyright of the entire matplotlib Development Team. If individual contributors want to maintain a record of what changes/contributions they have specific copyright on, they should indicate their copyright in the commit message of the change, when they commit the change to one of the matplotlib repositories.

The Matplotlib Development Team is the set of all contributors to the matplotlib project. A full list can be obtained from the git version control logs. License agreement for matplotlib 2.2.2

1. This LICENSE AGREEMENT is between the Matplotlib Development Team ("MDT"), and the Individual or Organization ("Licensee") accessing and otherwise using matplotlib software in source or binary form and its associated documentation.

2. Subject to the terms and conditions of this License Agreement, MDT hereby grants Licensee a nonexclusive, royalty-free, world-wide license to reproduce, analyze, test, perform and/or display publicly, prepare derivative works, distribute, and otherwise use matplotlib 2.2.2 alone or in any derivative version, provided, however, that MDT's License Agreement and MDT's notice of copyright, i.e., "Copyright (c) 2012-2013 Matplotlib Development Team; All Rights Reserved" are retained in matplotlib 2.2.2 alone or in any derivative version prepared by Licensee.

3. In the event Licensee prepares a derivative work that is based on or incorporates matplotlib 2.2.2 or any part thereof, and wants to make the derivative work available to others as provided herein, then Licensee hereby agrees to include in any such work a brief summary of the changes made to matplotlib 2.2.2.

4. MDT is making matplotlib 2.2.2 available to Licensee on an "AS IS" basis. MDT MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED. BY WAY OF EXAMPLE, BUT NOT LIMITATION, MDT MAKES NO AND DISCLAIMS ANY REPRESENTATION OR WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF MATPLOTLIB 2.2.2 WILL NOT INFRINGE ANY THIRD PARTY RIGHTS.

5. MDT SHALL NOT BE LIABLE TO LICENSEE OR ANY OTHER USERS OF MATPLOTLIB 2.2.2 FOR ANY INCIDENTAL, SPECIAL, OR CONSEQUENTIAL DAMAGES OR LOSS AS A RESULT OF MODIFYING, DISTRIBUTING, OR OTHERWISE USING MATPLOTLIB 2.2.2, OR ANY DERIVATIVE THEREOF, EVEN IF ADVISED OF THE POSSIBILITY THEREOF.

6. This License Agreement will automatically terminate upon a material breach of its terms and conditions.

7. Nothing in this License Agreement shall be deemed to create any relationship of agency, partnership, or joint venture between MDT and Licensee. This License Agreement does not grant permission to use MDT trademarks or trade name in a trademark sense to endorse or promote products or services of Licensee, or any third party.

8. By copying, installing or otherwise using matplotlib 2.2.2, Licensee agrees to be bound by the terms and conditions of this License Agreement. License agreement for matplotlib versions prior to 1.3.0

1. This LICENSE AGREEMENT is between John D. Hunter ("JDH"), and the Individual or Organization ("Licensee") accessing and otherwise using matplotlib software in source or binary form and its associated documentation.

2. Subject to the terms and conditions of this License Agreement, JDH hereby grants Licensee a nonexclusive, royalty-free, world-wide license to reproduce, analyze, test, perform and/or display publicly, prepare derivative works, distribute, and otherwise use matplotlib 2.2.2 alone or in any derivative version, provided, however, that JDH's License Agreement and JDH's notice of copyright, i.e., "Copyright (c) 2002-2009 John D. Hunter; All Rights Reserved" are retained in matplotlib 2.2.2 alone or in any derivative version prepared by Licensee.

3. In the event Licensee prepares a derivative work that is based on or incorporates matplotlib 2.2.2 or any part thereof, and wants to make the derivative work available to others as provided herein, then Licensee hereby agrees to include in any such work a brief summary of the changes made to matplotlib 2.2.2.

4. JDH is making matplotlib 2.2.2 available to Licensee on an "AS IS" basis. JDH MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED. BY WAY OF EXAMPLE, BUT NOT LIMITATION, JDH MAKES NO AND DISCLAIMS ANY REPRESENTATION OR WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF MATPLOTLIB 2.2.2 WILL NOT INFRINGE ANY THIRD PARTY RIGHTS.

