MSettle Version 8
EMBANKMENT DESIGN AND SOIL SETTLEMENT PREDICTION

MSettle

Version 8

Embankment Design and Soil Settlement Prediction

Deltares

Delft GeoSystems

MSettle Version 8.2

Embankment Design and Soil Settlement Prediction

Edited by:

M.A.T. Visschedijk, Deltares, the Netherlands

V. Trompille, Deltares, the Netherlands

With the co-operation of:

H. Best

E.J. den Haan

J.B. Sellmeijer

E. van Zantvoort

Deltares

Enabling Delta Life

Deltares, Delft, the Netherlands, 2009
Trademark
Copyright MSettle Version 8:
Deltares, Rotterdamseweg 185, 2629 HD Delft, Netherlands
E-mail: info@deltares.nl; Internet site: http://www.deltares.nl

ISBN/EAN: 978-90-810136-4-2

Photo's by: BeeldbankVenW.nl, Rijkswaterstaat

MSettle Version 8

EMBANKMENT DESIGN AND SOIL SETTLEMENT PREDICTION

MSettle

Version 8

Introduction

Tutorial

Reference

Background

Verification

Embankment Design
and Soil Settlement
Prediction

Deltares

Delft GeoSystems

1 General Information

1.1 Foreword

This is the user manual for MSettle, which is being developed by Delft GeoSystems, a Deltares company.
MSettle is a dedicated tool for predicting soil settlements by external loading. MSettle accurately and quickly determines the direct settlement, consolidation and creep along verticals in two-dimensional geometry. GeoDelft has been developing MSettle since 1992. Sponsorship from the Dutch Ministry of Transport, Public Works and Water Management (Rijkswaterstaat) and Senter/EZ (the latter through Delft Cluster projects and the GeoSafe project) has been vital for most model development and validation.

Easy and efficient
MSettle has proved itself to be a powerful tool in the everyday engineering practice of making settlement calculations. MSettle's graphical user interface allows both frequent and infrequent MSettle users to analyze regular settlement problems extremely quickly.

Complete functionality
MSettle provides a complete functionality for determining settlements for regular two-dimensional problems. Well-established and advanced models can be used to calculate primary settlement/swelling, consolidation and secondary creep, with possible influence of vertical drains. Different kinds of external loads can be applied: non-uniform, trapezoidal, circular, rectangular, uniform and water loads. Vertical drains (strips and planes) with optionally enforced consolidation by temporary dewatering or vacuum consolidation can be modelled. MSettle creates a comprehensive tabular and graphical output with settlements, stresses and pore pressures at the verticals that have to be defined. An automatic fit on measured settlements can be applied, in order to determine improved estimates of the final

settlement. Finally, the bandwidth and parameter sensitivity for total and residual settlements can be determined, including the effect of measurements.

Product integration
MSettle is an integrated component of the M-Series. Therefore, MSettle's soil parameters can be directly determined from test results by using MCompress. Furthermore relevant data can be exchanged with MGeobase (central project database) and MStab (stability analysis). MGeobase is used to create and maintain a central project database, containing data on the measurements, geometry and soil properties of several cross-sections. MGeobase can also be used to execute series of MSettle analyses along a location line. Besides the exchange of input data, MSettle can also export the settled geometry and excess pore pressures to MStab for stability analysis.

1.2 Features in standard module

MSettle was developed especially for geotechnical engineers. MSettle's graphical interactive interface requires just a short training period for novice users. This means that you can focus your skills directly on the input of sound geotechnical data and on the subsequent settlement calculation.

1.2.1 Soil profile

- *Multiple layers*
  The two-dimensional soil structure can be composed of several soil layers with an arbitrary shape and orientation. Each layer is connected to a particular soil type.
- *Verticals*
  By placing verticals in the geometry, you can define the co-ordinates for which output results will be shown. The position of the z co-ordinate is only relevant for circular or rectangular loads.
- *Soil properties*
  The well-established constitutive models are based on common soil parameters for virgin compression, unloading/reloading and secondary creep. Parameters of the different models can also be determined directly from the results of oedometer tests, using the MCompress program. Consolidation is either modelled by means of a consolidation coefficient or by means of permeability per layer.

1.2.2 Loads

Subsequent loads at different times can be applied. Initial loads will not cause consolidation or secondary creep. Stress distribution is taken into account, also in the soil weight loads.

- *Soil weight loads*
  Soil weight loads with uniform, trapezoidal and non-uniform shape of the soil

cross-section can be applied. MSettle can include an additional, deformation dependent load. This load is equal to the soil that must be added to maintain the defined top surface position. MSettle can take account of the settlement-dependent weight reduction by submerging. Embankment construction loading can be generated from simplified input, or from imported measured surface positions.

- *Distributed loads*
  Distributed loads with a circular or rectangular base can be applied.
- *Water loads*
  Changes in pore pressure distributions at different times can be defined.

1.2.3 Models

There are three constitutive models available in MSettle: NEN-Bjerrum, NEN-Koppejan and Isotache.

- *NEN-Bjerrum Cr/Cc/Ca*
  The NEN-Bjerrum model supports today's international de-facto standard for settlement predictions, as contained for example in the Dutch NEN standard [Lit 8]. The model uses common linear strain soil parameters (C_c, C_r, C_α). Linear strains are referred to the undeformed state, presuming that strains are sufficiently small. The theoretical basis of the underlying creep rate description is the isotache model, and often associated with the name Bjerrum [Lit 1].
- *Isotache a/b/c*
  The Isotache a/b/c model by Den Haan [Lit 7] enhances the NEN-Bjerrum model by using a so-called natural strain, which is referred to the deformed state. Usage of natural strain is expected to yield more realistic settlement curves in cases with large strains. The special natural strain parameters are furthermore more objective with respect to the stress and strain level.
- *NEN-Koppejan*
  Compared to the NEN-Bjerrum model, the traditional NEN-Koppejan model assumes an instantaneous contribution by primary settlement and is not capable of describing unloading/reloading behaviour. Furthermore, NEN-Koppejan uses different parameter definitions and assumes that secondary settlement is stress-dependent. The user can opt for a linear or natural strain assumption.

All three constitutive models can be combined with the Terzaghi or Darcy consolidation model. Both consolidation models are suited for all modern drainage systems. They support different types of vertical drains (strips, columns and screens), with optional enforced dewatering. For both models the influence of consolidation can be combined with user-defined piezometric levels defining the hydraulic field, optionally layer by layer and time-dependent.

- *Darcy*
  Darcy's general storage equation can be used for accurate determination of the influence of excess pore pressures on settlements of combined soil layers. The Darcy method calculates the excess pore pressure distributions at different time

points and derives the deformation during consolidation from the development of the true effective stress. The Darcy model in combination with the isotache models also allows for modelling the gradual decrease of effective weight during submerging of loading and layers.

- *Terzaghi*
  Terzaghi's one-dimensional theoretical solution for consolidation of elastic soil can be used to modify the drained settlement solution, in order to approximate the influence of excess pore pressure generation [Lit 3]. The combination with vertical drains can be considered as an extension to the Terzaghi-Barron-Carillo method [Lit 4], [Lit 5].

1.2.4 Results

- Following the analysis, MSettle can display results in tabular and graphical form.
- The tabular report contains an echo of the input data and both settlements and stresses per vertical.
- Settlements and stress components can be viewed graphically in time and along depth.
- A dissipation design graph can be viewed, showing the degree of consolidation by uniform loading for each layer.
- The settled geometry can be viewed or written to a geometry file.
- Finally, the settled geometry and excess pore pressures for a stability analysis with the MStab program can also be written.

1.3 Features in additional modules

1.3.1 Fits on settlement plate measurements

Measured settlements can be imported and used by MSettle to perform fits by automatic scaling of material parameters. This feature enables a more accurate estimate of the final and residual settlement.

1.3.2 Reliability analysis

A reliability analysis is available to determine the bandwidth and parameter sensitivity for total and residual settlements, including the increased reliability after a preliminary settlement plate fit.

1.3.3 Horizontal displacements

Horizontal displacements can be calculated according to De Leeuw tables [Lit 24]

1.4 History

MSettle has been developed by Deltares/GeoDelft. **Version 1.0** was first released in 1992 under the name of MZet. A simplified NEN-Bjerrum calculation method with limited applicability was added in 1993. Some new features, such as the option to save a settled geometry, were added in 1994. In 1995, the Koppejan method was adapted to allow loads to be added at different points in time. **Version 4.0** (1998) was the first Windows version of MZet. Its name was then changed to MSettle. In 1999 a first version of the a/b/c Isotache model was incorporated into MSettle **Version 5.0**.

Version 6.0 (2001) included an enhanced module for geometrical modelling, and improved versions of the user manual and on-line Help have been released.

Version 6.7 (2002) was the first modular release of MSettle, meaning that different modules can be purchased separately. The 6.7 version included separate 1D and 2D modules, simplified input of embankment construction by load generation, several improvements to the isotache model and its documentation, a choice between the Terzaghi and Darcy consolidation models, vertical drains (only for the Darcy model), and user-controlled variation of soil parameters in order to fit settlement plate curves.

Version 6.8 (2003) included a completely new formulation of the NEN-Bjerrum model and an enhanced report format. The new NEN-Bjerrum model still uses the common soil parameters C_c, C_r, C_α, but is now based on the same isotache formulation as the a/b/c/ model. The new formulation is therefore also suited for loading stages and un-/reloading sequences, which were not possible with the old formulation.

Version 7.1 (2004) featured the new combination of vertical drains with the Terzaghi consolidation model, coupled stability analysis with MStab and a new design graph for the degree of consolidation. Furthermore the chart data behind all graphs had been made available, for usage in spread sheets et cetera.

Version 7.3 (2006) offers an automatic settlement plate fit. It also includes the new reliability module. Furthermore, input of temporary loading has been simplified, the plot of transient settlements has been extended with a plot of the loading and the *Material* window has been redesigned.

- The settlement plate fit is now part of the *Calculation* menu [§ 10.3]. The usage of the manual fit has been simplified, and a robust automatic fit has been added. The *Use Fit parameters* option [§ 10.4] is available to generate modified results from a complete settlement analysis. Reading of measurement data is now also supported from files with tab delimited format (TXT), or comma (;) delimited format (CSV).
- An evaluation version of the *Reliability* module has been added [§ 10.4.2]. This module offers different methods to determine the bandwidth and the parameter

sensitivity, for the total settlement and the residual settlement. The initial bandwidth follows from the assumed standard deviation of the parameters. MSettle derives this uncertainty measure from defaults or from user input [§ 9.1.2]. A preceding settlement plate fit will affect the parameter uncertainty, and therefore the bandwidth of the predicted settlements.

- A graph of loading versus time has been attached to the graph of settlement versus time [§ 11.5, § 11.5.2].
- Input of temporary loading has been simplified by the introduction of an end time for non-uniform loading [§ 9.6.1].
- A graph of residual settlements versus different start times has been made available [§ 11.7].
- The *Material* window [§ 9.2] was redesigned, in order to separate the parameters for the soil model from the parameters for the consolidation model. An *equivalent age* indication of over-consolidation was added to the NEN-Bjerrum and Isotache models.

Version 8.2 was released in 2009. This version includes the following improvements and new features:

- The Darcy consolidation model has been strongly improved and is now the default consolidation model:
  - It is more accurate than the Terzaghi model;
  - It uses the same input as the Terzaghi model. This means that Darcy is now based on excess pore pressures instead of total pore pressures, and that direct input of the consolidation coefficient is allowed.
  - It consumes considerable less computation time than in the previous version, and features a significantly increased robustness. The latter means that previous numerical problems by spatial oscillations and by negative effective stresses are practically vanished.
  - Deformation of drained layers is now included.
  - *Submerging* modelling has been improved in combination with the Isotache and NEN-Bjerrum models: the effective weight of both non-uniform loads and soil layers changes gradually during submerging, by taken into account the actual settlement instead of the final settlement.

    See [§ 1.5.1] for a comparison between the new Darcy model and the Terzaghi model.
- Optional direct input of the *Preconsolidation pressure* in the *Material* window is available for the Isotache and NEN-Bjerrum models [§ 9.2.4, 9.2.5], in order to model special cases where a definition via POP or OCR is not sufficient.
- Vertical drains can be limited to a certain horizontal range. Furthermore the input has been simplified, both by introducing dedicated input for different drain types (strips, columns, sand screens) and dewatering methods and by supplying common defaults for applicable input parameters. [§ 9.4.2].
- The system for error messages and warnings has been improved, as well as the messages themselves [§ 11.2.7].

- Output of report and plots are now available in the English, French and Dutch languages [§ 8.2.4].
- Result graphs have been extended. With the Darcy model, MSettle gives results for different stress components in time and along the depth. With the Terzaghi model, the settlement-depth curve has been added [§ 11.5, 11.6].
- The *Reliability* module [§ 18.2] is upgraded from evaluation version to product version, including full verification.
- The *Horizontal Displacement* module [§ 18.3] based on De Leeuw tables [Lit 24] has been added.

1.5 Limitations

When working with MSettle, the following limitations apply.

- During vertical displacements calculation, MSettle assumes that horizontal displacements are zero. The horizontal displacements from the corresponding module will therefore not influence the vertical displacements calculation.
- For Terzaghi, the submerged weight is determined on the basis of final settlements. Furthermore, only the weight of non-uniform loads is reduced, e.g. not the weight of uniform loads or soil layers.
- For Darcy, the gradually changing submerged weight during the calculation is only calculated for non-uniform loads and soil layers, but not for uniform loads.
- The consolidation models do not explicitly describe horizontal flow. The horizontal flow to drains is modelled by a leakage term.
- The Terzaghi model does not calculate the actual effective pressures during consolidation, but is based on an approximate adjustment of settlements from a drained solution. See [§ 1.5.1].

1.5.1 Darcy vs. Terzaghi

The Darcy model uses a step-wise accurate numerical solution of effective stress and pore pressure at different points in time and space. The Terzaghi model uses a time-dependent "degree of consolidation" according to the Terzaghi theory [Lit 3], to adjust the drained settlement solution approximately for the effect of consolidation.

The Terzaghi model has a number of limitations, compared to the Darcy model.

- The settlement after completed consolidation with the Terzaghi model will always be equal to the settlement from a drained solution, even if unloading took place shortly after preceding loading.
- For the same reason, the updated pre-consolidation stress during reloading will be overestimated with Terzaghi if unloading took place before consolidation was finished.
- The combination of layers with different consolidation coefficients and the combination with vertical drains are also described more accurately with Darcy.

- The period of consolidation with Terzaghi will be equal during loading and un/reloading, while Darcy will show faster consolidation during un/reloading.
- The influence of vertical drains and dewatering is averaged along a full layer in combination with Terzaghi. This limitation is especially important for the layer in which the vertical drain ends.
- The Terzaghi model describes submerging by an initial load reduction, while the Darcy model in combination with the NEN-Bjerrum or Isotache model takes into account the gradual character of it.

Compared to the previous Darcy model, the Darcy model in version 8.2 consumes considerable less computation time than in the previous version, supports the same input as the Terzaghi model, features improved submerging modelling and a significantly increased robustness. A choice for the Darcy model is since release 8.2 recommended under most circumstances, as it combines the advantages of the Terzaghi model (fast, robust, convenient input) with improved accuracy.

1.5.2 NEN-Koppejan vs. NEN-Bjerrum/Isotache

The NEN-Koppejan model has been the traditional choice in the Netherlands for many years. The applicability of the Koppejan model is however limited, as it has not been designed to predict unloading/reloading. The Dutch geotechnical design codes currently prescribe a $C_c/C_r/C_\alpha$ method, just as other countries do. MSettle's isotache models with $C_c/C_r/C_\alpha$ or a/b/c parameters are capable of modelling both incremental loading and unloading/reloading. The other difference is that Koppejan assumes a stress dependent slope of the creep tail after virgin loading whereas the $C_c/C_r/C_\alpha$ model assumes that the slope after virgin loading is stress independent.

Key concept of both isotache models is a direct relationship between overconsolidation, creep rate and equivalent age. The only difference between these models is the usage of linear strain for the $C_c/C_r/C_\alpha$ model and natural strain for the a/b/c model.

1.6 Minimum System Requirements

The following minimum system requirements are needed in order to run and install the MSeries software, either from CD or by downloading from the Delft GeoSystems website via MS Internet Explorer:

- Windows 2003, Windows XP (service pack 2), Windows Vista
- Pc with 1 GHz Intel Pentium processor or equivalent
- 512 MB of RAM
- 400 MB free hard disk space
- SVGA video card, 1024 × 768 pixels, high colors (16 bits)
- CD-ROM player
- Microsoft Internet Explorer version 6.0 or higher (download from www.microsoft.nl).

1.7 Definitions and Symbols

n	Porosity
e_0	Initial void ratio: $$e_0 = \frac{n_0}{1 - n_0}$$
c_V	Vertical consolidation coefficient, one-dimensional
σ'	Effective vertical soil pressure
σ_p	Preconsolidation pressure (maximum vertical effective pressure experienced in the past)
σ_0	Initial vertical effective soil pressure
POP	Pre-overburden pressure: $POP = \sigma_p - \sigma_0'$
OCR	Overconsolidation ratio: $$OCR = \frac{\sigma_p}{\sigma_0'}$$
t	Time in days
h_0	Vertical height of layer or oedometer sample at the start of (un)loading
h_t	Vertical height of layer or oedometer sample at time t after (un)loading
Δh	Vertical settlement of layer or sample at time t: $\Delta h = h_t - h_0$
ε^C	Engineering vertical strain (Cauchy): $$\varepsilon^C = \frac{\Delta h}{h_0}$$
ε^H	Natural vertical strain (Hencky): $$\varepsilon^H = -\ln\left(\frac{h_0 - \Delta h}{h_0}\right) = -\ln\left(1 - \varepsilon^C\right)$$
$\dot{\varepsilon}$	Strain rate: $$\dot{\varepsilon} = \frac{d\varepsilon}{dt}$$
C_{sw}	Primary swelling index (unloading): $$C_{sw} = \left(1 + e_0\right)\frac{d\varepsilon}{d\log\sigma'} \quad \text{with} \quad \sigma' < \sigma_p$$
C_c	Primary compression index (virgin loading): $$C_c = \left(1 + e_0\right)\frac{d\varepsilon}{d\log\sigma'} \quad \text{with} \quad \sigma' > \sigma_p$$
C_α	Coefficient of secondary compression (strain based): $$C_\alpha = \frac{d\varepsilon}{d\log t}$$
a	(Isotache) Modified natural swelling index: $$a = \frac{C_{sw}\big|_{\varepsilon^H}}{\left(1 + e_0\right)\ln 10}$$
b	(Isotache) Modified natural compression index: $$b = \frac{C_c\big|_{\varepsilon^H}}{\left(1 + e_0\right)\ln 10}$$

c	(Isotache) Modified natural secondary compression constant: $c = \frac{C_\alpha\|_{\varepsilon^H}}{\ln 10}$
t_{age}	Initial equivalent age: • Isotache: $t_{age} = \tau_0 \; OCR^{\left(\frac{b-a}{c}\right)}$ • NEN-Bjerrum: $t_{age} = \tau_0 \; OCR^{\left(\frac{CR-RR}{C_\alpha}\right)}$
τ_0	Creep rate reference time
C_r	(NEN-Bjerrum) Reloading/Swelling index: $C_r = C_{sw}\|_{\varepsilon^C}$
CR	(NEN-Bjerrum) Compression ratio: $CR = \frac{C_c\|_{\varepsilon^C}}{1+e_0}$
RR	(NEN-Bjerrum) Reloading/Swelling ratio: $RR = \frac{C_r\|_{\varepsilon^C}}{1+e_0}$
C_p	(NEN-Koppejan) Primary compression coefficient below pre-consolidation: $C_p \approx \frac{(1+e_0)\ln 10}{C_{sw}}$ with $\sigma' < \sigma_p$
C_p'	(NEN-Koppejan) Primary compression coefficient above pre-consolidation: $C_p' \approx \frac{(1+e_0)\ln 10}{C_c}$ with $\sigma' > \sigma_p$
C_s	(NEN-Koppejan) Secular compression coefficient below pre-consolidation: $C_s = \ln\left(\frac{\sigma'}{\sigma_0}\right)\frac{\mathrm{d}\log t}{\mathrm{d}\varepsilon}$ with $\sigma' < \sigma_p$
C_s'	(NEN-Koppejan) Secular compression coefficient above pre-consolidation: $C_s' = \ln\left(\frac{\sigma'}{\sigma_p}\right)\frac{\mathrm{d}\log t}{\mathrm{d}\varepsilon}$ with $\sigma' > \sigma_p$
A_p	(NEN-Koppejan) Primary swelling coefficient: $A_p = \frac{(1+e_0)\ln 10}{C_{sw}}$ with $\sigma' < \sigma_0'$
A_s	(NEN-Koppejan) Secondary swelling coefficient: $A_s = \ln\left(\frac{\sigma'}{\sigma_0'}\right)\frac{d\log t}{d\varepsilon}$ with $\sigma' < \sigma_0'$
γ	Unit weight
φ	Water head
k_x, k_y	Darcy permeability in horizontal and vertical direction
C_k	The constant for strain dependent permeability
K_w	Bulk modulus of water

1.8 Getting Help

Help

From the *Help* menu, choose the *MSettle Help* option to open the *MSettle Help* window. For help about the window which is currently active, press F1 or click the *Help* button.

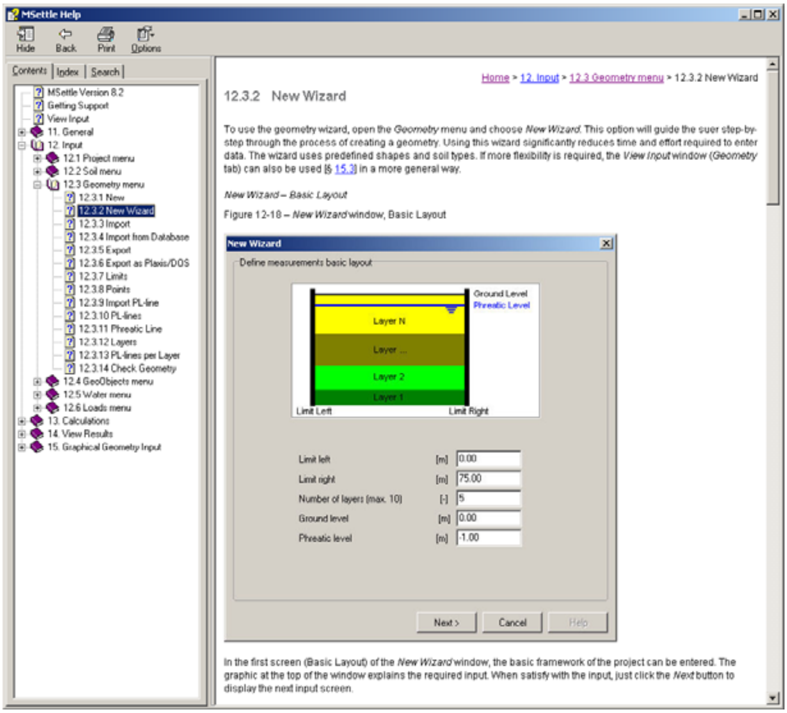
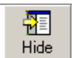
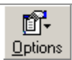


Figure 1-1 – *MSettle Help* window

In the *Help* window displayed (Figure 1-1), there are three ways (corresponding to the available tabs) to find a Help topic:

Contents	Contents	Click this button tab in the Help window for an overview of the Table of Contents.
Searching by word	Index	Click this button to search for Help topics on the basis of a specific word. MSettle will find the corresponding Help topic from the list of the Index section (see at the end of the document).
Searching by word (advanced)	Search	Click this button to search for Help topics on the basis of specific words. MSettle will find several corresponding Help topic that use those words in their description.

List Topics		In the *Search* tab, click this button to display a list of the Help topics generated on the basis of the specific word given.
Display		When a Help topic is selected, click this button to display its content.
Hide/Show	 	Click this button to alternatively hide or show the searching tabs *Content*, *Index* and *Search*.
Back		Click this button to go back to the previous selected Help topic.
Print		Click this button to print the contents of the window.
Options		Click this button to display the menu below (Figure 1-2).

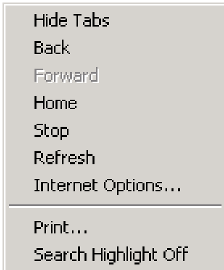


Figure 1-2 – Menu from the *Options* button of the *MSettle Help* window

Hide/Show Tabs	Select this option to alternatively hide or show the searching tabs *Content*, *Index* and *Search*.
Back	Select this option to go back to the previous selected Help topic.
Forward	Select this option to go forward to the preceding selected Help topic.
Home	Select this option to go to the default internet home page.
Stop	Select this option to stop searching.
Refresh	Select this option to refresh the content of the window.
Internet Options	Select this option to open the *Internet Options* window.
Print	Select this option to print the contents of the window
Search Highlight On/Off	Select this option to choose whether to highlight the search words wherever they appear in the displayed text.

The *MSettle Help* window contains only the Reference section of this manual.
To display and print the Help texts properly, the Symbol TrueType font must be installed.

1.9 Getting Support

If problems are encountered, the first step should be to consult the online Help at www.delftgeosystems.nl menu Software. On the left-hand side of the window (Figure 1-3), In 'FAQ' are listed the most frequently asked technical questions and their answers and in 'Known bugs' are listed the known bugs of the program.

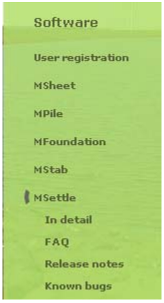


Figure 1-3 – 'Software' menu of the Delft GeoSystems website (www.delftgeosystems.nl)

If the solution cannot be found there, then the problem description can be e-mailed (preferred) or faxed to the Delft GeoSystems Support team. When sending a problem description, please add a full description of the working environment. To do this conveniently:

- Open the program.
- If possible, open a project that can illustrate the question.
- Choose the *Support* option in the *Help* menu. The *System Info* tab contains all relevant information about the system and the MSeries software. The *Problem Description* tab enables a description of the problem encountered to be added.

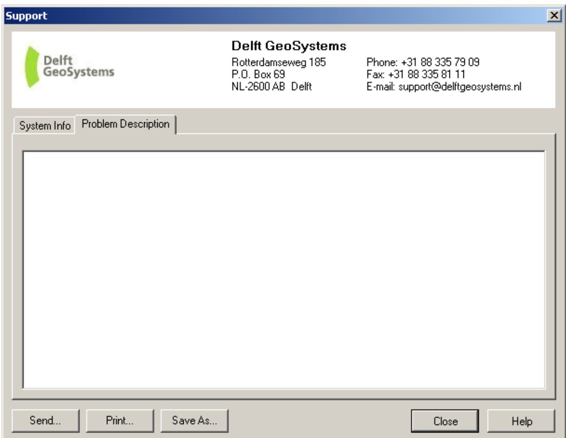


Figure 1-4 – *Support* window, *Problem Description* tab

- After clicking on the *Send* button, the *Send Support E-Mail* window opens, allowing sending current file as an attachment. Marked or not the *Attach current file to mail* checkbox and click *OK* to send it.

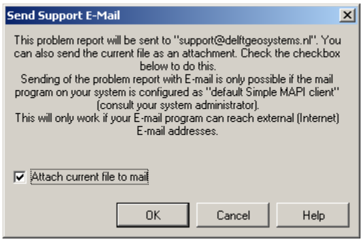


Figure 1-5 – *Send Support E-Mail* window

The problem report can either be saved to a file or sent to a printer or PC fax. The document can be emailed to support@delftgeosystems.nl or alternatively faxed to +31(0)88 335 8111.

1.10 Deltares

Since its foundation in 1934, GeoDelft has been one of the first and most renowned geotechnical engineering institutes of the world. On January 1st 2008, GeoDelft has merged with WL | Delft Hydraulics and some parts of Rijkswaterstaat and TNO into the new Deltares Institute on delta technology. Part of Deltares's role is still to obtain, generate and disseminate geotechnical know-how.. For more information on Deltares, visit the Deltares website: http://www.deltares.nl .

1.11 Delft GeoSystems

Delft GeoSystems was founded by GeoDelft in 2002. The company's objective is to convert Deltares's knowledge into practical geo-engineering services and software. Delft GeoSystems has developed a suite of software for geotechnical engineering. Besides software, Delft GeoSystems is involved in providing services such as hosting online monitoring platforms, hosting on-line delivery of site investigation, laboratory test results, etc. As part of this process Delft GeoSystems is progressively connecting these services to their software. This allows for more standardized use of information, and the interpretation and comparison of results. Most software is used as design software, following design standards. This however, does not guarantee a design that can be executed successfully in practice, so automated back-analyses using monitoring information are an important aspect in improving geotechnical engineering results.

Delft GeoSystems makes use of Deltares's intensive engagement in R&D for GeoBrain. GeoBrain's objective is to combine experience, expertise and numerical results into one forecast, using Artificial Intelligence, Neural Networks and Bayesian Belief Networks. For more information about Delft GeoSystems' geotechnical software, including download options, visit http://www.delftgeosystems.nl or choose the *Delft GeoSystems Website* option from the *Help* menu of MSettle.

1.12 Acknowledgements

The former Road and Hydraulic Engineering Division (Rijkswaterstaat/DWW) of the Dutch Ministry of Transport, Public Works and Water Management has sponsored the first development of MSettle.

The contribution from the EZ/Senter project GeoSafe on the reliability framework and the many contributions from the research program Delft Cluster are also gratefully acknowledged. These contributions were crucial for developing and evaluating the present set of well-established models.

2 Getting Started

This Getting Started chapter aims to familiarize the user with the structure and user interface of MSettle. The Tutorial section which follows uses a selection of case studies to introduce the program's functions.

2.1 Starting MSettle

To start MSettle, click *Start* on the Windows taskbar or double-click an MSettle input file that was generated during a previous session.

For an MSettle installation based on floating licenses, the *Modules* window may appear at start-up [§ 8.2.5]. Check that the correct modules are selected and click OK.

When MSettle is started from the Windows taskbar, the last project that was worked on will open automatically (unless the program has been configured otherwise in the *Program Options* window, reached from the *Tools* menu) and MSettle will display the main window [§ 2.2].

2.2 Main Window

When MSettle is started, the main window is displayed (Figure 2-1). This window contains a menu bar [§ 2.2.1], an icon bar [§ 2.2.2], a *View Input* window [§ 2.2.3] that displays the pre-selected or most recently accessed project, a *title panel* [§ 2.2.4] and a *status bar* [§ 2.2.5]. The caption of the main window of MSettle displays the program name, followed by the calculation model, the consolidation model and the strain type. When a new file is created, the default calculation model is *NEN-Bjerrum* (*Linear* strain), the default consolidation model is *Darcy* and the project name is *Project1*. The first time after installation of MSettle, the *View Input* window will be closed.

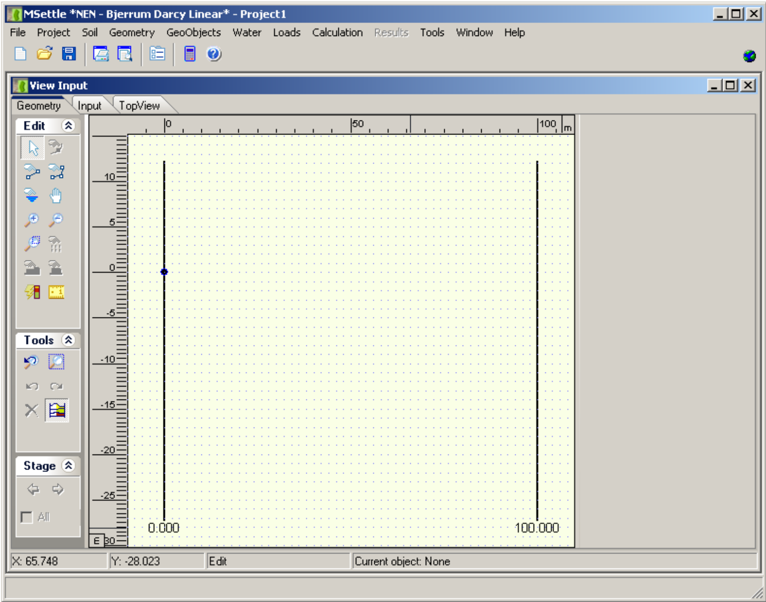


Figure 2-1 – MSettle main window

2.2.1 The menu bar

To access the MSettle menus, click the names on the menu bar.

Figure 2-2 – MSettle menu bar

The menus contain the following functions:

File	Standard Windows options for opening, saving and sending files as well as several MSettle options for exporting and printing active windows and reports [§ 8.1].
Project	Options for selecting the model types, defining project properties and viewing the input file [§ 9.1].
Soil	Options for defining the soil type properties [§ 9.2].
Geometry	Options for defining layers boundaries, soil types and piezometric lines [§ 9.3].
GeoObjects	Options for defining the verticals (X co-ordinates) for which results will be shown, the vertical drains and the pore pressure meters [§ 9.4].
Water	Input of water parameters [§ 9.5].

Loads	Input of external loads [§ 9.6].
Calculation	A wide range of calculation options to determine the settlements and stresses along the verticals [chapter 10].
Results	Options for displaying graphical or tabular output of the settlements and stresses per vertical [chapter 11].
Tools	Options for editing MSettle program defaults [§ 8.2].
Window	Default Windows options for arranging the MSettle windows and choosing the active window.
Help	Online Help [§ 2.1].

2.2.2 The icon bar

Use the buttons on the icon bar to quickly access frequently used functions (see below).

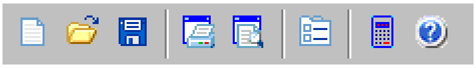
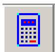


Figure 2-3 – MSettle icon bar

Click on the following buttons to activate the corresponding functions:

	Start a new MSettle project.
	Open the input file of an existing project.
	Save the input file of the current project.
	Print the contents of the active window.
	Display a print preview.
	Open the *Project Properties* window. Here you can enter the project title and other identification data, and determine the View Layout and Graph Settings for your project.
	Start the calculation.
	Display the contents of online Help.

2.2.3 View Input window

The *View Input* window displays the geometry and additional MSettle input of the current project. The window has the following three tabs:

- *Geometry*
  In this view it is possible to define, inspect and modify the positions and soil types of different layers. For more information about these general M-Series options for geometrical modelling, see the description of the *Geometry* menu [§ 9.3] or see [§ 12.4].

- *Input*
  In this view it is possible to define, inspect and modify the additional MSettle specific input. For more information on the available options, see below in this paragraph.
- *Top View*
  This tab shows the lateral and the top view of the inputted project.

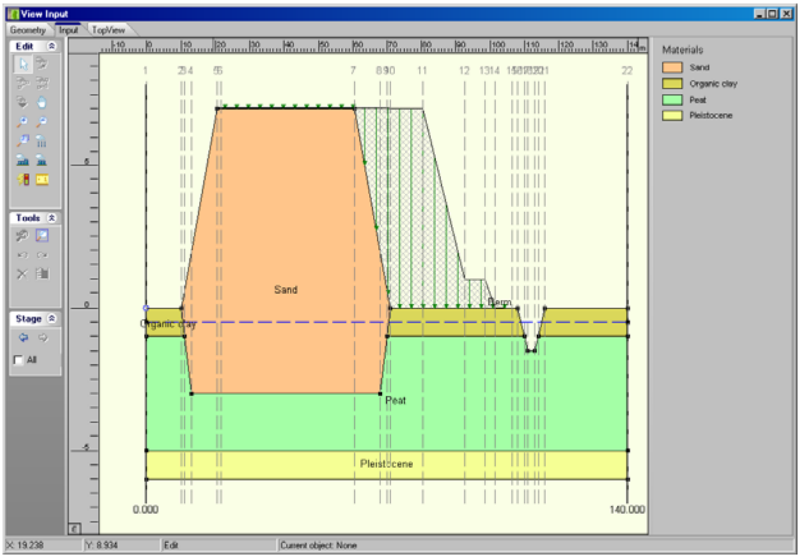


Figure 2-4 – *View Input* window, *Input* tab

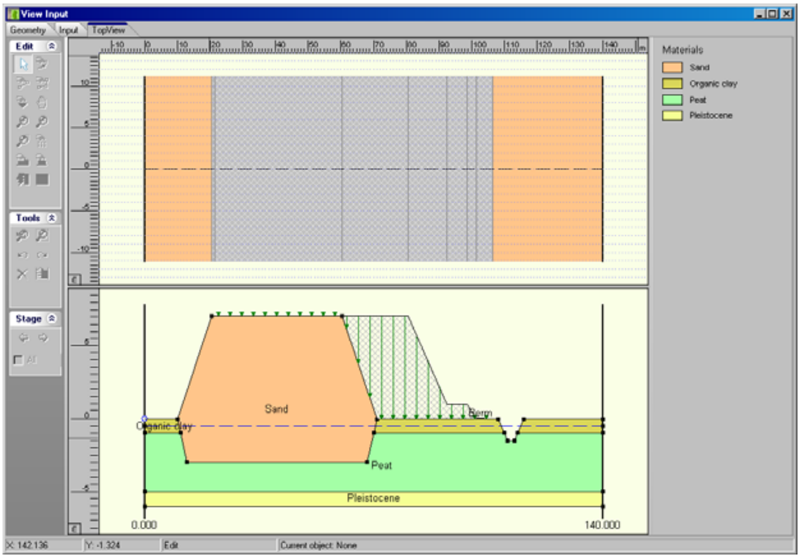


Figure 2-5 – *View Input* window, *Top View* tab

The panel on the left of the view contains buttons for entering data and controlling the graphical view. Click on the following buttons in the *Edit*, *Tools* or *Stage* panel to activate the corresponding functions:

Select and *Edit* mode

In this mode, the left-hand mouse button can be used to graphically select a previously defined grid, load, geotextile or forbidden line. Items can then be deleted or modified by dragging or resizing, or by clicking the right-hand mouse button and choosing an option from the menu displayed. Pressing the *Escape* key will return the user to this *Select* and *Edit* mode.

Add point(s) to boundary / PL-line

Click this button to add points to all types of lines (lines, polylines, boundary lines, PL-lines). By adding a point to a line, the existing line is split into two new lines. This provides more freedom when modifying the geometry.

Add single line(s)

Click this button to add single lines. When this button is selected, the first left-hand mouse click will add the info bar of the new line and a “rubber band” is displayed when the mouse is moved. The second left-hand mouse click defines the end point (and thus the final position) of the line. It is now possible to either go on clicking start and end points to define lines, or stop adding lines by selecting one of the other tool buttons, or by clicking the right-hand mouse button, or by pressing the *Escape* key.

Add polyline(s)

Click this button to add polylines. When this button is selected, the first left-hand mouse click adds the starting point of the new line and a “rubber band” is displayed when the mouse is moved. A second left-hand mouse click defines the end point (and thus the final position) of the first line in the polyline and activates the “rubber band” for the second line in the polyline. Every subsequent left-hand mouse click again defines a new end point of the next line in the polyline. It is possible to end a polyline by selecting one of the other tool buttons, or by clicking the right-hand mouse button, or by pressing the *Escape* key.

Add PL-line(s)

Click this button to add a piezometric level line (PL-line). Each PL-line must start at the left limit and end at the right limit. Furthermore, each consecutive point must have a strictly increasing X co-ordinate. Therefore, a PL-line must be defined from left to right, starting at the left limit and ending at the right limit. To enforce this, the program will always relocate the first point clicked (left-hand mouse button) to the left limit by moving it horizontally to this limit. If trying to define a point to the left of the previous point, the rubber band icon indicates that this is not possible. Subsequently clicking on the left side of the previous point, the new point will be added at the end of the rubber band icon instead of the position clicked.

Pan

Click this button to change the visible part of the drawing by clicking and dragging the mouse.

Zoom in

Click this button to enlarge the drawing, and then click the part of the drawing which is to be at the centre of the new image. Repeat if necessary.

Zoom out

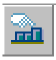
Click this button, and then click on the drawing to reduce the drawing size. Repeat if necessary.

Zoom rectangle

Click this button then click and drag a rectangle over the area to be enlarged. The selected area will be enlarged to fit the window. Repeat if necessary.

Add vertical

Click this button to graphically define the position of a vertical.



Add non-uniform load

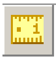
Click this button to display a window in which it is possible to add, modify or delete non-uniform loads per unit of area.

Add other load

Click this button to display a window in which it is possible to add, modify or delete trapezoidal, circular, rectangular or uniform loads.

Convert geometry to 1D

Click this button to convert geometry to 1D.



Measure the distance and slope between two points

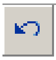
Click this button, then click the first point on the *View Input* window and place the cross on the second point. The distance and the slope between the two points can be read beside the second point. To turn this option off, click the escape key.

Undo zoom

Click this button to undo the zoom. If necessary, click several times to retrace each consecutive zoom-in step that was made.

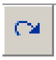
Zoom limits

Click this button to display the complete drawing.



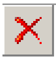
Undo

Click this button to undo the last change(s) made to the geometry



Redo

Click this button to redo the previous Undo action



Delete

Click this button to delete a selected element.

NOTE: This button is only available when an element is selected.

Automatic regeneration of geometry on/off

When selected, the program will automatically try to generate a new valid geometry whenever geometry modifications require this. During generation, (poly)lines (solid blue) are converted to boundaries (solid black), with interjacent layers. New layers receive a default material type. Existing layers keep the materials that were assigned to them. Invalid geometry parts are converted to construction elements. Automatic regeneration may slow down progress during input of complex geometry, because validity will be checked

	continuously.
	*Previous stage* Click this button to view the previous stage in the sequence of loading.
	*Next stage* Click this button to view the next stage in the sequence of loading.

2.2.4 Title panel

This panel situated at the bottom of the *View Input* window displays the project titles, as entered on the *Identification* tab in the *Project Properties* window [§ 9.1.3].

2.2.5 Status bar

This bar situated at the bottom of the main window displays a description of the selected icon of the icon bar [§ 2.2.2].

2.3 Files

*.sli	*MSettle input file (ASCII):* Contains all specific input for MSettle. After interactive generation, this file can be reused in subsequent MSettle analyses.
*.sls	*Setting file (ASCII):* Working file with settings data. This file doesn't contain any information that is relevant for the calculation, but only settings that apply to the representation of the data, such as the grid size.
*.geo	*Input file (ASCII):* Contains the (deformed) geometry data that can be shared with other M-Series programs.
*.sti	*Output file (ASCII):* File used by MSettle for a coupled stability analysis, with deformed geometry and excess pore pressures.
*.sld	*Dump file (ASCII):* Contains calculation results used for graphical and report output.
*.slo	*Obsolete file (ASCII):* Contains echo of input and tabular results.
*.err	*Error file (ASCII):* If there are any errors in the input, they are described in this file.
*.gef	*Geotechnical Exchange Format file (ASCII):* Contains measurements data.
*.slm	SLM file (ASCII): Input of settlement and surface measurements.

2.4 Tips and Tricks

2.4.1 Keyboard shortcuts

Use the keyboard shortcuts given in Table 2-1 to directly opening a window without selecting the option from the bar menu.

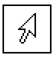
Table 2-1 – Keyboard shortcuts for MSettle

Keyboard shortcut	Opened window
Ctrl + N	New
Ctrl + S	Save
Ctrl + O	Open
F12	Save As
Ctrl + C	Copy Active Window to Clipboard
Ctrl + P	Print Report
Ctrl + M	Model
Ctrl + T	Materials
Ctrl + E	Verticals
F9	Start Calculation
Ctrl + R	Report
F1	MSettle Help

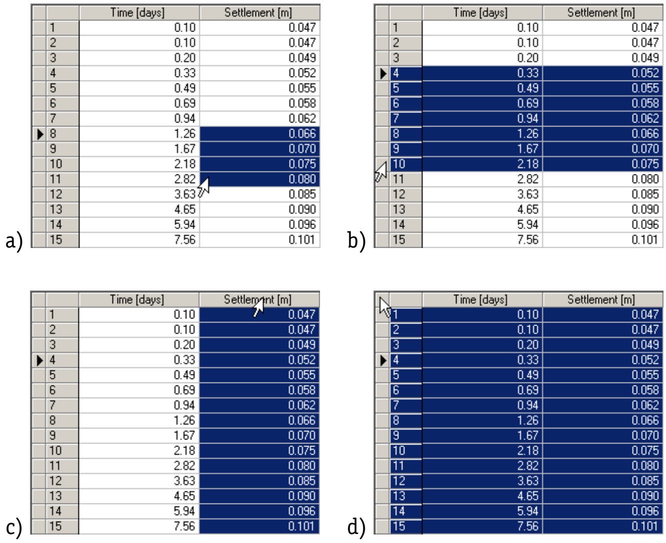
2.4.2 Exporting figures and reports

All figures in MSettle such as geometry and graphical output can be exported in WMF (Windows Meta Files) format. In the *File* menu, select the option *Export Active Window* to save the figures in a file. This file can be later imported in a Word document for example or added as annex in a report. The option *Copy Active Window to Clipboard* from the *File* menu can also be used to copy directly the figure in a Word document. The report can be entirely exported as PDF (Portable Document Format) or RTF (Rich Text Format) file. To look at a PDF file Adobe Reader can be used. A RTF file can be opened and edited with word processors like MS Word. Before exporting the report, a selection of the relevant parts can be done with the option *Report Selection* [§ 11.1].

2.4.3 Copying part of a table



It is possible to copy part of a table in another document, an Excel sheet for example. If the cursor is placed on the left-hand side of a cell of the table, the cursor changes in an arrow which points from bottom left to top right. Select a specific area by using the mouse (see Figure 2-6a). Then, using the copy button (or ctrl+C) this area can be copied.



a)

	Time [days]	Settlement [m]
1	0.10	0.047
2	0.10	0.047
3	0.20	0.049
4	0.33	0.052
5	0.49	0.055
6	0.69	0.058
7	0.94	0.062
8	1.26	0.066
9	1.67	0.070
10	2.18	0.075
11	2.82	0.080
12	3.63	0.085
13	4.65	0.090
14	5.94	0.096
15	7.56	0.101

b)

	Time [days]	Settlement [m]
1	0.10	0.047
2	0.10	0.047
3	0.20	0.049
4	0.33	0.052
5	0.49	0.055
6	0.69	0.058
7	0.94	0.062
8	1.26	0.066
9	1.67	0.070
10	2.18	0.075
11	2.82	0.080
12	3.63	0.085
13	4.65	0.090
14	5.94	0.096
15	7.56	0.101

c)

	Time [days]	Settlement [m]
1	0.10	0.047
2	0.10	0.047
3	0.20	0.049
4	0.33	0.052
5	0.49	0.055
6	0.69	0.058
7	0.94	0.062
8	1.26	0.066
9	1.67	0.070
10	2.18	0.075
11	2.82	0.080
12	3.63	0.085
13	4.65	0.090
14	5.94	0.096
15	7.56	0.101

d)

	Time [days]	Settlement [m]
1	0.10	0.047
2	0.10	0.047
3	0.20	0.049
4	0.33	0.052
5	0.49	0.055
6	0.69	0.058
7	0.94	0.062
8	1.26	0.066
9	1.67	0.070
10	2.18	0.075
11	2.82	0.080
12	3.63	0.085
13	4.65	0.090
14	5.94	0.096
15	7.56	0.101

Figure 2-6 – Selection of different parts of a table using the arrow cursor

To select a row, click on the cell before the row number (see b) in Figure 2-6). To select a column, click on the top cell of the column (see c) in Figure 2-6). To select the complete table, click on the top left cell (see d) in Figure 2-6).

In some tables the option *Copy* is also present at the left hand pane.

2.4.4 Continuous display of the results in time or depth

In the *Time-History* and/or *Depth-History* windows, by selecting the first *Time* or *Depth* step respectively at the top of the window and using the scroll button of the mouse, graphical results are displayed in a continuous way in time (from initial to final time) or in depth (from ground surface to the base).

MSettle Version 8
EMBANKMENT DESIGN AND SOIL SETTLEMENT PREDICTION

MSettle

Version 8

Introduction
Tutorial
Reference
Background
Verification

Embankment Design and Soil Settlement Prediction

Deltares

Delft GeoSystems

3 Tutorial 1: Building site preparation

This first tutorial illustrates the execution of a simple settlement analysis with loading and partial unloading. The NEN-Bjerrum soil model is used, in combination with two different consolidation models.

The objectives of this exercise are:

- to learn how to define:
  - layers and their properties,
  - an initial hydraulic pore pressure distribution,
  - non-uniform loads;
- to learn how to determine the total and residual settlement of consolidating soft soil by loading and partial unloading;
- to illustrate the behaviour of the NEN-Bjerrum isotache model for loading and unloading;
- to illustrate the differences between the Darcy and Terzaghi consolidation model.

For this example, the following MSettle modules are needed:

- MSettle (1D model with Terzaghi)
- 2D geometry model
- Darcy consolidation model

This tutorial is presented in the files Tutorial-1a.sli to Tutorial-1e.sli.

3.1 Introduction

A soft soil site has to be prepared for further residential construction activities, by adding a sand layer on top with a height of 1 meter. The subsoil consists of approximately 6 meters of overconsolidated clay on stiff sand. The available time for the construction preparation stage is 200 days. The construction activities thereafter will take 400 additional days. The maximum value for the allowed residual settlements in the period from 600 days to 10000 days is 10 cm. The thick layer of low permeable clay will consolidate slowly. Vertical drains are however not allowed along the full depth, because the clay layer must keep the sand aquifer sealed. A temporary additional loading of 1 m sand is therefore applied until 200 days, to reduce the residual settlement.

The position of layers and loads is shown in Figure 3-1. The initial surface is located at reference level. The phreatic level is located half a meter below the surface level. The value of the piezometric level in the pleistocene sand layer is at the surface level.

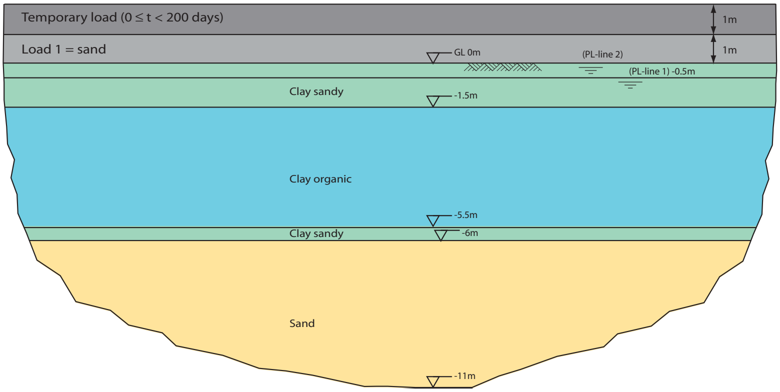


Figure 3-1 – Layers and loading (Tutorial 1)

The parameters of the three soil types are given in Table 3-1.

Table 3-1 – Soil type properties (Tutorial 1)

			Sand	Clay Organic	Clay Sandy
Saturated unit weight	γ_{sat}	[kN/m^3]	20	14	16
Unsaturated unit weight	γ_{unsat}	[kN/m^3]	18	14	16
Overconsolidation Ratio	OCR	[kPa]	1	2.69	1.66
Consolidation coefficient	C_v	[m^2/s]	Drained	4×10^{-8}	10^{-6}
Reloading/ Swelling ratio	$RR = C_r/(1+e_0)$	[-]	0.0001	0.03	0.0125
Compression ratio	$CR = C_c/(1+e_0)$	[-]	0.0023	0.23	0.15
Coeff. of secondary comp.	C_α	[-]	0	0.02	0.007

3.2 Project

3.2.1 Create New Project

Follow the steps below to start the creation of the geometry displayed in Figure 3-1:

1. Start MSettle from the Windows taskbar (Start/Programs/Delft GeoSystems/MSettle/MSettle).
2. Click *File* on the MSettle menu bar, and choose *New*.
3. Select *New geometry* and click *OK*.

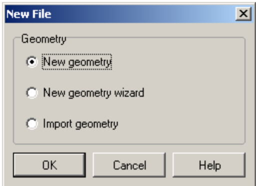


Figure 3-2 – *New File* window

The *View Input* window will appear, with an empty initial geometry (Figure 3-3).

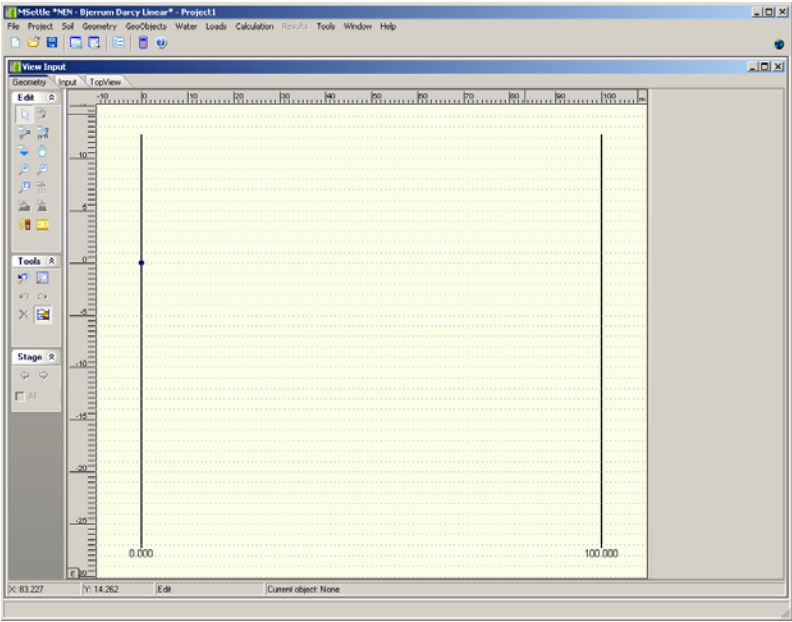


Figure 3-3 – *View Input* window

4. Click *Save as* in the *File* menu.
5. Enter <Tutorial-1a> as file name.
6. Click *Save*.

3.2.2 Project Properties

To give the project a meaningful description, follow the steps described below:

7. On the menu bar, click *Project* and then choose *Properties* to open the *Project Properties* window.
8. Fill in <Tutorial 1 for MSettle> and <Building site preparation> for *Title 1* and *Title 2* respectively in the *Identification* tab (Figure 3-4, left).

In the *View Input* tab, some default values are modified:

9. In the *View Input* tab, mark the *Points* checkbox of the *Labels* sub-window to display the point's number and select the option *As material names* of the *Layers* sub-window to display the name of the layers. Also mark the *Snap to grid* checkbox and decrease the *Grid distance* from 1 m to <0.5 m> to make easier the graphical defining the layer boundaries [§ 3.3] (Figure 3-4, right).
10. Click *OK*.

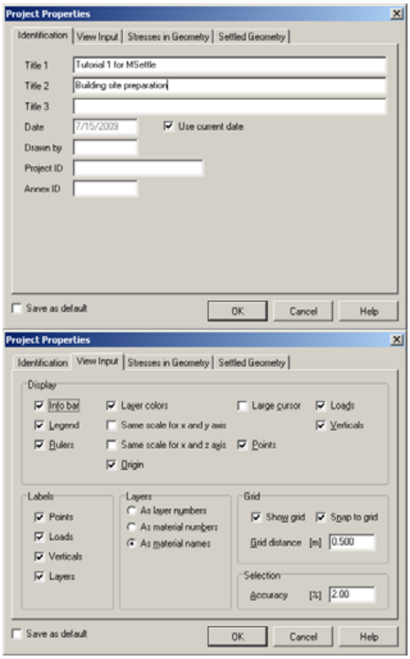


Figure 3-4 – *Project Properties* window, *Identification* tab (left) and *View Input* tab (right)

See *Project Properties* [§ 9.1.3] for a detailed description of this window.

3.3 Geometry

3.3.1 Layer boundaries

Layer boundaries need to be defined first. These boundaries have to run from the left to the right geometry limits. A combined graphical and numerical input will be used, as an alternative to fully numerical input of points and lines.

First the assignment of soil material to boundary lines must be deactivated, via the *Geometry* tab of the *View Input* window:

11. Click the *Automatic regeneration of geometry on/off* button  in the *Tools* panel on the left hand side.

Then the layer boundaries are added graphically at their approximate positions:

12. Click on the *Add single line(s)* button  in the *Edit* panel on the left hand side, and add the top and bottom lines respectively at approximate positions 0 and -11 meters using the cursor. Locate the cursor position outside the geometrical limits (the black vertical lines) when defining the start and end point of each line by clicking, in order to enforce the horizontal co-ordinates of these end points exactly at the geometry limits.
13. Click the *Zoom limits* button  of the *Tools* panel to enlarge the drawing.
14. Add the intermediate boundaries respectively at the following approximate positions: -6, -5.5 and then -1.5 meters, as explained in step 12.
15. Click the *Automatic Regeneration of Geometry* button in the *Tools* panel to generate soil layers between the boundaries.

3.3.2 Piezometric lines

As previously for the layer boundaries, the piezometric lines are added graphically at their approximate positions, via the *Geometry* tab of the *View Input* window:

16. Click on the *Add pl-line(s)* button  in the *Edit* panel, and add two piezometric level lines from the left to the right respectively at the approximate positions: -0.5 and 0 meters below surface level.

    The geometry given in Figure 3-5 should appear.

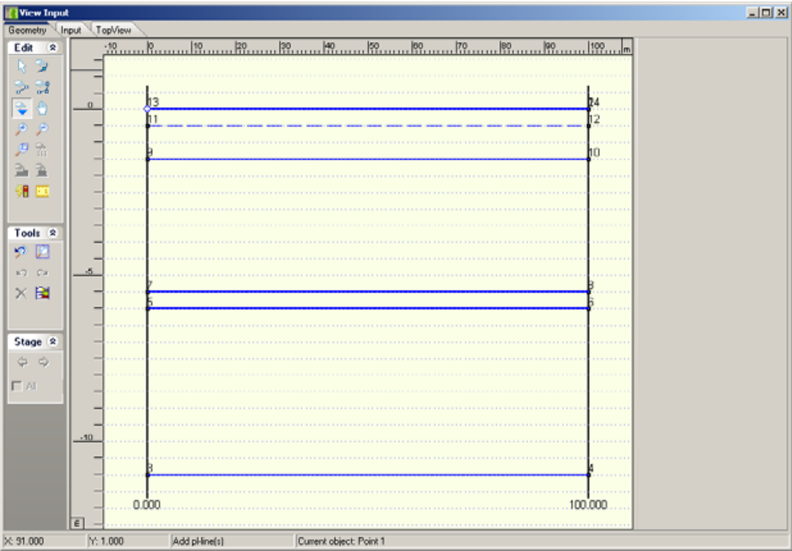
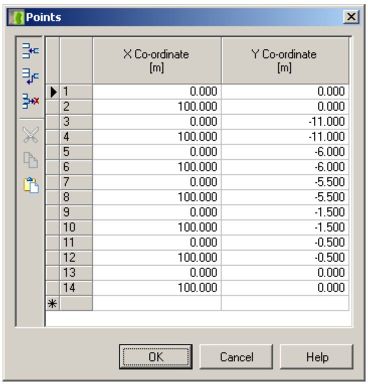


Figure 3-5 – *View Input* window, after input of single lines and piezometric lines

17. Click the *Automatic regeneration of geometry on/off* button  to generate soil layers between the boundaries.
18. Click *Geometry* on the menu bar and choose *Points*. Adjust the displayed approximate vertical values of the graphically created points to their exact values (Figure 3-6).
19. Click *OK* to confirm.



	X Co-ordinate [m]	Y Co-ordinate [m]
1	0.000	0.000
2	100.000	0.000
3	0.000	-11.000
4	100.000	-11.000
5	0.000	-6.000
6	100.000	-6.000
7	0.000	-5.500
8	100.000	-5.500
9	0.000	-1.500
10	100.000	-1.500
11	0.000	-0.500
12	100.000	-0.500
13	0.000	0.000
14	100.000	0.000

Figure 3-6 – *Points* window

3.3.3 Phreatic Line

20. Click *Geometry* on the menu bar, and choose *Phreatic Line*. Note that MSettle assumes the location of the phreatic line by default at the first defined piezometric level.

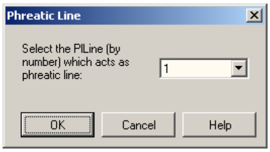


Figure 3-7 – *Phreatic Line* window

3.3.4 PL-lines per Layer

21. Click *Geometry* on the menu bar, and choose *PL-lines per Layer*.
22. Enter the PL-line numbers (1 for the phreatic line and 2 for the piezometric level in the sand layer) at the top and the bottom of the different layers. The piezometric level will vary linearly in the organic clay layer, due to its relatively low permeability compared to the surrounding sandy clay layers.
23. Click *OK* to confirm.

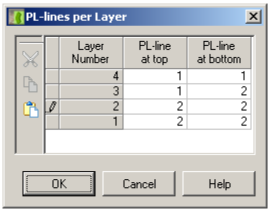


Figure 3-8 – *PL-lines per Layer* window

See *Geometry* menu [§ 9.3 and chapter 12] for a detailed description of geometry input.

3.4 Soil types and properties

24. Choose *Materials* from the *Soil* menu to open the *Materials* window.
25. Select *Soft Clay* in the material list at the left hand of the window. Click *Rename* and change *Soft Clay* into <Clay Organic>. Enter the soil properties according to Table 3-1. Click the *Compression* tab and the *Consolidation and unit weight* tab to switch between the input screens of the corresponding parameters.
26. Select *Sand* and mark the *Drained* checkbox. Enter the soil properties according to Table 3-1.
27. Select *Medium Clay* and rename it into <Clay Sandy>. Enter the soil properties according to Table 3-1. The final input for *Clay Sandy* is presented in Figure 3-9.

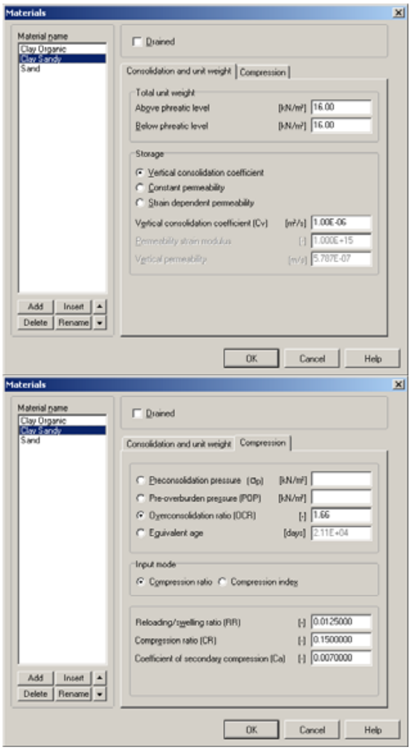


Figure 3-9 – *Materials* window for *Clay Sandy*

28. Optionally delete the unused default soil types, using the *Delete* button.
29. Click *OK* to confirm.

NOTE: No consolidation coefficient value is required if completely drained behaviour is assumed.

NOTE: It is possible to import soil properties from the MGeoBase database, see [§ 9.2.1]. To this end MGeoBase has to be installed.

See *Soil* menu [§ 9.2] for a detailed description of this window.

3.5 Layers

30. Choose *Layers* from the *Geometry* menu to open the *Layers* window.
31. Click the *Materials* tab and attach the added soil types to the previously generated layers, using the › button: <Clay Sandy> to layer <4> and <2>, <Clay Organic> to layer <3> and <Sand> to layer <1>.
32. Click *OK* to confirm.

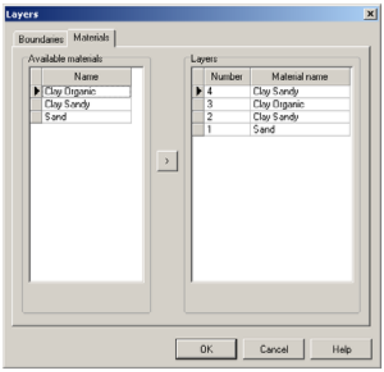


Figure 3-10 – *Layers* window, *Materials* tab

See *Layers* [§ 9.3.12] for a detailed description of this window.

3.6 Loads

The self-weight of the added sand layer is modeled as a non-uniform load.

33. From the *Loads* menu, choose *Non-Uniform Loads* to open the input window.
34. In the *Load name* sub-window, click *Add* and rename the new load to <Sand layer>. Enter the values for the first load as displayed in Figure 3-11.
35. Repeat this for the second load named <Temporary load>. Note that the temporary effect of this load is modeled by input of an *End time*. Also note that the second load starts from the defined position of the first load.
36. Click *OK* to confirm.

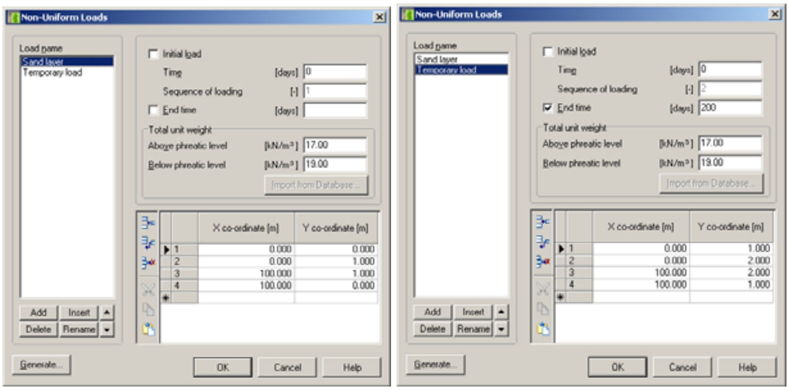


Figure 3-11 – *Non-Uniform Loads* window

The defined loads are depicted in the *Input* tab of the *View Input* window (Figure 3-12). The sequence of loading can be viewed by clicking the arrows in the *Stage* panel.

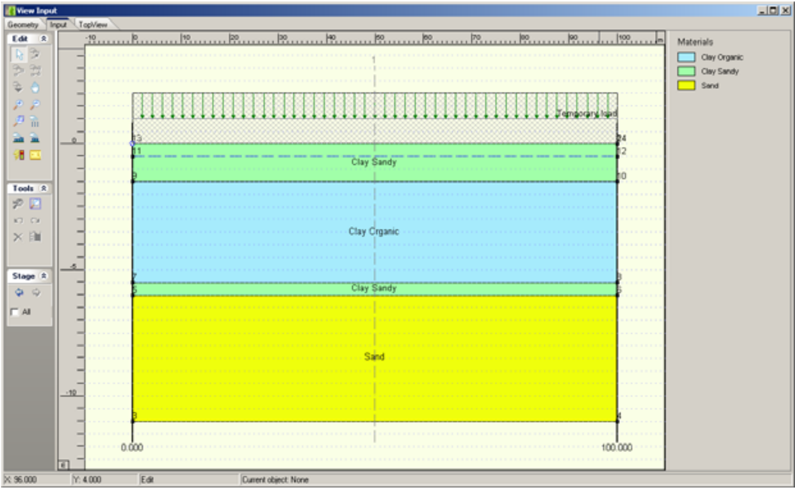


Figure 3-12 – *View Input* window, *Input* tab

See *Non-Uniform Loads* [§ 9.6.1] for a detailed description of this window.

3.7 Verticals

MSettle determines time-dependent settlements along one or more user-defined verticals. In this case (uniform loading) it is sufficient to define one vertical at the centre.

37. Choose *Verticals* from the *GeoObjects* menu to open the input window.

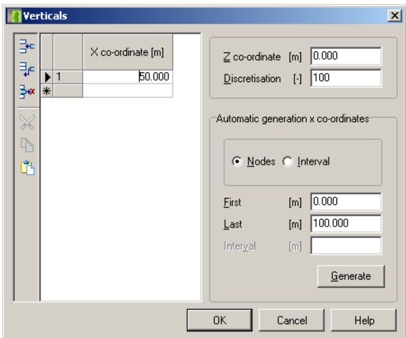


Figure 3-13 – *Verticals* window

38. Enter the X co-ordinate <50>.
39. Click *OK* to confirm.

The defined vertical is displayed together with the defined loads in the *Input* tab of the *View Input* window.

See *Verticals* [§ 9.4.1] for a detailed description of this window.

3.8 Calculation

3.8.1 Calculation Options

40. Choose *Options* from the *Calculation* menu.
41. In the *Calculation Options* window, mark the *Output of settlements by partial loading* checkbox.
42. Click *OK* to confirm.

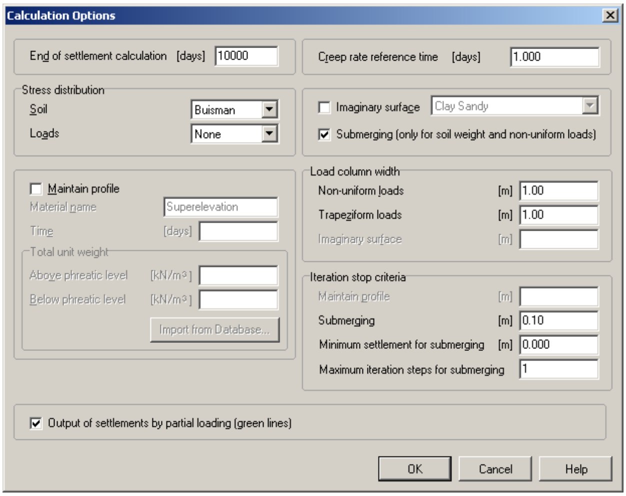


Figure 3-14 – *Calculation Options* window

See *Calculation Options* [§ 10.1] for a detailed description of this window.

3.8.2 Calculation Times

Tabular output of the intermediate and residual settlement in the *Report*, together with the graphical output of the residual settlement, will be displayed in user defined time points only.

43. Choose *Times* from the *Calculation* menu.

44. In the *Calculation Times* window enter the times according to Figure 3-15, using the *Add row* button.
45. Click *OK* to confirm.

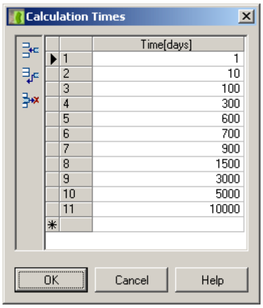


Figure 3-15 – *Calculation Times* window

See *Calculation Times* [§ 10.2] for a detailed description of this window.

3.8.3 Start Calculation

The calculation can now be started.

46. Choose *Start* from the *Calculation* menu or press the function key F9.
47. Mark the checkbox *Add dissipation calculation* to generate dissipation graphs (average degree of consolidation versus time) for the different layers.
48. Click *Start* to perform the calculation.

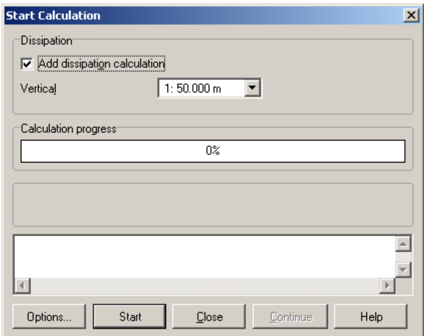


Figure 3-16 – *Start Calculation* window

3.9 Results basic analysis

Results can be viewed from the *Results* menu, after the calculation has finished. The following selected results will be presented hereafter:

- *Time-History* curve [§ 3.9.1]. Graphs of settlement and/or different stress components versus time.
- *Depth-History* curve [§ 3.9.2]. Graphs of settlement and/or different stress components along verticals.
- *Residual Settlement* [§ 3.9.3]. Graph of remaining settlements until the end time versus the start time of measurement.

See *View Results* [chapter 11] for a description of all available results.

3.9.1 Time-History

49. Choose the *Time-History* option in the *Results* menu. The graphs of effective stress versus time and settlement versus time are now displayed at the surface level. The green line indicates the virtual settlements that would occur after a certain loading stage, if no further loading or unloading would have been applied.

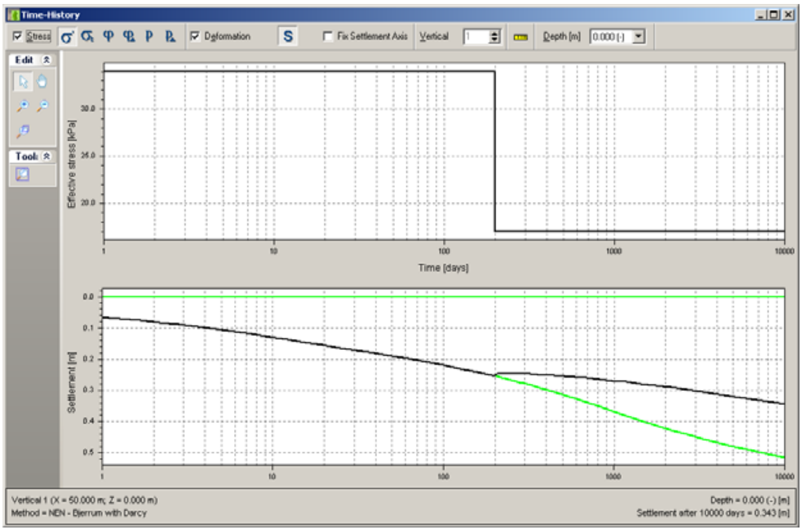


Figure 3-17 – *Time-History* window, Effective stress and Settlement at surface level

50. Click the right-hand mouse button in the *Settlement* graph and select *View Data*, to view the numerical data in the *Chart Data* window (Figure 3-20). This numerical data can also be copied for usage in for example spreadsheets. The predicted residual settlement between 600 days and 10000 days is 0.343 – 0.257 = 0.086 m.

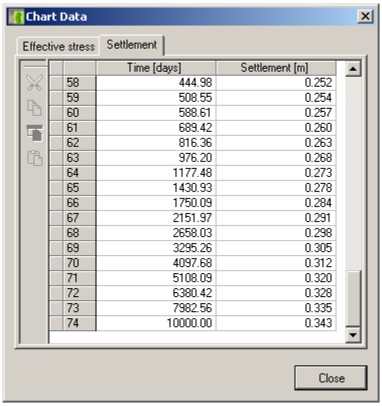


Chart Data

Effective stress | Settlement

	Time [days]	Settlement [m]
58	444.98	0.252
59	508.55	0.254
60	588.61	0.257
61	689.42	0.260
62	816.36	0.263
63	976.20	0.268
64	1177.48	0.273
65	1430.93	0.278
66	1750.09	0.284
67	2151.97	0.291
68	2658.03	0.298
69	3295.26	0.305
70	4097.68	0.312
71	5108.09	0.320
72	6380.42	0.328
73	7982.56	0.335
74	10000.00	0.343

Close

Figure 3-18 – *Chart Data* window, Surface settlement versus Time

51. Click the *Excess hydraulic head* icon , and change the *Depth* to <3.5 m>. The excess head at the centre of the layer *Clay Organic* reduces quite quickly in time during the first stage of loading, as the Darcy model automatically uses a smaller effective consolidation coefficient below the preconsolidation stress, compared to the input value for virgin loading. The effect of unloading on the excess head is clearly visible.

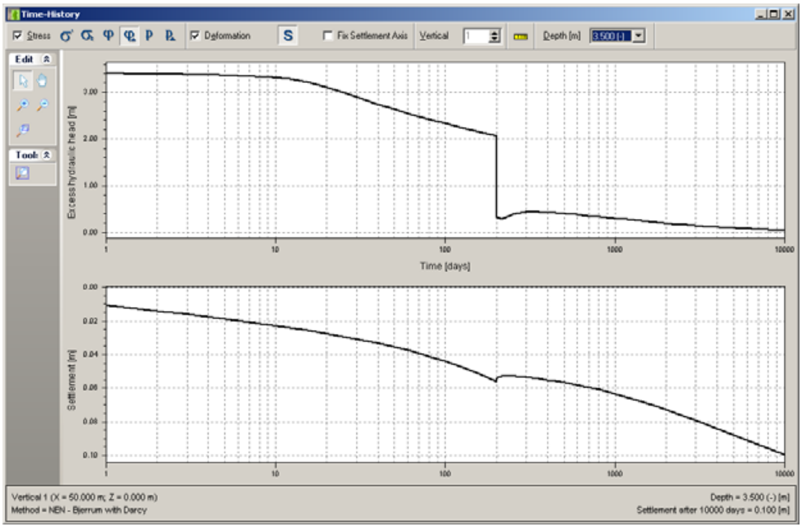


Figure 3-19 – *Time-History* window, Excess hydraulic head at depth 3.5 m

52. Try selecting different stress components at different depths. The development of effective stress in the drained sand layer for example, shows the effect of the submerging of the top layer due to settlement in time, leading to a gradually reducing effective weight.

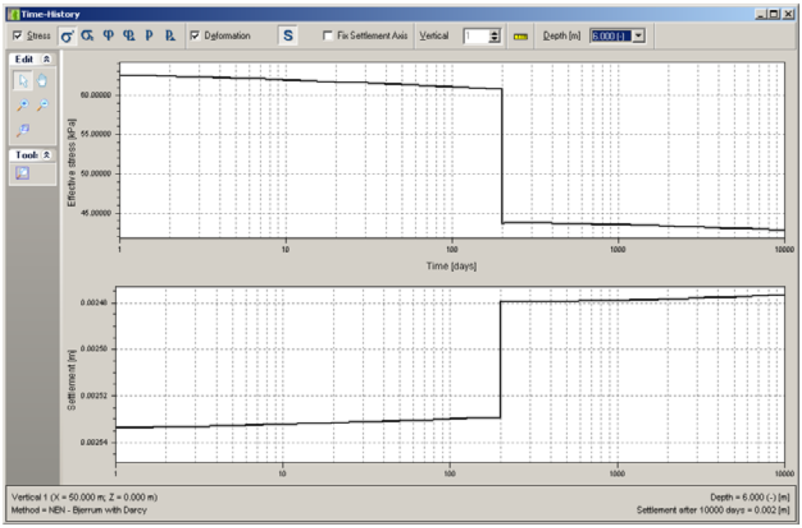


Figure 3-20 – *Time-History* window, Effective stress in the drained pleistocene sand, gradually decreasing by submerging of the top layer

3.9.2 Depth-History

53. Choose the *Depth-History* option in the Results menu. Select different stress components and browse through the stress distribution at different times by using the mouse scroll wheel, after clicking the *Depth* selection box. Figure 3-21 shows for example the excess head distribution before and directly after unloading at time is 200 days. Try also selecting different stress components at different times. MSettle always plots the values along the depth at their original location. The hydrostatic pore pressure contribution at a certain location will therefore increase by the settlement of that location.

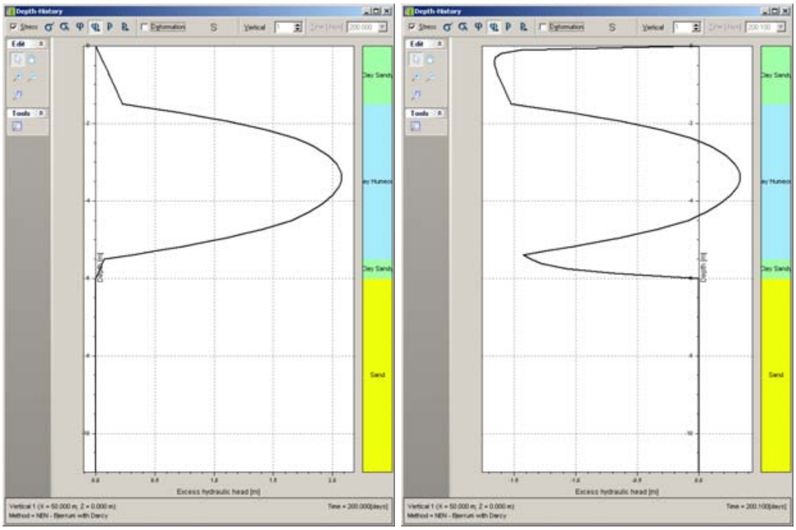


Figure 3-21 – *Depth-History* window, Excess head before and after unloading

3.9.3 Residual Settlement

54. Choose the *Residual Settlement* option in the *Results* menu. MSettle will present a graph with the settlement between a certain start time and the end time of the analyis (10000 days).

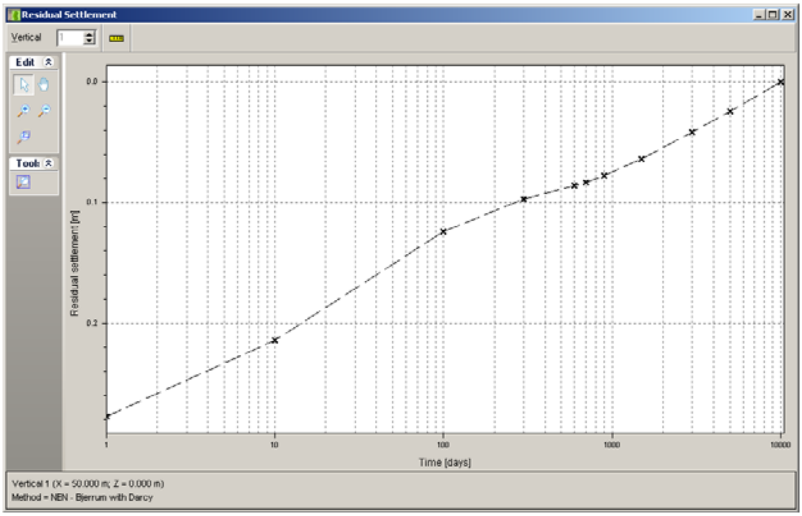


Figure 3-22 – *Residual Settlement* window

3.10 Influence of submerging

55. Choose *Save as* from the *File* menu, and create a copy of the input file with name <Tutorial-1b>.
56. Choose *Options* from the *Calculation* menu, and unmark the *Submerging* option.
57. Click *OK* to confirm.

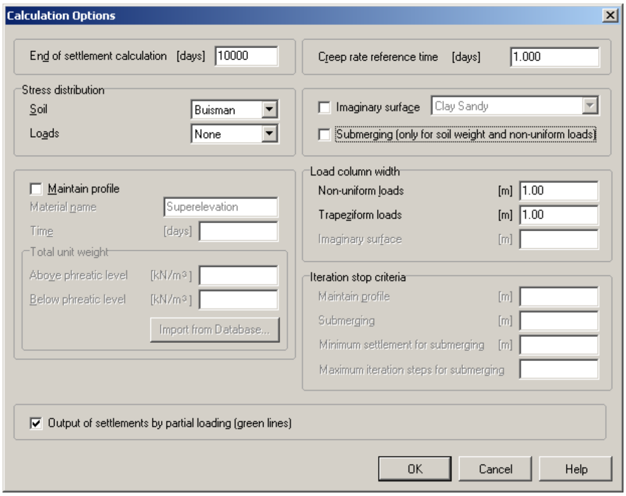


Figure 3-23 – *Calculation Options* window

58. Start the calculation, by choosing *Start* from the *Calculation* menu and then clicking *Start*.
59. After the calculation has finished, choose *Time History* from the *Results* menu and view the graph of the settlements versus time (Figure 3-24). Apparently, the submerging of the top layer reduces the final settlement from 0.381 meters to 0.343 meters.

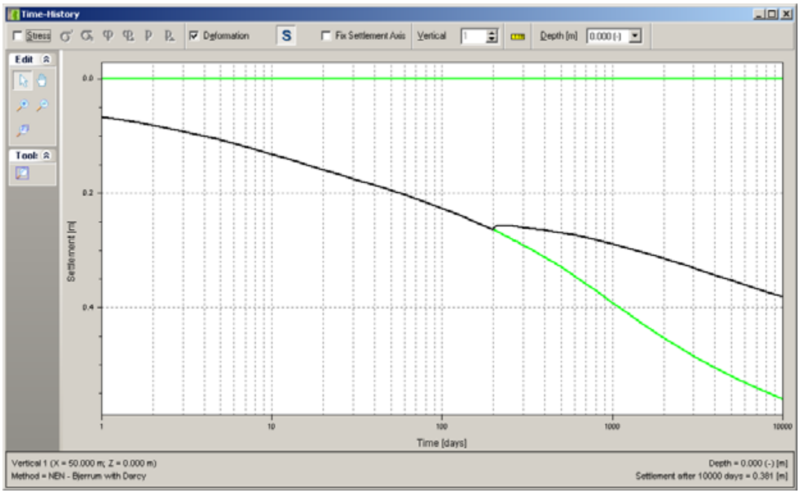


Figure 3-24 – *Time-History* window, Surface settlement with submerging switched off (Tutorial-1b)

3.11 Comparison of consolidation models

To illustrate the influence of the consolidation, two other calculations are performed:

- [§ 3.11.1] Using Terzaghi consolidation model (Tutorial-1c);
- [§ 3.11.2] Using drained layers (Tutorial-1d).

3.11.1 Terzaghi consolidation

Perform the following steps to compare the results from the Darcy model (with submerging switch off) with the result from the approximate Terzaghi model.

60. Choose *Model* from the *Project* menu, and select the *Terzaghi* consolidation model. Click *OK* to confirm.

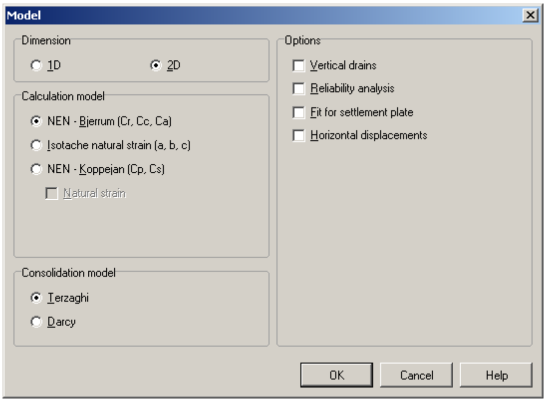


Figure 3-25 – *Model* window

61. Choose *Save as* from the *File* menu, and create a copy of the input file with name <Tutorial-1c>.
62. Choose *Calculation* from the *Project* menu, and click *Start*.
63. After the calculation, select *Time-History* from the *Results* menu (Figure 3-26).

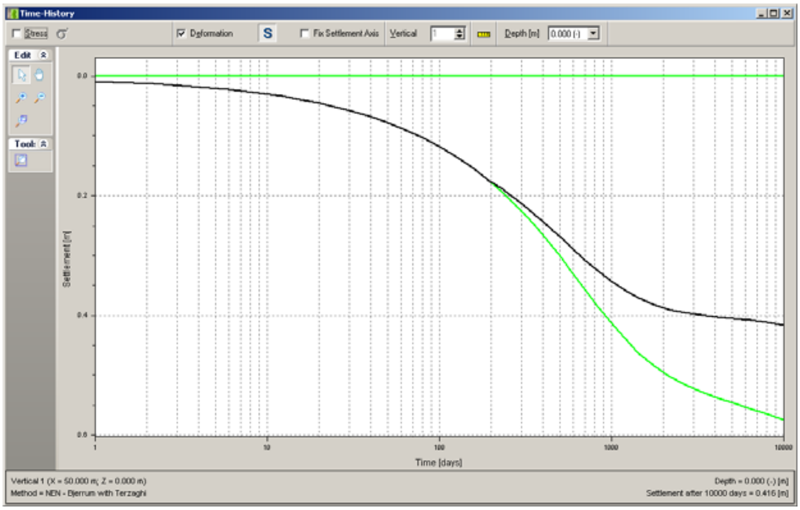


Figure 3-26 – *Time-History* window, Surface settlement for Terzaghi model and no submerging (Tutorial-1c)

64. Click the right-hand mouse button in the *Settlement* graph and select *View Data*, to view the numerical data in the *Chart Data* window (Figure 3-27). The predicted residual settlement between 600 days and 10000 days is now 0.416 - 0.287 = 0.129 m.

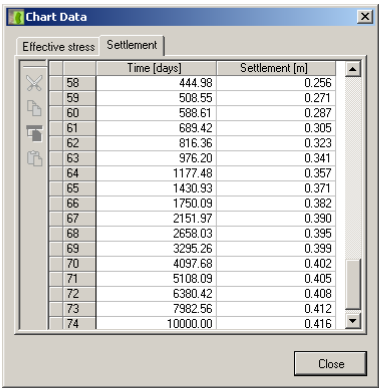


Figure 3-27 – *Chart Data* window, Surface settlement versus Time (Tutorial-1c)

Figure 3-24 (Tutorial-1b) and Figure 3-26 (Tutorial-1c) illustrate the differences between respectively the Darcy and the Terzaghi model. Both results are presented in the same graph in Figure 3-30. The Terzaghi solution consolidates considerably slower in the early stage of loading and after unloading. The reason is that the Terzaghi model simply multiplies the settlements from a drained solution with a "Degree of consolidation". The Terzaghi model therefore does not take into account the influence of the pore pressure development on the effective stress and also assumes the same consolidation period during virgin loading and during un/reloading.

To view the development of the degree of consolidation according to the Terzaghi model:

65. Select *Dissipations* from the *Results* menu.
66. In the drop-down menu at the left top of the window, select <Clay Organic> (Figure 3-28).

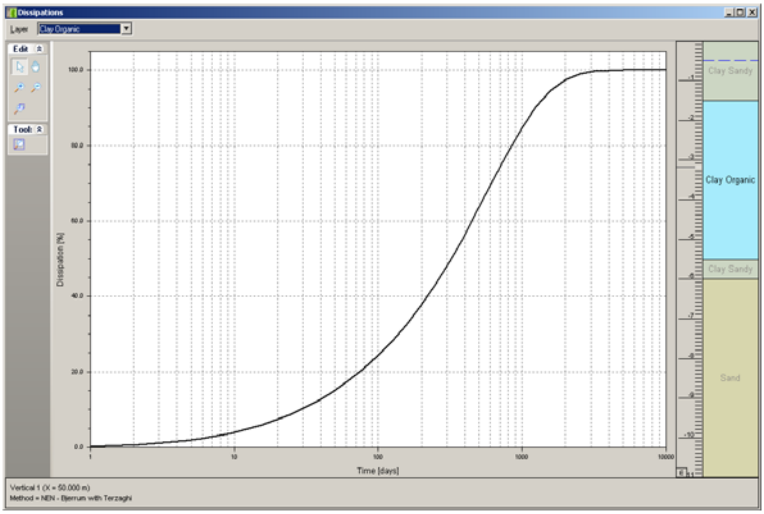


Figure 3-28 – *Dissipations* window, Degree of consolidation versus Time in *Clay Organic* layer for Terzaghi model and no submerging (Tutorial-1c)

3.11.2 Drained behaviour

67. Choose *Save as* from the *File* menu, and create a copy of the input file with name <Tutorial-1d>.
68. To view the drained solution, change the behavior of all layers to *Drained* in the *Materials* window, and run another calculation. Note that the final settlements from the drained solution are indeed exactly equal to the final settlements from the solution using Terzaghi consolidation.

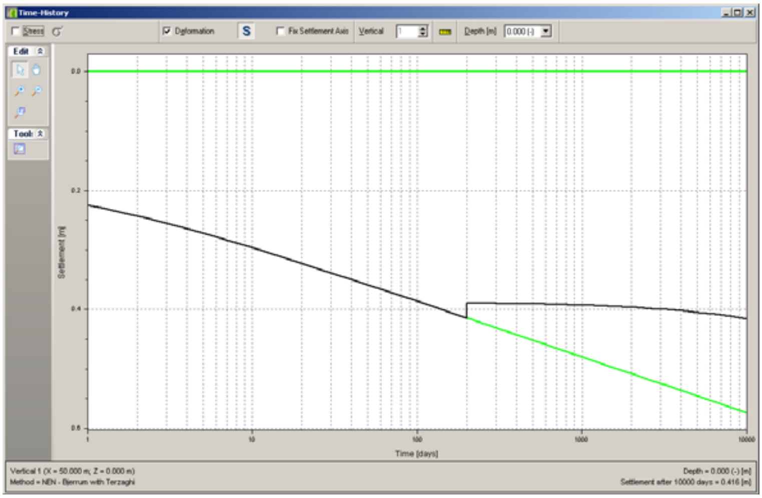


Figure 3-29 – *Time-History* window, Surface Settlements using Drained layers and no submerging (Tutorial-1d)

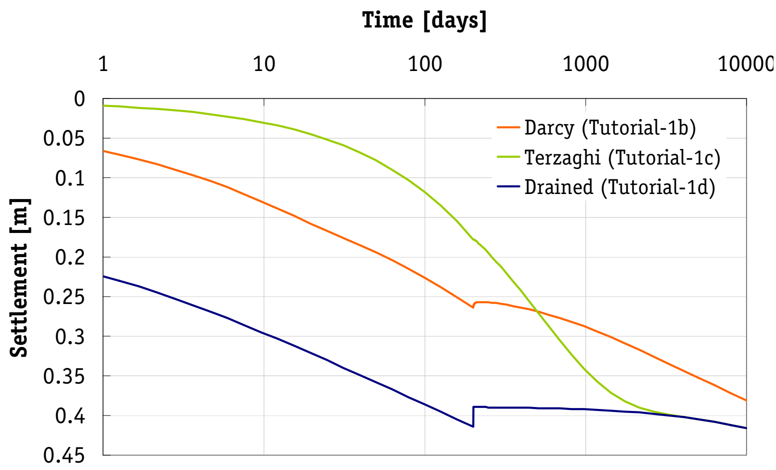


Figure 3-30 – Surface Settlements compared (no submerging)

3.12 Influence of initial overconsolidation

A well-known characteristic of soft soil is that primary and secondary (creep) deformation are larger after passing the initial vertical preconsolidation stress. This initial preconsolidation stress is in general above the field stress, due to the overconsolidation by creep and/or preloading in the past. Input of initial overconsolidation is usually done via either a *POP* value (the difference between preconsolidation stress and field stress) or via the *OCR* (the ratio between the preconsolidation stress and the field stress). Direct input of the preconsolidation stress is also possible. According to the isotache theory, the initial overconsolidation ratio affects the initial creep strain rate, expressed by $\frac{C_\alpha}{\ln(10)\, t_{equivalent}}$. The equivalent age ($t_{equivalent}$) in this expression is the theoretical soil age if the preconsolidation would have been caused completely by (secondary) creep, after a preceding virgin loading. In the *Materials* window, MSettle will show the corresponding input value of the equivalent age after input of OCR and vice versa.

69. Open <Tutorial-1b.sli> and save it as <Tutorial-1e> to switch back to the *Darcy* model with the *Submerging* option still switched off.
70. Choose *Materials* from the *Soil* menu, and enter the value of <200> days for the Equivalent age of both *Clay Sandy* and *Clay Organic*. After input of each age value, use the TAB key to view the corresponding *OCR* value. Click *OK* to confirm.

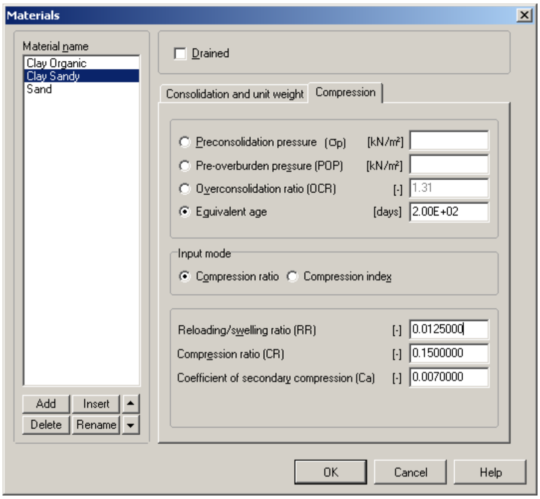


Figure 3-31 – *Materials* window with reduced OCR (Tutorial-1e)

71. Start the calculation, by choosing *Start* from the *Calculation* menu and then clicking *Start*. After the calculation has finished, choose *Time History* from the *Results* menu and view the graph of the settlements versus time (Figure 3-32).

Figure 3-33 illustrates that the settlements are significantly increased as a result of the OCR reduction.

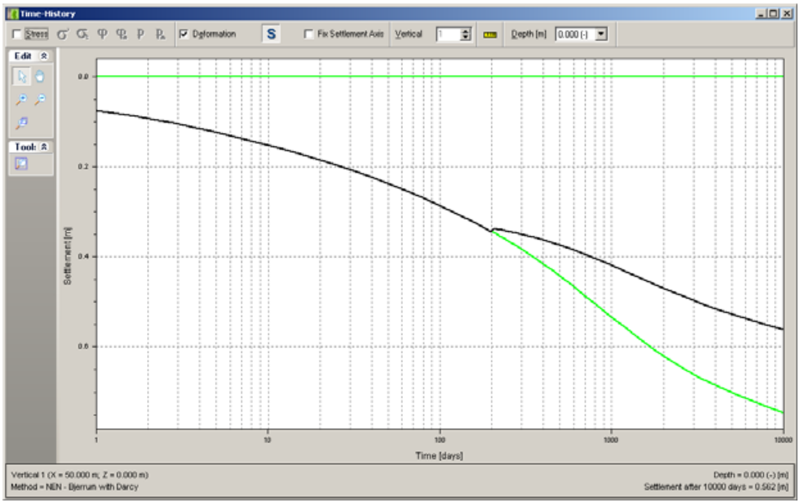


Figure 3-32 – *Time-History* window, Surface settlement with reduced OCR (Tutorial-1e)

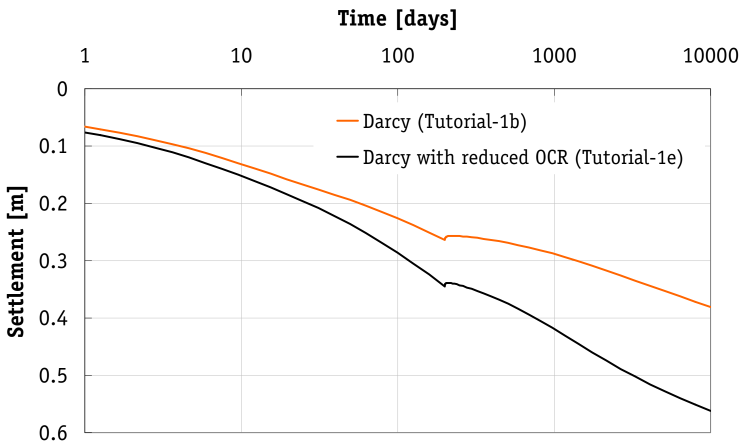


Figure 3-33 – Surface Settlements compared (no submerging)

72. Click the *Excess hydraulic head* icon  and change the *Depth* to <3.5 m> to view the excess head versus time at a depth of -3.5 meters (Figure 3-34). Note that the excess head now even increases slightly directly after the initial undrained response, before starting to dissipate. The reason of this additional excess head development is the large initial creep rate of the *Clay Organic* layer, in combination with its thickness and low permeability.

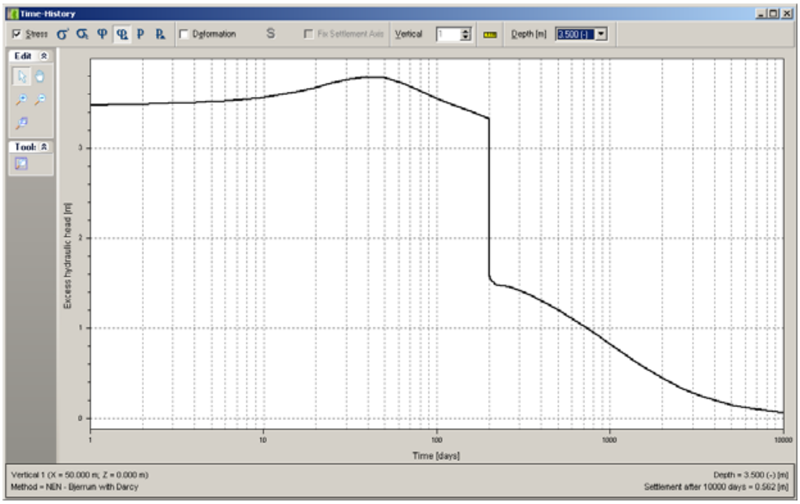


Figure 3-34 – *Time-History* window, Excess head (at depth 3.5 m) with reduced OCR (Tutorial-1e)

4

Tutorial 2: Embankment design with vertical drains

This is the first tutorial in a sequence of two on the construction of a high embankment for the Dutch A2 highway, at a viaduct crossing the N201 road nearby Vinkeveen.

This part illustrates the usage of the following MSettle features for embankment design and vertical strip drains, without and with enforced dewatering:

- The automatic determination of the required total soil raise by input of the final design level in combination with the settlement-dependent Maintain Profile load;
- Input of regular vertical strip drains, to speed up the consolidation process;
- The approximately allowed speed of loading, based on the required degree of consolidation for achieving the minimally required stability factor;
- Simplified input of loading stages at certain times, with the generate loads option;
- The determination of the needed additional temporary preloading and its duration, related to the requirements on the residual settlements;
- Input of enforced dewatering in combination with strip drains for the purpose of preloading;
- The determination of horizontal displacements according to De Leeuw theory;
- The determination of bandwidth in total and residual settlements from a reliability analysis.

The following MSettle modules are needed:

- MSettle (1D model with Terzaghi)
- 2D geometry model
- Darcy consolidation model
- Vertical drains module
- Horizontal displacements module

- Reliability module.

This tutorial is presented in the files Tutorial-2a.sli to Tutorial-2g.sli.

4.1 Introduction

The considered embankment has been constructed for a viaduct crossing of the Dutch A2 highway with the N201 road nearby Vinkeveen. The soft subsoil consists of approximately 5.5 m of peat, with a clay layer of 0.5 m on top. The initial surface level resides at approximately RL-1.85 m (RL = reference level) and the phreatic level resides at RL-2.2 m. The design level of the completed embankment at the time of delivery (1000 days) is at 6 m RL. The base width is 103 m and the top width is 32 m. See also the geometry in Figure 4-1.
The totally available embankment construction period is 840 days. The residual settlements after 900 days are not allowed to exceed 15 cm.

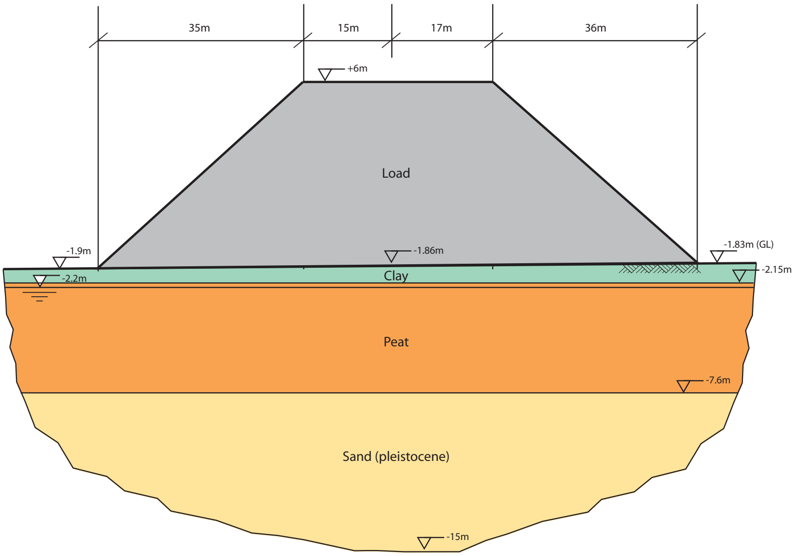


Figure 4-1 – Embankment geometry (Tutorials 2 and 3)

The soil properties for sand, peat and clay are given in respectively Table 4-1, Table 4-2 and Table 4-3.

Available from the lab were Koppejan parameters from 21 peat tests and 3 clay tests. The NEN-Bjerrum parameters have been derived from the Koppejan parameters for each oedometer test, using the conversion formulas (82) to (84), on page 316. The parameters for the a/b/c isotache model were then derived from the NEN-Bjerrum parameters for each oedometer test, using formulas (85) to (87), at the last but one stress level in the test.

The standard deviation of the local average, which is additional input for bandwidth determination, has been estimated by equation (1), assuming that 75 % of the natural variance within a layer occurs within one vertical.

$$S_{local} = \sqrt{\left(\frac{1}{N} + 0.25\right)\left(\frac{t_{0.975}}{u_{0.975}} S_{statistical}\right)^2} \tag{1}$$

where:

N	Number of samples
$S_{statistical}$	Statistical standard deviation
S_{local}	Approximated standard deviation of the local average
$t_{0.975}$	Distance t in a Student-t distribution at exceeding probability 2.5 %
$u_{0.975}$	Distance u in a Standard Normal distribution at exceeding probability 2.5 %.

Note that the compressibility for reloading and swelling is relatively high compared to the compressibility for virgin loading. This is because the reloading compressibility was determined in the lab from the branch below the initial preconsolidation stress, instead of using a separate unloading/reloading branch.

Table 4-1 – Sand properties (Tutorial 2)

Parameter		Unit	Mean
Sat. unit weight	γ_{sat}	[kN/m^3]	20
Unsat. unit weight	γ_{unsat}	[kN/m^3]	17
Consolidation coefficient	C_v	[10^{-8} m^2/s]	Drained
Ratio hor./vert. consolid. coeff.	C_h/C_v	[-]	-
Pre-overburden pressure	POP	[kN/m^2]	0
NEN-Koppejan parameters	C_p	[-]	10^9
	C_p'	[-]	10^9
	C_s	[-]	10^9
	C_s'	[-]	10^9
NEN-Bjerrum Isotache parameters (linear strain)	$RR = C_r/(1+e_0)$	[-]	0.0001
	$CR = C_c/(1+e_0)$	[-]	0.0023
	C_α	[-]	0
abc Isotache parameters (natural strain)	a	[-]	10^{-6}
	b	[-]	10^{-5}
	c	[-]	0

Table 4-2 – Peat properties (Tutorial 2)

Parameter	Unit	Mean	Standard Deviation	
			Statistic	Local average
γ_{sat}	[kN/m^3]	10.15	0.435	0.246
γ_{unsat}	[kN/m^3]	10.15	0.435	0.246
C_v	[10^{-8} m^2/s]	30.5	29.42	16.65
C_h/C_v	[-]	1	-	-
POP	[kN/m^2]	7.88	4.50	2.55
NEN-Koppejan:				
C_p	[-]	13.8	4.983	2.821
C_p'	[-]	5.95	1.483	0.840
C_s	[-]	10^9	-	-
C_s'	[-]	43.8	2.35	6.988
NEN-Bjerrum Isotache (linear strain):				
$RR = C_r/(1+e_0)$	[-]	0.1860	0.0558	0.0316
$CR = C_c/(1+e_0)$	[-]	0.409	0.074	0.0418
C_α	[-]	0.0312	0.006	0.0034
abc Isotache (natural strain):				
a	[-]	0.08517	0.027	0.015
b	[-]	0.2259	0.057	0.032
c	[-]	0.02126	0.006	0.003

Table 4-3 – Clay properties (Tutorial 2)

Parameter	Unit	Mean	Standard Deviation	
			Statistic	Local average
γ_{sat}	[kN/m^3]	13.94	0.588	0.985
γ_{unsat}	[kN/m^3]	13.94	0.588	0.985
C_v	[10^{-8} m^2/s]	2.47	2.02	3.38
C_h/C_v	[-]	1	-	-
POP	[kN/m^2]	5.12	2.05	3.44
NEN-Koppejan:				
C_p	[-]	18	4	6.71
C_p'	[-]	10	1.5	0.8
C_s	[-]	10^9	-	-
C_s'	[-]	54	17.4	29.2
NEN-Bjerrum Isotache (linear strain):				
$RR = C_r/(1+e_0)$	[-]	0.132	0.03	0.05
$CR = C_c/(1+e_0)$	[-]	0.237	-	0.06*
C_α	[-]	0.0262	0.006	0.011
abc Isotache (natural strain):				
a	[-]	0.05804	0.013	0.023
b	[-]	0.1096	-	0.02*
c	[-]	0.01363	0.0036	0.006

*Estimated, due to limited number of samples

4.2 Initial embankment design (Tutorial-2a)

The input of layers boundaries, piezometric lines, phreatic line and soil parameters have already been described in Tutorial 1 [chapter 3]. This section will describe all additional steps to determine:

- the required soil raise to arrive at the design level after settlement, using the *Maintain Profile* option;
- the approximately allowed speed of loading without and with vertical drains, by coupling to an MStab stability analysis.