5. JDH SHALL NOT BE LIABLE TO LICENSEE OR ANY OTHER USERS OF MATPLOTLIB 2.2.2 FOR ANY INCIDENTAL, SPECIAL, OR CONSEQUENTIAL DAMAGES OR LOSS AS A RESULT OF MODIFYING, DISTRIBUTING, OR OTHERWISE USING MATPLOTLIB 2.2.2, OR ANY DERIVATIVE THEREOF, EVEN IF ADVISED OF THE POSSIBILITY THEREOF.

6. This License Agreement will automatically terminate upon a material breach of its terms and conditions.

7. Nothing in this License Agreement shall be deemed to create any relationship of agency, partnership, or joint venture between JDH and Licensee. This License Agreement does not grant permission to use JDH trademarks or trade name in a trademark sense to endorse or promote products or services of Licensee, or any third party.

8. By copying, installing or otherwise using matplotlib 2.2.2, Licensee agrees to be bound by the terms and conditions of this License Agreement.

Numba

License:

Copyright (c) 2012, Anaconda, Inc. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF

MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

NumPy

NumPy license:

Copyright ©2005-2018, NumPy Developers. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of the NumPy Developers nor the names of any contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Pillow

License:

The Python Imaging Library (PIL) is

Copyright © 1997-2011 by Secret Labs AB Copyright © 1995-2011 by Fredrik Lundh

Pillow is the friendly PIL fork. It is

Copyright © 2010-2018 by Alex Clark and contributors

Like PIL, Pillow is licensed under the open source PIL Software License:

By obtaining, using, and/or copying this software and/or its associated documentation, you agree that you have read, understood, and will comply with the following terms and conditions:

Permission to use, copy, modify, and distribute this software and its associated documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appears in all copies, and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of Secret Labs AB or the author not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission.

SECRET LABS AB AND THE AUTHOR DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS. IN NO EVENT SHALL SECRET LABS AB OR THE AUTHOR BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

psutil

License:

psutil is distributed under BSD license reproduced below.

Copyright (c) 2009, Jay Loden, Dave Daeschler, Giampaolo Rodola' All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

* Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. * Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. * Neither the name of the psutil authors nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

PyCXX

Copyright:

Copyright (c) 1998 - 2007 The Regents of the University of California Produced at the Lawrence Livermore National Laboratory Written by Geoff Furnish, Paul F. Dubois, Barry A. Scott UCRL-CODE-227018 All rights reserved.

This file is part of PyCXX. For details, see http://cxx.sourceforge.net.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the disclaimer below. - Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the disclaimer (as noted below) in the documentation and/or materials provided with the distribution. - Neither the name of the UC/LLNL nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OF THE UNIVERSITY OF CALIFORNIA, THE U.S. DEPARTMENT OF ENERGY OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Additional BSD Notice

1. This notice is required to be provided under our contract with the U.S. Department of Energy (DOE). This work was produced at the University of California, Lawrence Livermore National Laboratory under Contract No. W-7405-ENG-48 with the DOE.

2. Neither the United States Government nor the University of California nor any of their employees, makes any warranty, express or implied, or assumes any liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately-owned rights.

3. Also, reference herein to any specific commercial products, process, or services by trade name, trademark, manufacturer or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or the University of California, and shall not be used for advertising or product endorsement purposes.

PyInstaller

License:

PyInstaller is distributed under the GPL License but with an exception that allows you to use it to build commercial products:

1. You may use PyInstaller to bundle commercial applications out of your source code. 2. The executable bundles generated by PyInstaller from your source code can be shipped with whatever license you want. 3. You may modify PyInstaller for your own needs but

changes to the PyInstaller source code fall under the terms of the GPL license. That is, if you distribute your modifications you must distribute them under GPL terms.

pyperclip

The 3-Clause BSD License:

Note: This license has also been called the "New BSD License" or "Modified BSD License". See also the 2-clause BSD License.

Copyright <YEAR> <COPYRIGHT HOLDER>

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

3. Neither the name of the copyright holder nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

PyQtGraph

License:

Copyright (c) 2012 University of North Carolina at Chapel Hill Luke Campagnola ('luke.campagnola@%s.com' % 'gmail')

The MIT License Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND,

EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

PySide

License: GNU Library or Lesser General Public License (LGPL):

Copyright © 2007 Free Software Foundation, Inc. <https://fsf.org/>

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

This version of the GNU Lesser General Public License incorporates the terms and conditions of version 3 of the GNU General Public License, supplemented by the additional permissions listed below. 0. Additional Definitions.

As used herein, "this License" refers to version 3 of the GNU Lesser General Public License, and the "GNU GPL" refers to version 3 of the GNU General Public License.

"The Library" refers to a covered work governed by this License, other than an Application or a Combined Work as defined below.

An "Application" is any work that makes use of an interface provided by the Library, but which is not otherwise based on the Library. Defining a subclass of a class defined by the Library is deemed a mode of using an interface provided by the Library.

A "Combined Work" is a work produced by combining or linking an Application with the Library. The particular version of the Library with which the Combined Work was made is also called the "Linked Version".

The "Minimal Corresponding Source" for a Combined Work means the Corresponding Source for the Combined Work, excluding any source code for portions of the Combined Work that, considered in isolation, are based on the Application, and not on the Linked Version.

The "Corresponding Application Code" for a Combined Work means the object code and/or source code for the Application, including any data and utility programs needed for reproducing the Combined Work from the Application, but excluding the System Libraries of the Combined Work. 1. Exception to Section 3 of the GNU GPL.

You may convey a covered work under sections 3 and 4 of this License without being bound by section 3 of the GNU GPL. 2. Conveying Modified Versions.

If you modify a copy of the Library, and, in your modifications, a facility refers to a function or data to be supplied by an Application that uses the facility (other than as an argument passed when the facility is invoked), then you may convey a copy of the modified version:

a) under this License, provided that you make a good faith effort to ensure that, in the event an Application does not supply the function or data, the facility still operates, and performs whatever part of its purpose remains meaningful, or b) under the GNU GPL, with none of the additional permissions of this License applicable to that copy.

3. Object Code Incorporating Material from Library Header Files.

The object code form of an Application may incorporate material from a header file that is part of the Library. You may convey such object code under terms of your choice, provided that, if the incorporated material is not limited to numerical parameters, data structure layouts and accessors, or small macros, inline functions and templates (ten or fewer lines in length), you do both of the following:

a) Give prominent notice with each copy of the object code that the Library is used in it and that the Library and its use are covered by this License. b) Accompany the object code with a copy of the GNU GPL and this license document.

4. Combined Works.

You may convey a Combined Work under terms of your choice that, taken together, effectively do not restrict modification of the portions of the Library contained in the Combined Work and reverse engineering for debugging such modifications, if you also do each of the following:

a) Give prominent notice with each copy of the Combined Work that the Library is used in it and that the Library and its use are covered by this License. b) Accompany the Combined Work with a copy of the GNU GPL and this license document. c) For a Combined Work that displays copyright notices during execution, include the copyright notice for the Library among these notices, as well as a reference directing the user to the copies of the GNU GPL and this license document. d) Do one of the following: 0) Convey the Minimal Corresponding Source under the terms of this License, and the Corresponding Application Code in a form suitable for, and under terms that permit, the user to recombine or relink the Application with a modified version of the Linked Version to produce a modified Combined Work, in the manner specified by section 6 of the GNU GPL for conveying Corresponding Source. 1) Use a suitable shared library mechanism for linking with the Library. A suitable mechanism is one that (a) uses at run time a copy of the Library already present on the user's computer system, and (b) will operate properly with a modified version of the Library that is interface-compatible with the Linked Version. e) Provide Installation Information, but only if you would otherwise be required to provide such information under section 6 of the GNU GPL, and only to the extent that such information is necessary to install and execute a modified version of the Combined Work produced by recombining or relinking the Application with a modified version of the Linked Version. (If you use option 4d0, the Installation Information must accompany the Minimal Corresponding Source and Corresponding Application Code. If you use option 4d1, you must provide the Installation Information in the manner specified by section 6 of the GNU GPL for conveying Corresponding Source.)

5. Combined Libraries.

You may place library facilities that are a work based on the Library side by side in a single library together with other library facilities that are not Applications and are not covered by this License, and convey such a combined library under terms of your choice, if you do both of the following:

a) Accompany the combined library with a copy of the same work based on the Library, uncombined with any other library facilities, conveyed under the terms of this License. b) Give prominent notice with the combined library that part of it is a work based on the Library, and explaining where to find the accompanying uncombined form of the same work.