1. In the *Open* window from the *File* menu, select <Tutorial-2.sli> from the *Examples* directory where the MSettle program was installed.
2. Save it as <Tutorial-2a>.

The *View Input* window (Figure 4-2) shows top-down the clay and peat layer. A drained sand layer has been added at the base, for the purpose of a coupled stability analysis.

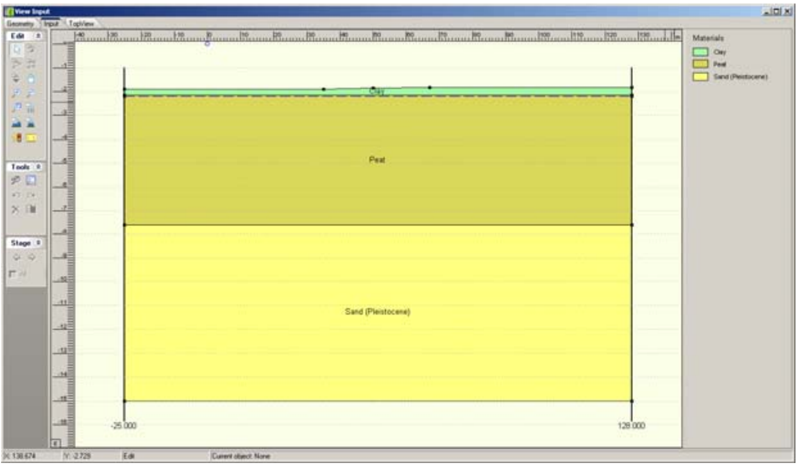


Figure 4-2 – *View Input* window, *Input* tab showing the soil layers

3. Open the *Non-Uniform Loads* window from the *Loads* menu.
4. Click *Add* to add a single load <Final Load>, and then enter the embankment profile co-ordinates, according to Figure 4-3. Also enter the unit weight above <18> and below <20> phreatic level as well as the time of loading <1>. Click *OK* to confirm.

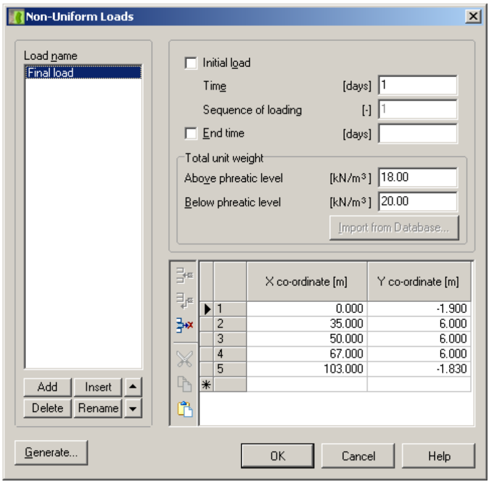


Figure 4-3 – *Non-Uniform Loads* window

5. Open the *Options* window from the *Calculation* menu, and mark the *Maintain Profile* checkbox. Enter day <1> as the start time for the additional load that will depend on the final settlement. Also enter the unit weight above <18> and below <20> phreatic level.
6. Click *OK* to confirm.

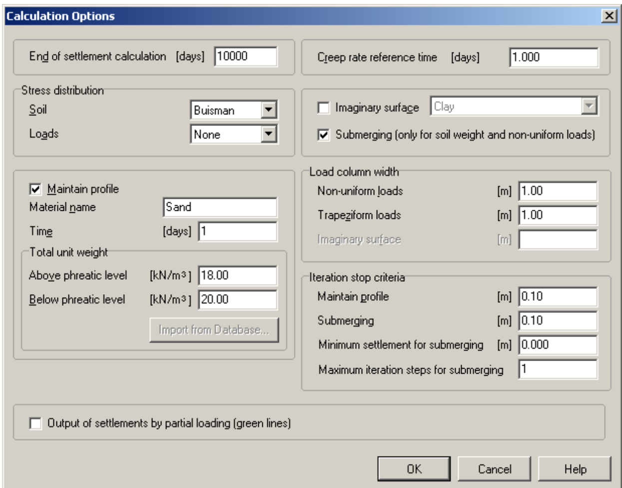


Figure 4-4 – *Calculation Options* window

7. Open the *Verticals window* from the *GeoObjects* menu.
8. Click *Generate* to generate verticals at all different horizontal positions of the nodes. MSettle will calculate the settlements in each of these verticals, and also use the settlements to update the geometry before export to a stability analysis.

9. Click *OK* to confirm.

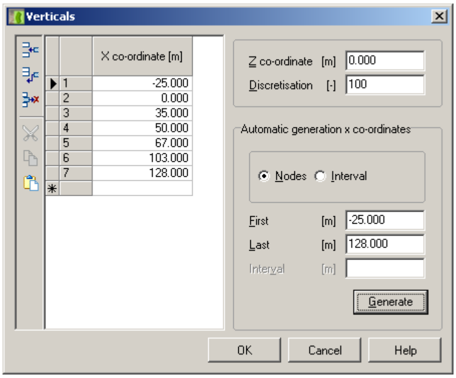


Figure 4-5 – *Verticals* window

The result (Figure 4-6) shows that vertical 4 is located in the centre of the embankment.

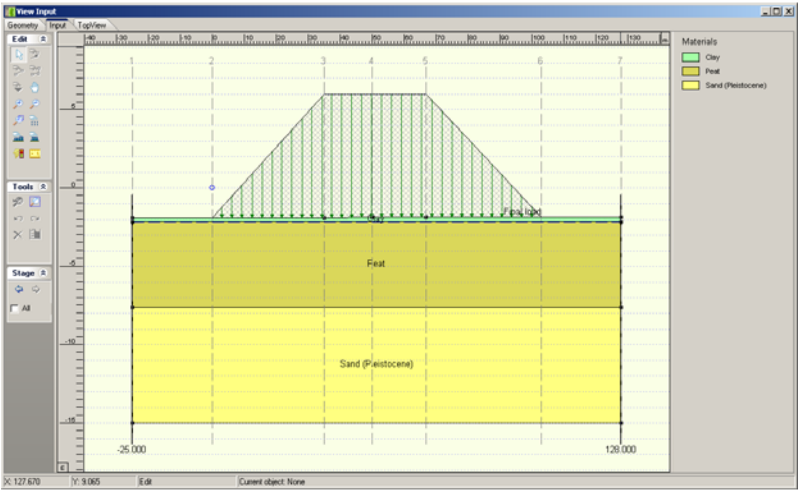


Figure 4-6 – *View Input* window, *Input* tab showing the generated verticals (Tutorial 2)

10. Open the *Start Calculation* window from the *Calculation* menu and click *Start*. MSettle will iteratively increase the load at 1 day, to arrive at an embankment top level of RL +6 m after 10000 days.
11. Open the *Time-History* window from the *Results* menu after the calculation has finished.
12. Select *Vertical* number <4> at the top of the window to view the settlements and effective stresses in vertical 4 at the subsoil surface level (Figure 4-7). The

reduction of effective stress at the subsoil surface level in time is caused by submerging. The final settlement by the *Maintain Profile* load is 3.672 m at 10000 days.

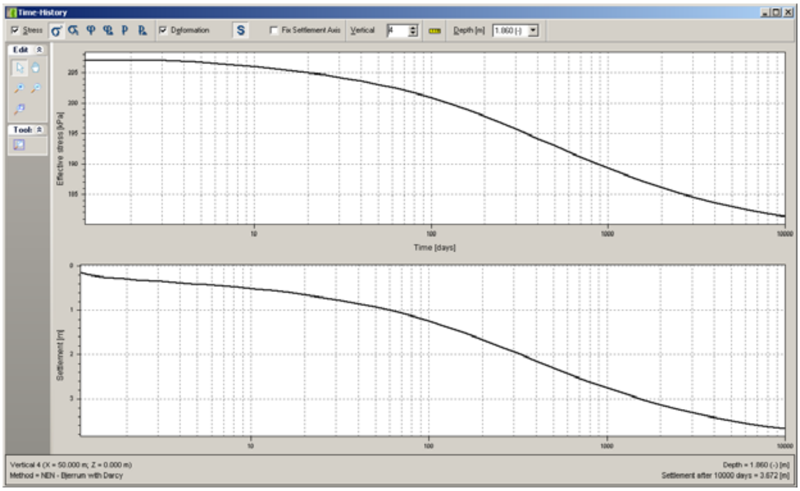


Figure 4-7 – *Time-History* window, Natural consolidation: Settlement and Effective stress vs. Time in vertical 4 (Tutorial 2a)

13. Click the *Excess hydraulic head* icon , and change the *Depth* to <-4.875 m> to view the excess head development in vertical 4, at a depth of RL -4.875 m (Figure 4-8). It is clear that drainage is required to speed up the consolidation process.

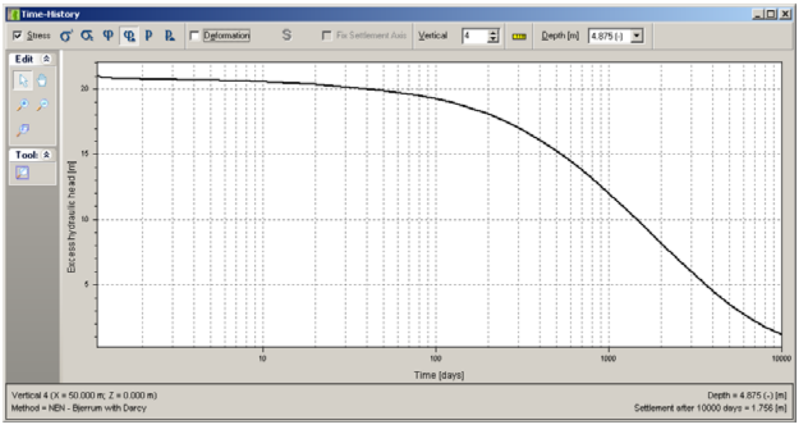


Figure 4-8 – *Time-History* window, Natural consolidation: Excess head vs. Time in vertical 4 at RL-4.875m (Tutorial 2a)

14. Finally, view the greenfield settlement in vertical 1 by selecting *Vertical* number <1> (Figure 4-9): approximately 0.08m in 10000 days. Greenfield settlements are part of the isotache concept (NEN-Bjerrum and a/b/c), and depend on the coefficient of secondary settlement and the initial equivalent age.

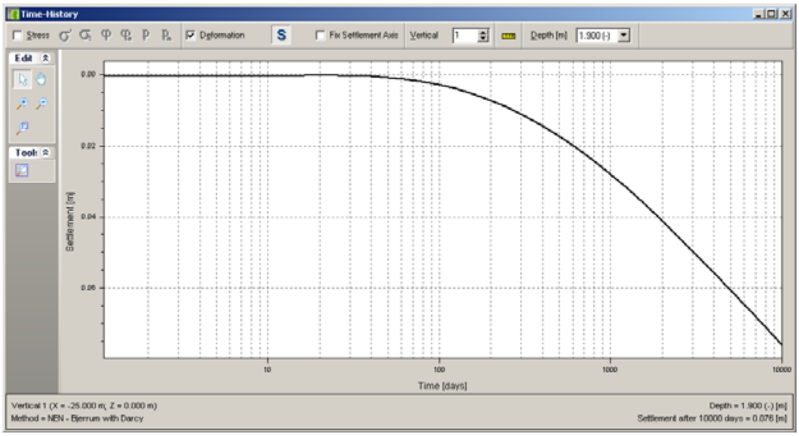


Figure 4-9 – *Time-History* window, Greenfield settlement in vertical 1 (Tutorial 2a)

4.3 Acceleration of the consolidation process by means of vertical drains (Tutorial-2b)

As shown in Figure 4-8, drainage is required to speed up the consolidation process.

4.3.1 Vertical Drains

15. Open the *Save As* window and save the current project as <Tutorial-2b>.
16. Open the *Model* window from the *Project* menu and select *Vertical drains*. Click *OK* to confirm.

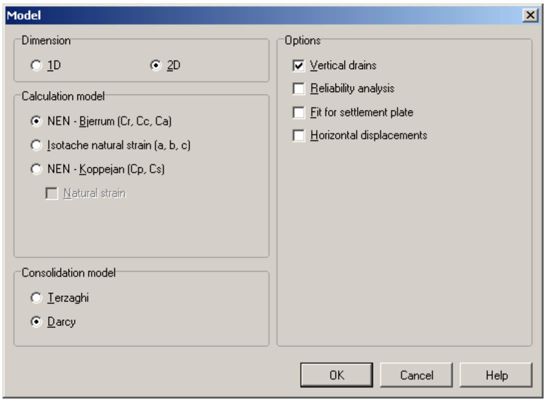


Figure 4-10 – *Model* window, Select vertical drain option (Tutorial-2b)

17. Open the *Vertical Drains* window from the *GeoObjects* menu. Note that the default drain type is a strip, with regular dimensions and a triangular spacing of 1 m.
18. Enter a bottom position of RL -7.5 m (close to the top of the sand layer) and narrow the initial *Horizontal Range* to match the two sides of the embankment base, from <0 m> to <103 m>.
19. Click *OK* to confirm.

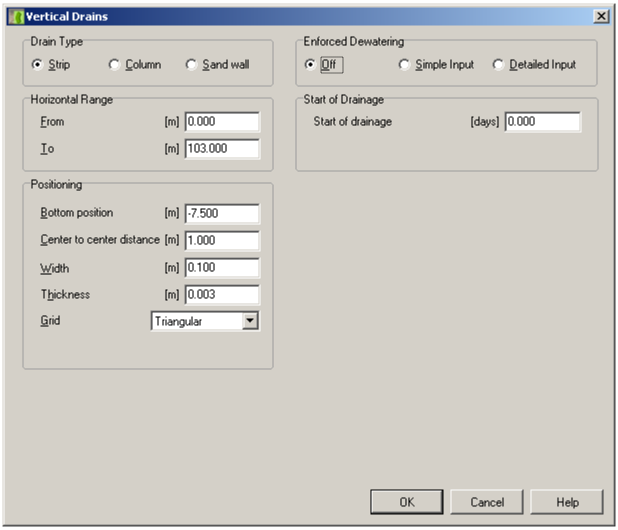


Figure 4-11 – *Vertical Drains* window (Tutorial-2b)

See *Vertical Drains* [§ 9.4.2] for a detailed description of this window.

4.3.2 Time-History results

20. Again open the *Start Calculation* window from the *Calculation* menu, and click *Start*.
21. After the calculation has finished, open the *Time-History* window from the *Results* menu. Select *Vertical* number <4> to view the settlements and effective stresses in vertical 4 at the subsoil surface level (Figure 4-12). The final settlement by the *Maintain Profile* load is now 3.775 m at 10000 days.

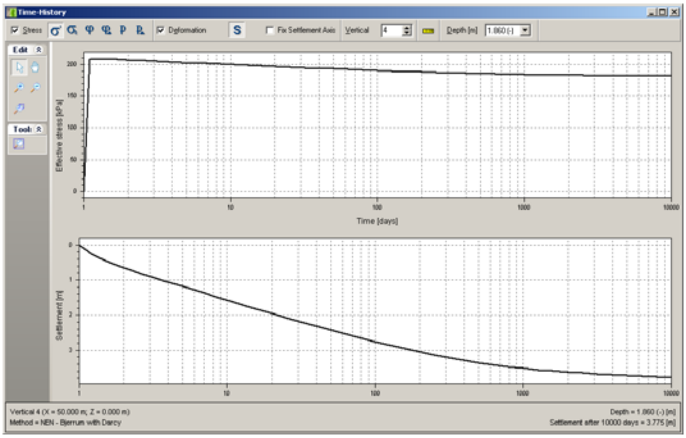


Figure 4-12 – *Time-History* window, Consolidation with vertical drains: Settlement and Effective stress vs. Time in vertical 4 (Tutorial-2b)

22. Click the *Excess hydraulic head* icon and change the *Depth* to <-4.875 m> to view the excess head development in vertical 4, at a depth of RL -4.875 m. The reduction of the consolidation period by the vertical drains is clearly visible.

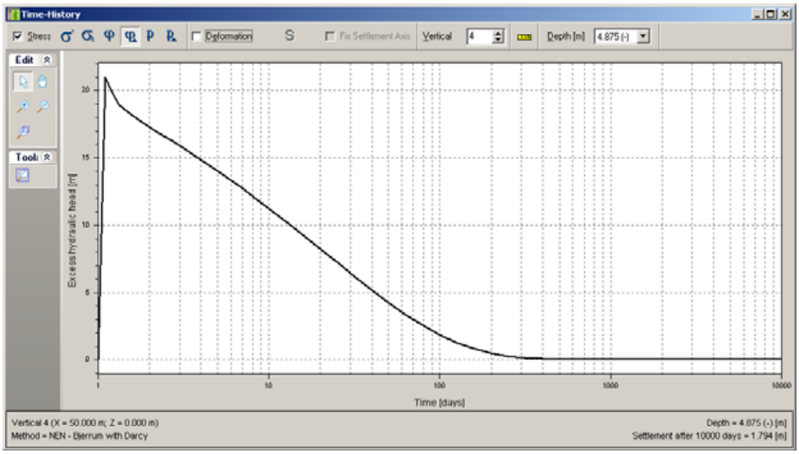


Figure 4-13 – *Time-History* window, Consolidation with vertical drains: Excess head vs. Time in vertical 4 at RL -4.875 m (Tutorial-2b)

4.3.3 Stability analysis with MStab

A coupled stability analysis of the total embankment raise at 50% of the final settlement will now be used for a quick approximation of the allowed rate of loading.

23. Open the *Write MStab Input File* window from the *Results* menu, and enter the input according to Figure 4-14. Select the *Add superelevation* option for addition of the special *Maintain Profile* load to the geometry.
24. Click *OK* and accept the default file name <Tutorial-2bAt50percent>.

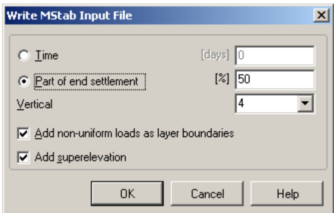


Figure 4-14 – *Write MStab Input File* window (Tutorial-2b)

When using MStab, this MStab input file can be opened, strength properties and grid can be added, and a stability analysis can be performed. The following steps describe how to perform the stability with the MStab program. However, if the access to this program is not possible, results can be directly seen in Figure 4-18.

25. Open the generated input file with MStab (Figure 4-15).

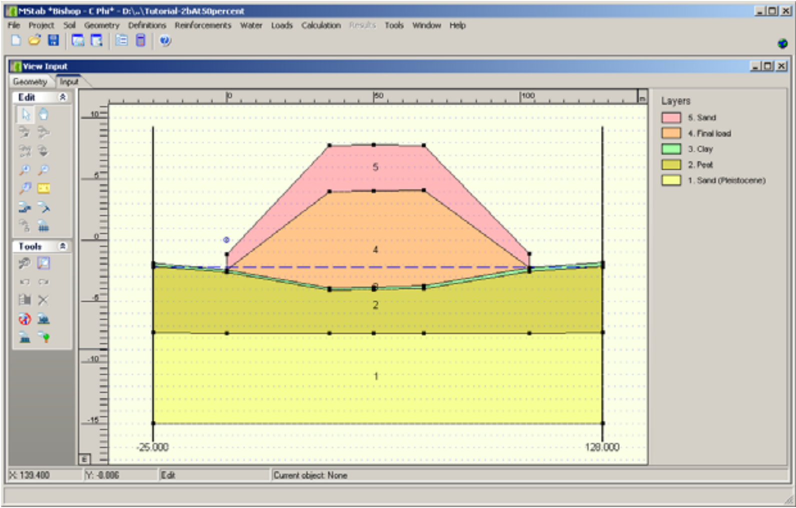


Figure 4-15 – *MStab View Input* window (Tutorial-2b)

26. In the *Materials* window from the *Soil* menu, add the cohesion and friction angle values for sand (<0>, <33>), peat (<7>, <25>) and clay (<2>, <29>).

NOTE: If the soil properties in the MSettle calculation were derived from an MGeobase database, then the strength properties will be already filled in the MStab input file.

27. Also add a slip circle range according to Figure 4-15 in the *Slip Circle Definition* window from the *Definitions* menu.

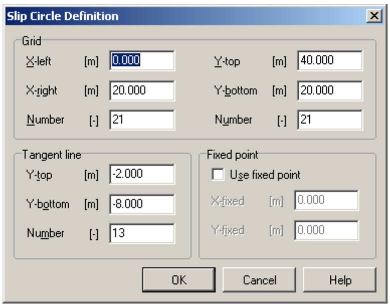


Figure 4-16 – *MStab Slip Circle Definition* window (Tutorial-2b)

The following step is to determine the required degree of consolidation in the *Clay* and *Peat* layers (layer 3 and 2) after addition of the embankment (layers 4 and 5), for a stability factor of 1.1 or more. This is done by trial and error.

28. Enter a trial value for the degree of consolidation (equal for clay and peat for simplicity reasons) via the *Degree of Consolation* window from the *Water* menu.

> Note that the generated input by MSettle already contains initial values, following from the calculated heads in time. Select *Start* from the *Calculation* menu to determine the associated stability factor.

After a few cycles, it will prove that the required stability factor is reached for a degree of consolidation larger than 45% (Figure 4-17) as the resulting stability factor is 1.11 (Figure 4-18).

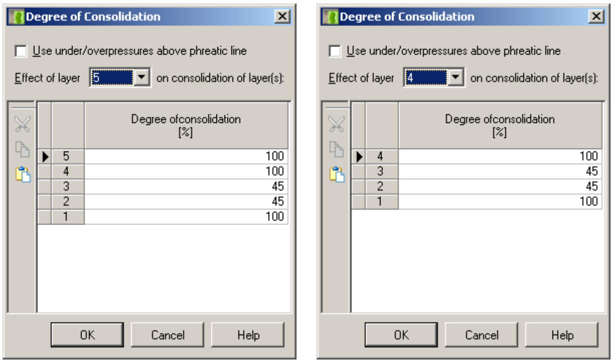


Figure 4-17 – *MStab Degree of Consolidation* window, manual input (Tutorial-2b)

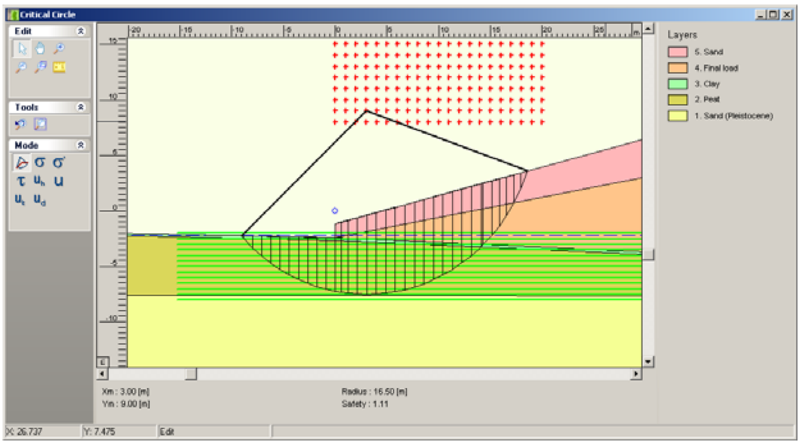


Figure 4-18 – MStab slip circle result (Tutorial-2b)

4.3.4 Dissipations results

As a rule of thumb, the minimum period for stable staged construction to the final height is twice the period needed for sufficient stability at 50% settlement after a one-off raise. During the previous step was shown that the stability in this case is sufficiently large at a 45% degree of consolidation. MSettle offers a convenient design graph of the degree of consolidation versus time, to find the associated time period.

29. Mark the *Add dissipation calculation* checkbox in the *Start Calculation* window and select *Vertical* <4: 50.000 m> (Figure 4-19) and click *Start* to create the dissipation graph.

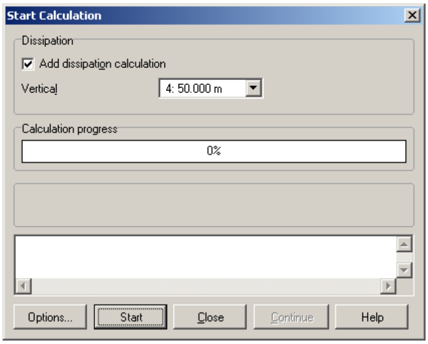


Figure 4-19 – *Start Calculation* window (Tutorial-2b)

30. Open the *Dissipations* window from the *Results* menu and select <Peat> from the drop-down menu (Figure 4-20).
31. Right click in the graph area (*Results/Dissipations)* to view the data numerically. Check that the 45% consolidation period is about 10 days for the initial drain distance (1 m). The total soil raise follows from the preceding *Maintain Profile* calculation (Figure 4-12) and is 7.86 + 3.78 = 11.64 m (7.86 m being the height of the *Final load* at vertical 4, see Figure 4-3). The approximately allowed rate of loading is therefore 0.5 × 11.64 m/10 days = 0.582 m/day.

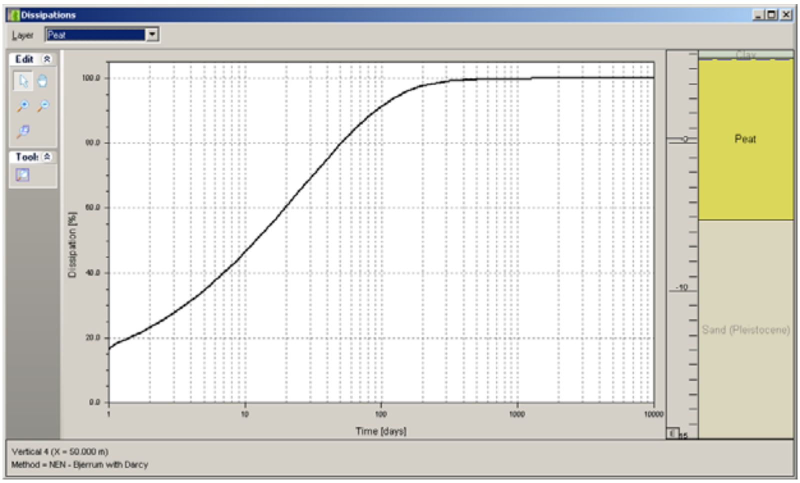


Figure 4-20 – *Dissipations* window, Degree of consolidation vs. Time in *Peat* at vertical 4, for grid distance 1 m (Tutorial-2b)

32. Determine the allowed rate also for other drain distances, by performing a new calculation after altering the *Center to center distance* input in the *Vertical Drains* window (*GeoObjects* menu). The allowed rate for a drain distance of 2 m is for example 0.5 × 11.64/50 = 0.116 m/day (Figure 4-21).

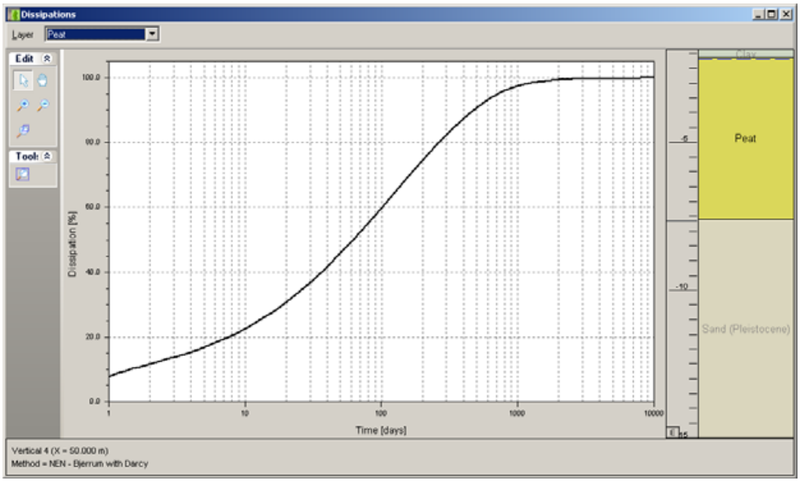


Figure 4-21 – *Dissipations* window, Degree of consolidation vs. Time in *Peat* at vertical 4, for grid distance 2 m (Tutorial-2b)

4.4 Staged loading (Tutorial-2c)

This section describes the input of staged loading and the subsequent calculation of the resulting (residual) settlements, using a triangular grid of strip drains. Starting point is the input with drains and loading as described in the previous section [§ 4.3]. The addition of temporary preloading and dewatering will be discussed in the next sections.

A period of 20 weeks in combination with 8 construction stages is chosen to raise the embankment to a final height of approximately 11.6 m above subsoil (Figure 4-22). This includes the construction of a working floor with a thickness of 1 m in the first stage.

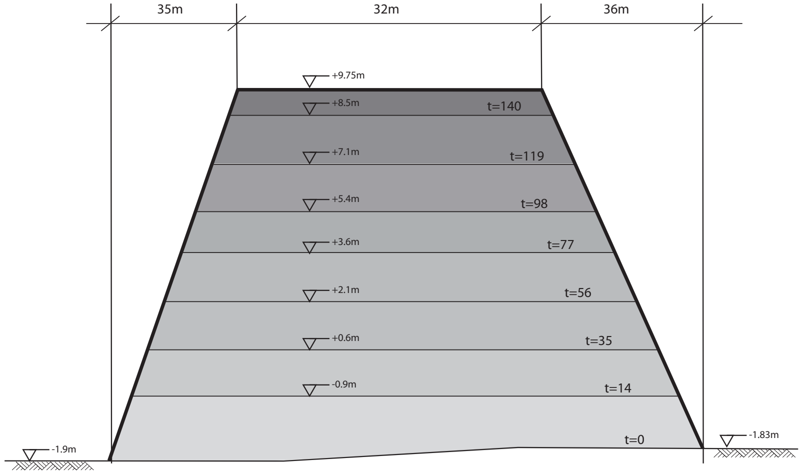


Figure 4-22 – 8-staged loading (Tutorial-2c)

33. Open the *Save As* window and save the current project (with a grid distance of 2 m) as <Tutorial-2b>.
34. Open the *Non-Uniform Loads* window from the *Loads* menu and remove the previously defined loading using the *Delete* button. Then click *Generate,* and enter the profile and stages according to Figure 4-23. Click *OK* to confirm.

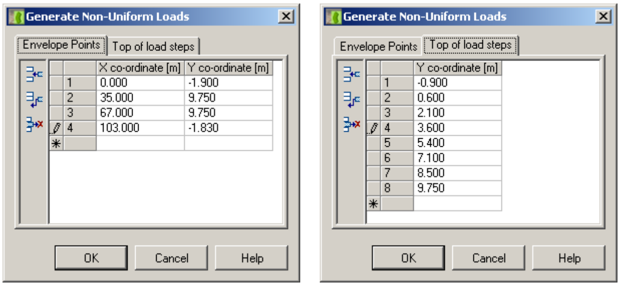


Figure 4-23 – *Generate Non-Uniform Loads* window (Tutorial-2c)

35. In the *Non-Uniform Loads* window, remove the abundant *Final load*. For each of the generated loads: add a unit weight *Above* and *Below phreatic surface* of respectively <18> and <20> and a *Time* of application of <0>, <14>, <35>, 56>, <77>, <98>, <119> and <140> days from *Generate load (1)* to *Generate load (8)*. The input for the last loading is shown in Figure 4-24.

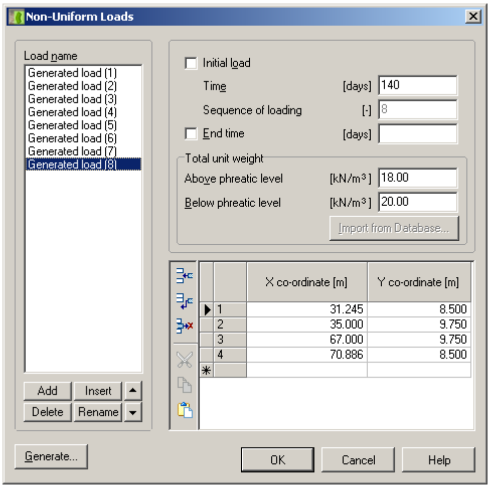


Figure 4-24 – *Non-Uniform Loads* window, Load 8 (Tutorial-2c)

The staged loading is now displayed in the *Input* tab of the *View Input* window. The *Zoom limits* button in the *Tools* panel can be used to optimize the limits of the drawing (Figure 4-25).

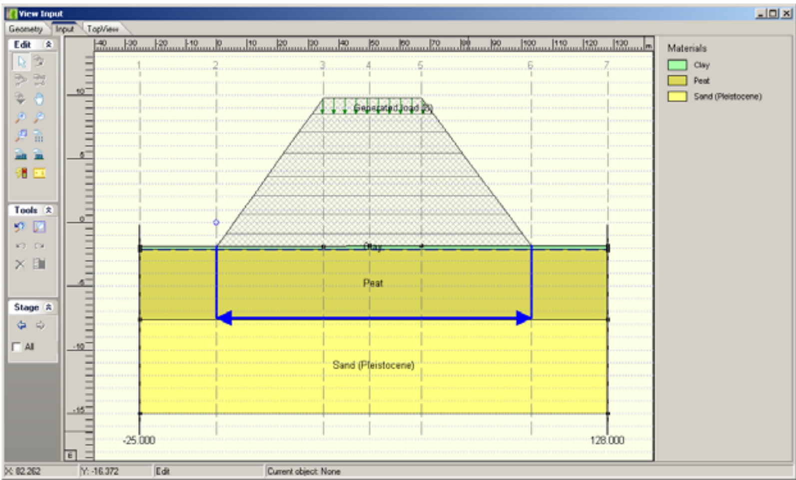


Figure 4-25 – *View Input* window, *Input* tab (Tutorial-2c)

36. Open the *Calculation Options* window from the *Calculation* menu, unmark the *Maintain Profile* option and click *OK* to confirm.
37. Open the *Calculation Times* window from the same menu and add a number of times for residual stress calculation, according to Figure 4-26.

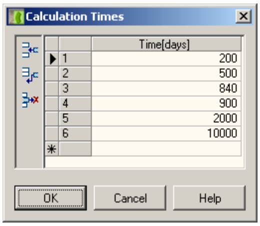


Figure 4-26 – *Calculation Times* window (Tutorial-2c)

38. Check that the drain distance is <2 m> in the *Vertical Drains* window and perform a first calculation in the *Start Calculation* window.
39. View the development of the total settlement (Figure 4-27), the excess head at *Depth* <-4.875 m> (Figure 4-28) and the residual settlement (Figure 4-29) through the *Results* menu, after selecting *Vertical* number <4> (i.e. horizontal co-ordinate 50 m). The residual settlement at 900 days is 0.278 m, while the allowed value is 0.15 m.

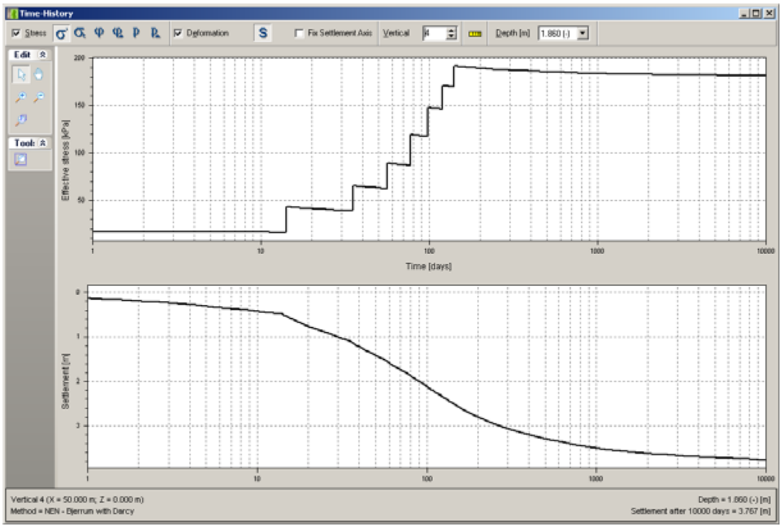


Figure 4-27 – *Time-History* window, Settlement and Effective stress vs. Time in vertical 4 for drain distance 2 m (Tutorial-2c)

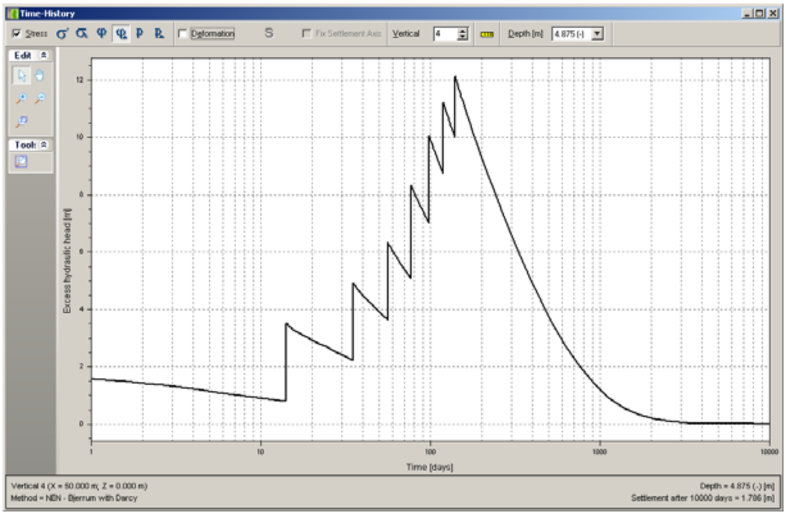


Figure 4-28 – *Time-History* window, Excess head vs. Time in vertical 4 at RL-4.875 m for drain distance 2 m (Tutorial-2c)

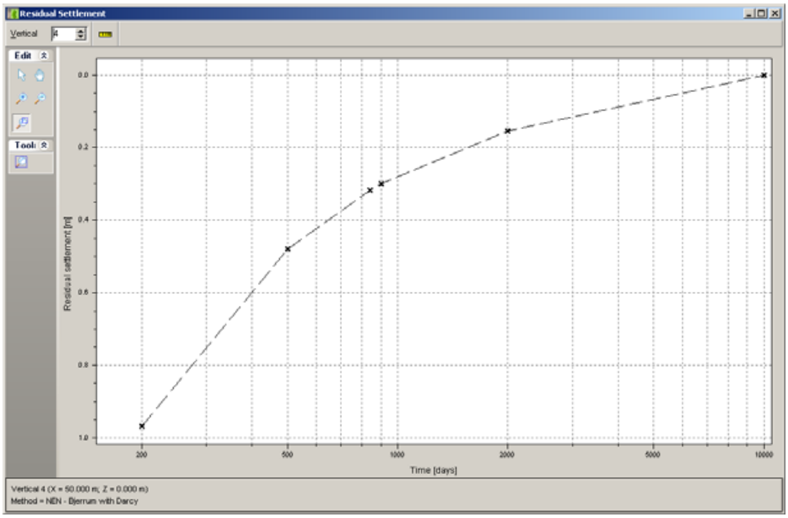


Figure 4-29 – *Residual Settlement* window for drain distance 2 m (Tutorial-2c)

40. Check yourself that a drain distance of 1 m reduces the residual settlements to 0.203 m (Figure 4-30), which is still more than allowed. Temporary preloading and/or dewatering will therefore be required, in combination with sufficiently fast dissipation of excess pore pressures.

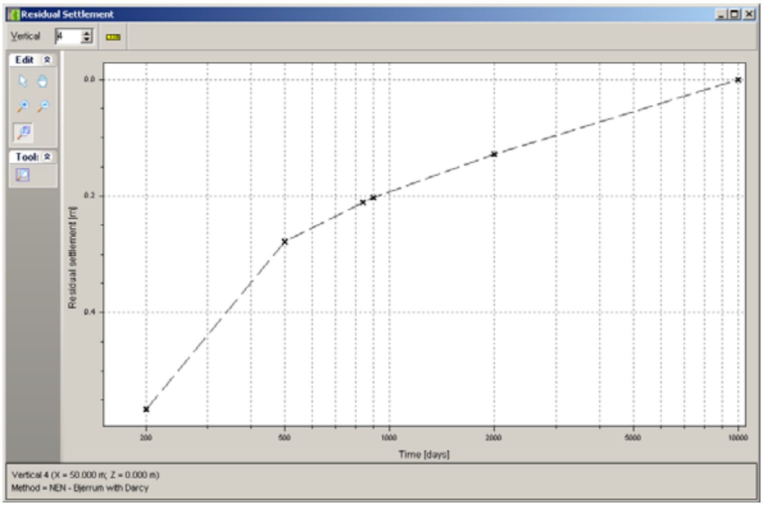


Figure 4-30 – *Residual Settlement* window for drain distance 1 m (Tutorial-2c)

4.5 Temporary preloading by soil raise (Tutorial-2d)

Precompression by a temporary increase of effective stress will reduce residual creep settlements. The Isotache models (NEN-Bjerrum, a/b/c) are capable of capturing this behavior.

41. Open the *Save As* window and save the current project (with a grid distance of 1 m) as <Tutorial-2d>.
42. Open the *Non-Uniform Loads* window from the *Loads* menu and add a temporary soil raise of 1 m from 161 to 840 days (*Loads/Nonuniform Loads*), according to Figure 4-31.
43. Perform a new calculation in the *Start Calculation* window.
44. After the calculation, view the development of total and residual settlements, and check that the residual settlement for vertical 4 at 900 days is now reduced to 0.145 m (Figure 4-32).

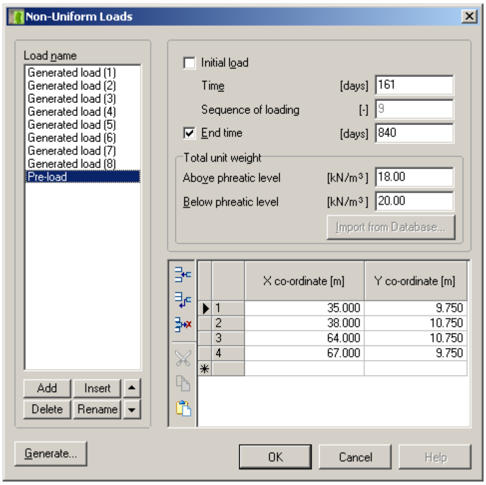


Figure 4-31 – *Non-Uniform Loads* window, Temporary preloading 1 m (Tutorial-2d)

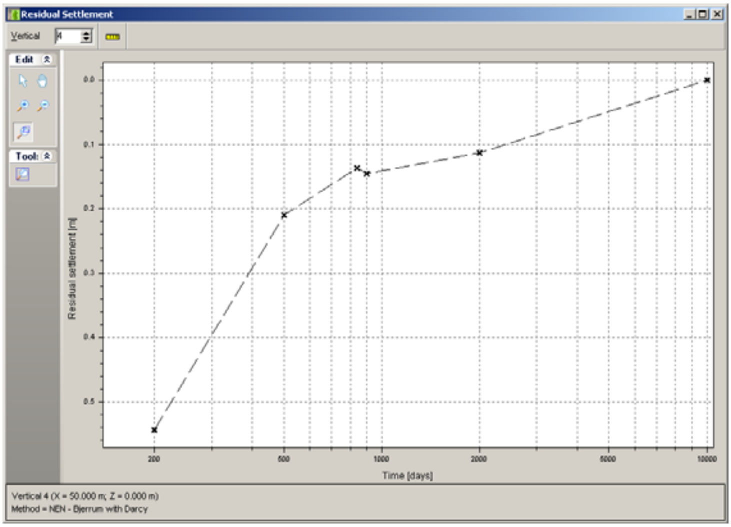


Figure 4-32 – *Residual Settlement* window (Tutorial-2d)

4.6 Additional enforced dewatering (Tutorial-2e)

Temporary preloading by enforced dewatering is an alternative for (part of the) temporary preloading by soil raise. MSettle supports different enforced dewatering methods, including Menard consolidation, IFCO (sand screens) and BeauDrain (strip drains). In this case, enforced dewatering of strip drains with rectangular grid (BeauDrain) has been combined with a small temporary soil raise of 0.5 m.

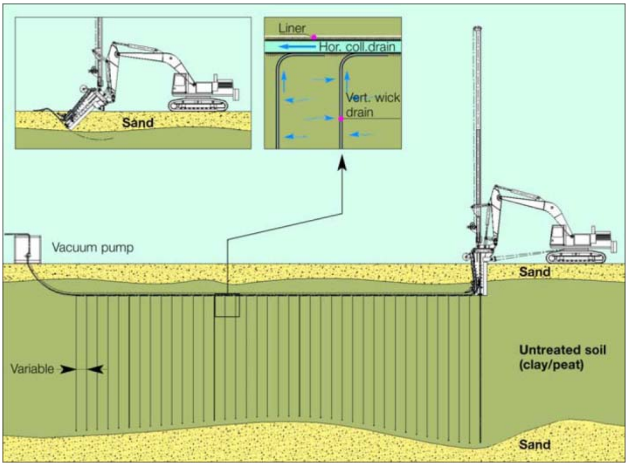


Figure 4-33 – Installation Beau Drain system (Tutorial-2e)

45. Open the *Save As* window and save the current project as <Tutorial-2e>.
46. Modify the temporary preloading in the *Non-Uniform Loads* window, according to Figure 4-34, and click *OK* to confirm.

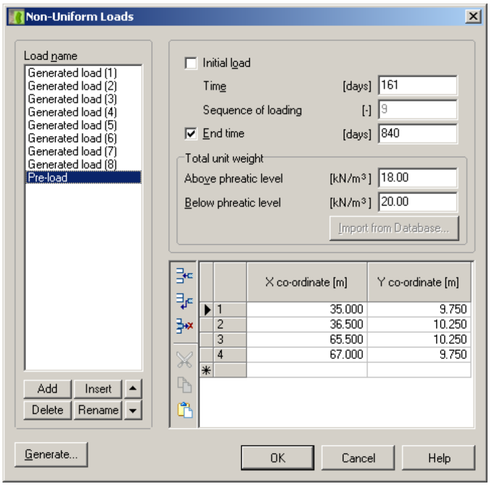


Figure 4-34 – *Non-Uniform Loads* window, Temporary preloading 0.5 m (Tutorial-2e)

47. Open the *Vertical Drains* window via the *GeoObjects* menu, change the drain spacing to a <Rectangular> grid (typical for Beau Drain), select the *Simple Input* option for *Enforced Dewatering*, add a *Begin time* for the pumping of <54> days, and add a *End time* of <438> days. Leave the value for the underpressure to the default of <35> kPa. The value of the water head in the drains during dewatering

should be chosen equal to the initial position of the horizontal drains, in this case at RL <-2.2> m as shown in Figure 4-35.

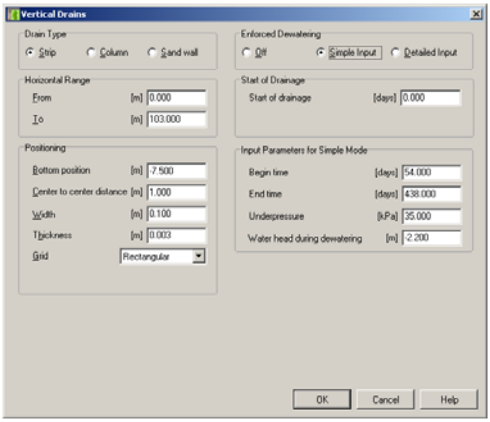


Figure 4-35 – *Vertical Drains* window, Enforced Dewatering input (Tutorial-2e)

48. Perform a new calculation in the *Start Calculation* window.
49. Verify that the residual settlement after 900 days is 0.140 m for vertical 4.
50. View the excess head versus time at vertical 4, RL-4.875 m (Figure 4-36). Note that the excess head is reduced considerably during enforced dewatering.

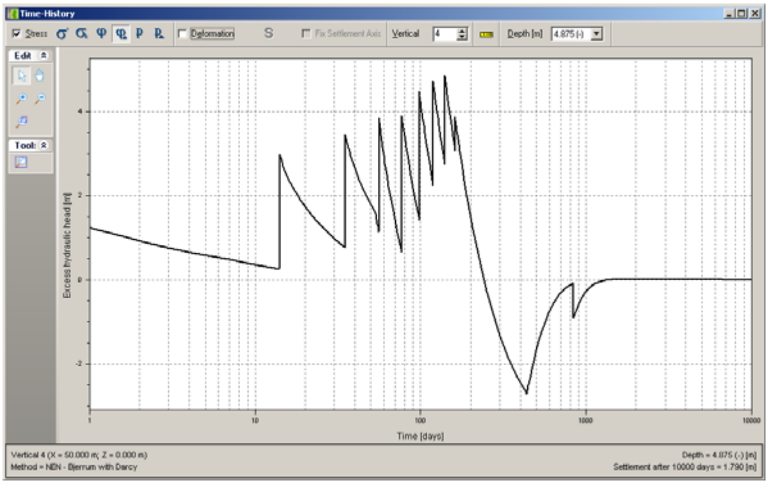


Figure 4-36 – *Time-History* window, Excess head vs. Time in vertical 4 at RL-4.875 m, with enforced dewatering (Tutorial-2e)

51. View also the effective stress versus time at vertical 4, RL-4.875 m (Figure 4-37). Before 438 days, the effective stress increases continuously, due to still

dissipating excess pore pressures. After the end of pumping, at 438 days, the effective stress decreases with approximately 35 kPa.

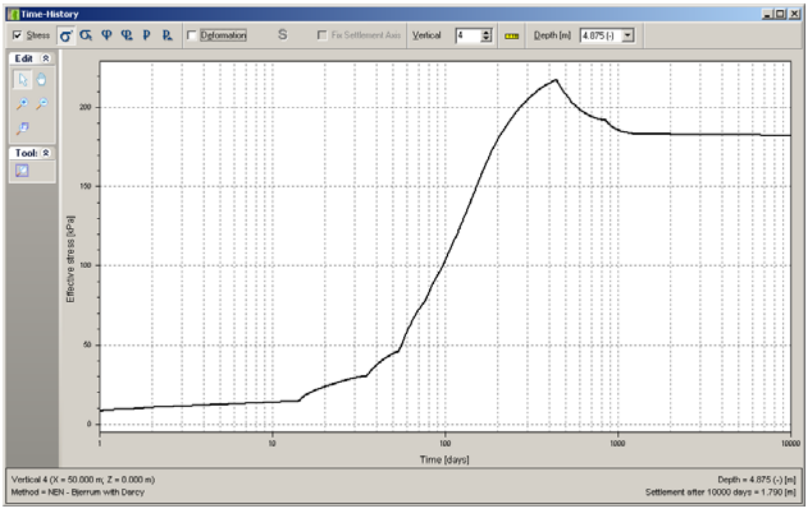


Figure 4-37 – *Time-History* window, Effective stress vs. Time in vertical 4 at RL-4.875 m, with enforced dewatering (Tutorial-2e)

4.7 Horizontal Displacements (Tutorial-2f)

The construction of the embankment can cause damaging horizontal displacements for existing constructions, especially piles. De Leeuw theory implemented in MSettle will be used hereafter to estimate those horizontal displacements.

4.7.1 Principles of De Leeuw method

The De Leeuw method [Lit 24] is based on the work of Van IJsseldijk (elastic soil) and Loof (elastic soil with stiff top layer) and estimates the horizontal displacements based on an elastic solution for a single elastic incompressible layer, characterized by the Young's modulus *E*. The method assumes that the horizontal deformations of the elastic layer are always constrained at the bottom by a stiff foundation layer. Optionally the deformations can also be constrained by a stiff layer at the top.

In this tutorial, the *Clay* and *Peat* layers are considered as elastic layers that will deform and the *Sand (Pleistocene)* layer is the foundation layer (Loof case).

4.7.2 Evaluation of the elasticity modulus

The Young's modulus of the elastic layer can be automatically estimated by MSettle from the average unit weight γ of the soft layers according to De Leeuw & Timmermans [§ 18.3.3].

An other method, called Betuweroute method, is used in this tutorial. The E-modulus is determined from the following equation:

$$(2) \qquad E = 1.25\, H \frac{\Delta\sigma}{\Delta s}$$

where:

H	Thickness of the elastic layer [m].
$\Delta\sigma$	Vertical stress increase of the elastic layer [kPa].
Δs	Settlement of the elastic layer [m].

To estimate the E-modulus from MSettle results, vertical 4 leading to maximum settlements is used: in the *Depth-History* window, relative final settlement of the *Clay* (between NAP -1.86 m and NAP -2.15 m) and *Peat* (between NAP -2.15 m and NAP -7.60 m) layers (i.e. elastic layers) is respectively 0.15 m and 3.62 m and the loading goes from -1.86 m (surface) to 9.75 m with a unit weight of 18 kN/m^3, which leads to a modulus of:

$$E = \begin{cases} 1.25 \times \left(-1.86 - \left(-2.15\right)\right) \dfrac{18 \times \left(9.75 - \left(-1.86\right)\right)}{0.15} = 505 \text{ kPa} & \text{for Clay} \\ 1.25 \times \left(-2.15 - \left(-7.60\right)\right) \dfrac{18 \times \left(9.75 - \left(-1.86\right)\right)}{3.62} = 393 \text{ kPa} & \text{for Peat} \end{cases}$$

4.7.3 Input for horizontal displacements

52. Open the *Save As* window and save the current project as <Tutorial-2f>.
53. Open the *Model* window via the *Project* menu, and mark the *Horizontal displacements* checkbox.