6. Revised Versions of the GNU Lesser General Public License.

The Free Software Foundation may publish revised and/or new versions of the GNU Lesser General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Library as you received it specifies that a certain numbered version of the GNU Lesser General Public License "or any later version" applies to it, you have the option of following the terms and conditions either of that published version or of any later version published by the Free Software Foundation. If the Library as you received it does not specify a version number of the GNU Lesser General Public License, you may choose any version of the GNU Lesser General Public License ever published by the Free Software Foundation.

If the Library as you received it specifies that a proxy can decide whether future versions of the GNU Lesser General Public License shall apply, that proxy's public statement of acceptance of any version is permanent authorization for you to choose that version for the Library.

Python 3.4

Python 3.4.0 license:

A. HISTORY OF THE SOFTWARE ==========================

Python was created in the early 1990s by Guido van Rossum at Stichting Mathematisch Centrum (CWI, see http://www.cwi.nl) in the Netherlands as a successor of a language called ABC. Guido remains Python's principal author, although it includes many contributions from others.

In 1995, Guido continued his work on Python at the Corporation for National Research Initiatives (CNRI, see http://www.cnri.reston.va.us) in Reston, Virginia where he released several versions of the software.

In May 2000, Guido and the Python core development team moved to BeOpen.com to form the BeOpen PythonLabs team. In October of the same year, the PythonLabs team moved to Digital Creations (now Zope Corporation, see http://www.zope.com). In 2001, the Python Software Foundation (PSF, see http://www.python.org/psf/) was formed, a non-profit organization created specifically to own Python-related Intellectual Property. Zope Corporation is a sponsoring member of the PSF.

All Python releases are Open Source (see http://www.opensource.org for the Open Source Definition). Historically, most, but not all, Python releases have also been GPL-compatible; the table below summarizes the various releases.

Release Derived Year Owner GPL- from compatible? (1)

0.9.0 thru 1.2 1991-1995 CWI yes
1.3 thru 1.5.2 1.2 1995-1999 CNRI yes
1.6 1.5.2 2000 CNRI no
2.0 1.6 2000 BeOpen.com no1.6.1 1.6 2001 CNRI yes (2)2.1 2.0+1.6.1 2001 PSF no2.0.1 2.0+1.6.1 2001 PSF yes2.1.1 2.1+2.0.1 2001 PSF yes2.2 2.1.1 2001 PSF yes2.1.2 2.1.1 2002 PSF yes2.1.3 2.1.2 2002 PSF yes2.2.1 2.2 2002 PSF yes2.2.2 2.2.1 2002 PSF yes2.2.3 2.2.2 2003 PSF yes2.3 2.2.2 2002-2003 PSF yes2.3.1 2.3 2002-2003 PSF yes2.3.2 2.3.1 2002-2003 PSF yes2.3.3 2.3.2 2002-2003 PSF yes2.3.4 2.3.3 2004

PSF yes2.3.5 2.3.4 2005 PSF yes2.4 2.3 2004 PSF yes2.4.1 2.4 2005 PSF yes2.4.2 2.4.1 2005 PSF yes2.4.3 2.4.2 2006 PSF yes2.4.4 2.4.3 2006 PSF yes2.5 2.4 2006 PSF yes2.5.1 2.5 2007 PSF yes2.5.2 2.5.1 2008 PSF yes2.5.3 2.5.2 2008 PSF yes2.6 2.5 2008 PSF yes2.6.1 2.6 2008 PSF yes2.6.2 2.6.1 2009 PSF yes2.6.3 2.6.2 2009 PSF yes2.6.4 2.6.3 2009 PSF yes2.6.5 2.6.4 2010 PSF yes3.0 2.6 2008 PSF yes3.0.1 3.0 2009 PSF yes3.1 3.0.1 2009 PSF yes3.1.1 3.1 2009 PSF yes3.1.2 3.1.1 2010 PSF yes3.1.3 3.1.2 2010 PSF yes3.1.4 3.1.3 2011 PSF yes3.1.5 3.1.4 2012 PSF yes3.2.1 3.2 2011 PSF yes3.2.2 3.2.1 2011 PSF yes3.2.3 3.2.2 2012 PSF yes3.3.0 3.2 2012 PSF yes3.3.1 3.3.0 2013 PSF yes3.3.2 3.3.1 2013 PSF yes3.4.0 3.3 2013 PSF yes

Footnotes:

(1) GPL-compatible doesn't mean that we're distributing Python under the GPL. All Python licenses, unlike the GPL, let you distribute a modified version without making your changes open source. The GPL-compatible licenses make it possible to combine Python with other software that is released under the GPL; the others don't.