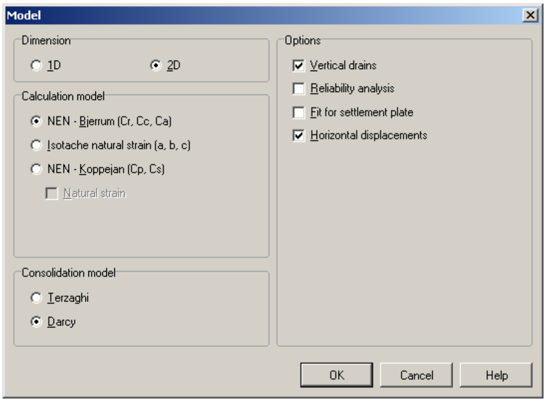


Figure 4-38 – *Model* window (Tutorial-2f)

54. Open the *Materials* window via the *Soil* menu, and select <Foundation> as *Layer behaviour* for *Sand (Pleistocene)* layer and <Elastic> for *Clay* and *Peat* layers (Figure 4-39). For the *Clay* and *Peat* layers with an elastic behaviour, enter a soil modulus of respectively <505 kPa> and <393 kPa> [§ 18.3.3].

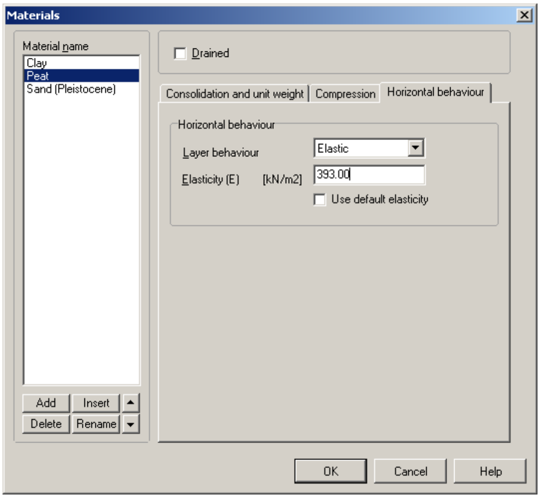


Figure 4-39 – *Materials* window (Tutorial-2f)

4.7.4 Calculated horizontal displacements

55. Open the *Start Calculation* window via the *Calculation* menu and click *Start* to start the calculation.
56. Open the *Depth-History* window via the *Results* menu. Unmarked the *Stress* checkbox and click on the *Horizontal Displacement* button in the *Deformation* field.
57. Select the different verticals to see the influence of the position.

Horizontal displacements in the stiff foundation (i.e. *Sand*) layer are nil as De Leeuw theory is based on elastic solution.
At the bottom of the *Depth-History* window, the resulting elasticity for the vertical is displayed (average elasticity between all elastic layers).
Horizontal displacements are maximum and equal for verticals 3 and 5 as they are both situated at the top level of the load (Figure 4-40). For vertical 4 situated at the middle of the loading, horizontal displacements are almost nil because of symmetry.

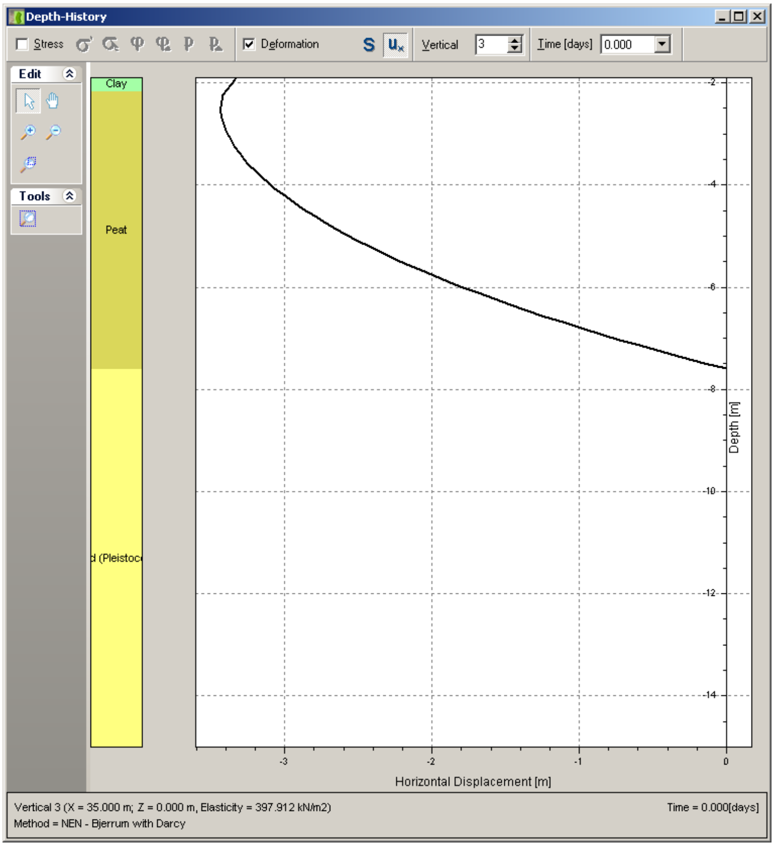


Figure 4-40 – *Depth-History* window, Horizontal Displacements at vertical 3 (Tutorial-2f)

4.8 Bandwidth Determination (Tutorial-2g)

MSettle's reliability module will be used hereafter to estimate the bandwidth in total and residual settlement, based on values for the standard deviation of soil parameters and layer positions. MSettle can either estimate standard deviations based on safe defaults for variation coefficients, or use direct input of the standard deviation. In this case, direct input has been applied, based on Equation (1).

NOTE: It is assumed in this case, that the thickness of the layers is large compared to the scale of vertical variability. Averaging in vertical direction is then allowed. The input value of the standard deviation of the local average in a vertical has been estimated from the total variance, by assuming a ratio of 1 to 4 between the variance of the local average in a vertical and the total variance from the lab tests.

NOTE: MSettle supports normal and lognormal distributions. Usage of a Student-t distribution is theoretically preferred in cases with a small number of lab tests. The additional uncertainty by small test numbers has been incorporated approximately in the standard deviation of a normal or lognormal distribution, by an exaggeration factor on the total variance.

NOTE: MSettle does not stochastically model the uncertainties following from limitations of the prediction model, the uncertainties in loading and the uncertainty in soil type. The expected bandwidth is in reality therefore presumably larger than the calculated bandwidth.

58. Open the *Save As* window and save the current project as <Tutorial-2g>.
59. Open the *Model* window via the *Project* menu, mark the *Reliability Analysis* checkbox and unmark the *Horizontal displacements* checkbox.

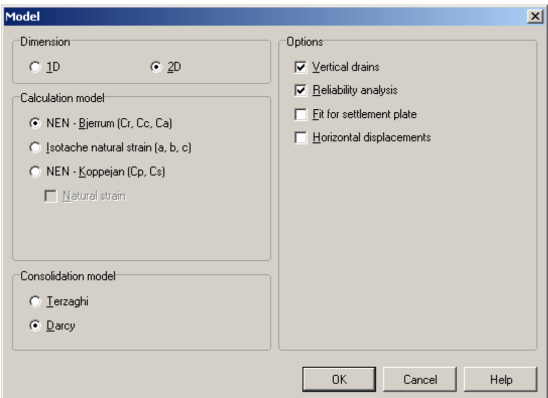


Figure 4-41 – *Model* window (Tutorial-2g)

60. Open the *Probabilistic Defaults* window via the *Project* menu, and select <Deterministic> for the standard deviation of the *Layer boundary*.

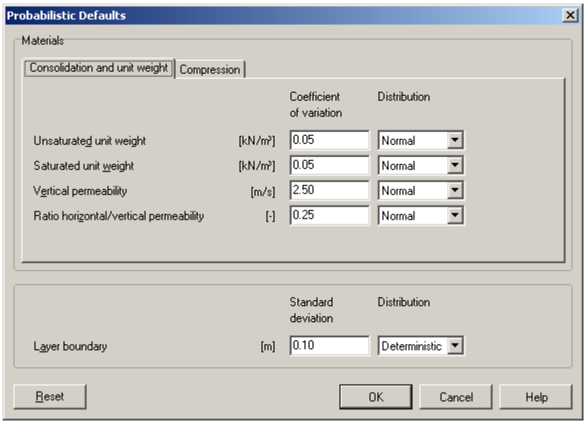


Figure 4-42 – *Probabilistic Defaults* window (Tutorial-2g)

61. Open the *Materials* window via the *Soil* menu. Unmark the *Probabilistic Defaults* checkbox for each soil type, and add the standard deviations and distributions, according to Figure 4-43 to Figure 4-45.

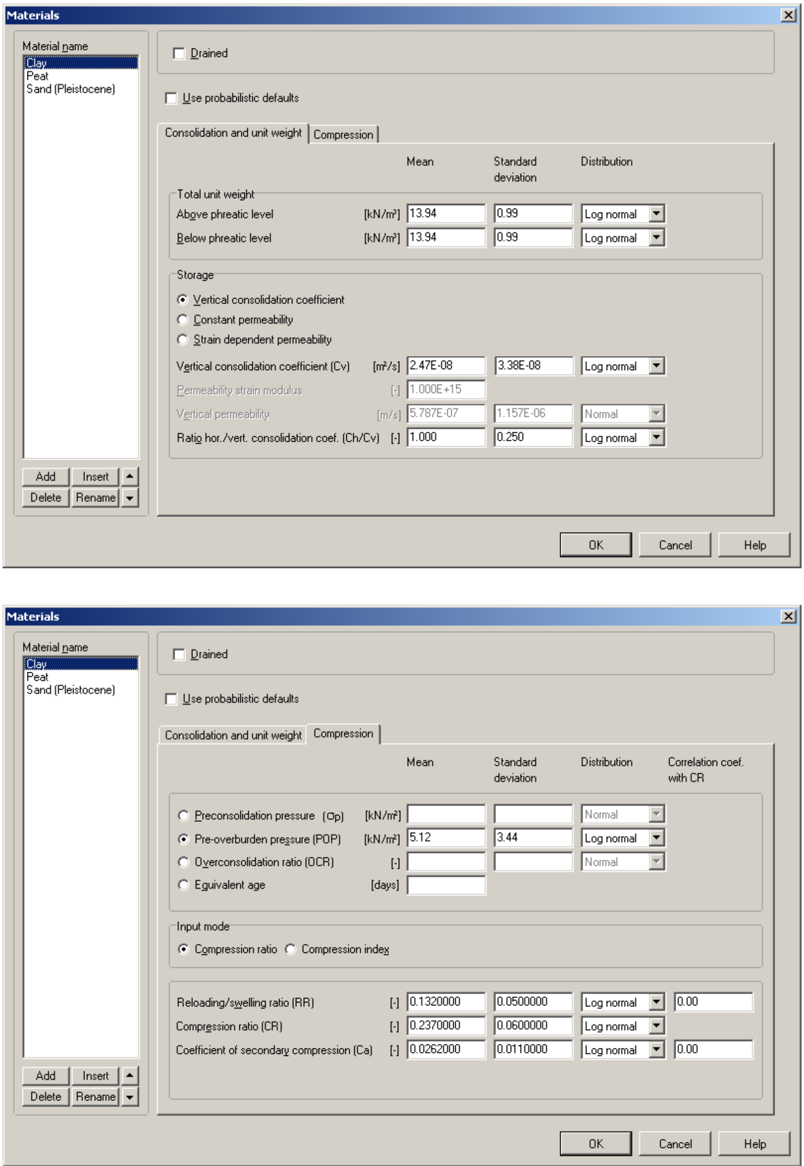


Figure 4-43 – *Materials* window for *Clay* (Tutorial-2g)

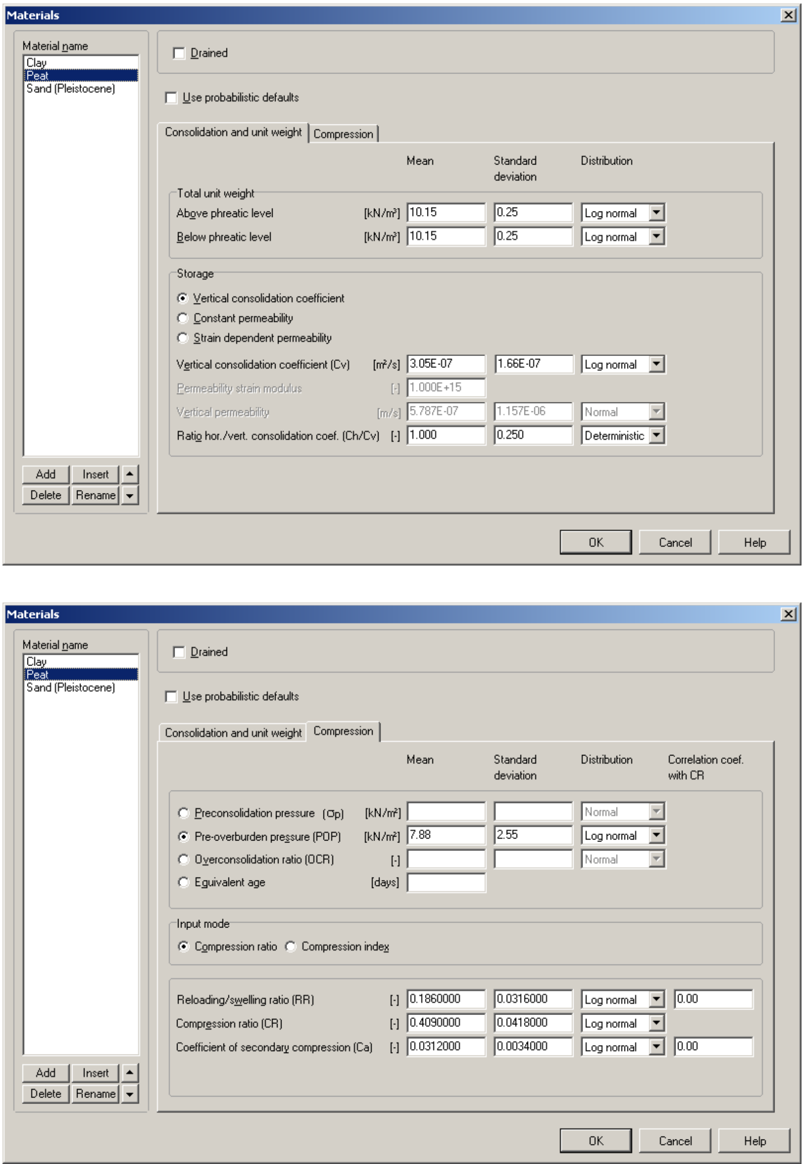


Figure 4-44 – *Materials* window for *Peat* (Tutorial-2g)

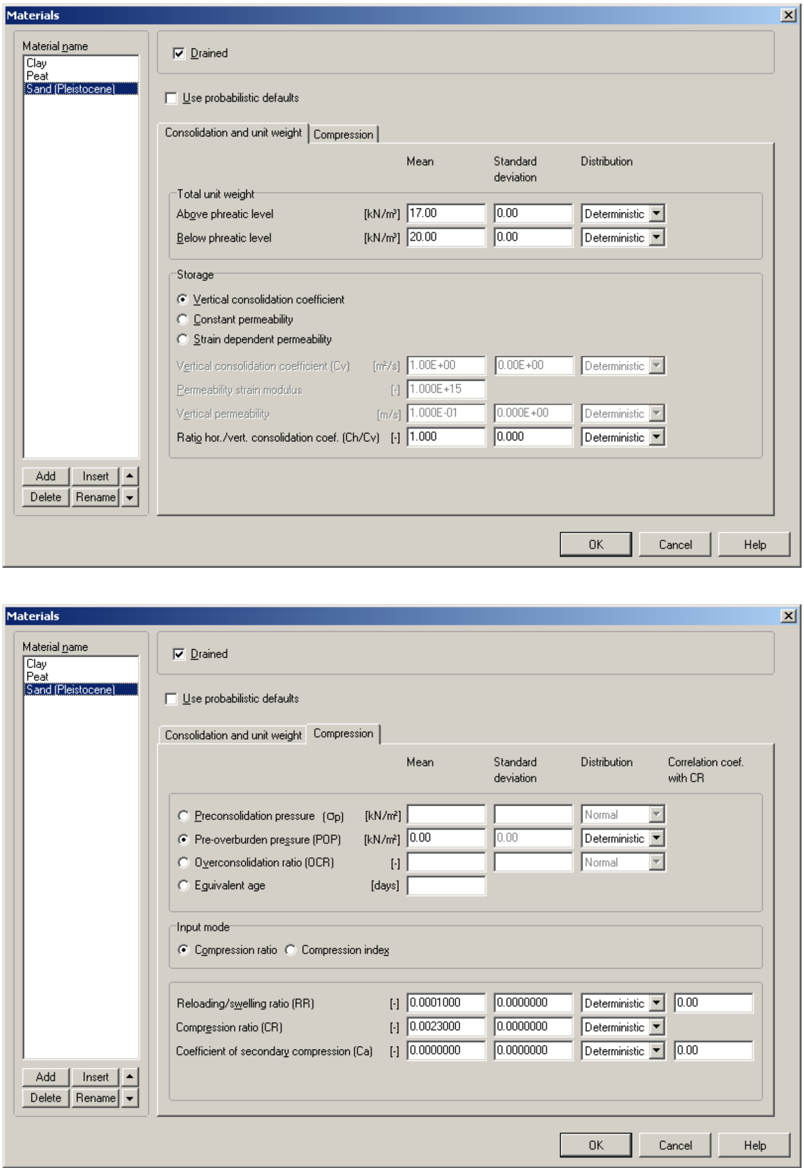


Figure 4-45 – *Materials* window for *Sand (Pleistocene)* (Tutorial-2g)

62. Open the *Calculation Times* window via the *Calculation* menu and add the times for bandwidth determination, according to Figure 4-46.

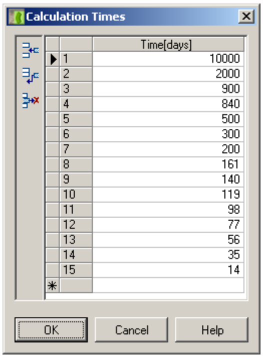


Figure 4-46 – *Calculation Times* window for Bandwidth determination (Tutorial-2g)

63. Open the *Start Calculation* window via the *Calculation* menu. Monte Carlo is the preferred method for robust determination of bandwidth in both total and residual settlements. Select *Monte Carlo* reliability analysis, select *Vertical* <4> at horizontal co-ordinate 50 for the settlement determination, enter <0.15 m> as *Allowed residual settlement*, and enter <200> as the *Maximum number of samples*. Unselect the *Add dissipation calculation* option. Click *Start* to start the Monte Carlo sampling.

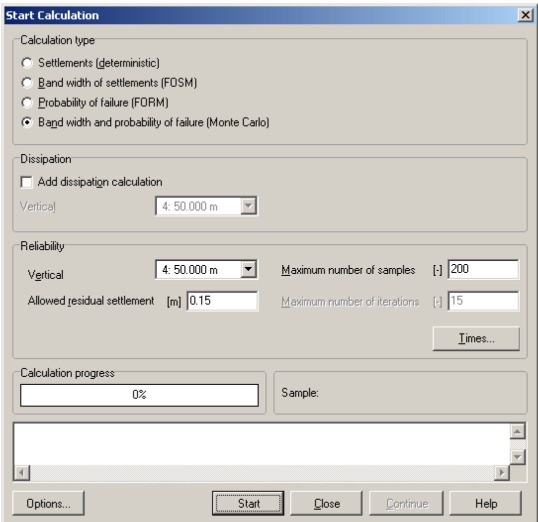


Figure 4-47 – *Start Calculation* window for *Monte Carlo* reliability analysis (Tutorial-2g)

64. After the analysis has finished, open the *Time-History (Reliability)* from the *Results* menu to view the bandwidth results (Figure 4-48). Monte Carlo results can vary slightly from analysis to analysis, because of the random drawing of soil parameters for the 200 samples. Using the right-hand mouse button, open the *Chart Data* window and check that the total settlement after 1000 days is

approximately 3.80 ± 0.61 m. Note that those values can vary from a calculation to another due to a different sampling for each calculation.

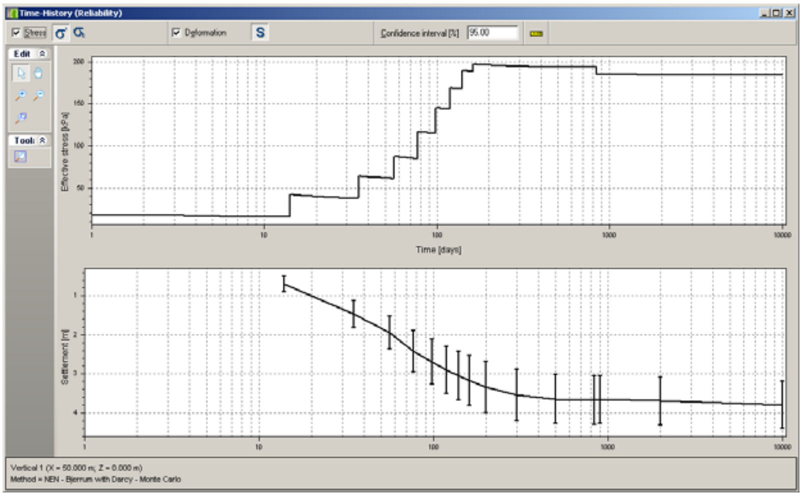


Figure 4-48 – *Time-History (Reliability)* window, Total settlement vs. Time with Band width for Monte Carlo method (Tutorial-2g)

65. Then open the *Residual Settlement (Reliability)* window from the *Results* menu (Figure 4-49). Using the right-hand mouse button, open the *Chart Data* window (Figure 4-50) and check that the residual settlement after 900 days is approximately 0.14 ± 0.06 m, with a failure probability (residual settlement larger than 0.15 m) of 56%. Note that those values can vary from a calculation to another due to a different sampling for each calculation. Note also that the mean final and residual settlements from a Monte Carlo analysis are larger than results from a deterministic calculation.

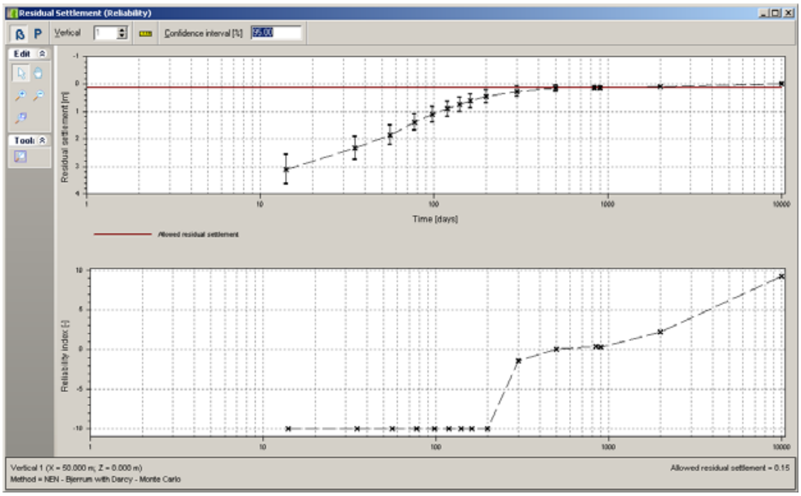
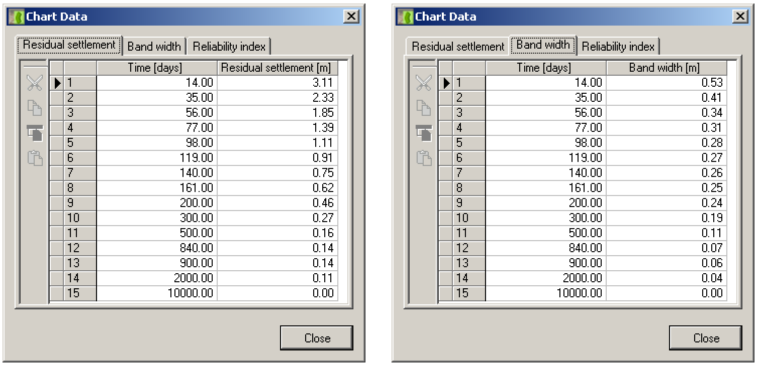


Figure 4-49 – *Residual Settlement (Reliability)* window (Tutorial-2g)



	Time [days]	Residual settlement [m]
1	14.00	3.11
2	35.00	2.33
3	56.00	1.85
4	77.00	1.39
5	98.00	1.11
6	119.00	0.91
7	140.00	0.75
8	161.00	0.62
9	200.00	0.46
10	300.00	0.27
11	500.00	0.16
12	840.00	0.14
13	900.00	0.14
14	2000.00	0.11
15	10000.00	0.00

	Time [days]	Band width [m]
1	14.00	0.53
2	35.00	0.41
3	56.00	0.34
4	77.00	0.31
5	98.00	0.28
6	119.00	0.27
7	140.00	0.26
8	161.00	0.25
9	200.00	0.24
10	300.00	0.19
11	500.00	0.11
12	840.00	0.07
13	900.00	0.06
14	2000.00	0.04
15	10000.00	0.00

Figure 4-50 – *Chart Data* window, *Residual settlement* and *Band width* tabs (Tutorial-2g)

4.9 Conclusion

This tutorial presents the different stages of a project leading to use vertical strip drains with enforced dewatering in combination with temporary preloading in order to accelerate the consolidation process and finally get acceptable residual settlements.

5 Tutorial 3: Settlement plate fit

This is the second tutorial in a sequence of two on the construction of a high embankment for the Dutch A2 highway, at a viaduct crossing with the N201 road nearby Vinkeveen. Vertical drains with enforced dewatering have been used to speed up the consolidation and to reduce the residual settlement. The first part [chapter 4] already illustrated MSettle's different features for the initial design.

The objectives of this exercise are:

- to perform a settlement plate fit after input of the actual loading stages;
- to perform a bandwidth determination, in order to improve the predictions and reduce the uncertainty during the construction stage.

The following MSettle modules are needed:

- MSettle (1D model with Terzaghi)
- 2D geometry model
- Darcy consolidation model
- Vertical drains module
- Fit for settlement plate module
- Reliability analysis module

This tutorial is presented on the files Tutorial-3a.sli to Tutorial-3c.sli and is based on measurement file Tutorial-3.txt.

5.1 Actual loading steps

Compared to the initial design calculation in the previous Tutorial-2f [chapter 4], a waiting period of 100 days has been introduced after construction of the working floor and the installation of the drains, and the additional period for the soil raise to maximum height has been extended to 264 days. The available construction period,

including the construction of the working floor, is now 940 days, and the residual settlements from 1000 days may not exceed 0.15 m.
The shape of the loading must also be adapted to fit with the actual loading stages. The 14 stages with their application time and geometry are given in Figure 5-1. The exact co-ordinates of each loading stage are given in Table 5-1.

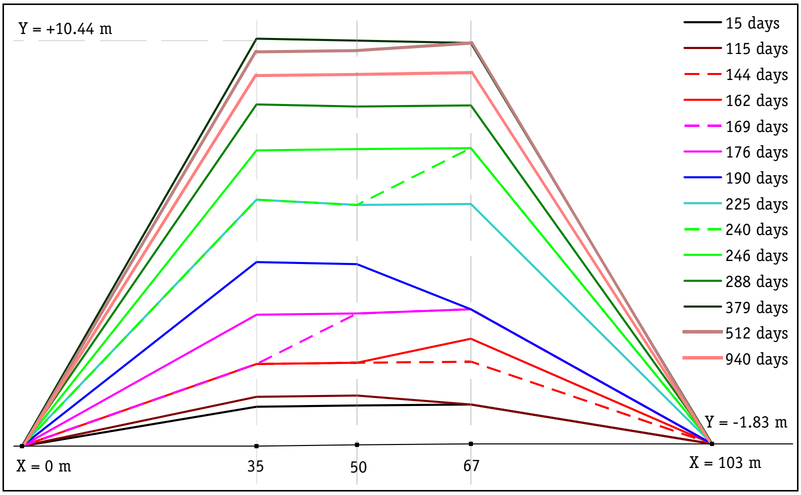


Figure 5-1 – Actual loading stages for Tutorial 3

Table 5-1 – Co-ordinates of the different loading stages (Tutorial 3)

Load name	Time [days]	Y co-ordinate [m] at … X = 0	X= 35	X = 50	X= 67	X = 103
15 days	15	-1.9	-0.7	-0.66	-0.63	-1.83
115 days	115	-1.9	-0.4	-0.36	-0.63	-1.83
144 days	144	-1.9	0.6	0.64	0.67	-1.83
162 days	162	-1.9	0.6	0.64	1.37	-1.83
169 days	169	-1.9	0.6	2.14	2.27	-1.83
176 days	176	-1.9	2.1	2.14	2.27	-1.83
190 days	190	-1.9	3.7	3.64	2.27	-1.83
225 days	225	-1.9	5.6	5.44	5.47	-1.83
240 days	240	-1.9	5.6	5.44	7.17	-1.83
246 days	246	-1.9	7.1	7.14	7.17	-1.83
288 days	288	-1.9	8.5	8.44	8.47	-1.83
379 days	379	-1.9	10.5	10.44	10.37	-1.83
512 days	512	-1.9	10.1	10.14	10.37	-1.83
940 days	940	-1.9	9.379	9.413	9.469	-1.83

5.2 Initial prediction (Tutorial-3a)

1. Open the initial input file <Tutorial-2e.sli>, containing already the input data for the subsoil, the drains with enforced dewatering and the measured loading.
2. Open the *Save As* window and save it as <Tutorial-3a>.
3. Open the *Non-Uniform Loads* window from the *Loads* menu and delete all existing loads using the *Delete* button.
4. Add a new load by clicking the *Add* button and rename it to <15 days>. Enter a *Time* of <15> days. Enter a *Total unit weight above* and *below phreatic level* of respectively <18> and <20> kN/m^3. Enter the co-ordinates of this first load as given in Table 5-1. This should result in the same window as Figure 5-2.

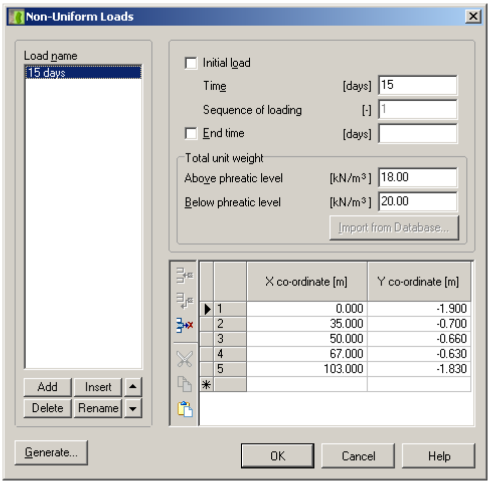


Figure 5-2 – *Non-Uniform Loads* window, First load

5. Then click 13 times on the *Add* button to input the 13 other loads. Modify the *Load name*, the *Time* and the *Y co-ordinate* of those 13 loads according to Table 5-1. For the two last loads <512 days> and <940 days> enter a negative *Total unit weight above* and *below phreatic level* to model the removing of the load, as illustrated in Figure 5-3.

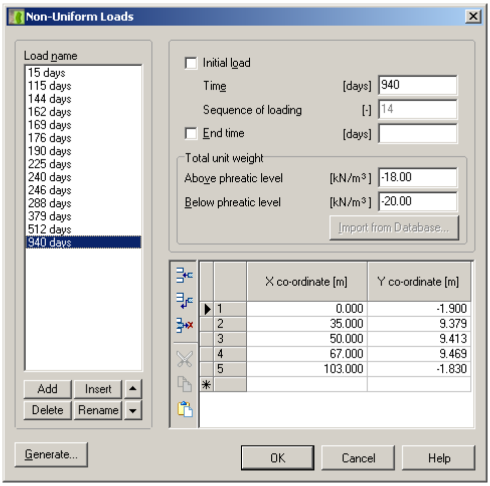


Figure 5-3 – *Non-Uniform Loads* window, Last load

6. Open the *Vertical Drains* window and increase the *Start of drainage* of 20 days and the *Begin* and *End time* of enforced dewatering of 100 days to get the same window as Figure 5-4. Click *OK* to confirm.

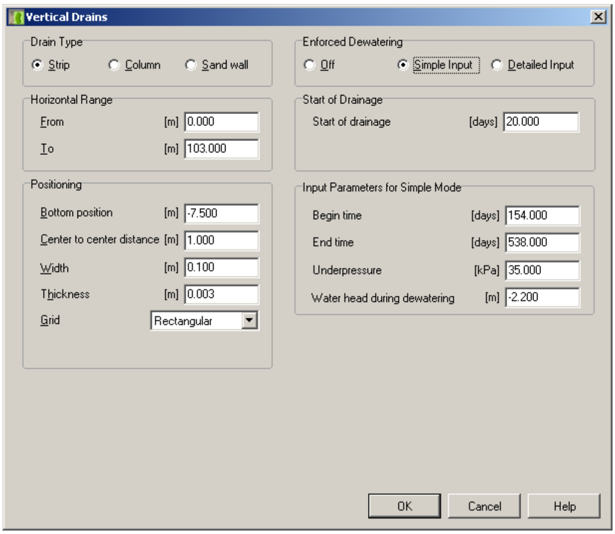


Figure 5-4 – *Vertical Drains* window

7. Open the *Calculation Times* window and modify the times according to Figure 5-5. Click *OK* to confirm.

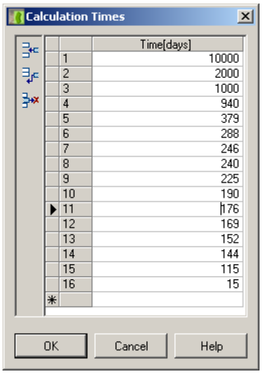


Figure 5-5 – *Calculation Times* window

8. Press the function key F9 to open the *Start Calculation* window.
9. View the transient settlement and effective loading at the surface level after selecting *Vertical* number <4> in the *Time-History* window from the *Results* menu (Figure 5-6) and check that the predicted final settlement is 3.747 m.

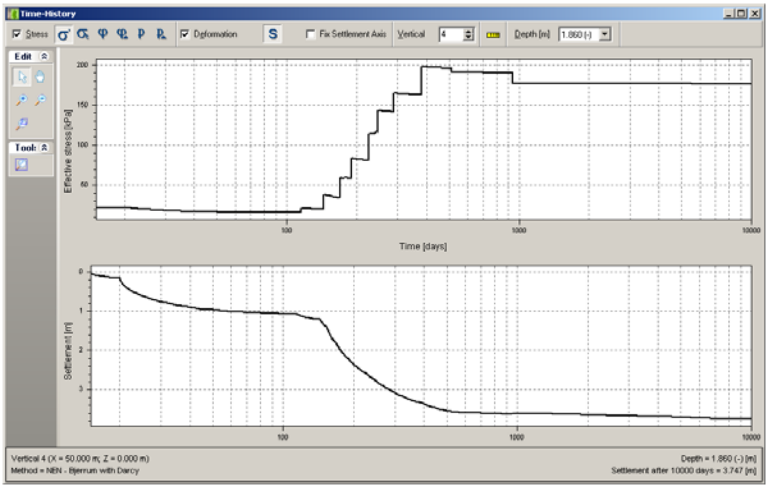


Figure 5-6 – *Time-History* window, Settlements and Effective stress at surface level vs. Time for vertical 4 (Tutorial-3a)

10. Open the *Residual Settlement* window and check that the predicted residual settlement after 1000 days for vertical 4 is about 0.13 m.

5.3 Settlement plate fit (Tutorial-3b)

11. Open the *Save As* window and save the current project as <Tutorial-3b>.
12. Open the *Model* window via the *Project* menu and mark the *Fit for settlement plate* checkbox (Figure 5-7).

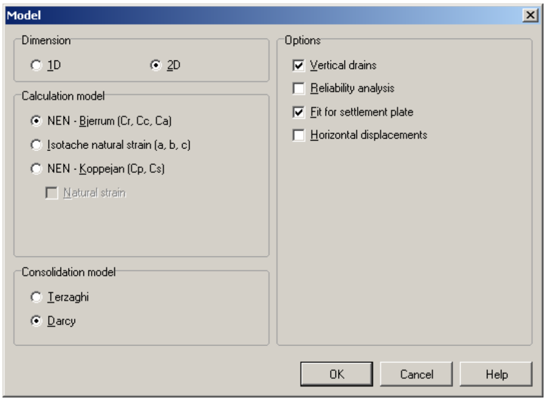


Figure 5-7 – *Model* window

13. Open the *Fit for Settlement Plate* window via the *Calculation* menu.
14. At the top of the window, select *Vertical* <4 at 50.000m>.
15. In the *Measurements* tab, click the *File Open* button and select <Tutorial-3.txt> from the *Examples* directory where the MSettle program was installed (Figure 5-15). Click *Open*.

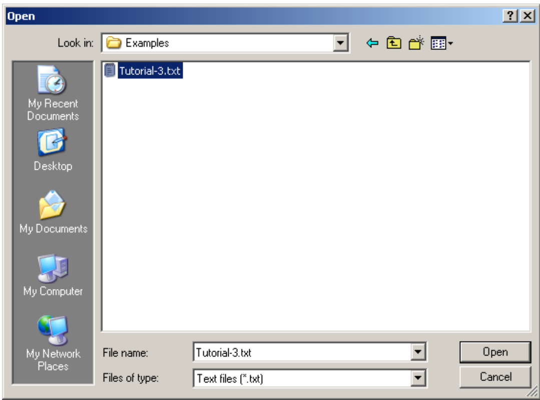


Figure 5-8 – *Open* window

NOTE: The text file named Tutorial-3.txt has a simple two-column number format (times and settlements), separated by tabs. It is possible in the input window to enter a shift in time or settlement.

The measurements are displayed in the *Measurements* tab of the *Fit for Settlement Plate* window (Figure 5-9). Separate weights can be attached to each of the measurements. The default weight is 1. A large weight to a certain measurement will increase its relative influence.

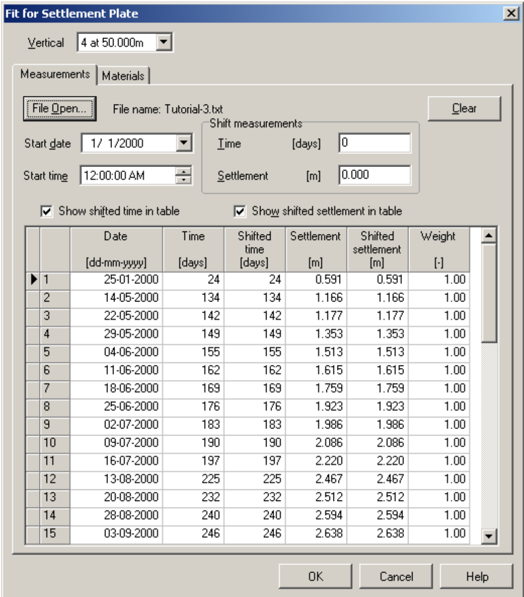


Figure 5-9 – *Fit for Settlement Plate* window, *Measurements* tab (Tutorial 3b)

16. Select the *Materials* tab. This tab offers options for automatic or manual adaptation of 5 special fit parameters as shown in Figure 5-10.

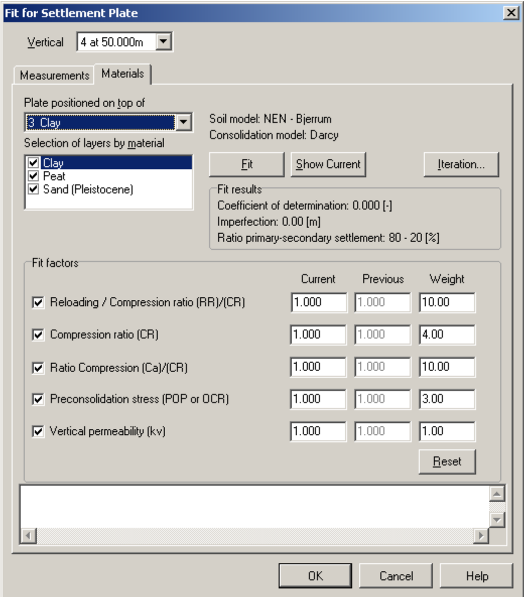


Figure 5-10 – *Fit for Settlement Plate* window, *Materials* tab (Tutorial 3b)

17. Click the *Show Current* button to compare the initial prediction with the actual measurements as shown in the *Time-History (Fit)* window that opens (Figure 5-11).

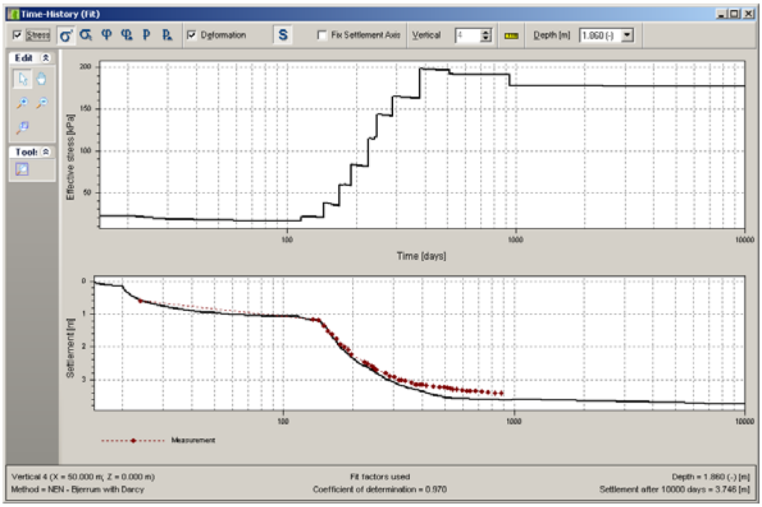


Figure 5-11 – *Time-History (Fit)* window, Initial prediction versus measurement, imperfection 0.19 m (Tutorial-3b)

In the *Materials* tab of the *Fit for Settlement Plate* window, MSettle also displays a so-called *Imperfection* value of 0.22 m (Figure 5-12). This is the root-mean-square deviation between prediction and settlement.

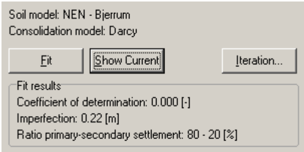


Figure 5-12 – *Fit for Settlement Plate* window, *Materials* tab, Details of the *Fit Results* (Tutorial-3b)

MSettle uses fit factors to multiply the following five soil parameters and ratio's for all layers or for user-selected layers:

- Cv or kv (consolidation)
- OCR or POP (preconsolidation)
- CR (primary virgin compressibility),
- ratio RR/CR (reloading compressibility relative to primary virgin compressibility)
- ratio Ca/CR (secondary compressibility relative to primary virgin compressibility)

It is possible to manually modify those single fit factors and see the effect on the total and residual settlements. For instance:

18. Set the multiplication factor on *CR* to <0.95> and click *Show Current* to view the prediction versus the measurement.

Now, an automatic iterative modification of the fit factors is performed:

19. Reset all fit factors to <1> in the *Materials* tab from the *Fit for Settlement Plate* window.
20. Click the *Iteration* button to open the *Iteration stop criteria* window and change the default iteration stop criteria to the values displayed in Figure 5-13. The *coefficient of determination* is defined as 1 minus the division of the square of the final imperfection by the square of the initial one. The *required iteration accuracy* is the minimally required improvement in the coefficient of determination per iteration. Click *OK* to confirm.

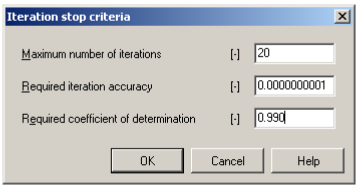


Figure 5-13 – *Iteration stop criteria* window (Tutorial-3b)

21. Click *Fit* to start the automatic iterative modification of the fit factors.

MSettle uses a robust weighted least squares procedure, which minimizes not only the deviation between prediction and settlement, but also the deviation between the initial and modified parameter. Separate weights can be attached to each of the fit factors. The default weights are suited for most purposes. A large weight on a fit factor will reduce the freedom to deviate from 1. The default weights are the largest for the two compressibility ratios, because a local variation in primary virgin compressibility is likely to be correlated to a similar variation in reloading and secondary compressibility.

The fit factors during the fit are displayed in the *Fit for Settlement Plate* window. An acceptable match between fit and measurements by modification of soil parameters might hide that model limitations and loading uncertainties are in reality sometimes also a major cause of deviations between the initial prediction and the measurements. Therefore, a fit result can only be trusted if the initial soil parameters were determined accurately and if the variation of the fit factors in different cross sections is realistic compared to the natural variability in the soil parameters.

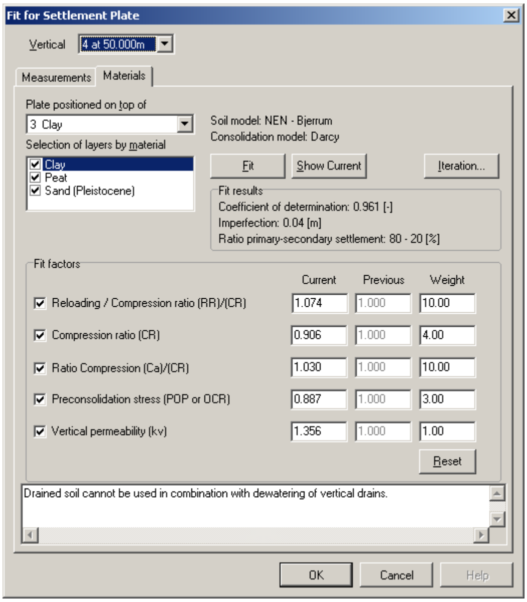


Figure 5-14 – *Fit for Settlement Plate* window, *Materials* tab, Fit factors after fit (Tutorial-3b)

22. After completion (Figure 5-14), click the *Show Current button* to view the final result, with an imperfection value of 0.04 m (Figure 5-15).

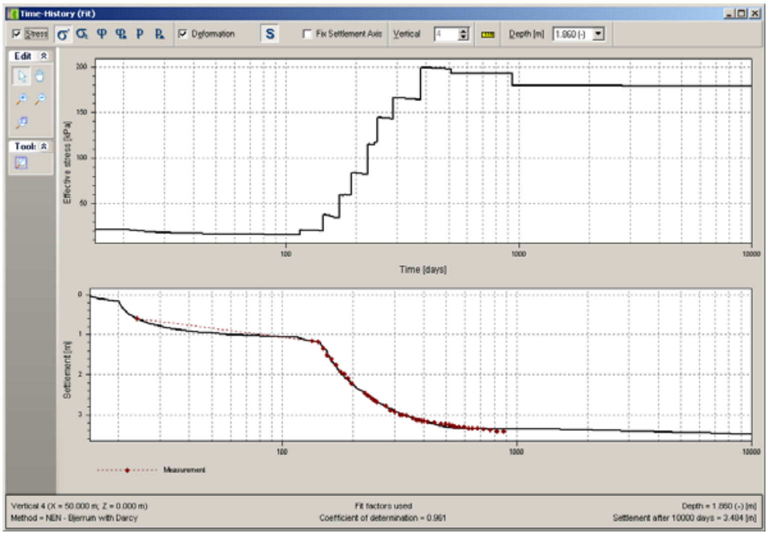


Figure 5-15 – *Time-History (Fit)* window, Prediction vs. measurement after fit, imperfection 0.04 m (Tutorial 3b)

23. Open the *Start Calculation* window and mark the *Use fit parameters* checkbox (Figure 5-16).

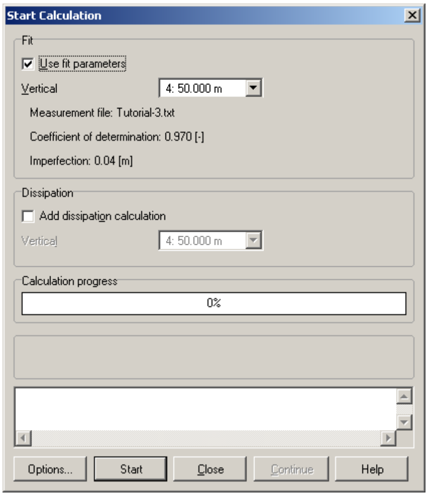


Figure 5-16 – *Start Calculation* window

24. Click *Start*.
25. Open the *Time-History* window from the *Results* menu and check that the total settlement in vertical 4 after 10000 days is 3.484 m, identical to Figure 5-15.

5.4 Band width after settlement plate fit (Tutorial-3c)

26. Open the *Save As* window and save the current project as <Tutorial-3c>.
27. Open the *Model* window and mark the *Reliability analysis* checkbox. See [§ 4.8] for the input of the stochastic soil data.
28. Open the *Start Calculation* window, and select the *Monte Carlo* analysis. Input of an *Imperfection* value is required for a reliability analysis with a preceding fit, to quantify limitations of the model and measurement errors, preventing a perfect fit and a perfect prediction of the remainder. The imperfection value resulting from the fit (0.04 m) needs to be multiplied with $\sqrt{(n-5)/(n-1)}$ to derive the input value of 0.05 m, where n equals the number of measurements (n = 43).
29. Click *Start*.

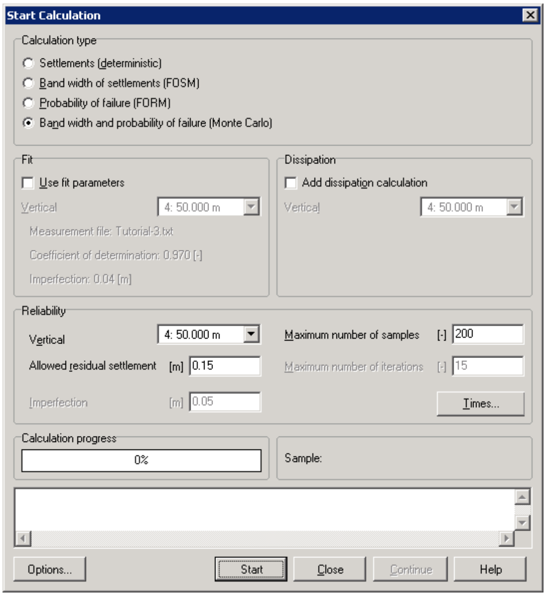


Figure 5-17 – *Start Calculation* window, Monte Carlo using fit parameters (Tutorial-3c)

MSettle will start with an update of the parameters dependencies (correlation matrix), followed by the actual Monte Carlo analysis with updated mean values and updated correlation matrix.

30. View the resulting settlement in the *Time-History (Reliability)* window and check that the final settlement at 10000 days is now approximately 3.49 ± 0.06 m (Figure 5-18).

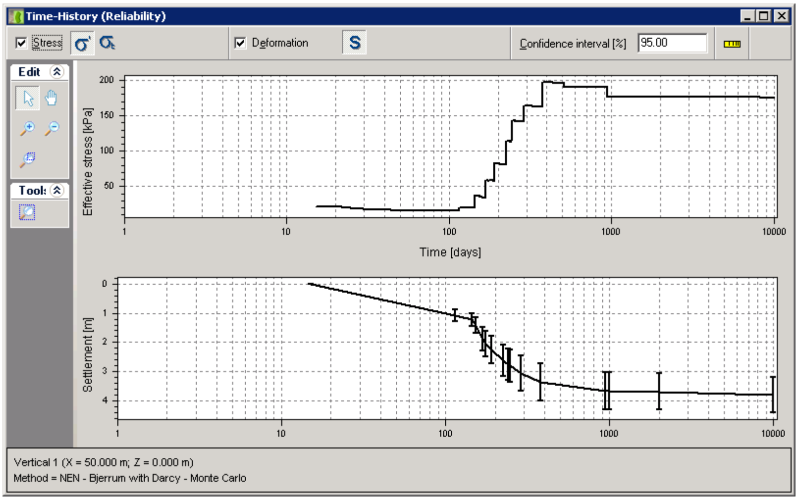


Figure 5-18 – *Time-History (Reliability)* window, Total settlement vs. Time with Band width for Monte Carlo method (Tutorial-3c)

31. Open the *Residual Settlements (Reliability)* window and check that the residual settlement after 1000 days is now approximately 0.13 ± 0.03 m (Figure 5-19), with a probability of 11% that the maximum of 0.15 m is exceeded.

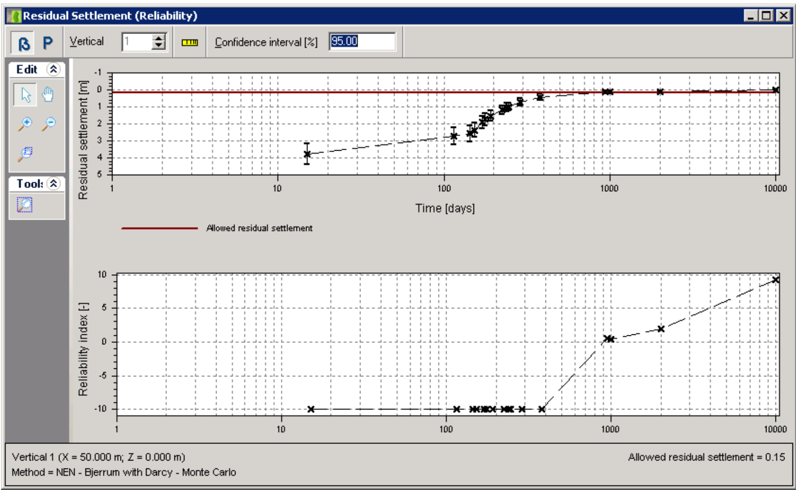


Figure 5-19 – *Residual Settlement (Reliability)* window (Tutorial-3c)

5.5 Conclusion

This tutorial illustrates that the initial uncertainty at the design stage can be reduced significantly during the construction stage, by using measurement data. Conditions for such a significant reduction are however that a large number of measurements is available, in combination with a low imperfection value (0.05 m or less).

6 Tutorial 4: Ground improvement

This tutorial illustrates the modelling of ground improvement using two different methods. To reduce the settlement by embankment construction, part of the original soil (peat) is first replaced by sand.

The objectives of this exercise are:

- To simulate ground improvement (replacing soft soil by a foundation layer of sand)
- To apply a load using different construction stages
- To analyze the settlement results by comparing both methods

For this example, the following MSettle modules are needed:

- MSettle (1D model with Terzaghi)
- 2D geometry model
- Darcy consolidation model

This tutorial is presented in the files Tutorial-4a.sli and Tutorial-4b.sli.

6.1 Introduction

This tutorial includes the ground improvement of part of the actual soil, and the construction of a road embankment including several stages.

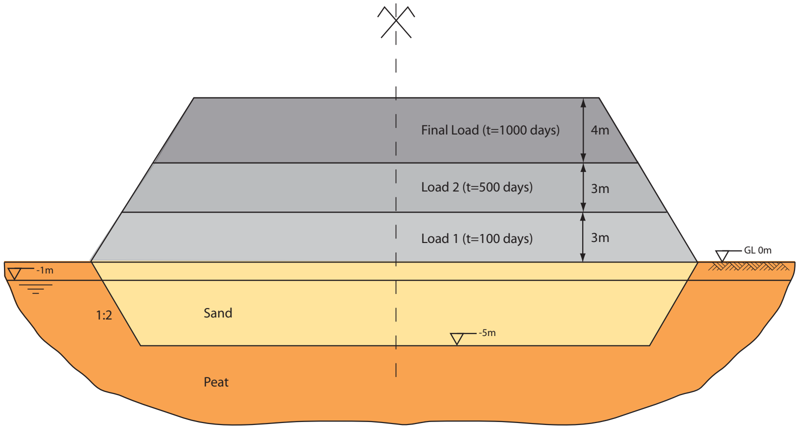


Figure 6-1 – Ground improvement and embankment construction in three stages (Tutorial 4)

Ground improvement
To reduce the settlement by embankment construction, part of the original soil (clay and peat) is first excavated and replaced by sand. There are two ways to simulate soil improvement in MSettle. Method 1 is modelling the excavated soil as initial load. This is the most straightforward method. Drawback is that MSettle will apply some unphysical load distribution for the initial load in horizontal direction; Method 2 is modelling the sand slab as a soil layer with reduced initial weight and additional loading. This enforces MSettle to calculate a proper initial stress distribution and also to calculation deformations and pore pressures in the foundation layer.

Both methods consist in:

- Method 1: excavated soil as an initial load (Tutorial-4a)
  - *Initial stage*: the part of the soil that will be replaced is modeled as an initial non-uniform load. The top surface of the soil layers is therefore located at the bottom of the part that will be excavated. An imaginary surface is defined at this bottom in order to achieve a proper initial stress distribution.
  - *Time t = 0 days*: the excavation is modeled by a reversed initial non-uniform load (negative unit weight) and the replacement by sand is modeled by applying a non-uniform load with the unit weight of sand.
- Method 2: new soil as an initial layer (Tutorial-4b)
  - *Initial stage*: the final foundation layer is already defined in the initial geometry. This layer has the mechanical properties of the improved soil but

the density of the original soil. In this way, proper initial stresses are created.

- *Time t = 0 days*: Replacement is modelled by a non-uniform load, with a unit weight equal to the difference between the sand and the original soil.
- *Time t = 100 days*: A nil load is added to redefine the initial level for subsequent embankment construction (i.e. non-uniform nil load with a top surface at the ground level). This nil load has a zero unsaturated unit weight. The saturated unit weight is equal to the unit weight of water, to neutralize the effect of possible submerging.

NOTE: Method 1 will disturb the real initial stress field due to load distribution.

Embankment

After the soil improvement, a road embankment of 10 m height is constructed including several stages:

- *Time t = 100 days*: first stage of the embankment construction (3 m height).
- *Time t = 500 days*: second stage of the embankment construction (3 m height).
- *Time t = 1000 days*: third stage of the embankment construction (4 m height).

For this tutorial, the a/b/c isotache model is used in combination with the Darcy consolidation model. The a/b/c isotache model enhances the NEN-Bjerrum isotache model, by using natural strain (based on deformed state) instead of linear strain (based on initial state). Natural strains can be advantageous to prevent unphysical large deformations. All parameters for the a/b/c Isotache model can be derived from common oedometer tests. The *OCR* (over-consolidation ratio) is the ratio between the initial vertical preconsolidation stress and the initial field stress. The amount of initial over-consolidation is an important value for the Isotache model, because it defines the initial creep rate that would occur without additional loading.

Table 6-1 – Soil type properties (Tutorial 4)

			Peat	Sand
Saturated unit weight	γ_{sat}	[kN/m^3]	15	17.5
Unsaturated unit weight	γ_{unssat}	[kN/m^3]	15	20
Overconsolidation ratio	*OCR*	[-]	1.1	1.1
Consolidation coefficient	C_v	[m^2/s]	5×10^{-8}	Drained
Reloading/Swelling constant	*a*	[-]	10^{-2}	10^{-6}
Primary compression constant	*b*	[-]	10^{-1}	2×10^{-6}
Secondary compression constant	*c*	[-]	5×10^{-3}	10^{-6}

6.2 Project

To create a new file, follow the steps described below:

1. Click *File* and choose *New* on the MSettle menu bar.
2. Select *New geometry* (Figure 6-2) to create the project geometry.
3. Click *OK*.

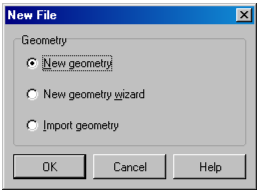


Figure 6-2 – *New File* window

4. Click *Save as* in the *File* menu.
5. Enter <Tutorial-4a> as file name.
6. Click *Save*.

6.2.1 Soil and Consolidation Models

The soil and consolidation models are to be set.

7. Choose *Model* from the *Project* menu to open the *Model* window.
8. Select the *Isotache* soil model and the *Darcy* consolidation model in *2D* geometry (Figure 6-3).
9. Click *OK* to confirm.

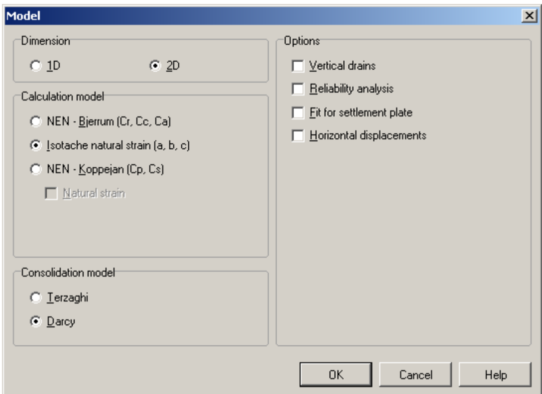


Figure 6-3 – *Model* window

6.2.2 Project Properties

To give the project a meaningful description, follow the steps described below:

10. On the menu bar, click *Project* and then choose *Properties* to open the *Project Properties* window.
11. Fill in <Tutorial 4 for MSettle> and <Ground improvement> for *Title 1* and *Title 2* respectively in the *Identification* tab.
12. In the *View Input* tab, mark the *Points* checkbox of the *Labels* sub-window in order to display the point's number and select the option *As material names* of the *Layers* sub-window in order to display the name of the layers
13. Click *OK*.

6.3 Geometry

In the *Geometry* menu, the geometry aspects of the project can be specified.

6.3.1 Limits

The boundaries of the calculation domain must be specified.

14. Choose *Limits* from the *Geometry* menu to open the *Geometry Limits* window.
15. Enter a *Boundary limit at left* of <-100 m> instead of 0 m.
16. Click *OK*.

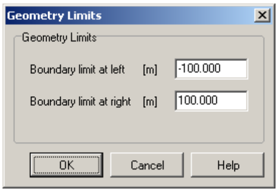


Figure 6-4 – *Geometry Limits* window

6.3.2 Points

All lines (phreatic line, piezometric line or/and boundary layer) in MSettle are connected between points. The different points are defined using the *Add row* button:

17. Choose *Points* from the *Geometry* menu to open the *Points* window.
18. Click the *Add row* button to enter the first point.
19. Click the X co-ordinate of point 1 and enter <-100>.
20. Click the Y co-ordinate of point 1 and enter <0>.
21. Repeat it for the other points (2 to 10) as shown in Figure 6-5.
22. Click *OK*.

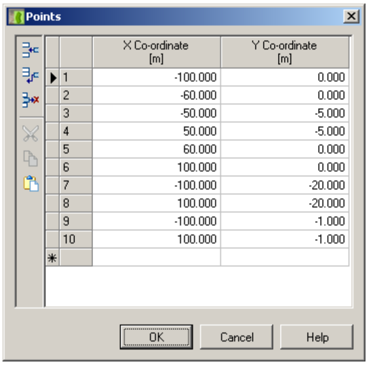


Figure 6-5 – *Points* window

The defined points can now be seen in the *View Input* window. The *Zoom limits* button in the *Tools* panel can be used to optimize the limits of the drawing.

6.3.3 PL-line / Phreatic line

To create the phreatic line, first a PL-line (piezometric level) must be defined:

23. Choose *Pl-lines* from the *Geometry* menu to open the *Pl-Lines* window.
24. Click the *Add* button to create PL-line number <1>.
25. Enter points number <9> and <10> in the *Point number* column at the right of the window (Figure 6-6).
26. Click *OK*.

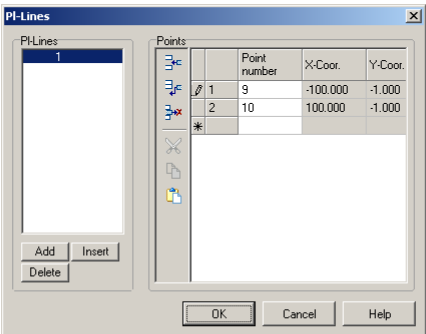


Figure 6-6 – *Pl-Lines* window

The defined phreatic line can now be seen in the *View Input* window.

NOTE: When at least one PL-line is defined in the *Pl-Lines* window, MSettle automatically defined PL-line number 1 to be the phreatic line, as can be seen in the *Phreatic Line* window from the *Geometry* menu (Figure 6-7).

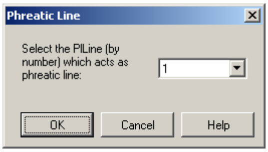


Figure 6-7 – *Phreatic line* window

6.3.4 Layers

After defining the points [§ 6.3.2], the actual layers can now be defined according to Figure 6-1.

27. On the menu bar, click *Geometry* and then choose *Layers*.
28. In the *Layers* window that appears, click the *Add* button to create boundary number <0>. Remember that layer number 0 is never a physical layer but defines the base of the project.
29. Enter points number <7> and <8> in the *Point number* column at the right of the window.
30. Add boundary number <1> by clicking the *Add* button and enter point's number <1>, <2>, <3>, <4>, <5> and <6>.

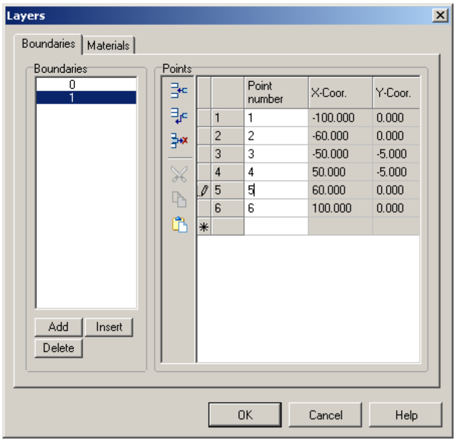


Figure 6-8 – *Layers* window, *Boundaries* tab

31. Select the *Materials* tab of the *Layers* window to define a soil type for each layer.

On the left of the window (Figure 6-9), a list containing default available materials is displayed.

32. Assign material *Peat* to layer number 1 as shown in Figure 6-9 by clicking the  button.
33. Click *OK* to confirm the input.

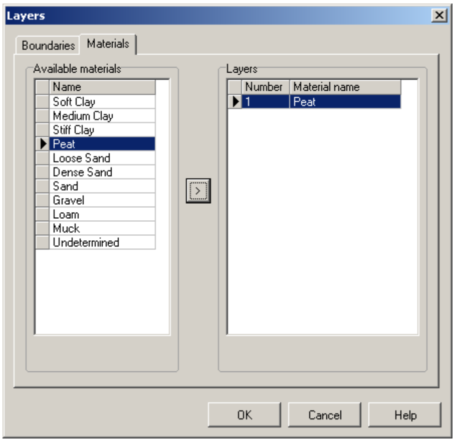


Figure 6-9 – *Layers* window, *Materials* tab

The defined layer and phreatic line can now be seen in the *View Input* window (Figure 6-10).

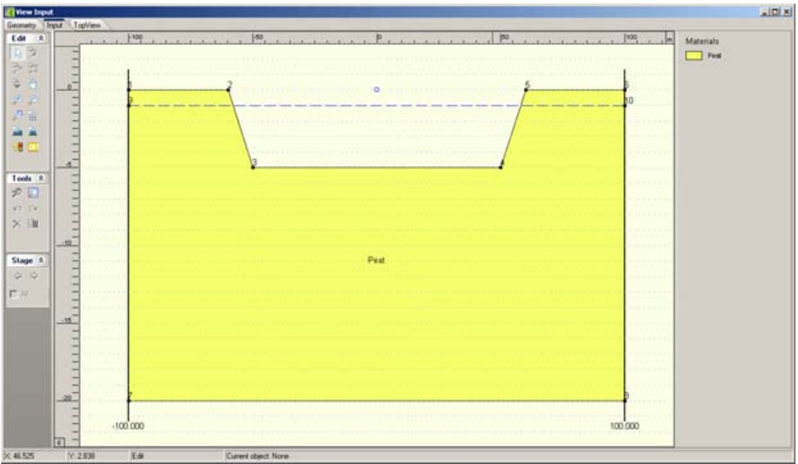


Figure 6-10 – *View Input* window, *Input* tab

6.4 Method 1 for ground improvement

6.4.1 Soil properties

In the *Soil* menu, the properties of the *Peat* layer given in Table 6-1 can be inputted.

34. Choose *Materials* from the *Soil* menu to open the *Materials* window.
35. Select *Peat* in the material list and enter the soil properties values of this layer as indicated in Table 6-1 in both tabs.
36. Click *OK* to confirm.

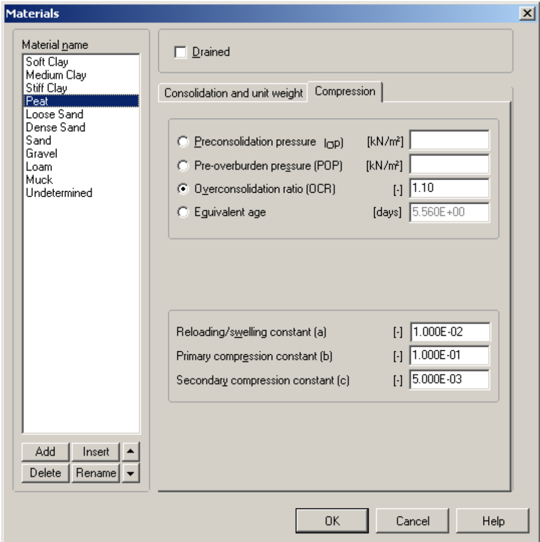


Figure 6-11 – *Materials* window, *Compression* tab for *Peat*

6.4.2 Loads

As explained in [§ 6.1], the soil that has to be excavated is modeled as an initial non-uniform load with the same unit weight as the *Peat* layer.

37. From the *Loads* menu, choose *Non-Uniform Loads* to open the input window.
38. In the *Load name* sub-window, click the *Add* button and rename the load with name <Initial soil>.
39. Mark the *Initial load* checkbox.
40. Enter a *Total unit weight* above and below phreatic level of <15> (as for *Peat* [§ 6.4.1]).
41. Enter two points using the *Add row*  button with X co-ordinate of <-60> and <60> and Y co-ordinate of <0> (see Figure 6-12).

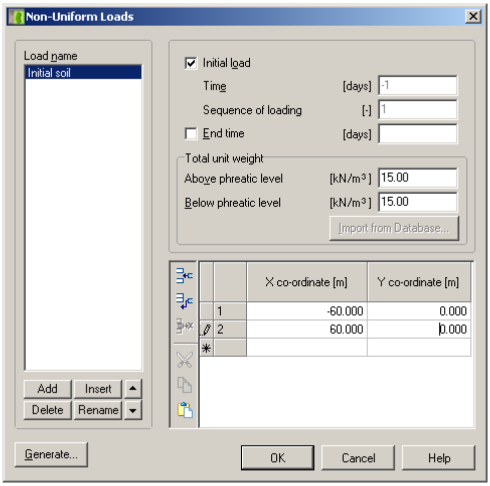


Figure 6-12 – *Non-Uniform Loads* window

As explained in [§ 6.1], at time 0 day, the excavation is modelled by simply adding a reversed initial non-uniform load (by means of a negative unit weight) and the refilling with sand material is modeled by applying a non-uniform load (with the same unit weight as the sand material).

42. Click the *Add* button and rename the load with name <Excavation>.
43. Unmark the *Initial load* checkbox.
44. Enter a *Time* of <0 days> and a *Total unit weight* above and below phreatic level of <-15>.
45. The bottom boundary of the excavation includes four points: select the second row and use the *Insert row* button to insert two rows between the two existing rows. Enter co-ordinates X of <-50> and Y of <-5> for point 2 and X of <50> and Y of <-5> for point 3 as shown in Figure 6-13 (left).
46. To model the refilling with sand material, select the load *Initial soil* previously defined and click the *Add* button. Rename the load with name <Improvement>.
47. Unmark the *Initial load* checkbox and enter a *Total unit weight* of <17.5> and <20> respectively above and below phreatic level. The co-ordinates don't need to be modified as the top boundary of the *Improvement* load is the same as the *Initial soil* load (Figure 6-13).

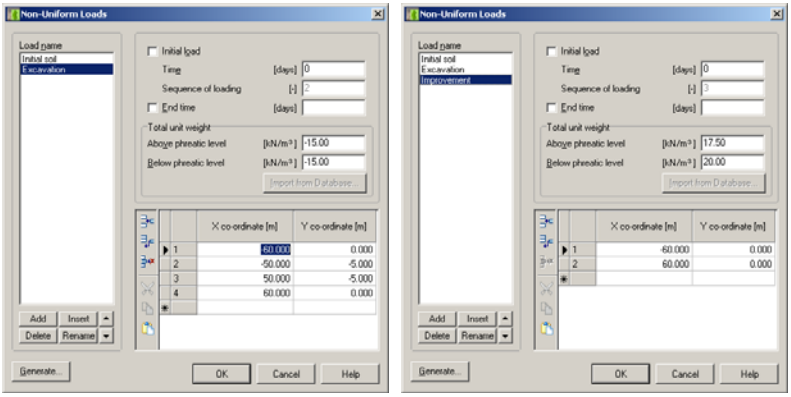


Figure 6-13 – *Non-Uniform Loads* window

After the soil improvement, now enter the three stages of the embankment construction by using the *Generate* button.

48. Click the *Generate* button at the bottom of the *Non-Uniform Loads* window to open the *Generate Non-Uniform Loads* window.
49. In the *Envelope Points* tab, enter the co-ordinates of the points that define the envelope of the road embankment, as given in Figure 6-14 to be in accordance with Figure 6-1.

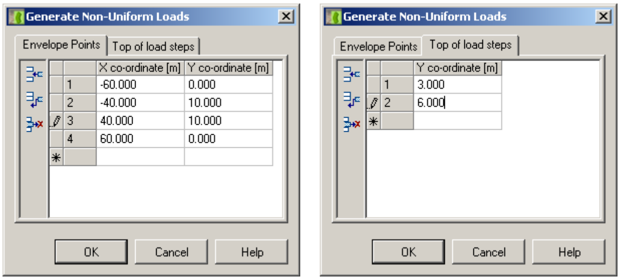


Figure 6-14 – *Generate Non-Uniform Loads* window

50. Select the *Top of load steps* tab and enter the two intermediate values at <3 m> and <6 m> (Figure 6-14).
51. Click *OK* to generate the loads.
52. Rename load *Generated load (1)* with name <Load 1> and enter a *Time* of <100 days>.
53. Rename load *Generated load (2)* with name <Load 2> and enter a *Time* of <500 days>.
54. Select *Final load* and enter a *Time* of <1000 days>.
55. Click *OK* to confirm.

The non-uniform loads are now displayed in the *Input* tab of the *View Input* window. The *Zoom limits* button in the *Tools* panel can be used to optimize the limits of the drawing (Figure 6-15).

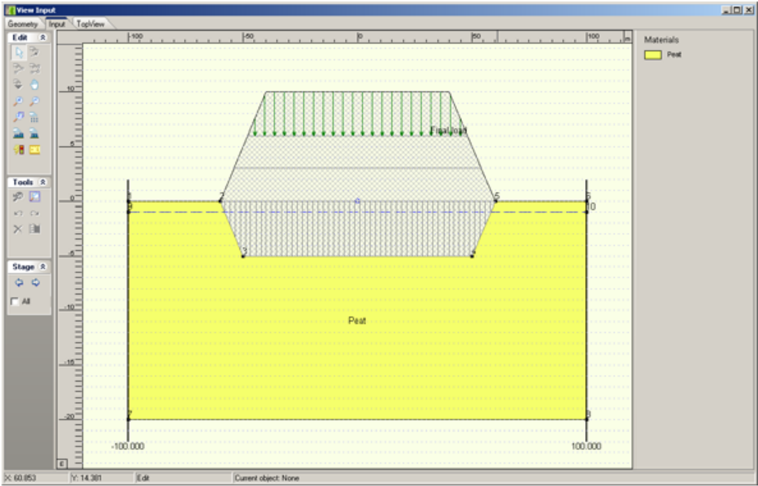


Figure 6-15 – *View Input* window, *Input* tab

To visualize the sequence of loading, use the *Previous stage* and *Next stage* buttons in the *Stage* panel.

6.4.3 Verticals

A sufficient number of verticals must be defined to get a good impression of the settlement distribution.

56. Choose *Verticals* from the *GeoObjects* menu to open the input window.
57. Select *Interval* in the *Automatic generation x co-ordinates* sub-window.
58. Enter a *First* and a *Last* point with X co-ordinate of respectively <0 m> and <60 m>, and enter an *Interval* of <10 m>. Because of symmetry, verticals are generated only for half part of the embankment.
59. Click the *Generate* button.
60. Click *OK* to confirm.

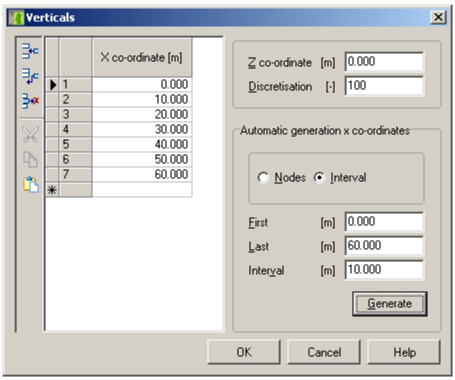


Figure 6-16 – *Verticals* window

6.4.4 Calculation Options

The top surface of the soil layers is located at the bottom of the excavation (i.e. top of the *Peat* layer). Therefore an imaginary surface is defined at this bottom in order to achieve a proper initial stress distribution.

61. Choose *Options* from the *Calculation* menu.
62. Mark the checkbox *Output of settlements by partial loading (green lines)* in order to view in the *Time-History* window the settlements due to each load-step [§ 6.4.5].
63. Mark the *Imaginary surface* checkbox.
64. Leave other options like submerging (decrease of effective load by submerging) to their default settings.
65. Click *OK* to confirm.

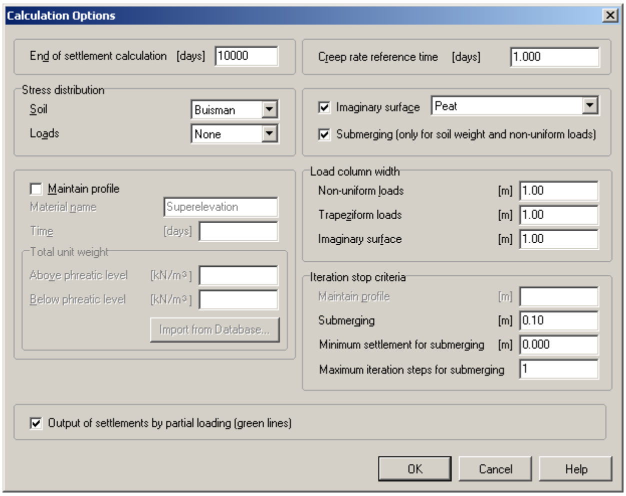


Figure 6-17 – *Calculation Options* window

6.4.5 Results of Method 1

66. Choose *Start* from the *Calculation* menu or press the function key F9.
67. Click *OK* to start the calculation.
68. Choose the *Time-History* option in the *Results* menu.
69. In the *Time-History* window displayed, inspect the results for each vertical using the scroll arrows of the *Vertical* box, at the top of the window. Vertical 1 at the axis of the embankment (Figure 6-18) gives the largest final settlements.

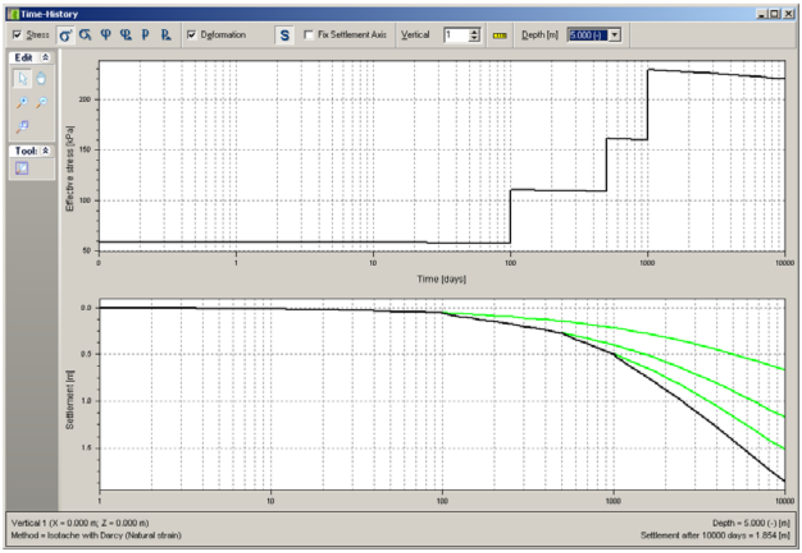


Figure 6-18 – *Time-History* window for vertical 1 (Tutorial-4a)

6.5 Method 2 for ground improvement

The second method models the sand foundation as an initial layer and uses an additional load to add the additional weight. Therefore a new *Sand* layer must be introduced in the project.

6.5.1 Defining the Sand layer

70. Click *Save As* in the file menu and save this tutorial as <Tutorial-4b>.
71. Click *Save*.
72. Select *Material* in the *Soil* menu to open the *Material* window.
73. Select the *Sand* material.
74. In the *Consolidation and unit weight* tab, mark the *Drained* checkbox as indicated in Table 6-1 for *Sand* but for the weight, enter the same unit weights (below and above the phreatic level) as the *Peat* layer (i.e. <15>).
75. In the *Compression* tab, enter the soil properties as indicated in Table 6-1 for *Sand*.
76. Click *OK*.
77. On the menu bar, click *Geometry* and then choose *Layers*.
78. In the *Layers* window that appears, click the *Add* button to create boundary number <2>.
79. Enter points number <1>, <2>, <5> and <6> in the *Point number* column at the right of the window.
80. In the *Materials* tab of the *Layers* window, assign the *Sand* material to *Layer* number 2 using the [>] button.

81. Click *OK* to confirm the input.

6.5.2 Modelling the soil improvement

As explained in [§ 6.1], at time t = 0 days, the additional density due to soil improvement is modelled as a non-uniform load (with an effective unit weight equal to the difference between the initial *Peat* material and the new *Sand* material):

- Above phreatic level: 17.5 – 15 = 2.5 kN/m^3;
- Below phreatic level: 20 - 15 + 9.81 = 14.81 kN/m^3.

82. From the *Loads* menu, choose *Non-Uniform Loads* to open the input window.
83. Delete the existing loads *Initial Soil* and *Excavation* by selecting them and clicking the *Delete* button.
84. Select the *Improvement* load and enter unit weights equal to the additional density: <2.5 kN/m^3> above and <14.81 kN/m^3> below the phreatic level.
85. In the co-ordinates table, enter the co-ordinates of the four points of the excavation boundary as given in Figure 6-19 (left)

A nil load must now be added at time 100 days to redefine the initial level for subsequent embankment construction (i.e. non-uniform nil load with a top surface at the ground level). This nil load has a zero unsaturated unit weight and a saturated unit weight equal to the unit weight of water, to neutralize the effect of possible submerging:

86. Select *Load 1* and click the *Insert* button.
87. Rename the load with <Step to surface> and enter unit weights of <0 kN/m^3> and <9.81 kN/m^3> respectively above and below the phreatic level.
88. In the co-ordinates table, delete points 2 and 3 using the *Delete row* button  in order to keep only the top surface boundary as shown in Figure 6-19 (right).
89. In the *Calculation Options* window, unmark the *Imaginary surface* checkbox.

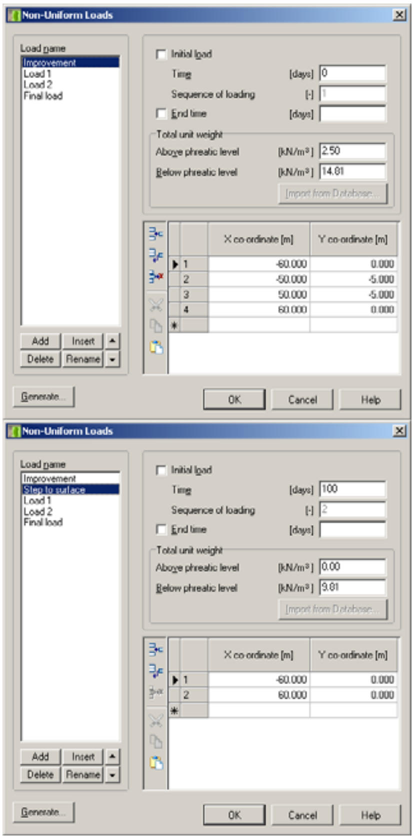


Figure 6-19 – *Non-Uniform Loads* window (Tutorial-4b)

6.5.3 Results of Method 2

90. Press the function key F9 to start the calculation.

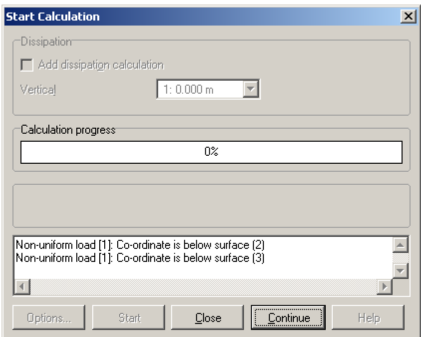


Figure 6-20 – *Start Calculation* window (Tutorial-4b)

As the *Improvement* load is below the ground surface, warning messages appear in the *Start Calculation* window (Figure 6-20).

91. Click the *Continue* button to continue the calculation.
92. Choose the *Time-History* option in the *Results* menu.
93. In the *Time-History* window displayed, inspect the results for each vertical using the scroll arrows of the *Vertical* box, at the top of the window. Note that vertical 1 (Figure 6-21) gives the more important final settlements.

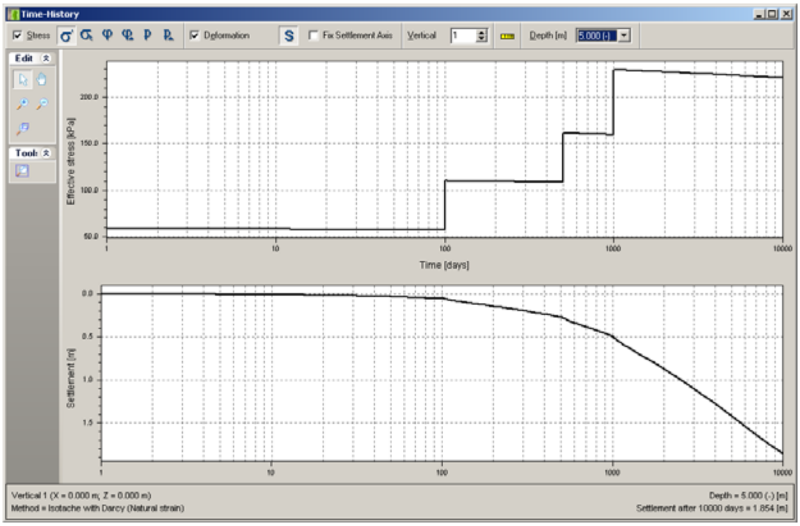


Figure 6-21 – *Time-History* window for vertical 1 (Tutorial-4b)

Practically no deformation occurs from depth 0 m to depth -5 m, because of the relatively low compressibility of the *Sand* layer (from depths 0 m to -5 m) To illustrate this:

94. Select depth <0.000 m> of the *Depth* box and then use the scroll button of the mouse to display in a continuous way the results at each depth.

Another way to illustrate this is to use the *Depth-History* window:

95. Open the *Depth-History* window from the *Results* menu.
96. Select the final time <10000 days> from the drop-down menu of the *Time* box.

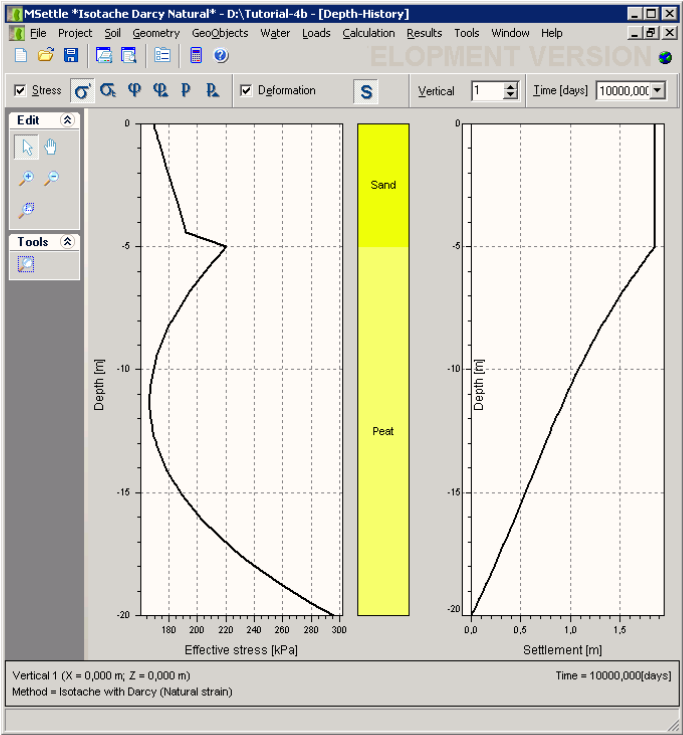


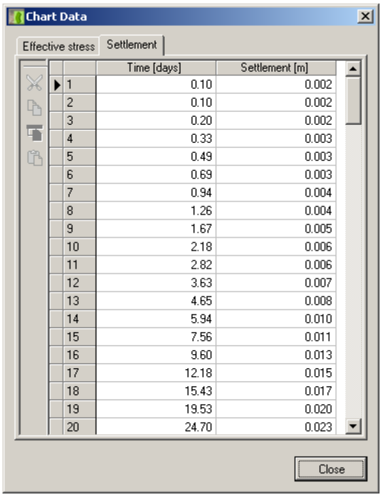
Figure 6-22 – *Depth-History* window (Tutorial-4b) after 10000 days

The settlement chart displayed (Figure 6-22) shows that almost no settlement occurs in the top sand layer called *Sand*. Note that excess pressures are still significant at 10000 days.

6.6 Comparison of both ground improvement methods

To compare the settlement and loading curves of both methods, the data from MSettle graphs are exported to spread sheets:

97. In the *Time-History* window, click with the right hand mouse button in the graph area.
98. Select *View Data*.
99. In the *Chart Data* window displayed (Figure 6-23), select the columns with the mouse.
100. Use the *Copy* button to copy the data to the Windows clipboard.



	Time [days]	Settlement [m]
1	0.10	0.002
2	0.10	0.002
3	0.20	0.002
4	0.33	0.003
5	0.49	0.003
6	0.69	0.003
7	0.94	0.004
8	1.26	0.004
9	1.67	0.005
10	2.18	0.006
11	2.82	0.006
12	3.63	0.007
13	4.65	0.008
14	5.94	0.010
15	7.56	0.011
16	9.60	0.013
17	12.18	0.015
18	15.43	0.017
19	19.53	0.020
20	24.70	0.023

Figure 6-23 – *Chart Data* window (vertical 1 of Tutorial-4b)

Using the steps described above, both chart data's (for both methods) can be pasted in a spreadsheet for direct comparison as shown in Figure 6-24 for settlement curve and Figure 6-25 for effective stress curve. Those figures show that both methods give approximately the same results in vertical 1.

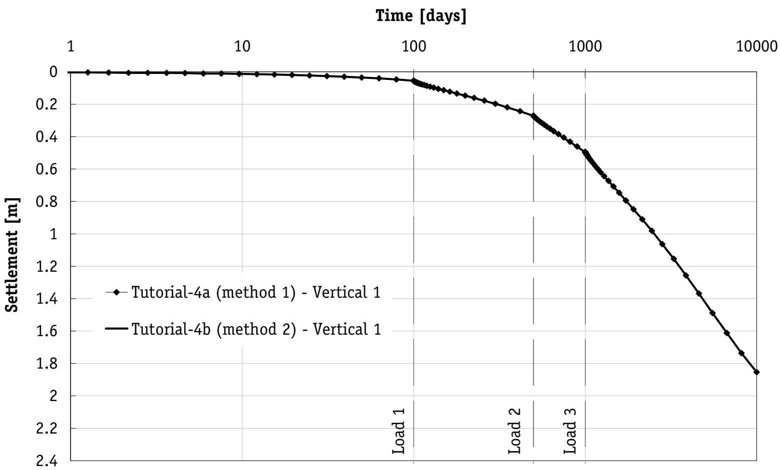


Figure 6-24 – Settlement vs. Time – Comparison between methods 1 and 2

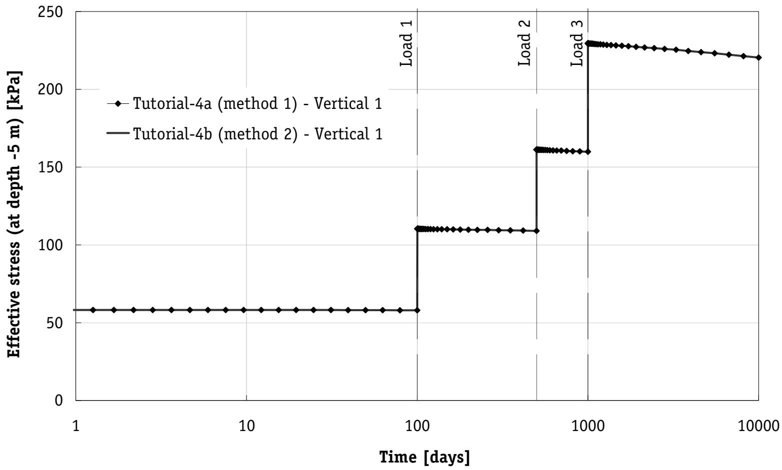


Figure 6-25 – Effective stress vs. Time – Comparison between methods 1 and 2

6.7 Conclusion

Two methods were demonstrated to model ground improvement with MSettle. Modeling of the ground improvement as an initial load is the most straightforward method. This method will however disturb the true initial stress distributions outside the centre of the embankment. Modeling of the ground improvement as an initial soil layer yields proper initial stresses. Results from both methods at the centre of the embankment are comparable for these embankment dimensions.

7

Tutorial 5: Enforced dewatering by sand screens (IFCO)

This tutorial illustrates the modelling of sand screens in combination with enforced dewatering (IFCO method) for the construction of a new Schiphol airport runway. This example has also been described in Dutch literature [Lit 15] and [Lit 16].

The objectives of this exercise are:

- To import the soil type properties from an MGeobase database;
- To model soil drainage by sand screens with enforced dewatering;
- To model ground improvement.

For this example, the following MSettle modules are needed:

- MSettle (1D model with Terzaghi)
- 2D geometry model
- Darcy consolidation model
- Vertical drains

This tutorial is presented in the files Tutorial-5a.sli to Tutorial-5c.sli.

7.1 Introduction

A new runway at a height of about 1.2 m above ground level has to be constructed. Sand screens with enforced dewatering (IFCO method) are used, because of the severe constraints on building time (short) and residual settlement (small). A general view of this project is shown in Figure 7-1.

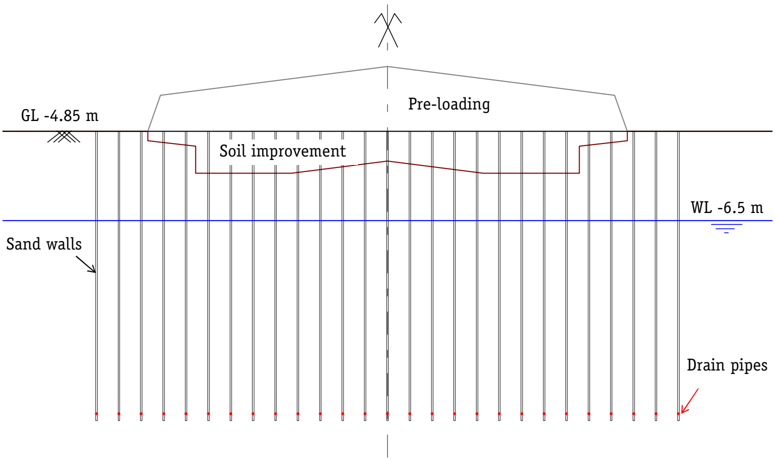


Figure 7-1 – General view with pre-loading and sand walls (Tutorial 5)

7.1.1 Excavation and loading stages

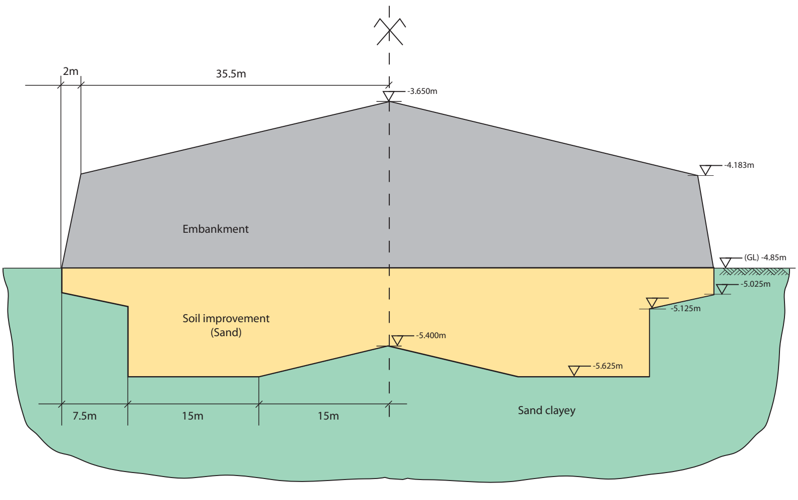


Figure 7-2 – Geometry of the excavation and pre-loading phases (Tutorial 5)

The following stages are modelled, up to and including the sand embankment construction.

- At time 0 day: Excavation of the subsoil, providing space for the foundation layer, until roughly 0.55 m below the ground level;
- At time 12 days: Filling of the foundation trench with sand;
- At time 19 days: Installation of sand screens and start of enforced dewatering;
- At time 39 days: embankment raise to a level of 1.2 m.

The added sand has an unsaturated and a saturated unit weight of respectively 17.5 and 20 kN/m^3.

7.1.2 Subsoil characterization

For the characterization of the subsoil, a boring is made nearby the studied location. Results are shown in Figure 7-3.

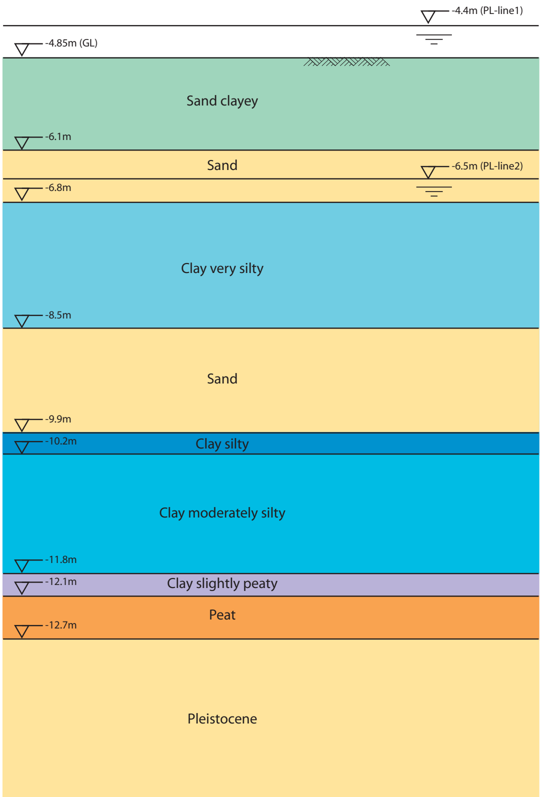


Figure 7-3 – Layers in the subsoil (Tutorial 5)

The compression related parameters of the six soft layers were determined from K_0-CRS (constant rate of strain) tests, each with an unloading/reloading branch. This test type allows a more accurate determination of the primary compression parameters and the preconsolidation stress, compared to an oedometer test. The resulting parameters are given in Table 7-1. Note that the POP value is very large for Dutch conditions.

Table 7-1 – Soil properties from K_0-CRS test (Tutorial 5)

		Pleisto-cene	Sand	Sand clayey	Clay very silty	Clay silty	Clay mod. silty	Clay slight. peaty	Peat
γ_{unssat}	[kN/m^3]	18	15.7	14.4	9.9	9.1	7.8	5.9	2.5
γ_{sat}	[kN/m^3]	20	19.5	18.7	16.0	15.5	14.4	13.3	10.5
$k_{v;0}$	[10^{-4} m/d]	-	1.3	7.2	2.7	0.6	7.0	0.53	7.9
C_k	[-]	-	0.01	0.082	0.353	0.396	0.209	0.316	0.213
k_h/k_v	[-]	-	1	1	1	1	1	1	4
a	[-]	10^{-6}	0.0002	0.0031	0.0085	0.0090	0.0134	0.0143	0.0211
b	[-]	2.10^{-6}	0.0419	0.0452	0.1197	0.1795	0.1825	0.2389	0.3225
c	[-]	0	0	0.0017	0.0025	0.0101	0.0109	0.0149	0.0187
POP	[kPa]	0	20	91.4	35.6	63.5	47.5	85.0	151.0

7.1.3 Drainage using sand screens and dewatering

The IFCO drainage method is based on the combination of sand screens with enforced dewatering during pumping. The enforced dewatering will cause temporary pre-loading by lowering of the water table and sometimes also by creating additional under pressure via sealing.

The sand screens are constructed roughly perpendicular to the axis of the runway, with a width of 0.25 m, a depth of 10.2 m below reference level and a distance of 3.5 m. Horizontal drain pipes are installed inside each screen at a depth of 10.075 m below reference level. A reduced pressure of 10 kPa is applied in the drain pipe during pumping.

Moreover, the runway is sealed from surrounding water and air pressure by means of bentonite shields and an impermeable foil. This way, an additional air underpressure of 30 kPa is created at the top of the trenches.

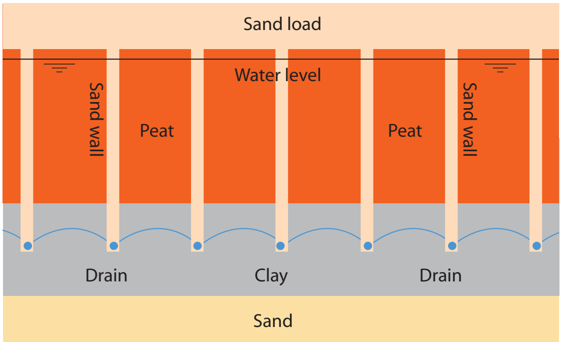


Figure 7-4 – IFCO system (sand walls)

7.2 Project

How to define the layers geometry and soil properties has been explained already in the previous tutorials. Use the different figures and data's given in [§ 7.1] to create the geometry and then proceed with [§ 7.5] for the description of the additional steps.

However, an alternative to the manual input is to import the geometry from a so-called GEO file [§ 7.2.1] and to import the soil properties from an MGeobase database [§ 7.3.1].

7.2.1 Importing an existing geometry

To import the geometry from a GEO file, follow the steps below.

1. In the *File* menu, select *New* to open the *New File* window (Figure 7-5).
2. Select the *Import geometry* option and click *OK*.

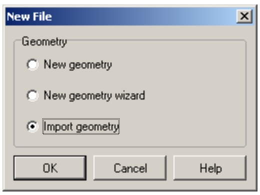


Figure 7-5 – *New File* window

3. In the *Import Geometry From* window displayed, select the GEO file named <Tutorial-5.geo> located in the *Examples* folder where the *MSettle* program was installed.
4. Click *OK*.

The predefined geometry is displayed in the *Geometry* tab of the *View Input* window (Figure 7-6). This imported geometry contains only the points, the layers boundary and the PL-lines, not the material types and properties. They will be imported from an MGeobase database [§ 7.3.1].

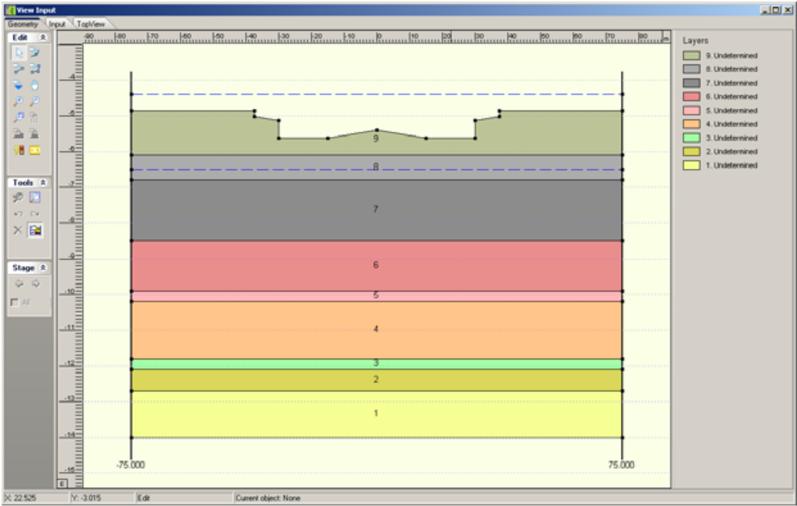


Figure 7-6 – *View Input* window, *Geometry* tab after importing geometry

5. Click *Save as* in the *File* menu, enter <Tutorial-5a> as file name and click *Save*.

7.2.2 Model

The soil and consolidation models, as well as the use of vertical drainage are to be set.

6. In the *Model* window from the *Project* menu, select the *Isotache* soil model and the *Darcy* consolidation model in *2D* geometry and mark the *Vertical drains* checkbox

7.3 Soil materials

The layers geometry is already modelled however the material properties, phreatic line and piezometric levels per layer still need to be defined.

7.3.1 Importing material properties from an MGeobase database

The parameters from Table 7-1 were saved in an MGeobase database. To import them, the location of this MGeobase database must be first specified:

7. In the *Program Options* window from the *Tools* menu, select the *Directories* tab.

8. Mark the *Use MGeobase database* checkbox and click the *Browse* button to specify the location of the MGeobase database with material data.
9. In the *Open project database* window displayed, select the MDB file named <Tutorial-5.mdb> located in the *Examples* folder where the *MSettle* program was installed.
10. Click *Open* and then *OK*.

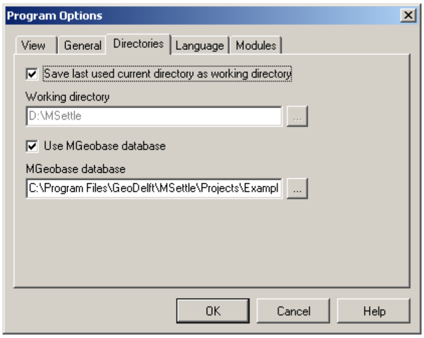


Figure 7-7 – *Program Options* window, *Directories* tab

The soil properties of each material given in Table 7-1 can now be imported from this MGeobase file:

11. Open the *Materials* window from the *Geometry* menu and select the *Database* tab.
12. Select *Pleistocene* in the material list of the *Database* tab and click the button to import this soil type (with associated properties) in the material list of the *Materials* window (Figure 7-8).
13. Repeat it for the 7 other materials.
14. In the *Parameters* tab, check that the imported properties are the same as in Table 7-1.
15. Click *OK*.