(2) According to Richard Stallman, 1.6.1 is not GPL-compatible, because its license has a choice of law clause. According to CNRI, however, Stallman's lawyer has told CNRI's lawyer that 1.6.1 is "not incompatible" with the GPL.

Thanks to the many outside volunteers who have worked under Guido's direction to make these releases possible.

B. TERMS AND CONDITIONS FOR ACCESSING OR OTHERWISE USING PYTHON
===

PYTHON SOFTWARE FOUNDATION LICENSE VERSION 2

1. This LICENSE AGREEMENT is between the Python Software Foundation ("PSF"), and the Individual or Organization ("Licensee") accessing and otherwise using this software ("Python") in source or binary form and its associated documentation.

2. Subject to the terms and conditions of this License Agreement, PSF hereby grants Licensee a nonexclusive, royalty-free, world-wide license to reproduce, analyze, test, perform and/or display publicly, prepare derivative works, distribute, and otherwise use Python alone or in any derivative version, provided, however, that PSF's License Agreement and PSF's notice of copyright, i.e., "Copyright (c) 2001, 2002, 2003, 2004, 2005, 2006, 2007, 2008, 2009, 2010, 2011, 2012 Python Software Foundation; All Rights Reserved" are retained in Python alone or in any derivative version prepared by Licensee.

3. In the event Licensee prepares a derivative work that is based on or incorporates Python or any part thereof, and wants to make the derivative work available to others as provided herein, then Licensee hereby agrees to include in any such work a brief summary of the changes made to Python.

4. PSF is making Python available to Licensee on an "AS IS" basis. PSF MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED. BY WAY OF EXAMPLE, BUT NOT LIMITATION, PSF MAKES NO AND DISCLAIMS ANY REPRESENTATION OR WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF PYTHON WILL NOT INFRINGE ANY THIRD PARTY RIGHTS.

5. PSF SHALL NOT BE LIABLE TO LICENSEE OR ANY OTHER USERS OF PYTHON

FOR ANY INCIDENTAL, SPECIAL, OR CONSEQUENTIAL DAMAGES OR LOSS AS A RESULT OF MODIFYING, DISTRIBUTING, OR OTHERWISE USING PYTHON, OR ANY DERIVATIVE THEREOF, EVEN IF ADVISED OF THE POSSIBILITY THEREOF.

6. This License Agreement will automatically terminate upon a material breach of its terms and conditions.

7. Nothing in this License Agreement shall be deemed to create any relationship of agency, partnership, or joint venture between PSF and Licensee. This License Agreement does not grant permission to use PSF trademarks or trade name in a trademark sense to endorse or promote products or services of Licensee, or any third party.

8. By copying, installing or otherwise using Python, Licensee agrees to be bound by the terms and conditions of this License Agreement.

BEOPEN.COM LICENSE AGREEMENT FOR PYTHON 2.0

——————————————-

BEOPEN PYTHON OPEN SOURCE LICENSE AGREEMENT VERSION 1

1. This LICENSE AGREEMENT is between BeOpen.com ("BeOpen"), having an office at 160 Saratoga Avenue, Santa Clara, CA 95051, and the Individual or Organization ("Licensee") accessing and otherwise using this software in source or binary form and its associated documentation ("the Software").

2. Subject to the terms and conditions of this BeOpen Python License Agreement, BeOpen hereby grants Licensee a non-exclusive, royalty-free, world-wide license to reproduce, analyze, test, perform and/or display publicly, prepare derivative works, distribute, and otherwise use the Software alone or in any derivative version, provided, however, that the BeOpen Python License is retained in the Software, alone or in any derivative version prepared by Licensee.