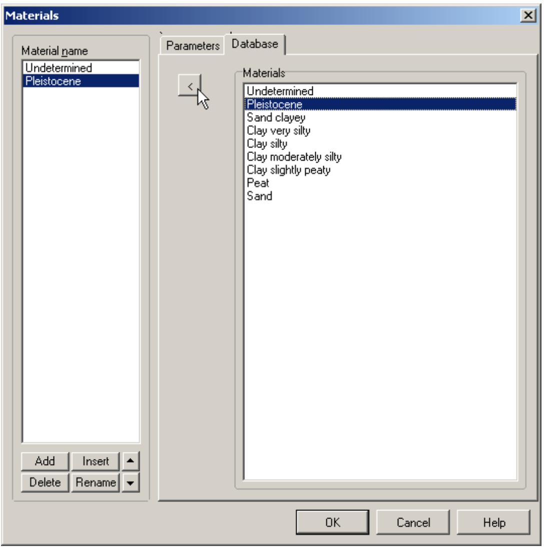


Figure 7-8 – *Materials* window, *Database* tab

7.3.2 Layers

To assign each material to a layer:

16. Select the *Materials* tab of the *Layers* window.
17. First select *Pleistocene* in the *Available materials* sub-window at the left and in the *Layers* sub-window at the right select *Number* <1>.
18. Then click the  button.
19. Repeat it for the eight other layers (nr. 2 to 9) as shown in Figure 7-9.
20. Click *OK* to confirm the input.

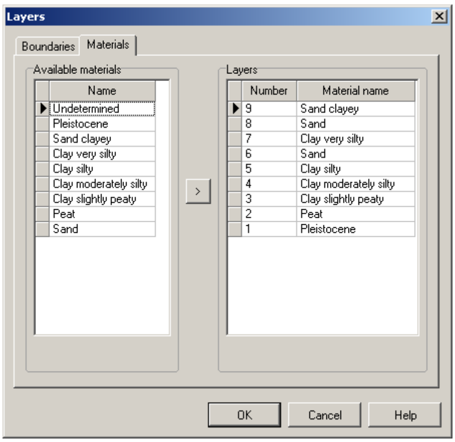


Figure 7-9 – *Layers* window, *Materials* tab

7.4 Piezometric Levels

7.4.1 Phreatic Line

21. In the *Phreatic Line* window from the *Geometry* menu, select PL-line number <2> at level -6.5 m as phreatic line.

7.4.2 PL-lines per Layer

In this project, the piezometric level at the ground surface corresponds with the phreatic line (i.e. PL-line number 1 at depth -6.5 m) and the piezometric level in the Pleistocene layer is at -4.4 m (i.e. PL-line number 2). In between, a linear distribution is assumed:

22. Open the *PL-lines per Layer* window from the *Geometry* menu and note that the eight layers are already defined with PL-line number 1 as default.
23. For layer 1 (i.e. Pleistocene), leave PL-line number <1> at both top and bottom.
24. For layer 8 (i.e. top layer), enter PL-line number <2> at the top.
25. Enter <99> in all other cells of the table to indicate a linear distribution (Figure 7-10): the interpolation will take place between the PL-line belonging to the first soil layer above with a real PL-line number (i.e. not equal to 99), and the PL-line belonging to the first soil layer below with a real PL-line number.

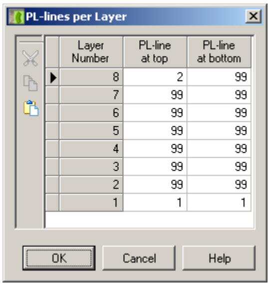


Figure 7-10 – *PL-lines per Layer* window

7.5 Loads

7.5.1 Modeling the soil improvement

The soil that has to be excavated is modeled as an initial non-uniform load with the same unit weight as the original layer (i.e. *Clay very silty 1*). This method is explained in detail in Tutorial 4 [§ 6.1]).

26. From the *Loads* menu, choose *Non-Uniform Loads* to open the input window.
27. In the *Load name* sub-window, click the *Add* button and rename the load to <Initial state>.
28. Mark the *Initial load* checkbox and enter a *Total unit weight above* and *below phreatic level* of respectively <14.4> and <18.7> kN/m^3 (same as for *Sand clayey*).

29. Enter two points using the *Add row* button with X co-ordinate of <-37.5> and <37.5> and Y co-ordinate of <-4.85>.

The excavation is modelled by simply adding a reversed initial non-uniform load at time 0, by means of a negative unit weight:

30. Click the *Add* button and rename the load to <Excavation>.
31. Unmark the *Initial load* checkbox.
32. Enter a *Time* of <0 days> and a *Total unit weight above* and *below phreatic level* of respectively <-14.4 and <-18.7> kN/m³.
33. Enter the co-ordinates of the excavation boundary given in Figure 7-11 (left).

The filling with sand material is modeled by applying a non-uniform load (with the same unit weight as the sand material) until the ground surface:

34. Select the previously defined load *Initial state,* and click the *Add* button. Rename the load to <Fill>.
35. Unmark the *Initial load* checkbox and enter a *Time* of <12> days.
36. Enter a *Total unit weight above* and *below phreatic level* of respectively <17.5> and <20> kN/m³. The co-ordinates don't need to be modified, as the top boundary of the *Fill* load is the same as the *Initial state* load (Figure 7-11, right).

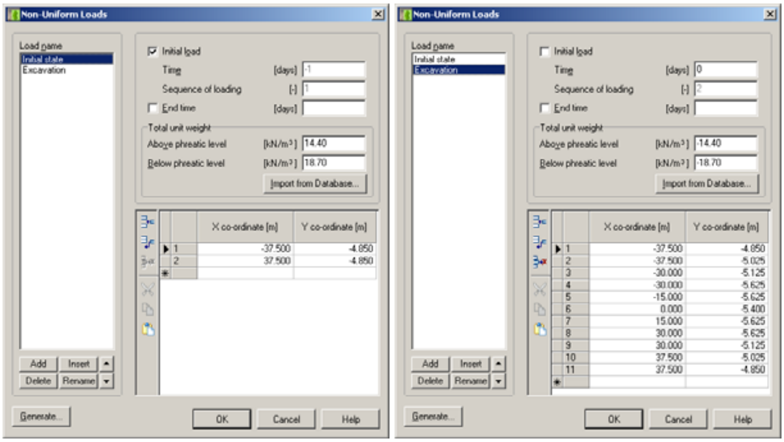


Figure 7-11 – *Non-Uniform Loads* window, *Initial state* and *Excavation* loads

7.5.2 Modelling the embankment construction

The sand embankment construction is modelled by applying a non-uniform load with the unit weight of sand and with the embankment profile:

37. Click the *Add* button. Rename the load to <Embankment>.
38. Enter a *Time* of <39> days.
39. Enter a *Total unit weight above* and *below phreatic level* of respectively <17.5> and <20> kN/m³. The position of the foil is given in the table of co-ordinates in Figure 7-12 (left).

40. Repeat it for the last load named <Embankment> using the values of Figure 7-12 (right).
41. Click *OK* to confirm.

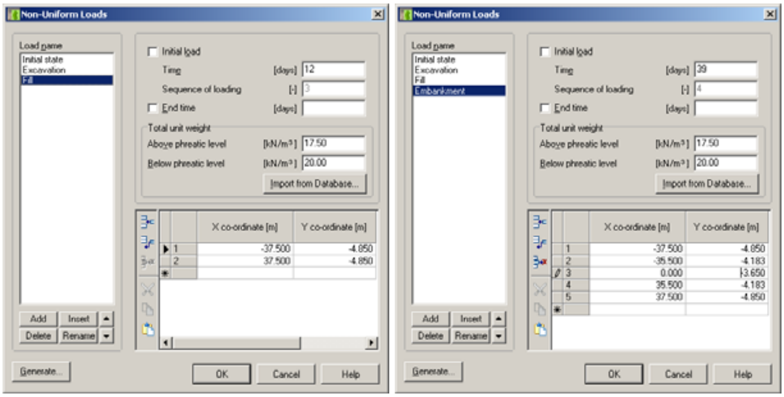


Figure 7-12 – *Non-Uniform Loads* window, *Fill* and *Embankment* loads

42. In the *View Input* window, select the *Input* tab to view the non-uniform loads and use the *Previous stage* and *Next stage* buttons in the *Stage* panel to visualize the sequence of loading.

7.6 Verticals

In this project only one calculation vertical is defined at the centre of the embankment.

43. Choose *Verticals* from the *GeoObjects* menu to open the input window.
44. Enter X co-ordinate of <0 m> and click *OK* to confirm.

7.7 Vertical Drains

Perform the following steps for definition of the sand screens.

45. In the *GeoObjects* menu, select *Vertical Drains* to display the corresponding window.
46. Select <Sand wall> as *Drain Type* and <Simple Input> of *Enforced Dewatering*.
47. Enter the values given in Figure 7-13 .

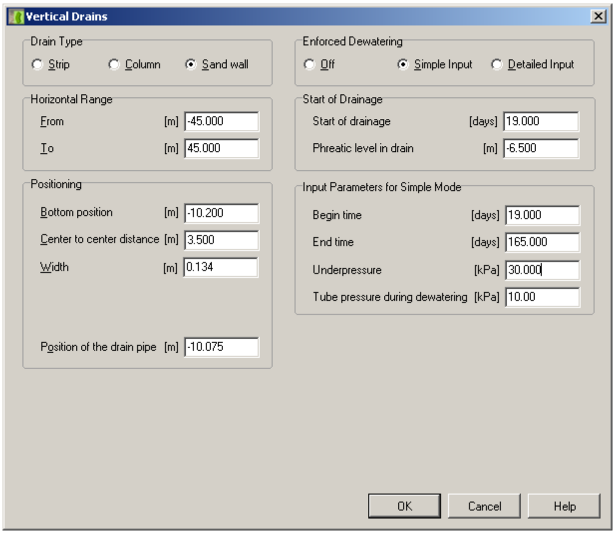


Figure 7-13 – *Vertical Drains* window for *Sand wall*

7.8 Calculation Times

48. Choose *Times* from the *Calculation* menu, and enter the times for calculation of residual settlements, according to Figure 7-14.

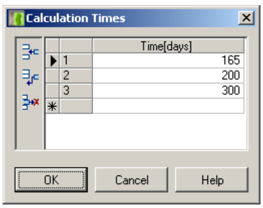


Figure 7-14 – *Calculation Times* window

7.9 Results

49. Press the function key F9 to start the calculation and click *Start*.

7.9.1 Settlements vs. time curve

50. Choose the *Time-History* option in the *Results* menu to view the settlements versus time (Figure 7-15). The final settlement is 0.189 m.

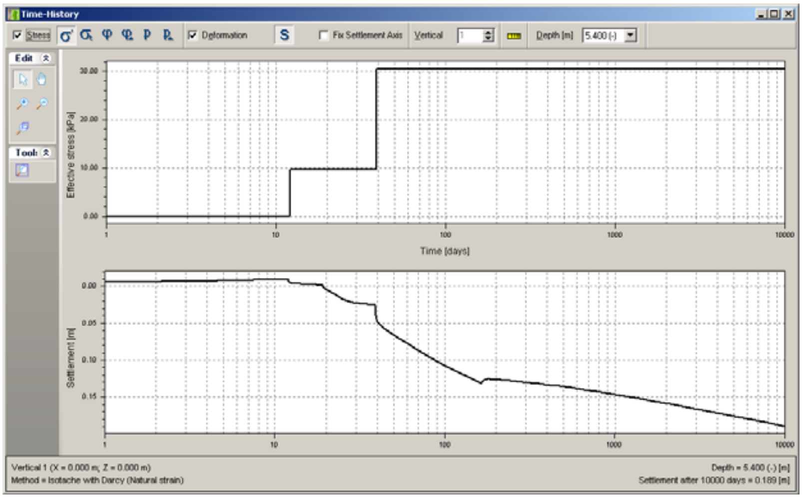


Figure 7-15 – *Time-History* window, dewatering with underpressure (Tutorial-5a)

7.9.2 Residual settlements vs. time curve

51. Choose the *Residual Settlement* option in the *Results* menu to view the residual settlements versus time (Figure 7-16).

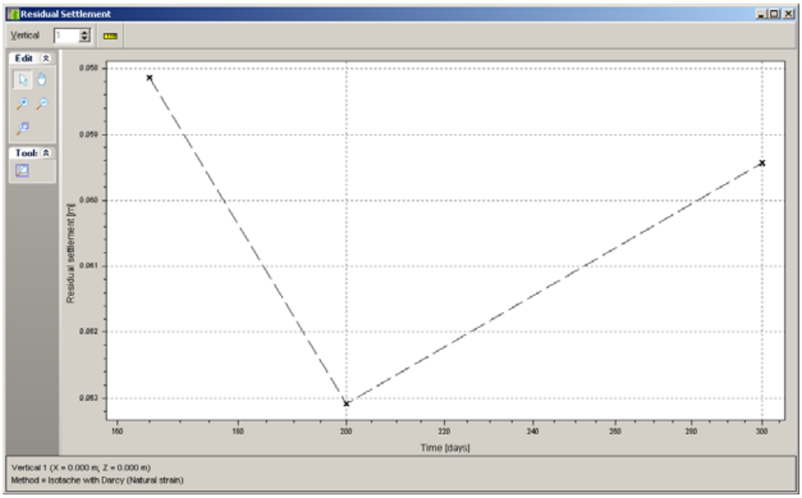


Figure 7-16 – *Residual Settlement* window, dewatering with underpressure (Tutorial-5a)

7.9.3 Excess hydraulic head vs. depth curve

52. Choose the *Depth-History* option in the *Results* menu to view the excess head along the depth at different times, at 10000 days for example (Figure 7-17).

Note that the apparent excess head at 10000 days is not caused by loading. This difference between the final and initial (user-defined) head distribution is caused by the effect of the sand screens. MSettle assumes after dewatering in the drains a hydrostatic pore pressure distribution below the user-defined position of the phreatic level.

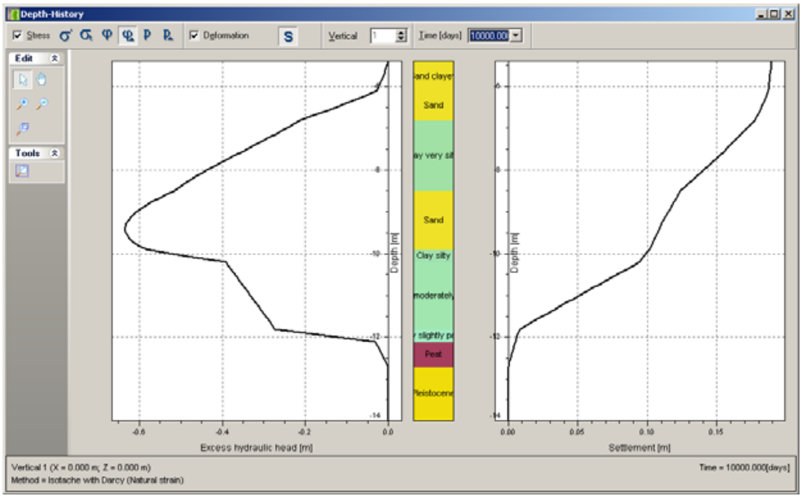


Figure 7-17 – *Depth-History* window, excess head at 10000 days (Tutorial-5a)

7.9.4 Effect of the enforced air underpressure (Tutorial-5b)

In case of perfect sealing at the top of the sand screens, the enforced air underpressure is equal to 30 kPa. A second calculation is performed, using a safe value of 0 kPa.

53. Save the current project as <Tutorial-5b>.
54. In the *Vertical Drains* window, enter an *Underpressure* of <0 kPa>.
55. Start the calculation via the *Calculation* menu.
56. Select *Time-History* in the *Result* menu to see the settlement results of this calculation without underpressure.

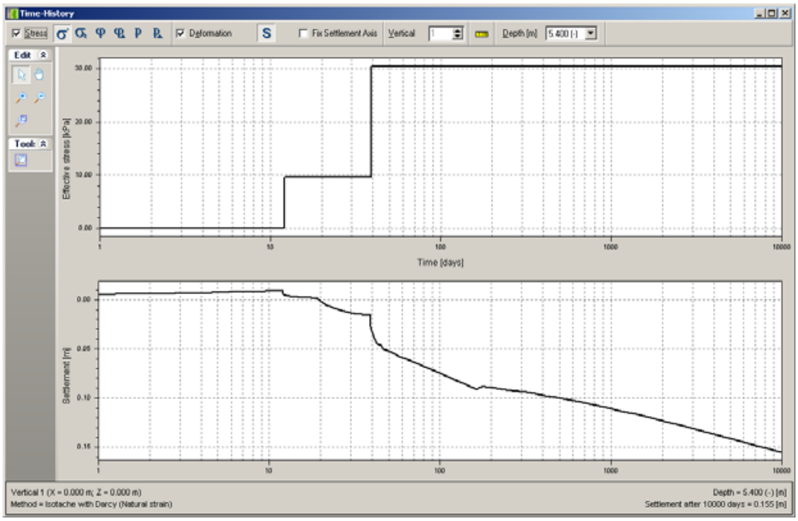


Figure 7-18 – *Time-History* window, dewatering without underpressure (Tutorial-5b)

The final settlement (0.155 m) is smaller compared to the case with underpressure (0.189 m).

7.9.5 Effect of dewatering (Tutorial-5c)

A last calculation is performed with dewatering turned off, to show its influence.

57. Save the current project as <Tutorial-5c>.
58. In the *Vertical Drains* window, turn the dewatering option off.
59. Start the calculation via the *Calculation* menu.
60. Select *Time-History* in the *Result* menu to see the total settlement results of this calculation without dewatering. The final settlement (0.132 m) is smaller compared to the case with dewatering.

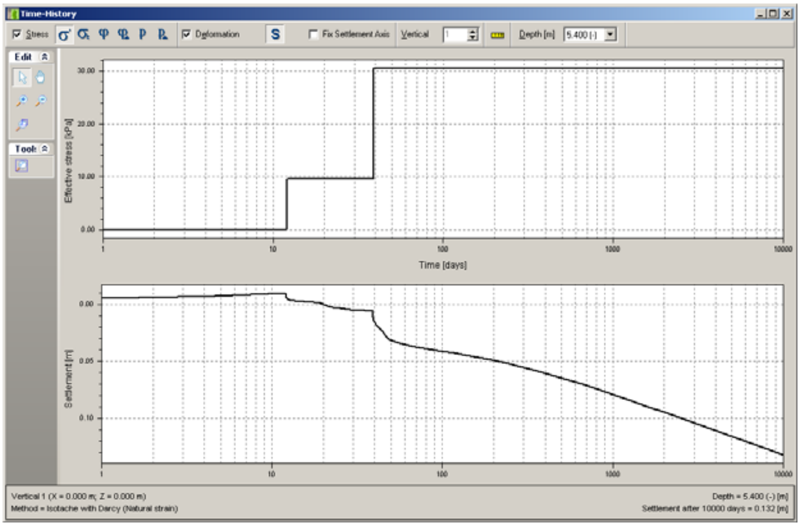


Figure 7-19 – *Time-History* window, no dewatering (Tutorial-5c)

61. Select *Residual Settlement* in the *Result* menu to see the residual settlement results of this calculation without dewatering. Note that the residual settlement after 200 days is hardly affected.

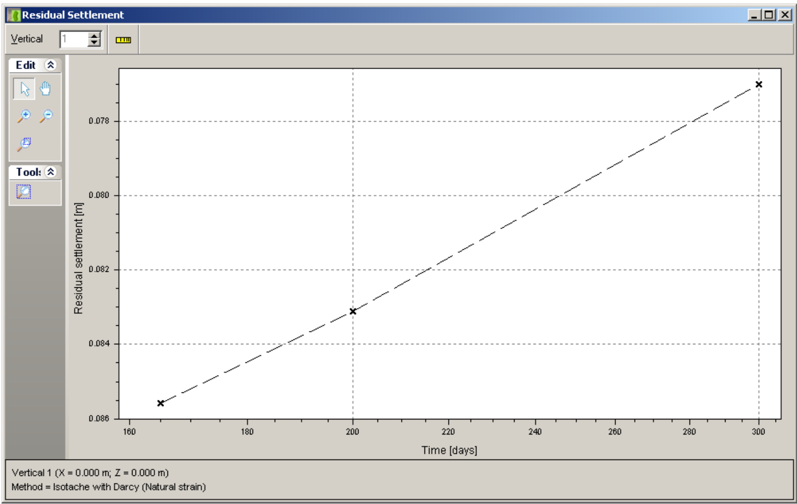


Figure 7-20 – *Residual Settlements* window, no enforced dewatering (Tutorial-5c)

7.10 Conclusion

In this tutorial the IFCO method (sand screens in combination with enforced dewatering) has been modelled. Three cases have been considered to see the influence of the enforced dewatering on the settlements, as illustrated by Figure 7-21.

- Case A: perfect sealing at the top (enforced air underpressure is 30 kPa);
- Case B: disfunctioning of the sealing (enforced air underpressure is 0 kPa);
- Case C: enforced dewatering is turned off.

It can be clearly seen that the enforced dewatering increase the final settlement, in other words reduce the residual settlements.

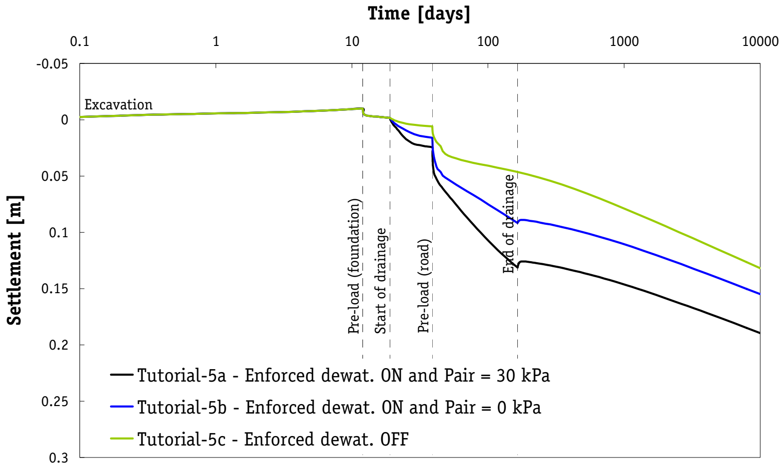


Figure 7-21 – Settlement results for different cases (Tutorial-5)

Introduction
Tutorial
Reference
Background
Verification

Embankment Design and Soil Settlement Prediction

Deltares

Delft GeoSystems

8 General

This part of the manual contains a detailed description of the available menu options for input, calculation and viewing results.

The examples in the tutorial section provide a convenient starting point for familiarization with the program.

8.1 File menu

Besides the familiar Windows options for opening and saving files, the File menu contains a number of options specific to MSettle.

- ***New***
  Select this option to display the *New File* window (Figure 8-1). Three choices are available to create a new geometry:
  - Select *New geometry* to display the *View Input* window, showing only the geometry limits (with their defaults values) of the geometry;
  - Select *New geometry wizard* to create a new geometry faster and easier using the wizard option (involving a step-by-step process for creating a geometry, see [§ 9.3.2]);
  - Select *Import geometry* to use an existing geometry.

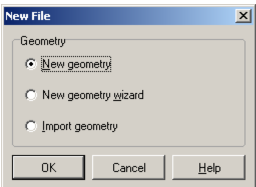


Figure 8-1 – *New File* window

- ***Copy Active Window to Clipboard***
  Use this option to copy the contents of the active window to the Windows clipboard so that they can be pasted into another application. The contents will be pasted in either text format or Windows Meta File format.
- ***Export Active Window***
  Use this option to export the contents of the active window as a Windows Meta File (*.wmf), a Drawing Exchange File (*.dxf) or a text file (*.txt). After clicking the *Save* button in the *Export to* window, the *Export complete* window opens displaying three choices:
  - *Open* to open the file containing the exported window;
  - *Open Folder* to open the folder where the file was saved;
  - *Close* to close the *Export complete* window.
- ***Export Report***
  This option allows the report to be exported in a different format, such as pdf or rtf.
- ***Page Setup***
  This option allows definition of the way MSettle plots and reports are to be printed. The printer, paper size, orientation and margins can be defined as well as whether and where axes are required for plots. Click *Autofit* to get MSettle to choose the best fit for the page.
- ***Print Preview Active Window***
  This option will display a print preview of the current contents of the *View Input* or *Results* window.
- ***Print Active Window***
  This option prints the current contents of the *View Input* or *Results* window.
- ***Print Preview Report***
  This option will display a print preview of the calculation report.
- ***Print Report***
  This option prints the calculation report.

8.2 Program Options menu

On the menu bar, click *Tools* and then choose *Program Options* to open the corresponding input window. In this window, the user can optionally define their own preferences for some of the program's default values through the following tabs:

- [§ 8.2.1] *View* tab
- [§ 8.2.2] *General* tab
- [§ 8.2.3] *Directories* tab
- [§ 8.2.4] *Language* tab
- [§ 8.2.5] *Modules* tab

8.2.1 View

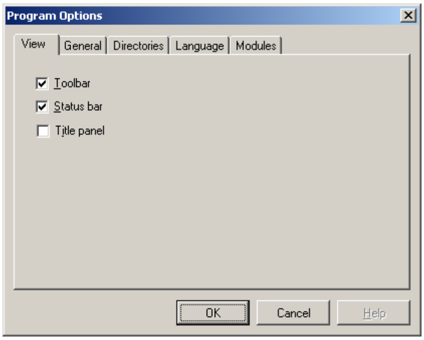


Figure 8-2 – *Program Options* window, *View* tab

Toolbar	Mark this checkbox to display the icon bar [§ 2.2.2] each time MSettle is started.
Status bar	Mark this checkbox to display the status bar [§ 2.2.5] each time MSettle is started.
Title panel	Mark the checkbox to display the project titles, as entered on the *Identification* tab in the *Project Properties* window, in a panel at the bottom of the *View Input* window.

8.2.2 General

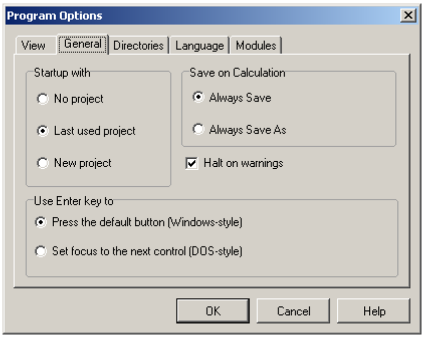


Figure 8-3 – *Program Options* window, *General* tab

Start-up with	Click one of these toggle buttons to determine whether a project should be opened or initiated when the program is started. *No project*: Each time MSettle is started, the buttons in the toolbar or the options in the File menu must be used to open an existing project or to start a new one. *Last used project*: Each time MSettle is started, the last project that has been worked on is opened automatically. *New project*: A new project is created. The user is offered three options at the start up of MSettle: New Geometry, new Geometry wizard and Import geometry. **NOTE:** The *Start-up with* option is ignored when MSettle is started by double-clicking on an input file.
Save on Calculation	The toggle buttons determine how input data is saved prior to calculation. The input data can either be saved automatically, using the same file name each time, or a file name can be specified each time the data is saved.
Halt on Warnings	Unmark this checkbox to prevent pausing the calculation in case of warnings.
Use Enter key to	Use the toggle buttons to determine the way the Enter key is used in the program: either as an equivalent of pressing the default button (Windows-style) or to shift the focus to the next item in a window (for users accustomed to the DOS version(s) of the program).

8.2.3 Directories

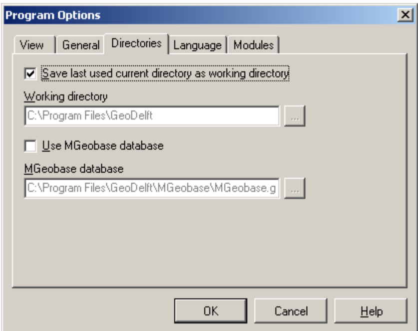


Figure 8-4 – *Program Options* window, *Directories* tab

Working directory	MSettle will start up with a working directory for selection and saving of files. Either choose to use the last used directory, or specify a fixed path.
MGeobase database	Here it is possible to assign a database location. This database (*.gdb or *.mdb) can be accessed with several options in MSettle to retrieve MSettle specific data from this file location.

8.2.4 Language

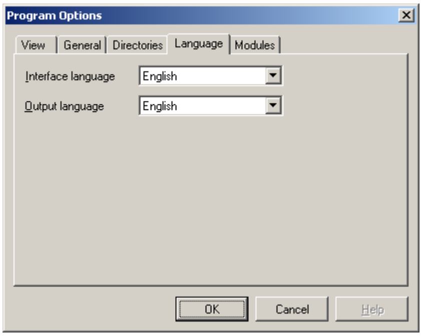


Figure 8-5 – *Program Options* window, *Language* tab

Select the language to be used in the MSettle windows and on printouts.

Interface language	Currently, the only available interface language is English.
Output language	Three output languages are supported: English, French and Dutch. The selected output language will be used in all exported reports and graphs.

8.2.5 Modules

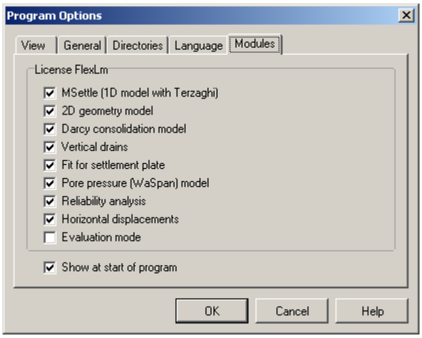


Figure 8-6 – *Program Options* window, *Modules* tab

For an MSettle installation based on floating licenses, the *Modules* tab can be used to claim a license for the particular modules that are to be used. If the Show at start of program checkbox is marked then this window will always be shown at start-up.

For an MSettle installation based on a license dongle, the *Modules* tab will just show the modules that may be used.

The *Vertical drains* option is only available in combination with 2D geometry.

9 Input

Before the analysis can be started, the data for layers, soil properties and loads need to be inputted.

9.1 Project menu

The Project menu can be used to set the model settings. The project preferences can be set, the default values of the probabilistic parameters can be entered and it is possible to view the input file.

9.1.1 Model

On the menu bar, click *Project* and then choose *Model* to open the input window. The available options will depend on the available modules [§ 8.2.5]. For an overview of different model limitations see [§ 1.5].

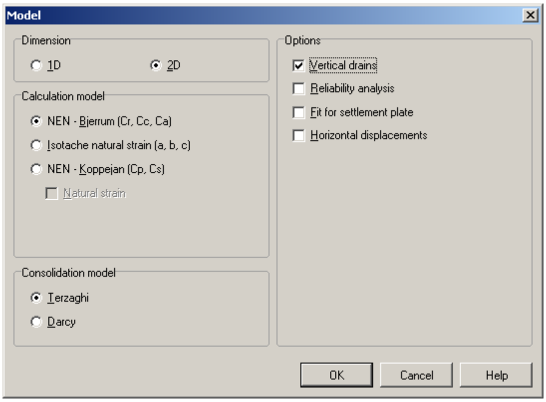


Figure 9-1 – *Model* window

Dimension	With 2D geometry the effect of different load types on multiple verticals in a two-dimensional geometry can be analyzed. With the reduced capabilities of 1D geometry the effect of uniform loading along one vertical can be analyzed.
Calculation model	The NEN-Bjerrum model [§ 16.1] uses the common parameters C_r, C_c and C_α and represents today's international de-facto standard. The model uses a linear strain assumption. The Isotache model [§ 16.2] is similar to the NEN-Bjerrum model, but uses the natural strain parameters a, b, c. Natural strain can be advantageous if large strains are expected. It makes parameters stress-objective and prevents prediction of unphysical large deformations. The traditional Dutch NEN-Koppejan model [§ 16.3] might be a logical choice if the model matches available historical parameters and user experience. Koppejan parameters are traditionally determined on a linear strain basis. The optional combination with natural strain theoretically requires that the parameters were also determined on the same basis.
Consolidation model	The Darcy model [§ 15.3] describes the influence of excess pore pressures on settlements most accurately. The approximate Terzaghi model [§ 15.2] is applicable in cases where the influence of consolidation is limited, for instance by application of vertical drains.
Vertical drains	Selection of this option enables additional modelling of vertical drains, with optionally enforced dewatering [§ 15.4].
Reliability Analysis	Selection of this option enables the determination of bandwidth in total and residual settlement, together with the determination of parameter sensitivity [§ 18].

Fit for settlement plate	Selection of this option enables the possibility to perform automatic fits on measured settlements by parameter scaling [§ 10.3]. Successful fits require a realistic prediction of the shape of the complete settlement curve. Combination with the Isotache and Darcy models is for this purpose most suited.
Horizontal displacements	Selection of this option enables the calculation of horizontal displacements according to De Leeuw tables [Lit 24].

9.1.2 Probabilistic Defaults

Input of probabilistic defaults is only required if *Reliability Analysis* has been selected in the *Model* window [§ 9.1.1]. On the menu bar, click *Project* and then choose *Probabilistic Defaults*, in order to modify the default settings for the uncertainty in soil parameters and in the layer boundary.

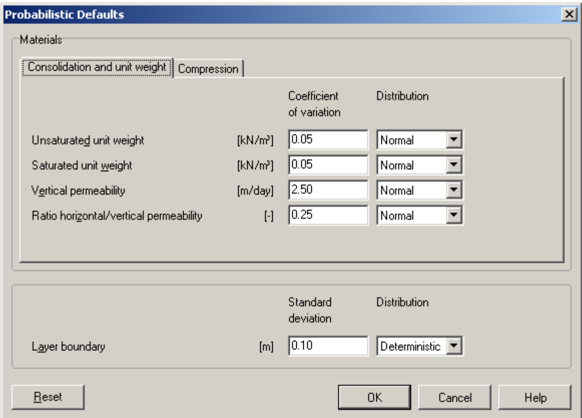


Figure 9-2 – *Probabilistic Defaults* window, *Consolidation and unit weight* tab

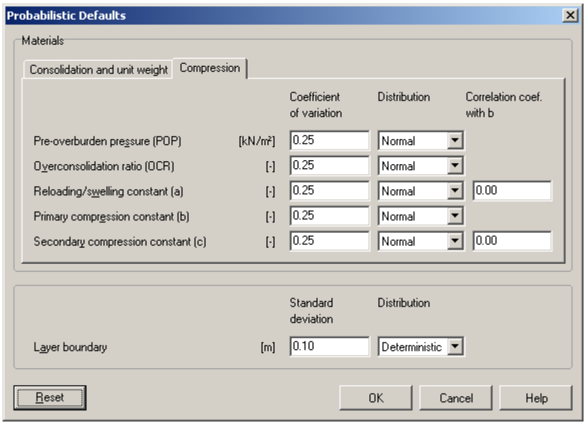


Figure 9-3 – *Probabilistic Defaults* window, *Compression* tab

Click this button to reset all values to the factory defaults.

Materials	
Coefficient of variation	The coefficient of variation times the mean value determines the default values for the standard deviation of stochastic soil parameters. Click the *Consolidation and unit weight* tab and the *Compression* tab to see all the available stochastic parameters for the selected material models. **NOTE:** The default values of the standard deviation for each material can be overruled in the *Materials* window [§ 9.2].
Distribution	Select either Normal, Lognormal or None. The Lognormal distribution will prevent values below zero. Choosing None means that MSettle will assume that this parameter is deterministic instead of stochastic.
Correlation coefficient with …	The correlation coefficient between the primary compression coefficient and the other compression parameters. A zero value indicates complete independency. Using a large nonzero value can cause numerical problems in combination with the probabilistic solution methods.

Layer boundary	
Standard deviation	The standard deviation of the boundaries between the different layers, if a stochastic distribution is used.
Distribution	

9.1.3 Project Properties

On the menu bar, click *Project* and then choose *Properties* to open the input window. The *Project Properties* window contains four tabs which allow the settings for the current project to be changed.

Project Properties – Identification
Use the *Identification* tab to specify the project identification data.

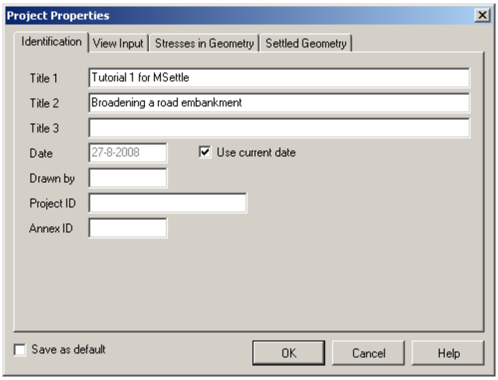


Figure 9-4 – *Project Properties* window, *Identification* tab

Titles	Use *Title 1* to give the calculation a unique, easily recognisable name. *Title 2* and *Title 3* can be added to indicate specific characteristics of the calculation. The three titles will be included on printed output.
Date	The date entered here will be used on printouts and graphic plots for this project. Either mark the *Use current date* checkbox on each printout or enter a specific date.
Drawn by	Enter the name of the user performing the calculation or generating the printout.
Project ID	Enter your project identification number.
Annex ID	Specify the annex number of the printout.

Mark the checkbox *Save as default* to use the current settings every time MSettle is started or a new project is created.

Project Properties – View Input
Use the *View Input* tab to specify the availability of components and the layout settings of the *View Input* window [§ 2.2.3].

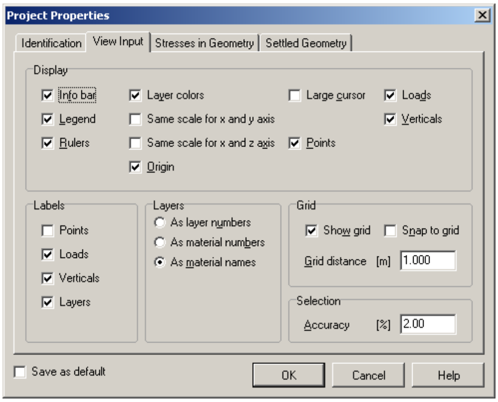


Figure 9-5 - *Project Properties* window, *View Input* tab

Display	
Info bar	Enable this checkbox to display the information bar at the bottom of the *View Input* window.
Legend	Enable this checkbox to display the legend.
Rulers	Enable this checkbox to display the rulers.
Layer colors	Enable this checkbox to display the layers in different colors.
Same scale for x and y axis	Enable this checkbox to display the x and y axis with the same scale in the top view.
Same scale for x and z axis	Enable this checkbox to display the x and z (i.e. vertical) axis with the same scale.
Origin	Enable this checkbox to draw a circle at the origin.
Large cursor	Enable this checkbox to use the large cursor instead of the small one.
Points	Enable this checkbox to display the points.
Loads	Enable this checkbox to display the loads.
Verticals	Enable this checkbox to display the verticals.

Labels	
Points	Enable this checkbox to display the point labels.
Loads	Enable this checkbox to display the load labels.
Verticals	Enable this checkbox to display the vertical labels.
Layers	Enable this checkbox to display the layer labels.

Layers

This option can only be used if the checkbox *Layers* has been marked. Choose how the layers are indicated: by number, by material number or by material name. This choice determines the layer coloring as well. If *As material numbers* or *As material names* is selected, all layers with the same material are drawn with the same color.

Grid	
Show Grid	Enable this checkbox to display the grid points.
Snap to Grid	Enable this checkbox to ensure that objects align to the grid automatically when they are moved or positioned in a graph.
Grid Distance	Enter the distance between two grid points.

Selection	
Accuracy	Mouse selection accuracy, define a large value for a large selection area.

Project Properties – Stresses in Geometry

Use the *Stresses in Geometry* tab to define the appearance of the *Stresses in Geometry* results window [§ 11.3].

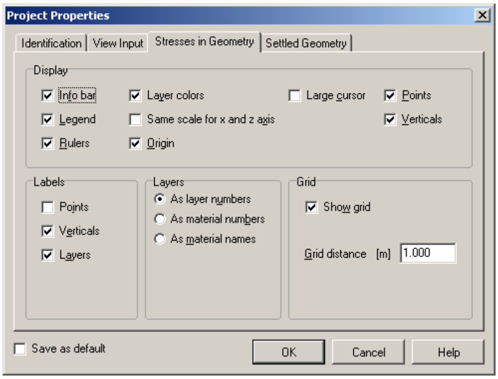


Figure 9-6 – *Project Properties* window, *Stresses in Geometry* tab

Display	
Info bar	Enable this checkbox to display the information bar at the bottom of the *View Input* window.
Legend	Enable this checkbox to display the legend.
Rulers	Enable this checkbox to display the rulers.
Layer colors	Enable this checkbox to display the layers in different colors.
Same scale for x and y axis	Enable this checkbox to display the x and y axis with the same scale.
Origin	Enable this checkbox to draw a circle at the origin.
Large cursor	Enable this checkbox to use the large cursor instead of the small one.
Points	Enable this checkbox to display the points.
Verticals	Enable this checkbox to display the verticals.

Labels	
Points	Enable this checkbox to display the point labels.
Verticals	Enable this checkbox to display the vertical labels.
Layers	Enable this checkbox to display the layer labels.

Layers
This option can only be used if the checkbox *Layers* has been marked. Choose how the layers are indicated: by number, by material number or by material name. This choice determines the layer coloring as well. If *As material numbers* or *As material names* is selected, all layers with the same material are drawn with the same colour.

Grid	
Show grid	Enable this checkbox to display the grid points.
Grid distance	Enter the distance between two grid points.

Project Properties – Settled Geometry
Use the *Settled Geometry* tab to set the appearance of the *Settled Geometry* window [§ 11.8].

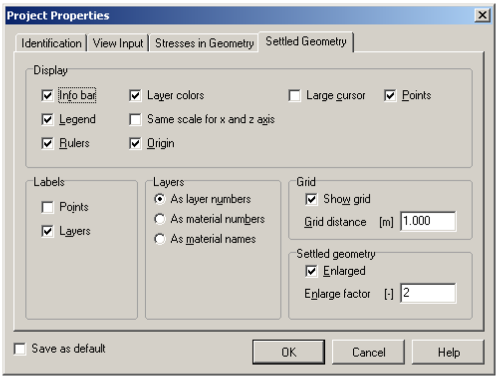


Figure 9-7 – *Project Properties* window, *Settled Geometry* tab

Display	
Infobar	Enable this checkbox to display the information bar at the bottom of the *View Input* window.
Legend	Enable this checkbox to display the legend.
Layer colors	Enable this checkbox to display the layers in different colors.
Rulers	Enable this checkbox to display the rulers.
Same scale for x and z axis	Enable this checkbox to display the x and z axis with the same scale.
Origin	Enable this checkbox to draw a circle at the origin.
Large cursor	Enable this checkbox to use the large cursor instead of the small one.
Points	Enable this checkbox to display the points.

Labels	
Points	Enable this checkbox to display the point labels.
Layers	Enable this checkbox to display the layer labels.

Layers
When the option Layers is checked, choose how the layer are indicated: by number, by material number or by material name. This choice determines the layer coloring as well. If you select As material numbers or As material names, all layers with the same material are drawn with the same colour.

Grid	
Show grid	Enable this checkbox to display the grid points.
Grid distance	Enter the distance between two grid points.

Settled geometry	
Enlarged	Enable this checkbox to use the enlarge factor.
Enlarge factor	Enter a factor to enlarge the drawing of the settled geometry.

9.1.4 View Input File

On the menu bar, click *Project* and then choose *View Input File* to open the Input File window where an overview of the input data is displayed. Click on the *Print Active Window* icon to print this file.

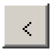
9.2 Soil menu

On the menu bar, click *Soil* and then select *Materials* to open an input window in which the soil type properties can be defined. The properties can either be imported directly from an MGeobase database (*Database* tab), or be inputted manually (*Parameters* tab):

- Import from database [§ 9.2.1];
- Manual input of Terzaghi parameters [§ 9.2.2];
- Manual input of Darcy parameters [§ 9.2.3];
- Manual input of Isotache parameters [§ 9.2.4];
- Manual input of NEN-Bjerrum parameters [§ 9.2.5];
- Manual input of NEN-Koppejan parameters [§ 9.2.6];
- Additional input for reliability analysis [§ 9.2.7];
- Additional input for horizontal displacement calculation [§ 9.2.8];

9.2.1 Materials – Database

The *Database* tab in the *Materials* window is only available if a location of an MGeobase database was specified in the *Directories* tab of the *Program Options* window [§ 8.2.3].



Select the *Database* tab in the *Materials* window to see the available soil types. Select a soil type, and use the *Import* button to import the soil type with associated properties.

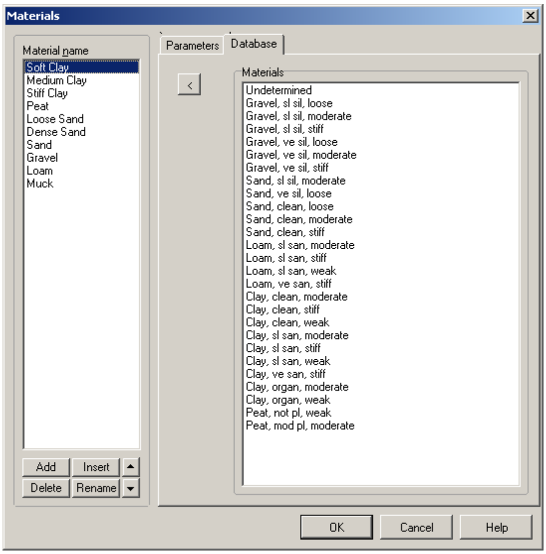


Figure 9-8 - *Materials* window, *Database* tab

9.2.2 Materials – Parameters Terzaghi

If the Terzaghi consolidation model was selected in the *Model* window [§ 9.1.1], then the Terzaghi parameters can be specified in the *Consolidation and unit weight* tab of the *Materials* window (Figure 9-9).

The Terzaghi model determines the approximate influence of consolidation, by modification of the theoretical drained settlements using a so-called degree of consolidation C_v. See [§ 1.5.1] for a comparison with the Darcy model, and see [§ 15.2] for background information.

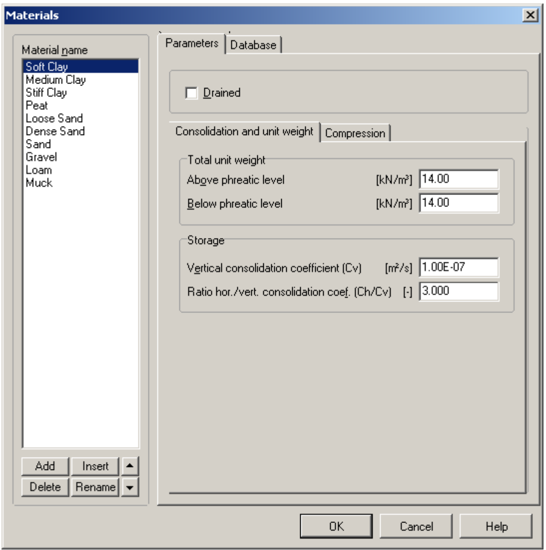


Figure 9-9 – *Materials* window, *Consolidation and unit weight* tab for Terzaghi model

Drained	Mark this checkbox to specify that the layer acts as a drained boundary for clusters of consolidation layers.
Total unit weight above phreatic level	The unit weight of the unsaturated soil above the user-defined phreatic line.
Total unit weight below phreatic level	The unit weight of the saturated soil below the user-defined phreatic line.
Vertical consolidation coefficient	Terzaghi's well-known consolidation coefficient for flow in vertical direction.
Ratio hor./vert. consolidation coef.	Only for vertical drainage [§ 9.1.1]: the ratio between the horizontal and vertical consolidation coefficients.

9.2.3 Materials – Parameters Darcy

If the Darcy consolidation model was selected in the *Model* window [§ 9.1.1], the Terzaghi parameters can be specified in the *Consolidation and unit weight* tab of the *Materials* window (Figure 9-10).

The improved and accurate Darcy model is the preferred consolidation model since release 8.2. Darcy solves numerically the transient development of excess heads along verticals and allows for a gradually developing effect of submerging on effective loading. The Darcy model is able to use the same input parameters as the Terzaghi model.

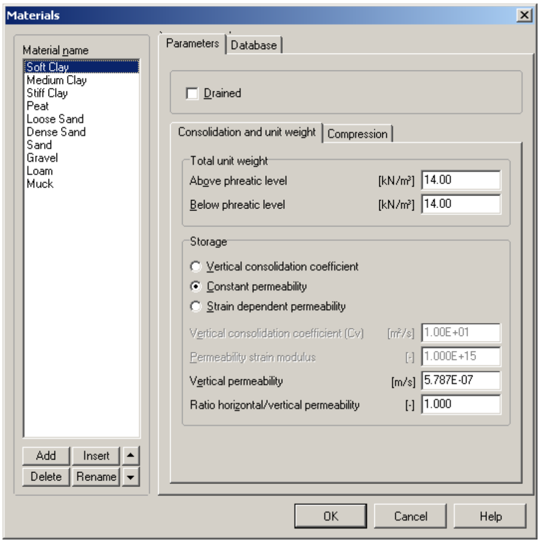


Figure 9-10 – *Materials* window, *Consolidation and unit weight* tab for Darcy model

Drained	Mark this checkbox to specify that the layer acts as a drained boundary for clusters of consolidation layers.
Total unit weight above phreatic level	The unit weight of the unsaturated soil above the user-defined phreatic line.
Total unit weight below phreatic level	The unit weight of the saturated soil below the user-defined phreatic line.

Storage	There are three ways to define the vertical permeability k_V (see the Darcy storage equation (24) on page 288): - *Vertical consolidation coefficient*: MSettle will deduct a strain dependent k_V at each location from the vertical consolidation coefficient for virgin loading, using equation (26) on page 289. - *Constant permeability*: direct input of k_V. - *Strain dependent permeability*: k_V is a strain dependent permeability according to equation (25) page 288.
Vertical consolidation coefficient Cv	The consolidation coefficient for flow in vertical direction.
Permeability strain modulus	The permeability strain modulus is the ratio C_k / (1 + e_0) where C_k is the permeability strain factor and e_0 the initial void ratio. The permeability strain modulus proves to be equal to the NEN-Bjerrum primary consolidation parameter *CR*
Vertical permeability	The initial value of the vertical permeability at undeformed state.
Ratio horizontal/vertical permeability	The ratio between the horizontal and vertical permeabilities, used by MSettle for vertical drainage modelling [§ 9.1.1].
Ratio hor./vert. consolidation coef.	The ratio between the horizontal and vertical consolidation coefficient, used by MSettle for vertical drainage modelling [§ 9.1.1].

9.2.4 Materials – Parameters Isotache

If the Isotache calculation model was selected in the *Model* window [§ 9.1.1], then the Isotache parameters can be specified in the *Compression* tab of the *Materials* window (Figure 9-11).

MSettle's a/b/c Isotache model [§ 16.2] is based on natural strain, and uses a rate type formulation. This means that all inelastic compression is assumed to result from visco-plastic creep. The model is superior in cases with large strains and is able to describe not only virgin loading but also unloading and reloading. The objective natural parameters can be derived simply from common oedometer tests [§ 17.4], or from compression parameters for other models [§ 17.7].

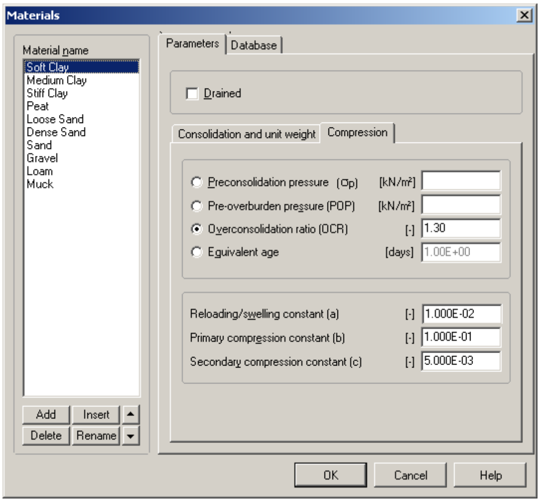


Figure 9-11 – *Materials* window, *Compression* tab for Isotache model

Preconsolidation pressure (σp)	Preconsolidation pressure in the middle of a layer. The preconsolidation pressure is the highest vertical stress experienced in the past. MSettle will use a vertical gradient equal to the initial stress gradient.
Pre Overburden Pressure (POP)	The Pre-Overburden Pressure (*POP*) is defined as the preconsolidation pressure minus the initial in-situ vertical effective stress.
Overconsolidation ratio (OCR)	The Overconsolidation Ratio (*OCR*) is defined as the ratio of preconsolidation pressure and in-situ vertical effective stress. The corresponding equivalent age (according to equation (53) page 303) is shown in grey in the *Equivalent age* field. This enables to check if the combination of the *OCR* value with the compression parameters a, b, and c is realistic.
Equivalent age	The equivalent age is an alternative input option for the overconsolidation ratio. It expresses the required time after virgin loading, if the overconsolidation would have been caused by ageing only. The corresponding *OCR* (according to equation (53) page 303) is shown in grey in the *Overconsolidation ratio* field.
Reloading/swelling constant (a)	The Isotache reloading/swelling constant *a* relates natural strain during recompression or swell to the change of vertical effective stress.
Primary compression constant (b)	The Isotache primary compression constant *b* relates natural strain during virgin loading to the change of vertical effective stress.

Secondary compression constant (c)	The Isotache secondary compression constant relates natural strain to the change of time. A zero value indicates non-creeping soil.

NOTE: *OCR*, *POP* or *Equivalent age*, together with the compression parameters *a*, *b* and *c*, determine the initial creep rate. See [§ 17.2] for background information.

9.2.5 Materials – Parameters NEN-Bjerrum

If the NEN-Bjerrum calculation model was selected in the *Model* window [§ 9.1.1], the NEN-Bjerrum parameters can be specified in the *Compression* tab of the *Materials* window (Figure 9-12).

The NEN-Bjerrum model [§ 16.1] is based on linear strain, and uses the same rate type formulation as the a/b/c Isotache model. The common NEN-Bjerrum soil parameters C_c, C_r and C_α can be derived simply from oedometer tests [§ 17.3]. Applicability of linear strain requires that parameters are determined at the appropriate stress level.

The NEN-Bjerrum compression parameters can either be inputted as ratios (Figure 9-12) or as indices (Figure 9-13).

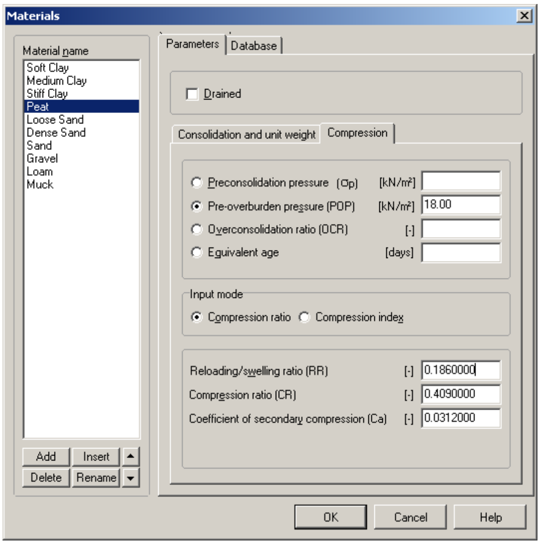


Figure 9-12 – *Materials* window, *Compression* tab for NEN-Bjerrum model (Input as ratio)

Preconsolidation pressure (σp)	Preconsolidation pressure in the middle of a layer. The stress gradient is equal to the initial stress gradient [§ 10.1.2]. The preconsolidation pressure is the highest vertical stress experienced in the past.
Pre Overburden Pressure (POP)	The Pre-Overburden Pressure (*POP*) is defined as the preconsolidation pressure minus the initial in-situ vertical effective stress.
Overconsolidation ratio (OCR)	The Overconsolidation Ratio (*OCR*) is defined as the ratio of preconsolidation pressure and in-situ vertical effective stress. Pressing the TAB key will show the corresponding equivalent age, according to equation (53) of page 303. This enables you to check if the combination of the *OCR* value with the compression parameters is realistic.
Equivalent age	The equivalent age is an alternative input option for the overconsolidation ratio. It expresses the required time after virgin loading, if the overconsolidation would have been caused by ageing only. Pressing the TAB key will show the corresponding *OCR*, according to equation (53) of page 303.
Reloading/Swelling ratio (RR	The reloading/swelling ratio is used to calculate the primary settlement below preconsolidation stress. The parameter relates the linear strain to the logarithm of stress during un-reloading.
Compression ratio (CR)	The compression ratio is used to calculate the primary settlement above preconsolidation stress. The parameter relates the linear strain to the logarithm of stress during virgin loading.
Coefficient of secondary compression (Ca)	The secondary compression coefficient is used to calculate the secondary (time dependent) settlement. The parameter relates the linear strain to the logarithm of time after virgin loading. A zero value indicates non-creeping soil.

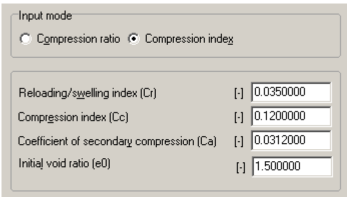


Figure 9-13 – *Materials* window, *Compression* tab for NEN-Bjerrum model (Input as index)

NOTE: *OCR*, *POP* or *Equivalent age* together with the compression parameters determine the initial creep rate. See [§ 17.2] for background information.

Reloading/Swelling index (Cr)	The reloading/swelling index is used to calculate the primary settlement below preconsolidation stress. The parameter relates the void ratio to the logarithm of stress during un-reloading.
Compression index (Cc)	The compression index is used to calculate the primary settlement above preconsolidation stress. The parameter relates the void ratio to the logarithm of stress during virgin loading.
Coefficient of secondary compression(Ca)	The secondary compression coefficient is used to calculate the secondary (time dependent) settlement. The parameter relates the linear strain to the logarithm of time after virgin loading. A zero value indicates non-creeping soil.
Initial void ratio (e0)	The initial void ratio is used by MSettle to convert the compression indices into the compression ratios.

9.2.6 Materials – Parameters NEN-Koppejan

If the NEN-Koppejan calculation model was selected in the *Model* window [§ 9.1.1], the NEN-Koppejan parameters can be specified in the *Compression* tab of the *Materials* window (Figure 9-14).

NEN-Koppejan's model [§ 16.3] is based on separate primary (instantaneous) and secondary (creep) contributions to the settlement. The model should be used prudently in case of load removal, because of its limitations. Another major difference with the NEN-Bjerrum model is the assumed stress-dependency of secondary settlements. The classic NEN-Koppejan model is based on linear strain. MSettle offers an optional extension to natural strain [§ 16.3.3].

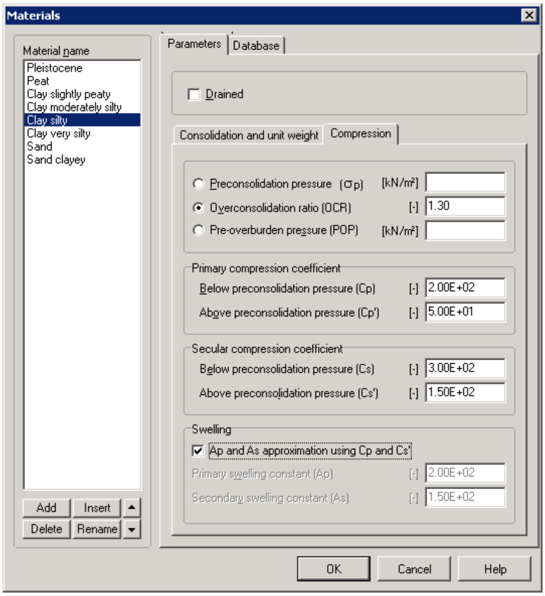


Figure 9-14 – *Materials* window, *Compression* tab for NEN-Koppejan model

Preconsolidation pressure (σp)	Preconsolidation pressure in the middle of a layer. The preconsolidation pressure is the highest vertical stress experienced in the past. By default the stress gradient is equal to the initial stress gradient, however the NEN-Koppejan model allows to defined other types of distribution and update of the preconsolidation stress via the *Calculation Options* window [§ 10.1.2]: constant or parallel to the effective stress and constant or update at each load-step.
Overconsolidation ratio (OCR)	The ratio between preconsolidation pressure and initial vertical stress
Pre Overburden Pressure (POP)	The Pre-Overburden Pressure (*POP*) is defined as the preconsolidation pressure minus the initial in-situ vertical effective stress.
Primary compression coefficient below preconsolidation pressure (Cp)	The primary compression coefficient is used to calculate the primary settlement.
Primary compression coefficient above preconsolidation pressure (Cp')	The primary compression coefficient is used to calculate the primary settlement.

Secular compression coefficient below preconsolidation pressure (Cs)	The secular compression coefficient is used to calculate the secondary (time dependent) settlement.
Secular compression coefficient above preconsolidation pressure (Cs')	The secular compression coefficient is used to calculate the secondary (time dependent) settlement.
Primary swelling constant (Ap)	The primary swelling constant for unloading.
Secondary swelling constant (As)	The secondary swelling constant for unloading. A large value of A_s implies that there will be no effect of load removal on creep. A large value is therefore only valid for cases with initial unloading.

9.2.7 Materials – Reliability Analysis

The input of reliability analysis parameters in the *Materials* window is only available if the *Reliability analysis* checkbox in the *Model* window [§ 9.1.1] was marked.

Unmark the *Use probabilistic defaults* checkbox to overrule the default values for the standard deviation, the stochastic distribution and the correlation between soil parameters in a certain layer as defined in the *Probabilistic Defaults* window [§ 9.1.2]. See [§ 18.2] for background on reliability and sensitivity analysis.

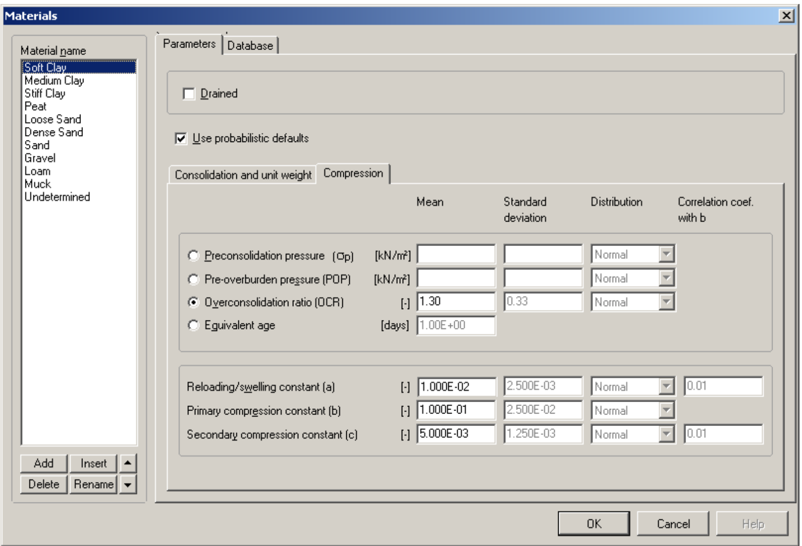


Figure 9-15 – *Materials* window, *Compression* tab for reliability analysis

9.2.8 Materials – Horizontal Displacements

The *Horizontal displacements* tab in the *Materials* window (Figure 9-16) is only available if the *Horizontal displacements* checkbox in the *Model* window [§ 9.1.1] was marked.
The calculation of horizontal displacements is based on De Leeuw theory [Lit 24]. For background information, see [§ 18.3].

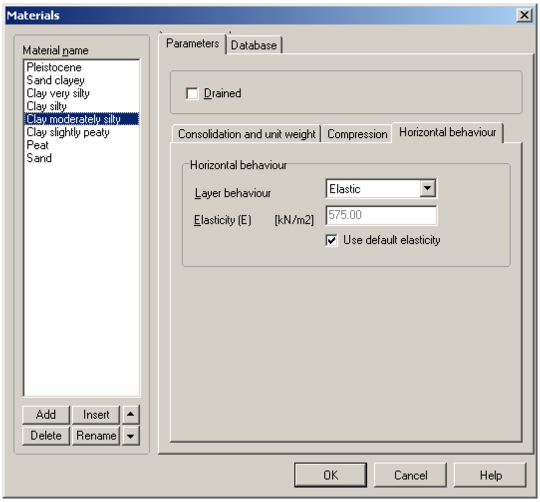


Figure 9-16 – *Materials* window, *Horizontal displacements* tab

Layer behaviour	The behaviour (*Stiff*, *Elastic* or *Foundation*) of the layer must be specified. De Leeuw theory assumes an elastic incompressible cluster of layers based on foundation layer(s) and eventually covered with stiff layer(s). Therefore, only the system of layers presented in the figure below is allowed where: • Elastic and foundation layer should be present at least one time; • Stiff layer (if present) should not be positioned below elastic or foundation layer Other systems will lead to fatal error during calculation. 
Elasticity (E)	Enter the elastic modulus of the elastic soil layer. Mark the Use default elasticity option to use the elasticity automatically calculated by MSettle according to De Leeuw and Timmermans (based on the dry unit weight).

9.3 Geometry menu

On the menu bar, click *Geometry* to display the menu options. These options are explained in the following sections.

- New [§ 9.3.1]. Start creating a new geometry manually.
- New Wizard [§ 9.3.2]. Create a new geometry using a wizard.
- Import [§ 9.3.3]. Import a (settled) geometry file in the M-Series exchange format.
- Import from database [§ 9.3.4]. Import a geometry from an MGeobase database.
- Export [§ 9.3.5]. Save a geometry file for exchange with other MSeries programs.
- Export as Plaxis/Dos [§ 9.3.6]. Save a geometry file in a different format.
- Limits [§ 9.3.7]. Set the range of the horizontal co-ordinates.
- Points [§ 9.3.8]. Add or manipulate points.
- Import PL-line [§ 9.3.9]. Import piezometric level lines from an existing MPL file.
- PL-lines [§ 9.3.10]. Add or manipulate piezometric level lines.
- Phreatic line [§ 9.3.11]. Define phreatic level lines.
- Layers [§ 9.3.12]. Define or modify layer boundaries and corresponding soil types.

- PL-lines per layer [§ 9.3.13]. Select the piezometric level line at the bottom and top of each layer.
- Check geometry [§ 9.3.14]. Check the validity of the geometry.

9.3.1 New

Select this option to display the *View Input* window (*Geometry* tab), showing only the geometry limits (with their default values) of the geometry. It is possible to now start modelling the geometry.

However, it is possible to create a new geometry faster and easier using the Geometry Wizard. This wizard involves a step-by-step process for creating a geometry.

9.3.2 New Wizard

To use the geometry wizard, open the *Geometry* menu and choose *New Wizard*. This option will guide the user step-by-step through the process of creating a geometry. Using this wizard significantly reduces time and effort required to enter data. The wizard uses predefined shapes and soil types. If more flexibility is required, the *View Input* window (*Geometry* tab) can also be used [§ 12.3] in a more general way.

New Wizard – Basic Layout

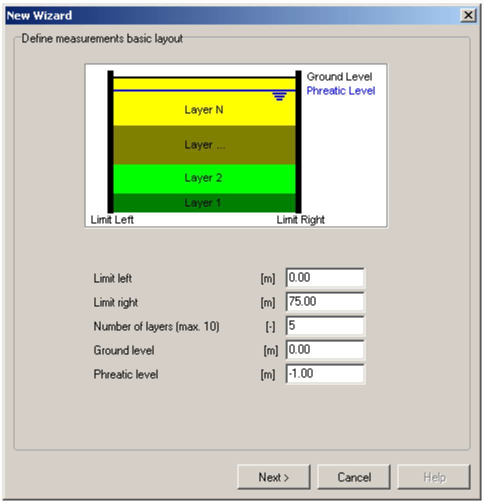


Figure 9-17 – *New Wizard* window, Basic Layout

In the first screen (Basic Layout) of the *New Wizard* window, the basic framework of the project can be entered. The graphic at the top of the window explains the required input. When satisfy with the input, just click the *Next* button to display the next input screen.

New Wizard – Shape Selection

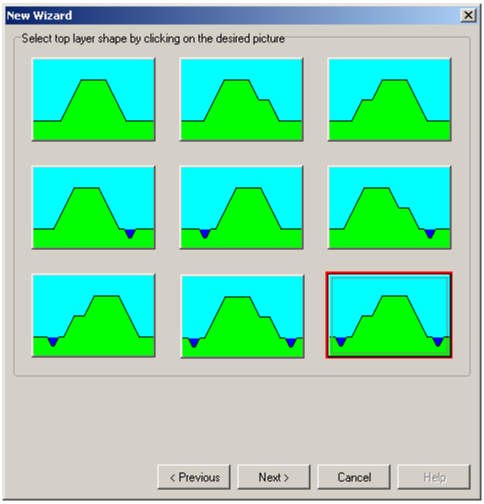


Figure 9-18 – *New Wizard* window, Top Layer Shape screen

In the second screen (Top Layer Shape) of the *New Wizard* window, one of nine default top-layer shapes can be selected. A red frame indicates the selected shape. Click the *Previous* button to return to the Basic Layout screen, or the *Next* button to display the next input screen with shape-specific input data.

New Wizard – Shape Definition

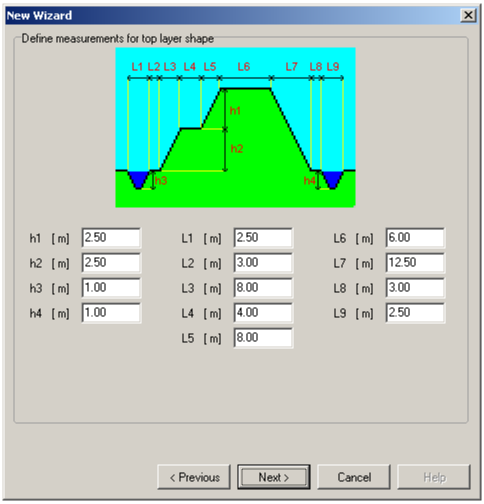


Figure 9-19 – *New Wizard* window, Top Layer Specification screen

In the third screen (Top Layer Specification) of the *New Wizard* window, the sizes for the selected top layer shape can be specified.

New Wizard – Material types

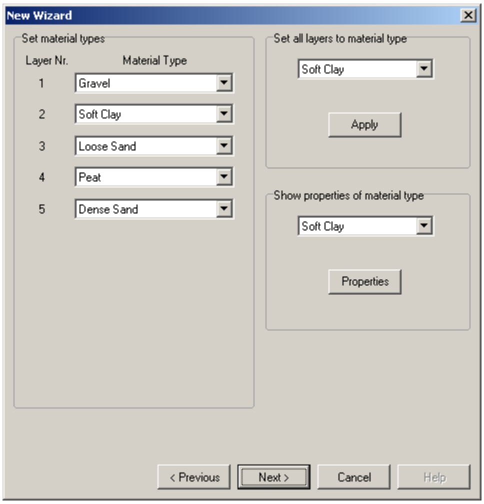


Figure 9-20 – *New Wizard* window, Material types screen

In the fourth screen (Material Types) of the New Wizard window, the materials used for the layers in the project can be specified. The number of layers was defined in the first screen (Basic Layout). The materials that can be chosen from are predefined and given in Table 9-1.

Table 9-1 – Predefined materials in MSettle

Material type	Unsaturated weight [kN/m^3]	Saturated weight [kN/m^3]
Muck	11	11
Peat	12	12
Soft Clay	14	14
Medium Clay	17	17
Stiff Clay	19	19
Loose Sand	17	19
Dense Sand	19	21
Sand	18	20
Gravel	18	20
Loam	20	20

The materials for each layer can be selected individually (using the selection boxes at the left-hand side of the screen) or one material for each layer can be selected at once (using the selection box at the top right of the screen). The parameters of each material can also be reviewed.

New Wizard – Summary

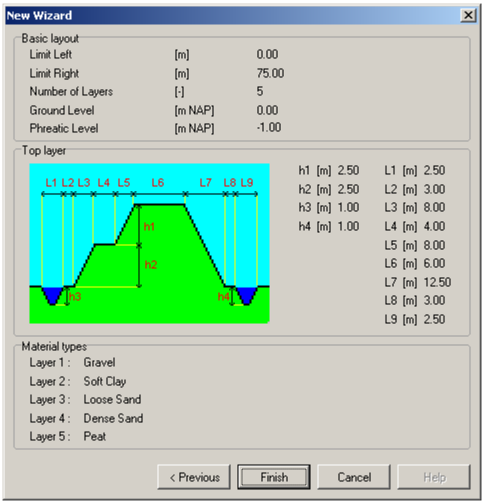


Figure 9-21 – *New Wizard* window, Summary screen

The last screen (Summary) of the *New Wizard* window displays an overview of the data entered in the previous wizard screens. If necessary, click *Previous* to go back to any screen and change the data as required. Click *Finish* to confirm the input and display the geometry in the *View Input Geometry* window. In this window, the geometry can be edited or completed graphically as described in [§ 12.3]. Of course, the *Geometry* menu options can also be used for this purpose [§ 9.3].

If the input contains errors, the *Error Report* window opens (when clicking the *Finish* button) showing the list of encountered errors and giving for each of them a solution. Click *Close* to close the *Error Report* window and use the *Previous* button of the *New Wizard* window to change the data as required.

9.3.3 Import

This option displays a standard file dialog for selecting an existing geometry stored in a geometry file, or in an existing input file for MSettle, MStab, MDrill or MSeep. For a full description of these programs and how to obtain them, visit http://www.delftgeosystems.nl.

When selecting the geometry, it is imported into the current project, replacing the current geometry. The imported geometry is displayed in the *View Input* window (*Geometry* tab). It is also possible to use this option to analyze the settled geometry at different stages, as all other input is retained.

9.3.4 Import from Database

This option displays the *Select geometry* dialog for importing a geometry from an existing MGeobase database.

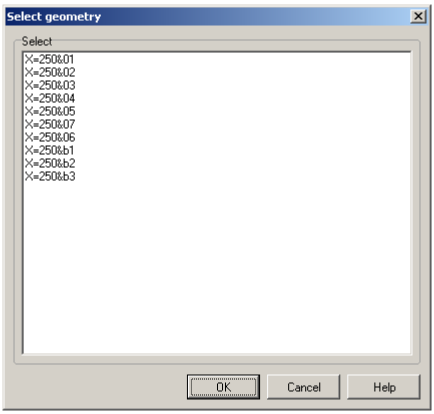


Figure 9-22 – *Select geometry* window

Again, the imported geometry will replace the current one and will be displayed in the *View Input* window (*Geometry* tab).

NOTE: This option is only available when the correct database directory has been specified using the *Directories* tab in the *Program Options* window (see [§ 8.2.3]). For more information on MGeobase, visit http://www.delftgeosystems.nl.

9.3.5 Export

This option displays a standard *Save As* dialog that enables to choose a directory and a filename in which to save the current geometry. The file will be saved in the standard geometry format for the M-Series. Files in this format can be used in a multitude of M-Series programs, such as MStab, MSettle, MSeep and MDrill. For a full description of these programs and how to obtain them, visit http://www.delftgeosystems.nl.

9.3.6 Export as Plaxis/DOS

This option displays the *Save As Plaxis/DOS* dialog that enables to choose a directory and a filename in which to save the current geometry. The file will be saved using the old DOS-style geometry format for the M-Series. Files in this format can be used by the finite element program Plaxis and in old DOS-based versions of M-Series programs such as MStab (DOS) and MZet (DOS).

Saving files of this type will only succeed, however, if the stringent demands imposed by the old DOS style are satisfied:

- number of layers ≤ 20
- number of PL-lines ≤ 20
- number of lines per boundary < 50
- total number of points ≤ 500

To be able to differentiate between an old DOS-style file and a normal geometry file, the file dialog that prompts for a new filename for the old DOS-style geometry file provides a default file name, prefixing the current name with a 'D'.

9.3.7 Limits

Use this option to edit the geometry limits.

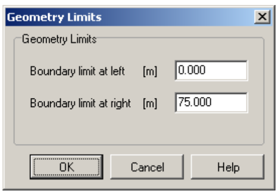


Figure 9-23 – *Geometry Limits* window

A limit is a vertical boundary defining the 'end' at either the left or right side of the geometry. It is defined by an X co-ordinate only.

NOTE: A limit is the only type of element that cannot be deleted. The values entered here are ignored if they resulted in an invalid geometry.

9.3.8 Points

Use this option to add or edit points that can be used as part of layer boundaries or PL-lines.

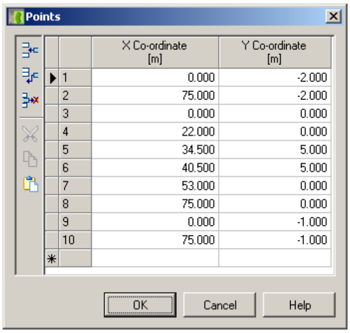


Figure 9-24 – *Points* window

A point is a basic geometry element defined by its co-ordinates. Since the geometry is restricted to two dimensions, it allows defining an X and Y co-ordinate only.

NOTE: When a point is to be deleted, MSettle will check whether the point is used as part of a PL-line or layer boundary. If so, a message will be displayed.

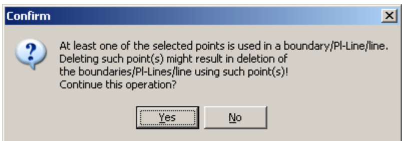


Figure 9-25 – *Confirm* window for deleting used points

When *Yes* is clicked, all layer boundaries and/or PL-lines using the point will also be deleted.
Every change made using this window (Figure 9-24) will only be displayed in the underlying *View Input* window (*Geometry* tab) after closing this window using the *OK* button. When this button is clicked, a validity check is performed on the geometry. Any errors encountered during this check are displayed in a separate window. These errors must be corrected before you can close this window using the *OK* button. Of

course, it is always possible to close the window using the *Cancel* button, but this will discard all changes.

9.3.9 Import PL-line

Use this option to display the *Import PL-line* dialog for importing a Piezometric Level lines (PL-lines) from an existing MPL file. For more information about PL-lines, refer to [§ 9.3.10].

9.3.10 PL-lines

Use this option to add or edit Piezometric Level lines (PL-lines) to be used in the geometry. A PL-line represents the pore pressures in the soil. A project can contain several PL-lines as different soil layers can have different piezometric levels. In [§ 9.3.13] it is described how different PL-lines are assigned to different layers.

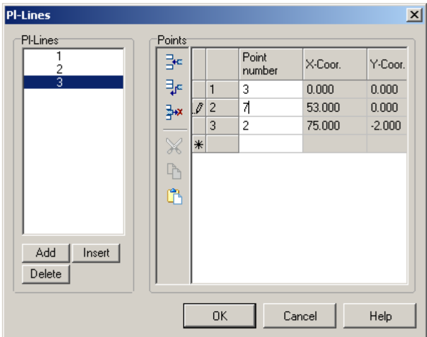


Figure 9-26 – *PL-Lines* window

In the lower left part of the window, it is possible to use the buttons to *Add*, *Insert* and *Delete* PL-lines. The selection box can be used to navigate between PL-lines that have already been defined.
Use the table to add/edit the points identifying the PL-lines. It is only possible to select points that are not attached to layer boundaries [§ 9.3.12].

NOTE: It is only possible to manipulate the *Point number* column – that is, the co-ordinate columns are purely for informative purposes. To manipulate the co-ordinates of the points, select the *Points* option from the *Geometry* menu (see [§ 9.3.8]).

Every change made using this window will only be displayed in the underlying *View Input* window (*Geometry* tab) after closing this window using the *OK* button. When clicking this button, a validity check is performed on the geometry. Any errors encountered during this check are displayed in a separate window. These errors must be corrected before this window can be closed using the *OK* button. Of course, it is always possible to close the window using the *Cancel* button, but this will discard all changes.

9.3.11 Phreatic Line

Use this option to select the PL-line that acts as a phreatic line. The phreatic line (or groundwater level) is used to mark the border between dry and wet soil.

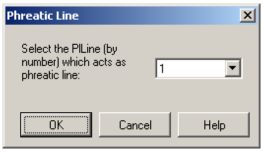


Figure 9-27 – *Phreatic Line* window

Select the appropriate line number from the dropdown list and click the *OK* button. At least one PL-line must be defined to be able to pick a Phreatic Line here.

9.3.12 Layers

This option enables to add or edit layers to be used in the geometry. A layer is defined by its boundaries and its material. Use the *Boundaries* tab (seen here in Figure 9-28) to define the boundaries for all layers by choosing the points that identify each boundary.

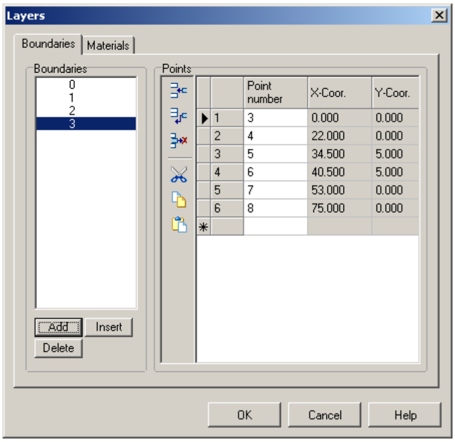


Figure 9-28 – *Layers* window, *Boundaries* tab

On the left-hand side of the window, it is possible to add, insert, delete or select a boundary. In the table on the right, it is possible to modify or add the points that identify the selected boundary.

NOTE: It is only possible to select points that are not attached to PL-lines [§ 9.3.10].

NOTE: It is only possible to manipulate the *Point number* column, because the co-ordinate columns are purely for informative purposes. To manipulate the co-ordinates of the points, select the *Points* option in the *Geometry* menu (see [§ 9.3.8]).

NOTE: When inserting or adding a boundary, all points of the previous boundary (if this exists) are automatically copied. By default, the material of a new layer is set equal to the material of the existing layer just beneath it.

The *Materials* tab enables to assign materials to the layers.

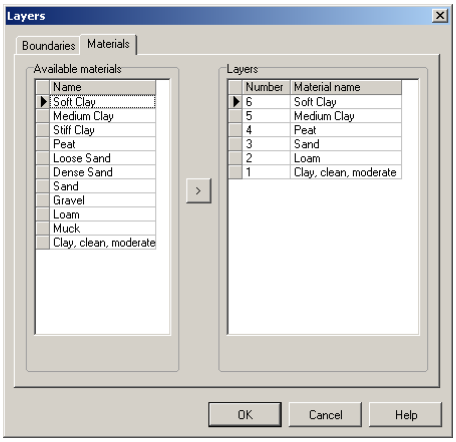


Figure 9-29 – *Layers* window, *Materials* tab

On the left of the screen, a list containing all defined materials (see the *Materials* option in the *Soil* menu [§ 9.2]) is displayed. On the right, a list of all defined layers together with their assigned materials (if available) is displayed.
The layers are listed from top to bottom as displayed in the *View Input* window (*Geometry* tab).

To assign a material to a layer, first select that layer on the right of the window. Then select the required material on the left of the window. Finally, click the *Assign* button.

Every change made using this window will only be displayed in the underlying *View Input* window (*Geometry* tab) after this window is closed using the *OK* button. When clicking this button, a validity check is performed on the geometry. If errors are encountered, a dialog window asks if auto-correction should be tried. Remaining errors are reported and can be corrected manually. The error correction is confirmed by clicking the *OK* button and discarded by clicking the *Cancel* button.

9.3.13 PL-lines per Layer

Use this option to define the top and bottom PL-lines for the defined layers. The PL-lines represent the hydrostatic heads at the boundaries of soil layers. For each soil layer, two PL-line number can be entered – one that corresponds to the top of the soil layer, and one that corresponds to the bottom. Therefore, different PL-lines can be defined for the top and the bottom of each soil layer. To do this, select the appropriate PL-line at top / PL-line at bottom field and enter the appropriate number. MSettle has reserved two numbers for special cases: 0 and 99.

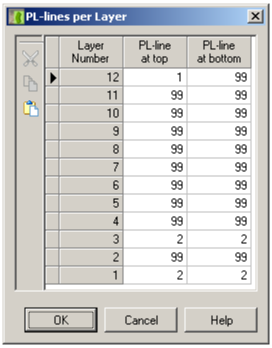


Figure 9-30 – *PL-lines per Layer* window

The PL-lines represent the pore pressure in a soil layer. For every soil layer (except the bottom layer), two PL-line numbers can be entered – one that corresponds to the top of the soil layer, and one that corresponds to the bottom. For the bottom soil layer, no second PL-line number is required. For this layer a hydrostatic increase of the pore pressure is automatically assumed from the pore pressure at the top of the layer downwards.

The following values can be used as PL-line numbers (N):

0 < N < 99	The number corresponds to one of the PL-lines defined during the geometry input. Capillary water pressures are not used – that is, if a negative water pressure is calculated for a point above the phreatic line, the water pressure in that point is defined as 0.
N = 0	Each point within the layer has a water pressure equal to 0 (Define 0 for PL-line at top of layer).
N = 99	It is possible to have a number of overlying soil layers with a non-hydrostatic pore pressure (for example, a number of layers consisting of cohesive soil). In this case, a large number of PL-lines would have to be calculated, one or two for each layer. To avoid this, M-Series software is able to interpolate across layer boundaries. For layers with a non-hydrostatic pore pressure, 99 can be entered as the PL-line number. For this layer, the interpolation will take place between the PL-line belonging to the first soil layer above with a real PL-line number, and the PL-line belonging to the first soil layer below with a real PL-line number. The first and the last soil layer must therefore always have a real PL number. **NOTE:** A real PL-line number is not equal to 99.

Water pressures above the phreatic line are set to zero.

When clicking the *OK* button, a validity check is performed on the geometry. Any errors encountered during this check are reported. A dialog window enables to disregard or correct the errors. The error correction is confirmed by clicking the *OK* button and discarded by clicking the *Cancel* button.

9.3.14 Check Geometry

Select this option to verify the validity of the geometry. All requirements are checked. If the geometry complies with all the requirements, a message will confirm this.

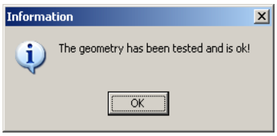


Figure 9-31 – *Information* window on confirmation of a valid geometry

If any errors are encountered during this check, they are displayed in a separate window.

9.4 GeoObjects menu

On the menu bar, click *GeoObjects* to display a menu containing:

- Verticals [§ 9.4.1].
- Vertical drains [§ 9.4.2]

9.4.1 Verticals

In the *Verticals* input window, the (horizontal) X co-ordinate for each vertical must be defined or generated. MSettle will calculate settlements along each of these verticals. At least one vertical is necessary to make a calculation. The position of the (out-of-plane) Z co-ordinate is only relevant for circular or rectangular loads.
It is possible to get MSettle to automatically generate verticals in all nodes of the geometry and non-uniform loads. At these points, verticals are required to view the settled geometry after calculation or to write the settled geometry to a file. In addition, it is possible to generate a range of verticals with an interval.

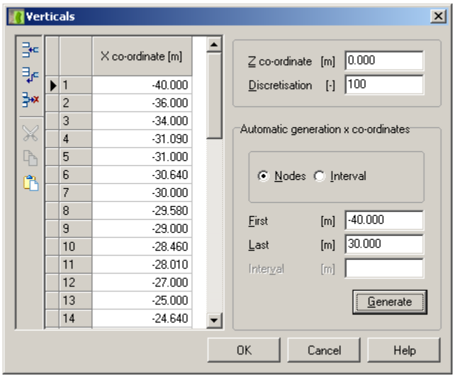


Figure 9-32 – *Verticals* window

X co-ordinate	Defines the places in geometry in x direction where the settlement will be calculated.
Z co-ordinate	Defines the place in geometry in z direction where the settlement will be calculated. This is only relevant for circular or rectangular loads. The z co-ordinate is equal for all verticals.
Discretisation	(Only available for Darcy consolidation model, see [§ 9.1.1]). The total number of elements [§ 15.3.4] per layer.
Automatic generation of X co-ordinates	Use the toggle buttons to specify whether MSettle must generate verticals in every geometry node or with an interval.
First	The start of the range for which verticals must be generated.
Last	The end of the range for which verticals must be generated.
Interval	The distance between two generated verticals.
	Click on the *Generate* button to execute the automatic generation of verticals.

9.4.2 Vertical Drains

The *Vertical Drains* window is only available if the corresponding option has been marked in the *Model* window [§ 9.1.1].

At the top left of the input window, select a strip, column or sand wall drain type (Figure 9-33).

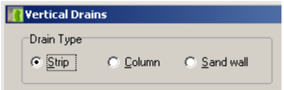


Figure 9-33 – *Vertical Drains* window (*Drain Type* sub-window)

MSettle extends the one-dimensional solution of the pore pressure distribution with a so-called leakage term. Enforced consolidation by dewatering (BeauDrain, IFCO, PTD) or vacuum consolidation can also be modelled. For background, see [§ 15.4].

Vertical Drains – Line shaped drains (Strip and Column)

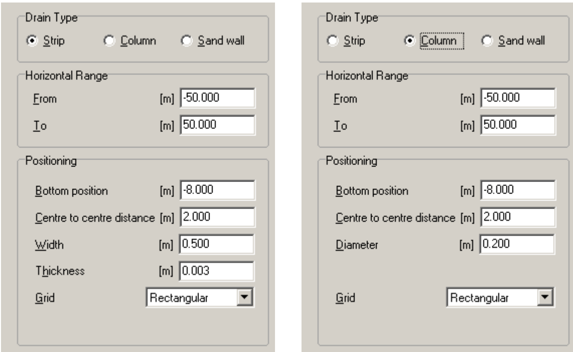


Figure 9-34 – *Vertical Drains* window, *Strip* and *Column* drains (*Positioning* input)

Horizontal Range	Enter the left (*From*) and right (*To*) limits of the drained area. This area is represented by a blue arrow in the *View Input* window (*Input* tab) [§ 2.2.3].
Bottom position	The (vertical) Y co-ordinate of the bottom end of the vertical drain. The *Bottom Position* is represented by a blue arrow in the *View Input* window (*Input* tab) [§ 2.2.3].
Centre to centre distance	The actual spacing between the drains.
Diameter	The diameter of the *Column* drain.
Width	The actual width of the *Strip* drain.
Thickness	The actual thickness of the *Strip* drain.

Grid	In the drop down menu, select the geometry of grid: *Undetermined*, *Rectangular* or *Triangular*.

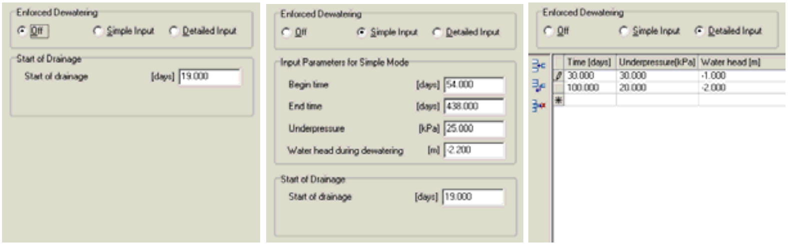


Figure 9-35 – *Vertical Drains* window, *Strip* and *Column* drains (*Enforced Dewatering* input)

Enforced Dewatering with strips or columns: Off

Start of drainage	The time t at which the drain becomes active. MSettle assumes that the water head in the drain equals the phreatic level [§ 9.3.11]

Enforced Dewatering with strips or columns: Simple Input

Start of drainage	The time at which the drain becomes active.
Begin time	The time at which dewatering (i.e. a certain water level and air pressure) starts.
End time	The time at which dewatering stops. Before and after enforced dewatering, MSettle assumes that the water head in the drain equals the phreatic level [§ 9.3.11]
Underpressure	The enforced underpressure p_{air} during dewatering. Usual values for enforced dewatering methods vary between 35 and 50 kPa [Lit 20].
Water head during dewatering	The vertical level where the negative pore pressure equals the enforced underpressure during dewatering. In case of enforced dewatering on top, this level is equal to the top level of the drain. In case of vacuum consolidation, the level is equal to the impermeable cover of the drainage layer, measured at the location where the underpressure is applied. **NOTE:** The input value is the position where the water pressure equals the applied underpressure, and therefore not the position where the water level equals the atmospheric pressure.
Start of drainage	The time t at which the drain becomes active. MSettle assumes that the water head in the drain equals the phreatic level [§ 9.3.11]

Enforced Dewatering with strips or columns: Detailed Input

Time	The time at which dewatering (i.e. a certain water level and air pressure) is active.
Underpressure	This value is zero for vertical drains without enforced underpressure. In case of enforced dewatering or vacuum consolidation on top, it represents the enforced underpressure p_{air} at time t. Usual values for enforced dewatering methods vary between 35 and 50 kPa [Lit 20].
Water head	The vertical level where the negative pore pressure equals the enforced underpressure during dewatering. In case of enforced dewatering on top, the level is equal to the top level of the drain. In case of vacuum consolidation, the level is equal to the impermeable cover of the drainage layer, measured at the location where the underpressure is applied.

Vertical Drains – Sand wall

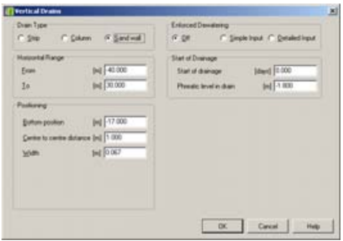


Figure 9-36 – *Vertical Drains* window, *Sand wall* (*Positioning* input)

Bottom position	The vertical co-ordinate of the bottom end of the granular wall.
Centre to centre distance	The centre to centre distance between the granular walls.
Width	The width of the granular wall.
Position of the drain pipe	Only for enforced dewatering: The vertical co-ordinate of the drainage tube at the bottom of the vertical drain z_{pipe}.

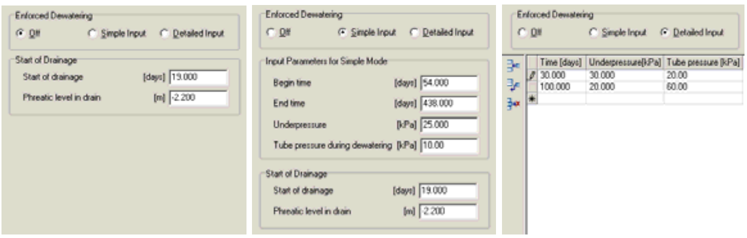


Figure 9-37 – *Vertical Drains* window, *Sand wall* (*Enforced Dewatering* input)

Enforced Dewatering with sand walls: Off

Start of drainage	The time t at which the drain becomes active.
Phreatic level in drain	The water head in the drain during drainage.

Enforced Dewatering with sand walls: Simple Input

Start of drainage	The time at which the drain becomes active.
Phreatic level in drain	The water head in the drain during drainage.
Begin time	The time at which dewatering (i.e. a certain tube pressure and air pressure) starts.
End time	The time at which dewatering stops. Before and after enforced dewatering, MSettle assumes that the water head in the drain equals the phreatic level [§ 9.3.11]
Underpressure	The enforced underpressure p_{air} during dewatering. This value can vary between 0 and 30 kPa, if an impermeable cover is applied on top [Lit 20].
Tube pressure	The water pressure p_{pipe} in the drainage tube during dewatering. A common input value during enforced dewatering is 10 kPa [Lit 20].

Enforced Dewatering with sand walls: Detailed Input

Time	The time at which dewatering (i.e. a certain water level and air pressure) is active.
Underpressure	The enforced underpressure p_{air} at time t. This value can vary between 0 and 30 kPa, if an impermeable cover is applied on top [Lit 20].
Tube pressure	The water pressure p_{pipe} in the drainage tube at time t. A common input value during enforced dewatering is 10 kPa [Lit 20]. Without enforced dewatering, you must determine this pressure from the assumed position of the free phreatic level in the granular wall.

9.5 Water menu

On the menu bar, click *Water* and choose *Properties* to open the *Water Properties* window [§ 9.5.1].

9.5.1 Water Properties

In this window, the unit weight of water can be specified.

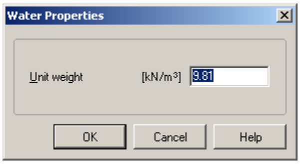


Figure 9-38 - *Water Properties* window

Unit weight	Unit weight of water. The default is 9.81 kN/m^3.

9.6 Loads menu

On the menu bar, click *Loads* to display the following menu options:

- Non-Uniform Loads [§ 9.6.1], to input non-uniform loads;
- Water Loads [§ 9.6.2], to input hydraulic pore pressure changes excluding the excess component;
- Other Loads [§ 9.6.3], to input loads with:
  - trapeziform cross-section
  - circular base
  - rectangular base
  - uniform cross-section

9.6.1 Non-Uniform Loads

Choose the *Non-Uniform Loads* option in the *Loads* menu to open an input window in which non-uniform loads can be defined. Use the panel on the left to add loads and enter the required parameters for each load.
MSettle assumes that a non-uniform load is caused by soil self weight. Therefore, the top surface of that load must be defined. The sequence of loading also must be defined. MSettle assumes that the base of a non-uniform load is equal to the top surface of the previous non-uniform load, in case of load increase.
See [§ 13.1] for background information, and see *Calculation Options* [§ 10.1] for related important options, such as maintain profile, load submerging and stress distribution in loads.

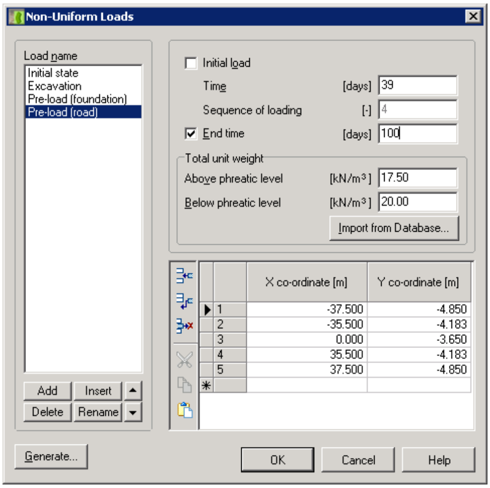


Figure 9-39 – *Non-Uniform Loads* window

Initial load	Enable this box if the load affects only the initial stresses and if the load does not cause any creep or consolidation. MSettle sets the time of application at -1.
Time	The number of days before the load will be applied. The time must correspond to the sequence of loading. For initial loads, the time is set to -1.
Sequence of loading	The sequence of loading must match the time at which the loads will be applied. To change the sequence of loading, change the order of the loads in the list by moving them up or down.
End time	The time at which a temporary load is removed.
Total unit weight above the phreatic level	The unit weight of the unsaturated soil above the phreatic line. Use negative values in case of unloading.
Total unit weight below the phreatic level	The unit weight of the saturated soil below the phreatic line. Use negative values in case of unloading.
X co-ordinate	X co-ordinate (horizontal) of points that define the surface of the load. The X co-ordinates must be ascending. The first and last co-ordinate must be located on the surface of the last defined load.
Y co-ordinate	Y co-ordinate (vertical) of points that define the surface of the load. The first and last co-ordinate must be located on the surface of the last defined load.

The Import from Database... button allows to connect material properties from a soil type to a load. This button can only be clicked if a location of an MGeobase database was specified in the *Program Options* window [§ 8.2.3]. MSettle will derive the saturated and unsaturated unit weight from the selected soil type. MSettle will also derive the strength properties from the database, when writing an MStab input file for a stability analysis [§ 11.10].

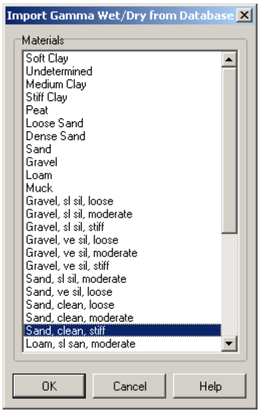


Figure 9-40 – *Import Gamma Wet/Dry from Database* window

After selecting a material from the database, MSettle changes the name of the selected uniform load into the material name. If a uniform load with this name already exists, the name is extended with a number between parentheses (see example of Figure 9-39 where the material *Sand, clean, stiff* was selected twice). The uniform load can be renamed after importing it from the database. However, if done, MStab will not recognize the material from an input file that was generated by MSettle.

Click the Generate button to generate stepwise loading from input of the final surface position and the position of the top at the end of each load step. The final surface position is inputted in the *Envelope Points* tab and the vertical levels of the top of each intermediate load steps are inputted in the *Heights* tab (see Figure 9-41).

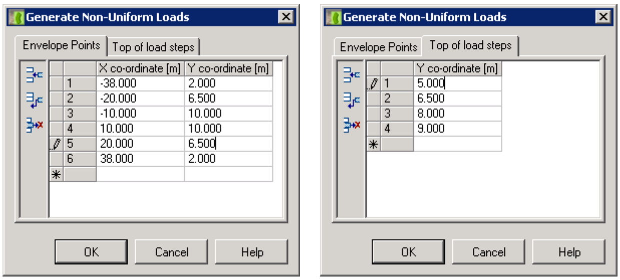


Figure 9-41 – *Generate Non-Uniform Loads* window

X co-ordinate	X co-ordinate (horizontal) of points that define the final load surface. The X co-ordinates must be ascending. The first and last co-ordinate must be located either on the initial ground surface, or on the surface of the last defined load.
Y co-ordinate	Y co-ordinate (vertical) of points that define the surface of the load. The first and last co-ordinate must be located on either on the initial ground surface, or on the surface of the last defined load.
Top of load steps	The vertical levels of the top of the added soil, during subsequent load steps.

9.6.2 Water Loads

Choose the *Water Loads* option in the *Loads* menu to open an input window in which changes in pore pressure during time can be defined. Use the panel on the left to add water loads, and select the active PL-lines at top and bottom of each layer. For background information, see [§ 15.1.1].
MSettle assumes that the initial PL-lines are defined during geometry creation [§ 9.3.10, 9.3.11, 9.3.13].

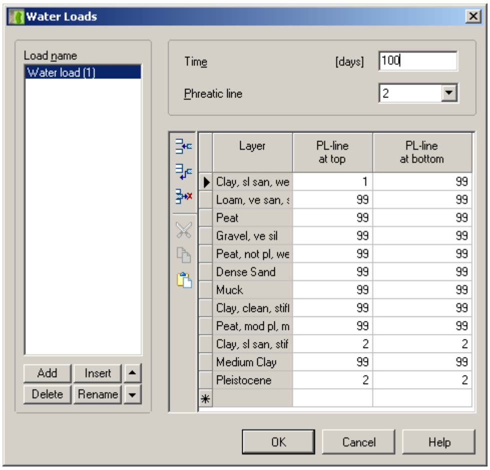


Figure 9-42 – *Water Loads* window

Time	The number of days before the load will be applied. During one time interval, only one water load can be specified.
Phreatic line	In this field, select which PL-line will function as the phreatic line. The phreatic line (or groundwater level) marks the border between dry and wet soil.
Layer	MSettle automatically enters the names of the layers.

PL-line at top	The PL-line that corresponds with the top of the layer (see [§ 9.3.10]). Use number 99 to get MSettle to perform an interpolation between adjacent layers, and use number 0 for unsaturated soil.
PL-line at bottom	The PL-line that corresponds with the bottom of the layer.

9.6.3 Other Loads

Choose the *Other Loads* option in the *Loads* menu to open an input window in which predefined shapes of soil loads can be selected. Use the panel on the left to add loads, and enter the required parameters for each load.
The following shapes are available:

- trapeziform cross-section;
- circular base;
- rectangular base;
- uniform cross-section.

Trapeziform Loads
MSettle assumes that trapeziform loads are caused by soil self weight. See [§ 13.2] for background information.

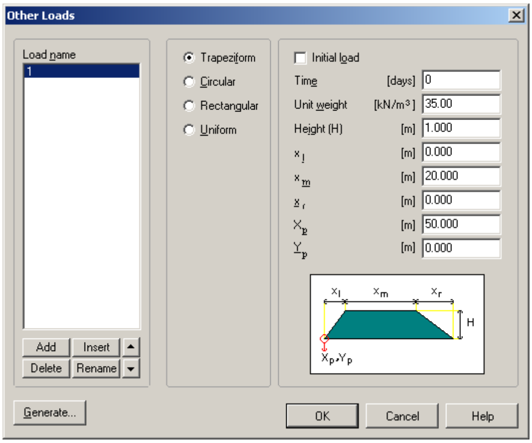


Figure 9-43 – *Other Loads* window with *Trapeziform* load

Initial load	Enable this box if the load affects only the initial stresses and if the load should not cause any creep or consolidation. MSettle sets the time of application at -1.
Time	The number of days before the load will be applied. For initial loads, the time is set to -1.
Unit weight	The weight of the load per m³. For unloading, a negative value can be entered. Zero is not allowed.

Height	Height of the load. For an inverted trapezium, enter a negative height.
x_l	Length of the left part of the load.
x_m	Length of the middle part of the load.
x_r	Length of the right part of the load. The total length of the (three) parts must be greater than zero.
X_p	X co-ordinate of the starting point (left side) of the load.
Y_p	Y co-ordinate of the starting point (left side) of the load.

Circular Loads

Loads with circular base may act on or in the geometry. See [§ 13.3] for background information.

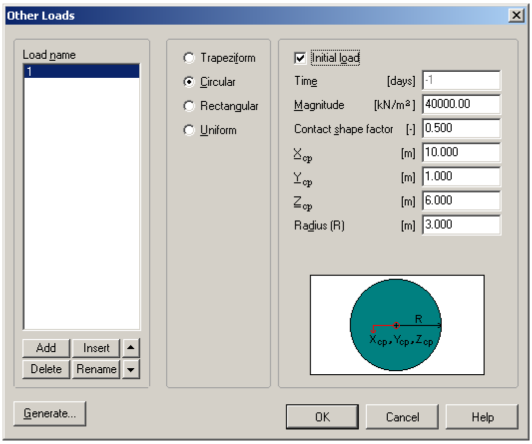


Figure 9-44 – *Other Loads* window with *Circular* load

Initial load	Enable this box if the load affects only the initial stresses and if the load should not cause any creep or consolidation. MSettle sets the time of application at -1.
Time	The number of days before the load will be applied. For initial loads, the time is set to -1.
Magnitude	The magnitude of the load. For unloading, a negative value can be entered. Zero is not allowed.
Contact shape factor	The shape factor α is used to specify the shape of the contact pressure. If $\alpha = 1$, the contact pressure is constant (represents flexible footing). If $\alpha = 0$, a parabolic distribution is used with 0 kN/m² in the centre, and twice the magnitude at the edge (represents rigid footing).
X_{cp}	X co-ordinate of the middle point of the circle.
Y_{cp}	Y co-ordinate of the middle point of the circle.
Z_{cp}	Z co-ordinate of the middle point of the circle.
Radius	The radius of the circle.

Rectangular Loads

Loads with rectangular base may act on or in the geometry. See [§ 13.4] for background information.

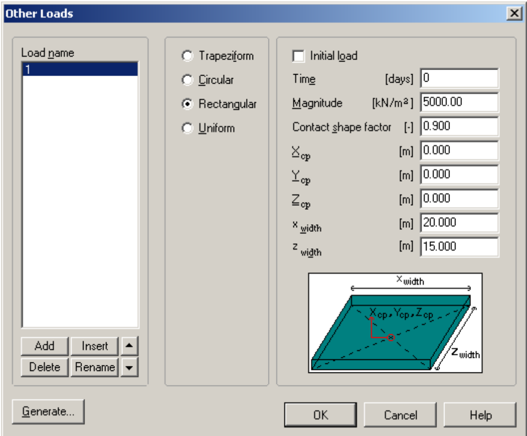


Figure 9-45 – *Other Loads* window with *Rectangular* load

Initial load	Enable this box if the load affects only the initial stresses and if the load should not cause any creep or consolidation. MSettle sets the time of application at -1.
Time	The number of days before the load will be applied. For initial loads, time is set to -1.
Magnitude	The magnitude of the load. For unloading, a negative value can be entered. Zero is not allowed.
Contact shape factor	The shape factor α is used to specify the shape of the contact pressure. If $\alpha = 1$, the contact pressure is constant (represents flexible footing). If $\alpha = 0$, a parabolic distribution is used with 0 kN/m² in the centre, and three times the magnitude at the edge (represents rigid footing).
X_{cp}	X co-ordinate of the middle point of the rectangle.
Y_{cp}	Y co-ordinate of the middle point of the rectangle.
Z_{cp}	Z co-ordinate of the middle point of the rectangle.
x_{width}	The dimension of the rectangle in x direction. It must be greater than zero.
z_{width}	The dimension of the rectangle in z direction. It must be greater than zero.