3. BeOpen is making the Software available to Licensee on an "AS IS" basis. BEOPEN MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED. BY WAY OF EXAMPLE, BUT NOT LIMITATION, BEOPEN MAKES NO AND DISCLAIMS ANY REPRESENTATION OR WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF THE SOFTWARE WILL NOT INFRINGE ANY THIRD PARTY RIGHTS.

4. BEOPEN SHALL NOT BE LIABLE TO LICENSEE OR ANY OTHER USERS OF THE SOFTWARE FOR ANY INCIDENTAL, SPECIAL, OR CONSEQUENTIAL DAMAGES OR LOSS AS A RESULT OF USING, MODIFYING OR DISTRIBUTING THE SOFTWARE, OR ANY DERIVATIVE THEREOF, EVEN IF ADVISED OF THE POSSIBILITY THEREOF.

5. This License Agreement will automatically terminate upon a material breach of its terms and conditions.

6. This License Agreement shall be governed by and interpreted in all respects by the law of the State of California, excluding conflict of law provisions. Nothing in this License Agreement shall be deemed to create any relationship of agency, partnership, or joint venture between BeOpen and Licensee. This License Agreement does not grant permission to use BeOpen trademarks or trade names in a trademark sense to endorse or promote products or services of Licensee, or any third party. As an exception, the "BeOpen Python" logos available at http://www.pythonlabs.com/logos.html may be used

according to the permissions granted on that web page.

7. By copying, installing or otherwise using the software, Licensee agrees to be bound by the terms and conditions of this License Agreement.

CNRI LICENSE AGREEMENT FOR PYTHON 1.6.1 ————————————————

1. This LICENSE AGREEMENT is between the Corporation for National Research Initiatives, having an office at 1895 Preston White Drive, Reston, VA 20191 ("CNRI"), and the Individual or Organization ("Licensee") accessing and otherwise using Python 1.6.1 software in source or binary form and its associated documentation.

2. Subject to the terms and conditions of this License Agreement, CNRI hereby grants Licensee a nonexclusive, royalty-free, world-wide license to reproduce, analyze, test, perform and/or display publicly, prepare derivative works, distribute, and otherwise use Python 1.6.1 alone or in any derivative version, provided, however, that CNRI's License Agreement and CNRI's notice of copyright, i.e., "Copyright (c) 1995-2001 Corporation for National Research Initiatives; All Rights Reserved" are retained in Python 1.6.1 alone or in any derivative version prepared by Licensee. Alternately, in lieu of CNRI's License Agreement, Licensee may substitute the following text (omitting the quotes): "Python 1.6.1 is made available subject to the terms and conditions in CNRI's License Agreement. This Agreement together with Python 1.6.1 may be located on the Internet using the following unique, persistent identifier (known as a handle): 1895.22/1013. This Agreement may also be obtained from a proxy server on the Internet using the following URL: http://hdl.handle.net/1895.22/1013".

3. In the event Licensee prepares a derivative work that is based on or incorporates Python 1.6.1 or any part thereof, and wants to make the derivative work available to others as provided herein, then Licensee hereby agrees to include in any such work a brief summary of the changes made to Python 1.6.1.

4. CNRI is making Python 1.6.1 available to Licensee on an "AS IS" basis. CNRI MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED. BY WAY OF EXAMPLE, BUT NOT LIMITATION, CNRI MAKES NO AND DISCLAIMS ANY REPRESENTATION OR WARRANTY OF MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE OR THAT THE USE OF PYTHON 1.6.1 WILL NOT INFRINGE ANY THIRD PARTY RIGHTS.

5. CNRI SHALL NOT BE LIABLE TO LICENSEE OR ANY OTHER USERS OF PYTHON 1.6.1 FOR ANY INCIDENTAL, SPECIAL, OR CONSEQUENTIAL DAMAGES OR LOSS AS A RESULT OF MODIFYING, DISTRIBUTING, OR OTHERWISE USING PYTHON 1.6.1, OR ANY DERIVATIVE THEREOF, EVEN IF ADVISED OF THE POSSIBILITY THEREOF.

6. This License Agreement will automatically terminate upon a material breach of its terms and conditions.

7. This License Agreement shall be governed by the federal intellectual property law of the United States, including without limitation the federal copyright law, and, to the extent such U.S. federal law does not apply, by the law of the Commonwealth of Virginia, excluding Virginia's conflict of law provisions. Notwithstanding the foregoing, with regard to derivative works based on Python 1.6.1 that incorporate non-separable material that was previously distributed under the GNU General Public License (GPL), the law of the Commonwealth of Virginia shall govern this License Agreement only as to issues arising

under or with respect to Paragraphs 4, 5, and 7 of this License Agreement. Nothing in this License Agreement shall be deemed to create any relationship of agency, partnership, or joint venture between CNRI and Licensee. This License Agreement does not grant permission to use CNRI trademarks or trade name in a trademark sense to endorse or promote products or services of Licensee, or any third party.

8. By clicking on the "ACCEPT" button where indicated, or by copying, installing or otherwise using Python 1.6.1, Licensee agrees to be bound by the terms and conditions of this License Agreement.

ACCEPT

CWI LICENSE AGREEMENT FOR PYTHON 0.9.0 THROUGH 1.2

Copyright (c) 1991 - 1995, Stichting Mathematisch Centrum Amsterdam, The Netherlands. All rights reserved.

Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation, and that the name of Stichting Mathematisch Centrum or CWI not be used in advertising or publicity pertaining to distribution of the software without specific, written prior permission.

STICHTING MATHEMATISCH CENTRUM DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS, IN NO EVENT SHALL STICHTING MATHEMATISCH CENTRUM BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

SciPy

SciPy license:

Copyright © 2001, 2002 Enthought, Inc. All rights reserved.

Copyright © 2003-2013 SciPy Developers. All rights reserved.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of Enthought nor the names of the SciPy Developers may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE

DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

SciTE

License for Scintilla and SciTE:

Copyright 1998-2002 by Neil Hodgson <neilh@scintilla.org>

All Rights Reserved

Permission to use, copy, modify, and distribute this software and its documentation for any purpose and without fee is hereby granted, provided that the above copyright notice appear in all copies and that both that copyright notice and this permission notice appear in supporting documentation.

NEIL HODGSON DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE, INCLUDING ALL IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS, IN NO EVENT SHALL NEIL HODGSON BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE USE OR PERFORMANCE OF THIS SOFTWARE.

APPENDIX B - WARNINGS AND ERRORS

B.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.

B.1.1 GENERATE

Table B.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* mode	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 *Soil* mode. Please correct them to continue
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* mode	Error	There are no soil layers present in the *Soil* 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)	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* mode	Warning	The recommended maximum pile length is 70% of the soil depth, specified in the *Soil* mode. In the following models, this value is exceeded. Results may be incorrect
The prescribed displacement $v_{max,z=h}$ equals zero	Warning	The value assigned to the parameter $v_{max,z=h}$ is zero. Assign a higher value for valid calculation results
A high value is assigned to the prescribed displacement $v_{max,z=h}$, which exceeds the recommended value. Refer to Section 4.3 for more information.	Warning	A value lower than <x> m (GeoDS_1), <x> m (GeoDS_2), etc. is suggested to be used for the parameter $v_{max,z=h}$. The currently assigned value may result in excessive lateral displacement at ground level. This may lead to a long computation time
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 <x> m (GeoDS_1), <x> m (GeoDS_2), etc. is suggested to be used for the parameter $v_{max,z=h}$. The currently assigned value may result in insufficient lateral displacement at ground level. See Section 4.3 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 B.2: Checks and feedback for model generation for Clay

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 B.3: Checks and feedback for model generation for Sand

φ' = 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
$G_0^{ref} > 128000$ 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
$1 < G_0^{ref}/G_{ur}^{ref} < 20$ where $G_{ur}^{ref} = E_{ur}^{ref}/(2 \cdot (1 + \nu_{ur}))$	Error	The value of the small strain shear modulus is outside the allowable range. Please use a different value
G_0^{ref} or ϕ' too low	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 ϕ' too high	Error	The value of the small strain shear modulus or of the friction angle is too high. Please use lower values

B.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 B.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, ν)	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

B.1.3 CALCULATE

Table B.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

B.1.4 PARAMETERISE

Before parameterisation begins, PLAXIS MoDeTo might give the following error and warning if the corresponding condition is met:

Table B.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 B.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
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

B.2 ANALYSIS MODE

The 1D FE model gives these following errors, warnings and success messages in the *Analysis* mode: (Table B.8):

Table B.8: 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
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