Uniform Loads

MSettle assumes that uniform loads are caused by soil self weight. See [§ 13.5] for background information. The input can be done manually or by automatic generation from measured surface positions.

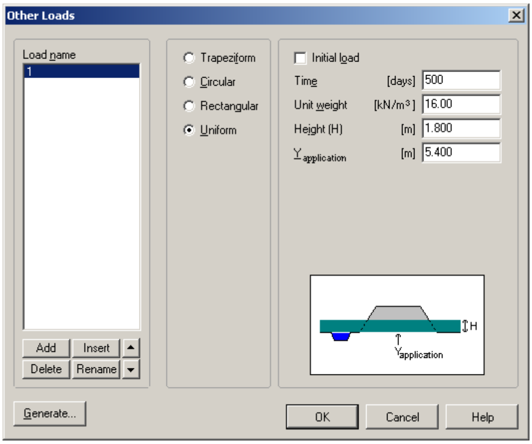
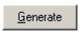


Figure 9-46 – *Other Loads* window with *Uniform* load

Initial load	Enable this box if the load affects only the initial stresses and if the load should not cause any creep or consolidation. MSettle sets the time of application at -1.
Time	The number of days before the load will be applied. For initial loads, the time is set to -1.
Unit weight	The weight of the load per m³. For unloading, a negative value can be entered. Zero is not allowed.
Height	Height of the load, relative to Yapplication.
$Y_{application}$	Y co-ordinate of the level of application.

Click the 

button to generate uniform loads from imported (SLM or GEF file) or manually specified surface positions. See Figure 9-47.

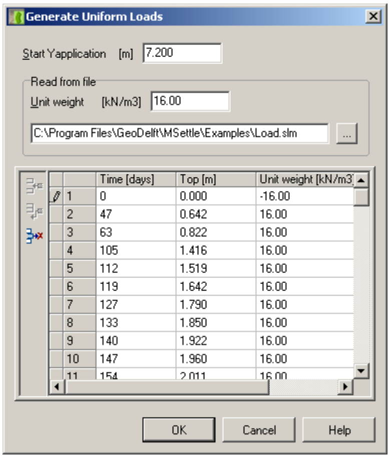


Figure 9-47 – *Generate Uniform Loads* window

Start Yapplication	Vertical co-ordinate of the level of application of the first load.
Browse	Select a file with measured surface positions (GEF or SLM) to generate the loading table automatically.
Time	The number of days before the load will be applied.
Top	New surface position.
Unit weight	The weight of the load per m³.

10 Calculations

On the menu bar, click *Calculation* to display the following menu options:

- *Options* [§ 10.1], to define various general options.
- *Times* [§ 10.2] to define time points for tabular output of remaining settlements.
- *Fit for Settlement Plate* [§ 10.3], to perform a fit on measured settlements.
- *Start* [§ 10.4], to start a regular or a reliability analysis.
- *Batch Calculation* [§ 10.5], successive calculations for different input files.

10.1 Calculation Options

In this window, a wide range of specific calculation options can be modified depending on the geometry dimension and the calculation model:

- Input fields for 1D geometry [§ 10.1.1].
- Input fields for 2D geometry [§ 10.1.2].

10.1.1 Calculation Options – 1D geometry

If a 1D dimension option was selected in the *Model* window [§ 10.1.2], the *Calculation Options* window contained only few input fields which depend on the calculation model.

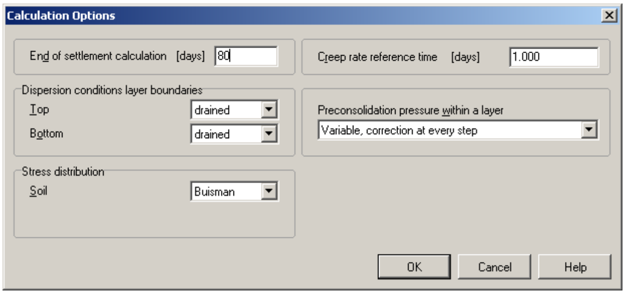


Figure 10-1 – *Calculation Options* window for 1D geometry

Dispersion conditions layer boundaries	(This parameter is required only for Terzaghi consolidation model). Use this option to influence the drainage length of the soil layers. Drainage can be introduced by selecting a drained bottom or top layer boundary. The selected drainage method will be summarised in the tabular report. For background information on Terzaghi drainage conditions, see [§ 15.2.3].
Stress distribution Soil	Distribution of the stresses in the underground can be calculated according to Buisman or Boussinesq. Boussinesq can be applied only for the trapeziform and non-uniform loads. For other kind of loads, Buisman will be used. For background information, see [§ 14.1]. Buisman: concentration index 3 Boussinesq: concentration index 4
End of settlement calculation	Enter the number of days after which the transient settlement is expected to have ended. **NOTE:** Consolidation is only included in the time-settlement curves and not in the individually reported final settlements.
Creep rate reference time	The value of the reference time τ_0 for the creep part. In practice, this value can be interpreted as the ratio between 1 day and the unit of time in the calculation. This means that a large value should be used when simulating short term settlements, with time steps smaller than 1 day, like in oedometer tests. **NOTE:** A value other than 1 day requires consistent input of all other time-dependent values [§ 17.1.2].

Preconsolidation pressure within a layer	This parameter is required only for the NEN-Koppejan model. Choose between a constant and a variable preconsolidation pressure in the layers. When variable (default), the input value is applied to the middle of the layer. Within the layer, the gradient of the preconsolidation pressure is equal to the gradient of the initial vertical effective stress. In this case, the *Pre Overburden Pressure* equals the difference between the preconsolidation pressure and the vertical effective stress at middle of the layer [§ 17.2]. There are two additional options available for updating the preconsolidation stress: Correction at every timestep (default). Adjustments will be performed after each load step. Terzaghi assumes that excess pore pressures are dissipated before a new load step starts. This is certainly not the case for small time increments between load steps. Correction at time=0 [days]. All preconsolidation stresses are adjusted to the maximum of the initial value or the corresponding effective stress.

10.1.2 Calculation Options – 2D geometry

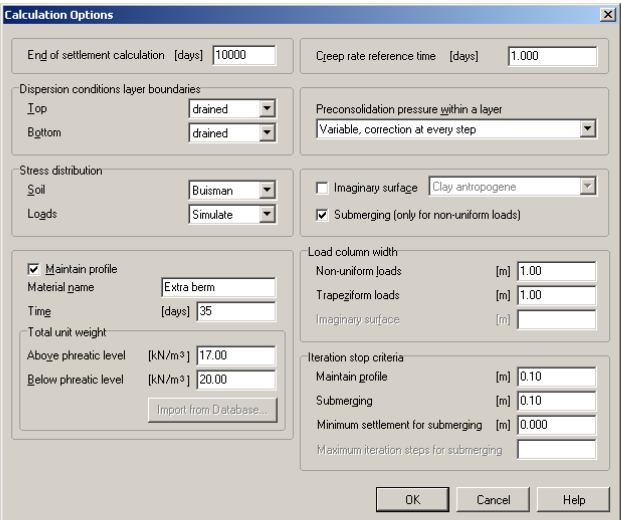


Figure 10-2 – *Calculation Options* window for 2D geometry

See [§ 10.1.1] for a description of the general input fields that are shared with a 1D geometry.

Stress distribution in loads	When this option is chosen, MSettle will incorporate the effect of stress distribution inside non-uniform loads. MSettle will divide the load in columns (see the *Load column width* option described below) and then divide each column into pieces with a height of approximately one meter.
Maintain profile	Enable this checkbox to get MSettle to add a special superelevation load. This load will keep the top surface at the defined level during deformation analysis. See [§ 13.6] for background information.
Material name	This is the name of the special superelevation load that is added to the geometry using the *Add Superelevation* option in the *Write Settled Geometry* window. This option can be accessed through the *Results* menu.
Time	Enter the number of days before the superelevation load will be applied. The superelevation load has to be applied during the last load step. If the pointer is located at the input field, a hint will indicate the current last load step.
Total unit weight above phreatic level	The unit weight of the unsaturated superelevation load above the water level.
Total unit weight below phreatic level	The unit weight of the saturated superelevation load below the water level.
Imaginary surface	When this checkbox is enabled, a so-called imaginary surface can be defined to model the stress distribution in the case of an initially non-horizontal surface. MSettle will calculate the spatial stress distribution in the layers below this surface, caused by the weight of the initial layers above the surface. When you do this, you must also select the layer whose top acts as the imaginary surface. For background information, see [§ 14.5].
Submerging	Enable this checkbox to take submerging of non-uniform soil weight loads [§ 9.6.1] into account in your calculations. The option is switched on by default. MSettle takes submerging into account approximately, by a once-off modification of the self weight of nonuniform loads, based on the final settlements. See [§ 13.7.1] for background information With Darcy in combination with the NEN-Bjerrum or Isotache model, MSettle will gradually adapt the effective weigth of layers and nonuniform loads in time, as function of the actual settlement. See [§ 13.7.2] for background information.

Load column width	MSettle can subdivide trapeziform loads or non-uniform loads into columns [§ 14.2]. The default value for the width of these columns is one meter, but it is possible to change these defaults. A small width will increase the accuracy, while a large width will increase the calculation speed.
Iteration stop criteria – Maintain Profile	The *Maintain Profile* iteration will stop when the difference in the calculated final settlements between two iterations becomes less than the specified value.
Iteration stop criteria – Submerging	The *Submerging* iteration will stop when the difference in settlements between the iterations becomes less than the specified value. With Terzaghi or NEN-Koppejan (i.e. approximate submerging model), iteration is performed on the final settlements. With Darcy in combination with the NEN-Bjerrum or Isotache model (i.e. accurate submerging model), iterations may occur within time steps, in case of large increments.
Minimum settlement for submerging	When submerging is used, the increment of the settlement is yet unknown. An estimate of the settlement is made by means of an iterative procedure. The iteration is stopped if the settlement is less than the given value.
Maximum iteration steps for submerging	Only for Darcy in combination with the NEN-Bjerrum or Isotache model (i.e. accurate submerging model): the maximum number of submerging iterations within a step. A value of 1 means no iterative correction per step. The iteration is stopped after this fixed number of iterations.

10.2 Calculation Times

The *Calculation Times* window allows input of time points at which MSettle will calculate tabular output of total and residual settlements and graphical output of residual settlement. See [§ 11.2.5].

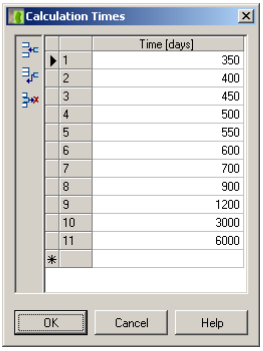


Figure 10-3 – *Calculation Times* window

10.3 Fit for Settlement Plate

The *Fit for settlement plate* option in the *Calculation* menu is available only if it has been selected previously in the *Model* window [§ 9.1.1].
Choose this option to improve the match between predicted and measured settlements, by manual or automatic scaling of soil properties. A close fit will improve the continued prediction of final and residual settlements. Usage is only possible after full input of geometry [§ 9.3, § 9.4], material properties [§ 9.2], loading [§ 9.6] and calculation options [§ 10.1].
MSettle performs the automatic fit by means of an iterative weighted least squares procedure, which minimizes both the difference between measurement and prediction, and the difference between the original and the adapted value of the parameters. During each iteration, MSettle linearizes the influence of parameter modifications, by first determining the settlement variations caused by very small parameter changes. See [§ 18.1] for background.
The *Fit for Settlement plate* window contains two tabs:

- The *Measurements* tab, for definition of the measured settlements [§ 10.3.1]
- The *Materials* tab, for execution of the fit of the prediction on measurements [§ 10.3.2]

10.3.1 Fit for Settlement Plate – Measurements

The *Measurements* tab of the *Fit for Settlement Plate* window enables the selection of the file with measured settlements and the optional input of a shift in the time or the settlement.

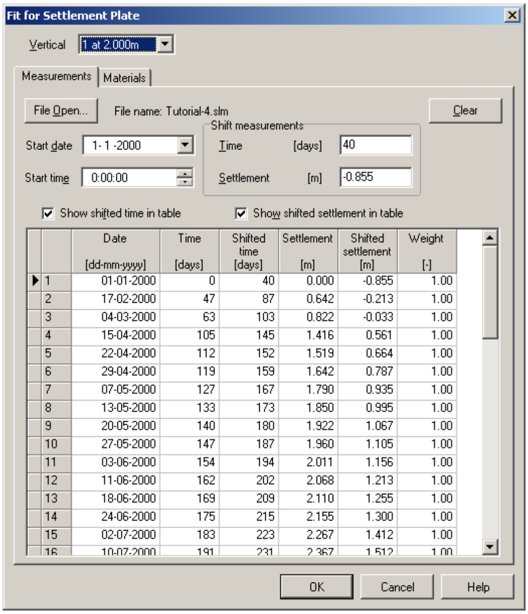


Figure 10-4 – *Fit for Settlement Plate* window, *Measurements* tab

Vertical	Select the vertical where the settlement plate was measured.
File Open...	Selection of the file containing pairs of time and measured settlement. You can use the self describing Geotechnical Exchange format (GEF), the tab delimited format (TXT), or the comma (;) delimited format (CSV). MSettle will neglect all lines in the delimited format files before the first line with numbers. MSettle also supports the old SLM format, for compatibility reasons.
Clear	Clear measurement data.
Start date	Optional input of the start date (not for GEF).
Start time	Optional input of the start time (not for GEF).
Shift measurements	Apply a shift to the time and/or the settement. Select the Show shifted time in table and the Show shifted settlement in table in order to show the shifted values that MSettle will use.
Date	The date of measurement. Not used by MSettle.
Time	The original time of measurement, relative to the start date and start time.
Shifted time	The shifted time of measurement, as used by MSettle.
Settlement	The original measured settlement.
Shifted settlement	The shifted measured settlements, as used by MSettle.
Weight	An influencing factor for automatic fitting. You can use large values for certain parts of the curve (for example the creep tail) to ensure that this part is fitted most closely.

10.3.2 Fit for Settlement Plate – Materials

The *Materials* tab of the *Fit for Settlement Plate* window enables the execution of a fit of the prediction on the measured settlements, at a certain position in a certain vertical.

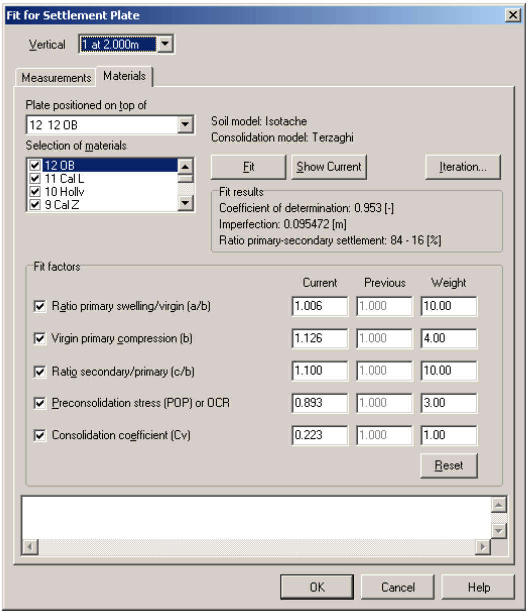


Figure 10-5 – *Fit for Settlement Plate* window, *Materials* tab

Plate positioned on top of	Select the layer which top defines the vertical location of the settlement plate. By default the top layer is selected.
Selection of material	Select the soil types for which you allow scaling of soil parameters. By default all layers are selected.

Fit factors	Select the parameters for which you allow scaling by hand or by automatic fitting. By default all scaling parameters are selected. **NOTE:** The scaling factors for un/reloading and secondary compression are not applied to the parameters themselves, but to the ratio of that parameter with the parameter for primary virgin compression (b, CR, $1/C_p'$). Increasing the parameters for primary virgin compression will therefore yield more settlement in all parts of the curve. Increasing the two ratios will yield a separate increase in respectively the primary settlement during un/reloading and the secondary (creep) settlement. Increasing the preconsolidation stress (*POP*, *OCR*) will reduce the settlements at low loading levels. Increasing the permeabilities or consolidation coefficients will speed up the settlement process, by reducing the consolidation period.
Iteration...	Click this button to change the default stop criteria for the iteration process during automatic fitting (Figure 10-6). Besides the maximum number of iterations, you can also define the target value for the coefficient of determination. Finally you can specify the minimally required improvement of this coefficient during a single iteration (iteration accuracy).
Fit	Click this button to perform automatic fitting by iterations. If the match after a single fit is not yet satisfactory, you can click the button again for continued iterations. A progress window indicates the goodness of fit during the iteration process (Figure 10-7). This information is also displayed in the main window.
Current	In case of fitting by hand: enter the scaling factors. In case of automatic fitting: the calculated scaling factors of the last iteration will be displayed.
Previous	The scaling factors at the start of the last fit.
Weight	An influencing factor for automatic fitting. A low value means that the corresponding scaling factor can change easily during the iteration process. Use a high value when the initial parameter values or ratios are considered reliable and a low value when these values are considered uncertain. The default values are usually sufficient.
Reset	Click this button to reset all weights to their defaults.
Show Current	Show the graph and data of the fit that is based on the Current factor values. See Figure 10-8. Afterwards, this graph is also available from the *Result* menu (Figure 10-9).
Coefficient of determination	The correlation factor for the last fit. A value close to 1 indicates a good match between measurement and prediction.
Imperfection	An average value for the differences between all measurements and predictions [m]. A value close to 0 indicates a good match between measurement and prediction.

Ratio primary-secondary settlement	The ratio between the primary settlement and the secondary settlement (due to creep).

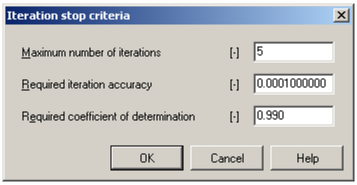


Figure 10-6 – *Iteration stop criteria* window

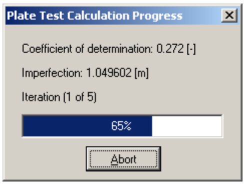


Figure 10-7 – *Plate Test Calculation Progress* window

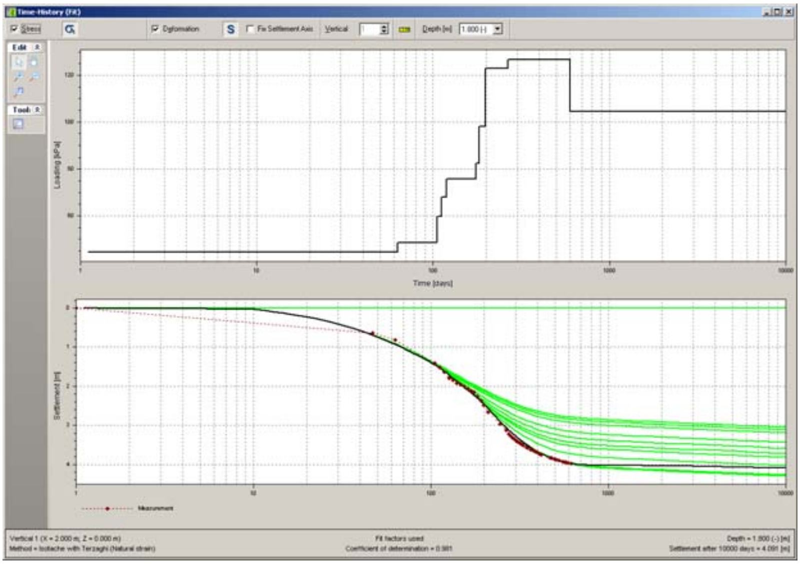


Figure 10-8 – *Time-History (Fit)* window

NOTE: Right click in the *Time-History* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

After a fit, the *Results* menu will show all the available results for the selected vertical, using the scaled parameters (Figure 10-9).

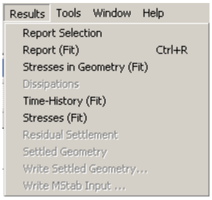


Figure 10-9 – Available results after a fit

NOTE: To apply the scaled parameters to all verticals and to generate other types of calculation results, select the *Use fit parameters* option in the *Start Calculation* window [§ 10.4].

10.4 Start Calculation

To start the actual calculation, choose the *Start* option in the *Calculation* menu.

The *Options* button allows to chose the calculation options (if not already done) by opening the *Calculation Options* window [§ 10.1].

When the calculation is started, MSettle will first check if the input contains any (fatal) errors. If the input contains errors, they are reported in the *Error Messages* window [§ 10.4.3] and they must be corrected. If the input contains no errors, the calculation will start.

MSettle can also generate (non-fatal) warning messages if the input is unrealistic or can be improved. You can either choose to *Close* the *Start Calculation* window without performing a calculation and change the input according to the warning messages or to *Continue* the calculation without taking into account the warning messages. In this case, the warning messages will be also printed in the *Report* [§ 11.2.7]. Unmark the *Halt on Warnings* checkbox in the *Program Options* window [§ 8.2.2], in case you want MSettle to proceed after warnings without pausing.

The screen displays a progress overview. The calculation can be aborted by clicking the *Abort* button. Therefore, no results in the *Results* menu will be available.

Two kinds of calculation are available:

- a regular (deterministic) analysis [§ 10.4.1];
- a reliability and sensitivity analysis [§ 10.4.2].

10.4.1 Regular (deterministic) analysis

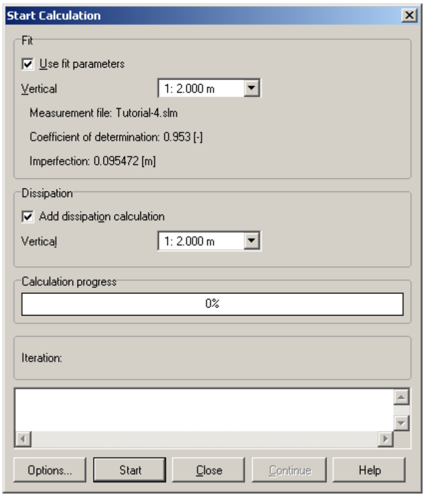


Figure 10-10 – *Start Calculation* window for a regular analysis

Use fit parameters	Select this option to use the previously determined scaling factors from a settlement plate fit for the settlement prediction along all verticals [§ 10.3]. **NOTE:** The selected *Vertical* must be the same as the vertical used in the *Fit for Settlement Plate* window [§ 10.3] otherwise the calculation will be a regular calculation without scaling factors. Moreover, the *Show Current* in the *Fit for Settlement Plate* window [§ 10.3] puts the scaling factors only on the materials that are selected, while the regular calculation with option *Use fit parameters* selected puts the scaling factors on all materials. Therefore results can differ when comparing both calculations.
Add dissipation calculation	Perform a dissipation calculation for a unit load along a selected vertical, before starting the actual calculation. MSettle will use the results of this calculation for the dissipation graph [§ 11.4], and for the export of an MStab file [§ 11.10]. The selection list shows all available verticals by number and by horizontal co-ordinate.

10.4.2 Reliability and sensitivity analysis

The *Start Calculation* window contains special options for reliability and sensitivity analysis, when the *Reliability* option in the *Model* window is selected [§ 9.1.1].

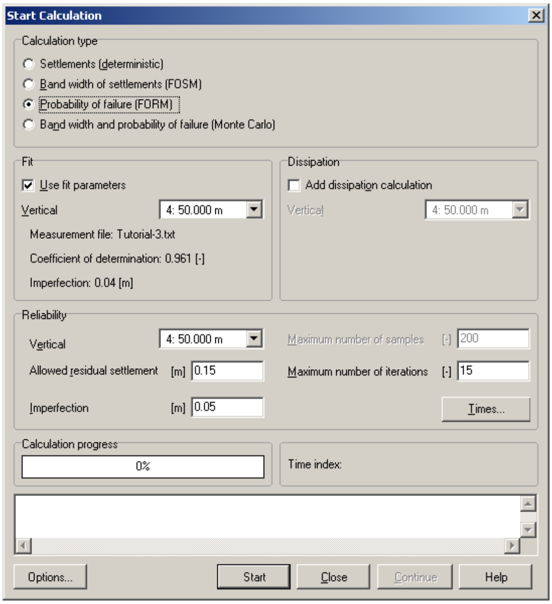


Figure 10-11 – *Start Calculation* window for a reliability and sensitivity analysis

See [§ 10.4.1] for a description of the options that are shared with a regular (deterministic) analysis. The description of the additional options for a reliability and sensitivity analysis follows hereafter. See [§ 18.2] for background information.

Calculation type	Select one of the following methods: - *Deterministic*: a regular deterministic settlement analysis along all verticals, based on fixed mean values of the parameters. - *FOSM* (First Order Second Moment): Quick and approximate determination of the bandwidth and the influencing factors (parameter sensitivity) for the total settlements along one vertical. The determination is executed at user defined time points and at the time points of measurements. Calculation time will increase with an increasing number of stochastic parameters. - *FORM* (First Order Reliability Method). Iterative determination of the reliability index, bandwidth and influencing factors for the residual settlement along one vertical. A separate FORM analysis is performed for each residual settlement that starts from each different user defined time point. Calculation time will increase with an increasing number of stochastic parameters, user defined

	time points and iterations. Furthermore, the FORM method is only conditionally stable. - *Monte Carlo* (recommended). Determination of the bandwidth for the total settlements along one vertical, and also of the reliability index and bandwidth for the residual settlements, by repetitive execution of settlement analyses (sampling). Each sample is executed with random parameter values, derived from the stochastic distributions. Calculation time will increase with the number of samples. Accurate Monte Carlo analysis requires a large number of samples, if many stochastic parameters are involved.
Use fit parameters	Select this option to: Use the previously determined scaling factors from a settlement plate fit for the settlement analysis in all verticals [§ 10.3] to determine updated mean values of the settlement. Automatically introduce correlations between the different stochastic parameters via Bayesian updating, based on the influence of the parameters on the predicted settlement at the times of measurement, and based on the input value of the imperfection. The updated correlations will usually yield a reduction of the calculated bandwidth in the total and residual settlements.
Vertical	Select the vertical for the reliability analysis.
Allowed residual settlement	For *FORM* and *Monte Carlo* methods, the allowed residual settlement represented in the *Residual Settlements (Reliability)* window [§ 11.13].
Imperfection	In case the *Use Fit* option is selected, this value represents the combined inaccuracy in the measurements and in the prediction model. A larger value implies less influence of the measurements on the Bayesian update of the correlations between uncertain parameters.
Maximum number of samples	The number of samples that the Monte Carlo method will use.
Maximum number of iterations	The maximum number of iterations for the FORM method.
	Click this button to open the *Calculation Times* window (see Figure 10-3 in [§ 10.2]). In this window, the times for the calculation of bandwidth, reliability index and influencing factors can be defined. **NOTE:** If the *Use Fit* option is selected, MSettle will already calculate the bandwidth and the influencing factors for the total settlement at the times of measurement.

10.4.3 Error Messages (before calculation)

If errors are found in the input, no calculation can be performed and MSettle opens the *Error Messages* window displaying more details about the error(s). Those errors must be corrected before performing a new calculation. To keep the messages, they must be printed because they will be overwritten the next time a calculation is started.

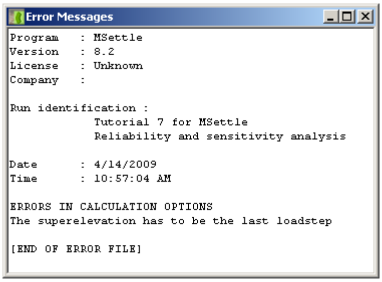


Figure 10-12 – *Error Messages* window

10.4.4 Warnings and Error Messages during calculation

Warnings and fatal errors might be displayed in the messages pane at the bottom of the *Start Calculation* window [§ 10.4], after clicking the *Start* button. These messages are also available in the report. The calculation will be paused or stopped. Fatal errors need to be corrected before the analysis can be executed. Warnings can be discarded, by clicking *Continue*. A pause after warnings can be prevented, by unselecting the the *Halt on Warnings* checkbox in the *Program Options* window [§ 8.2.2].

10.5 Batch Calculation

MSettle offers the possibility to perform calculations in batch which means successive calculations for different input files. This can be usefull for time consuming calculations (probabilistic calculations for example).
To do so, MSettle program must be started from the *Run* window by specifying its location followed by '/b', as shown in Figure 10-13.

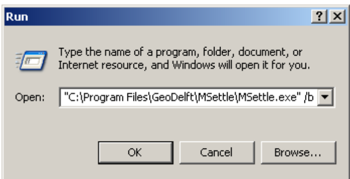


Figure 10-13 – *Run* window

Then the *Start Batch Calculation* window opens where the location of the files must be specified (Figure 10-14).

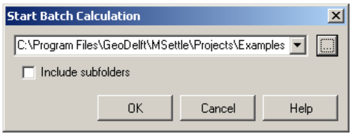


Figure 10-14 – *Start Batch Calculation* window

MSettle will run the specified files successively. The calculation progress can be viewed at the top of the *MSettle Calculation* window (Figure 10-15).

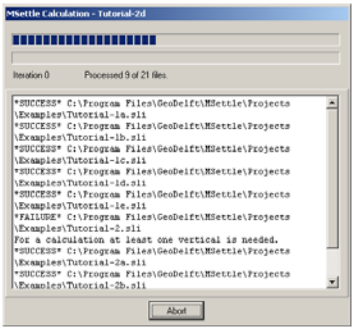


Figure 10-15 – *MSettle Calculation* window during batch calculation

11 View Results

On the menu bar, click *Results* to display the following menu options:

- *Report Selection* [§ 11.1], to select the content of the tabular report.
- *Report* [§ 11.2], to view a tabular report with selected content.
- *Stresses in Geometry* [§ 11.3], to graphically view the initial or final stress per vertical.
- *Dissipations* [§ 11.4], to view the degree of consolidation per layer as a function of time
- *Time-History Curves* for Terzaghi [§ 11.5.1] or Darcy [§ 11.5.2] to view graphs of data versus time per vertical.
- *Depth-History Curves* for Terzaghi [§ 11.6.1] or Darcy [§11.6.2] to view graphs of data along verticals.
- *Residual Settlement* [§ 11.7] to view a graph of the residual settlement starting from different time points
- *Settled Geometry* [§ 11.8], to graphically view the settled geometry within the original geometry.
- *Write Settled Geometry* [§ 11.9], to write the settled geometry to a new geometry file.
- *Write MStab Input* [§ 11.10], to write a MStab input with degrees of consolidation and with settled geometry.
- A special *Fit for Settlement Plate* analysis or *Reliability* analysis will yield the applicable results for just one vertical. Finally, the following special results are available after a reliability analysis:
- *Time-History (Reliability)* [§ 11.11], to view the total settlements together with the bandwidth, for the FOSM and the Monte Carlo method.
- *Influencing factors(Reliability)* [§ 11.12], to view the relative sensitivity of the total settlements (FOSM method) or the residual settlements (FORM method) to variations of uncertain parameters.
- *Residual Settlement (Reliability)* [§ 11.13], to view the residual settlement with bandwidth and reliability index, for the FORM and the Monte Carlo method.

11.1 Report Selection

On the menu bar, click *Results* and then choose *Report Selection* to open the *Report Selection* window (Figure 11-1) where the report content can be selected.

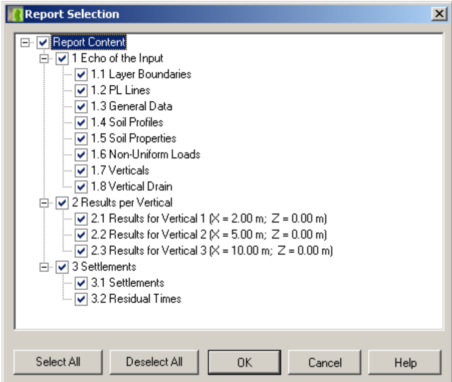


Figure 11-1 – *Report Selection* window

11.2 Report

On the menu bar, click *Results* and then choose *Report* to view a window displaying a table of the most recent analysis results.

Click the *Print* button to print the report or use the *Export Report* option from the *File* menu, in order to export the report in RTF, PDF, or HTML format. The content depends on the report selection [§ 11.1]. It can consist of:

- General section
- Program name and version, update, company name, license and copy number
- Title of the problem
- Names of the files used
- Echo of the input
- Stresses per vertical for Terzaghi model [§ 11.2.1] in the case of a long report
- Settlements per vertical for the Terzaghi model [§ 11.2.2] in the case of a long report
- Stresses and settlements per vertical for the Darcy model [§ 11.2.3] in the case of a long report
- Settlements [§ 11.2.4] and remaining settlements [§ 11.2.5]
- Maintain profile [§ 11.2.6] if the *Maintain Profile* option was used.

11.2.1 Stresses per vertical (Terzaghi)

In case of Terzaghi consolidation model, a stress table will be available for each selected vertical for initial and final states.

Report

Page 5 of 10

2 Results per Vertical

2.1 Results for Vertical 1 (X = 2.00 m; Z = 0.00 m)

Depth	Initial stress			Final stress		
	S-total	S-water	S-eff.	S-total	S-water	S-eff.
[m]	[kN/m²]	[kN/m²]	[kN/m²]	[kN/m²]	[kN/m²]	[kN/m²]
Layer 6						
-1.80	0.000	0.000	0.000	132.450	27.952	104.497
-1.90	1.460	1.020	0.440	133.209	28.258	104.951
-2.00	2.920	2.040	0.880	133.995	28.593	105.403
-2.10	4.380	3.060	1.320	134.796	28.943	105.853
-2.20	5.840	4.080	1.760	135.604	29.303	106.301
-2.30	7.300	5.100	2.200	136.419	29.672	106.747
-2.40	8.760	6.121	2.639	137.239	30.047	107.192
-2.50	10.220	7.141	3.079	138.063	30.428	107.635
-2.60	11.680	8.161	3.519	138.890	30.814	108.076
-2.70	13.140	9.181	3.959	139.720	31.204	108.516
-2.80	14.600	10.201	4.399	140.553	31.599	108.954
-3.35	22.630	15.812	6.818	145.170	33.838	111.332
-4.10	33.580	23.462	10.118	151.526	37.029	114.497
-4.90	45.260	31.623	13.637	158.345	40.569	117.777
Layer 5						
-4.90	45.260	31.623	13.637	158.345	40.569	117.777
-5.65	53.552	39.243	14.309	160.532	42.393	118.139
-6.35	61.322	46.384	14.938	162.552	44.144	118.408
-7.09	69.613	54.004	15.609	164.684	46.058	118.626
-7.79	77.383	61.145	16.239	166.663	47.892	118.770
Layer 4						
-7.79	77.384	61.145	16.239	166.663	47.893	118.770
-8.75	92.631	70.866	21.765	176.482	52.926	123.557
-9.75	108.631	81.067	27.564	186.815	58.320	128.496

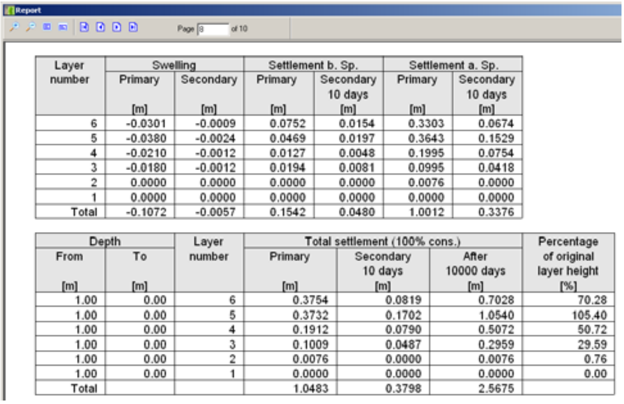
Figure 11-2 – *Report* window – Stresses per vertical (Terzaghi)

The following is an explanation of the column headings:

Depth	[m]	Depth of the point (= Y co-ordinate).
Initial Stress:		
- *S-total*	[kN/m²]	Initial total stress.
- *S-water*	[kN/m²]	Initial water pressure (hydrostatic and excess overpressure and underpressure).
- *S-eff.*	[kN/m²]	Initial effective stress.
Final Stress:		
- *S-total*	[kN/m²]	Final total stress.
- *S-water*	[kN/m²]	Final water pressure.
- *S-eff.*	[kN/m²]	Final effective stress.

11.2.2 Settlements per vertical (NEN-Koppejan with Terzaghi)

In case of NEN-Koppejan calculation model combined with Terzaghi consolidation model, two tables are printed for each selected vertical, as shown in Figure 11-3.



Layer number	Swelling		Settlement b. Sp.		Settlement a. Sp.	
	Primary [m]	Secondary [m]	Primary [m]	Secondary 10 days [m]	Primary [m]	Secondary 10 days [m]
6	-0.0301	-0.0009	0.0752	0.0154	0.3303	0.0674
5	-0.0380	-0.0024	0.0469	0.0197	0.3643	0.1529
4	-0.0210	-0.0012	0.0127	0.0048	0.1995	0.0754
3	-0.0180	-0.0012	0.0194	0.0081	0.0995	0.0418
2	0.0000	0.0000	0.0000	0.0000	0.0076	0.0000
1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Total	-0.1072	-0.0057	0.1542	0.0480	1.0012	0.3376

Depth		Layer number	Total settlement (100% cons.)			Percentage of original layer height
From [m]	To [m]		Primary [m]	Secondary 10 days [m]	After 10000 days [m]	[%]
1.00	0.00	6	0.3754	0.0819	0.7028	70.28
1.00	0.00	5	0.3732	0.1702	1.0540	105.40
1.00	0.00	4	0.1912	0.0790	0.5072	50.72
1.00	0.00	3	0.1009	0.0487	0.2959	29.59
1.00	0.00	2	0.0076	0.0000	0.0076	0.76
1.00	0.00	1	0.0000	0.0000	0.0000	0.00
Total			1.0483	0.3798	2.5675	

Figure 11-3 – *Report* window – Settlement per vertical (NEN-Koppejan with Terzaghi)

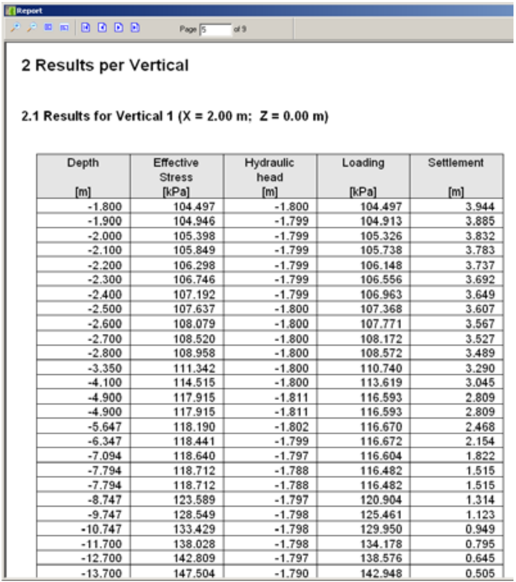
The following is an explanation of the column headings:

Layer number		Layer number.
Depth		
- *From*	[m]	Y co-ordinate at the top of the layer.
- *To*	[m]	Y co-ordinate at the bottom of the layer.
Swelling		
- *Primary*	[m]	Primary swelling.
- *Secondary*	[m]	Secondary swelling.
Settlement b. Sp (= settlement before preconsolidation stress)		
- *Primary*	[m]	Primary settlement.
- *Secondary 10 days*	[m]	Secondary settlement after 10 days.
Settlement a. Sp (= settlement after preconsolidation stress)		
- *Primary*	[m]	Primary settlement.
- *Secondary 10 days*	[m]	Secondary settlement after 10 days.
Total settlement (100% cons.)		
- *Primary*	[m]	Primary settlement.
- *Secondary 10 days*	[m]	Secondary settlement after 10 days.
- *After 10000 days*	[m]	Secondary settlement after 10000 days.
Percentage of original layer height	[%]	Percentage of the settlement relative to the original layer height.

NOTE: The settlements displayed in these tables are based on 100% consolidation.

11.2.3 Stresses, heads and settlements per vertical (Darcy)

A table with stresses and settlements is displayed in the report for selected verticals.



2 Results per Vertical

2.1 Results for Vertical 1 (X = 2.00 m; Z = 0.00 m)

Depth [m]	Effective Stress [kPa]	Hydraulic head [m]	Loading [kPa]	Settlement [m]
-1.800	104.497	-1.800	104.497	3.944
-1.900	104.946	-1.799	104.913	3.885
-2.000	105.398	-1.799	105.326	3.832
-2.100	105.849	-1.799	105.738	3.783
-2.200	106.298	-1.799	106.148	3.737
-2.300	106.746	-1.799	106.556	3.692
-2.400	107.192	-1.799	106.963	3.649
-2.500	107.637	-1.800	107.368	3.607
-2.600	108.079	-1.800	107.771	3.567
-2.700	108.520	-1.800	108.172	3.527
-2.800	108.958	-1.800	108.572	3.489
-3.350	111.342	-1.800	110.740	3.290
-4.100	114.515	-1.800	113.619	3.045
-4.900	117.915	-1.811	116.593	2.809
-4.900	117.915	-1.811	116.593	2.809
-5.647	118.190	-1.802	116.670	2.468
-6.347	118.441	-1.799	116.672	2.154
-7.094	118.640	-1.797	116.604	1.822
-7.794	118.712	-1.788	116.482	1.515
-7.794	118.712	-1.788	116.482	1.515
-8.747	123.589	-1.797	120.904	1.314
-9.747	128.549	-1.798	125.461	1.123
-10.747	133.429	-1.798	129.950	0.949
-11.700	138.028	-1.798	134.178	0.795
-12.700	142.809	-1.797	138.576	0.645
-13.700	147.504	-1.790	142.948	0.505

Figure 11-4 – *Report* window, *Results per Vertical* section (Darcy)

The following is an explanation of the column headings:

Depth	[m]	Vertical position (= Y co-ordinate).
Effective stress	[kPa]	Effective soil stress.
Hydraulic head	[m]	Full hydraulic head.
Loading	[kPa]	Top loading subjected to stress distribution.
Settlement	[m]	Settlement.

11.2.4 Settlements

In the *Settlements* section of the *Report* window, a short table displays the total settlement at the end of the calculation for each vertical.

Report

Page 8 of 11

3 Settlements

3.1 Settlements

Vertical number	X co-ordinate [m]	Surface level [m]	Settlement [m]
1	-10.00	-1.80	3.877
2	-5.00	-1.80	3.931
3	0.00	-1.80	3.951
4	2.00	-1.80	3.944
5	5.00	-1.80	3.908
6	10.00	-1.80	3.715

Figure 11-5 – *Report* window – Settlements

11.2.5 Residual Settlements

The *Residual Times* section of the Report window gives the output of the settlement for each vertical at all times that were specified in the *Calculation Times* window [§ 10.2]. Besides the settlement itself, the value of the remainder of the final settlement, and the reached percentage of the final settlement are also given.

Report

Page 8 of 9

3.2 Residual Times

Vertical number	Time [days]	Settlement [m]	Part of final settlement [%]	Residual settlements [m]
1	10000	3.877	100.000	0.000
	3000	3.822	98.573	0.055
	1100	3.794	97.853	0.083
	500	3.777	97.417	0.100
	300	3.578	92.286	0.299
	200	2.874	74.120	1.003
	80	1.290	33.259	2.588
	30	0.538	13.883	3.339
2	10000	3.931	100.000	0.000
	3000	3.874	98.550	0.057
	1100	3.844	97.795	0.087
	500	3.825	97.318	0.105
	300	3.623	92.163	0.308
	200	2.877	73.185	1.054
	80	1.290	32.819	2.641
	30	0.539	13.701	3.392
3	10000	3.951	100.000	0.000
	3000	3.900	98.710	0.051
	1100	3.875	98.078	0.076
	500	3.861	97.725	0.090
	300	3.652	92.434	0.299
	200	2.872	72.682	1.079
	80	1.290	32.640	2.661
	30	0.538	13.625	3.413
4	10000	3.944	100.000	0.000
	3000	3.895	98.765	0.049
	1100	3.872	98.175	0.072
	500	3.860	97.866	0.084
	300	3.648	92.493	0.296
	200	2.864	72.619	1.080
	80	1.286	32.611	2.658

Figure 11-6 – *Report* window – Residual settlements

11.2.6 Maintain Profile

If the *Maintain Profile* option was used, the extra amount of soil to be added is displayed in the *Maintain Profile Calculation Results* section of the *Report* window.

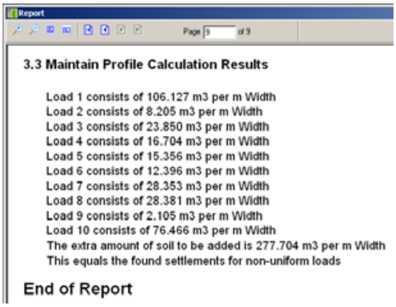


Figure 11-7 – *Report* window – Maintain Profile Calculation Results

11.2.7 Warnings and errors

Finally, if (non-fatal) warning/error messages were generated during the calculation and displayed in the *Start Calculation* window [§ 10.4.4], they can be found in this section of the report.

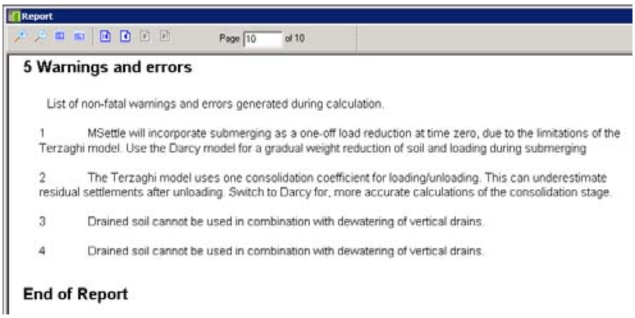


Figure 11-8 – *Report* window – Warnings and errors

11.3 Stresses in Geometry

Choose the *Stresses in Geometry* option in the *Results* menu to display the initial or final stress per vertical drawn in the geometry. The blue part represents the water pressure and the dark green part represents the additional effective stress. Use the *Stresses in Geometry* tab in the *Project Properties* menu to change visibility settings. This window can also be displayed by clicking the right-mouse button anywhere in the drawing and then choosing *View Preferences* from the pop-up menu.

Use the *Pan* and *Zoom* buttons to select the visible part.

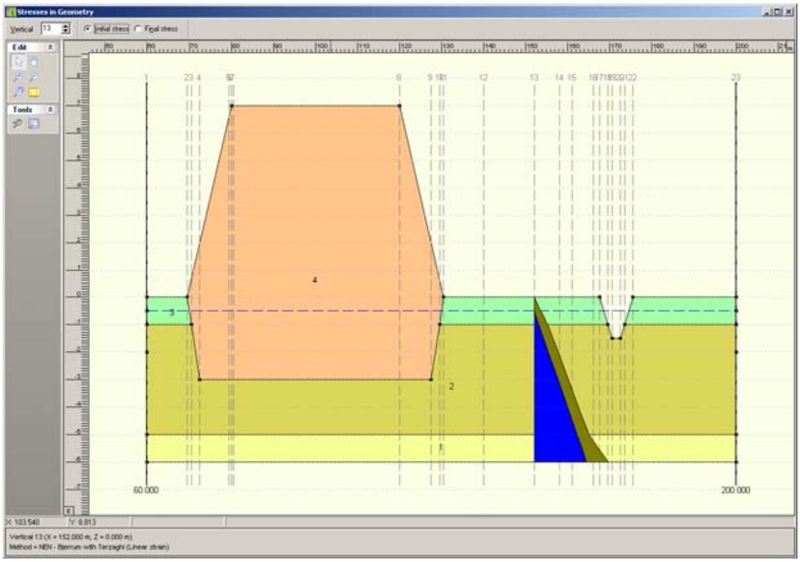


Figure 11-9 – *Stresses in Geometry* window

11.4 Dissipations

This option is available only if the *Add dissipation calculation* option in the *Start Calculation* window was selected [§ 10.4].
Choose the *Dissipations* option in the *Results* menu to display a graph of the average degree of consolidation versus the time, for a selected layer.
This graph can be used in combination with a stability analysis to estimate the allowed loading speed.

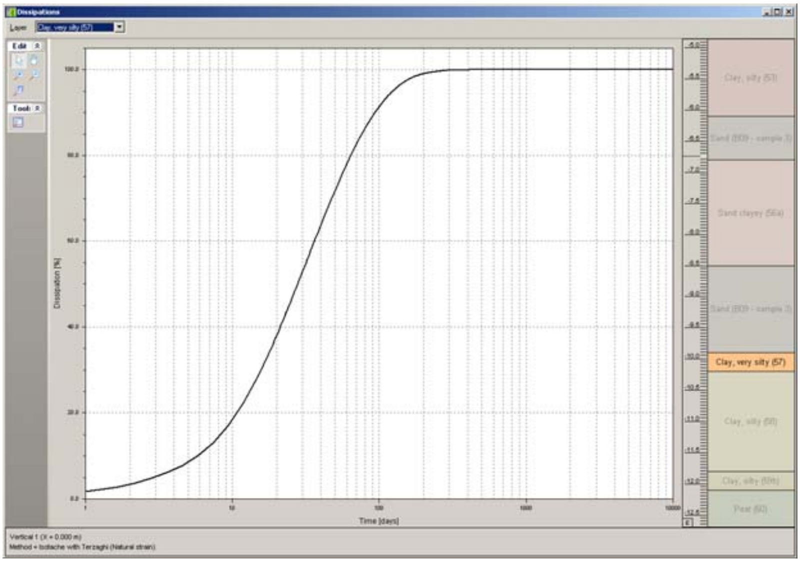


Figure 11-10 – *Dissipations* window

On the right hand side of the window, MSettle shows a graphical representation of the soil profile along the vertical.
A layer name can be select from the drop down list to see the results of the dissipation calculation for another layer. A new calculation must be performed to see the dissipation results for another vertical [§ 10.4].

NOTE: Click the right hand mouse button in the *Dissipations* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.5 Time-History

Choose the *Time-History* option in the *Results* menu to open the *Time-History* window. Depending on the selected consolidation model, the displayed window will be different:

- Refer to [§ 11.5.1] for Terzaghi consolidation model;
- Refer to [§ 11.5.2] for Darcy consolidation model.

11.5.1 Time-History – Terzaghi

For Terzaghi consolidation, the *Time-History* window displays graphs of the settlement and total loading versus time as shown in Figure 11-11.

- Click with the right hand mouse button inside the graph, in order to view and copy the chart data.

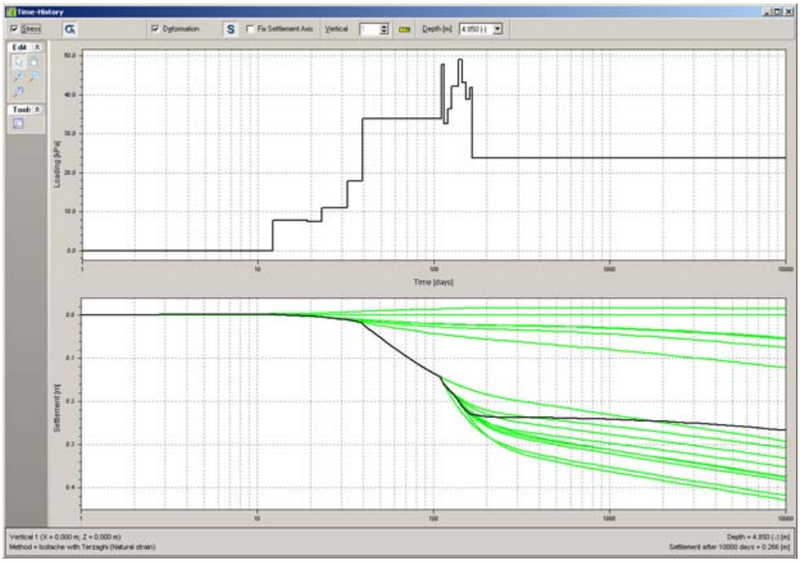
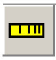


Figure 11-11 – *Time-History* window for Terzaghi consolidation

Stress	Enable this checkbox to display the graph of loading in time.
Deformation	Enable this checkbox to display the graph of settlement in time.
Fix Settlement Axis	Enable this checkbox to fix the range of the vertical axis of the graph of settlement whatever the selected time step.
	Click this button to switch from logarithmic to linear scale or vice versa.
Vertical	Type the vertical number that must be displayed or click the arrow-up and arrow-down keys to scroll through the available verticals.
Depth	Select a depth from the drop-down list. When typing the first digit of a desired depth, the next available depth starting with that digit is displayed. Use the arrow-down keys to scroll through the available depths.

Use the *Pan* and *Zoom* buttons to select the visible part.

At surface level, MSettle will plot also green lines in case of multiple load steps. These green lines indicate the predicted settlement that would occur if no further load steps were applied.

NOTE: Click the right hand mouse button in the *Time-History* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.5.2 Time-History – Darcy

For the Darcy model, the *Time-History* window displays graphs of settlements and stresses in time per vertical at a particular depth as shown in Figure 11-12.

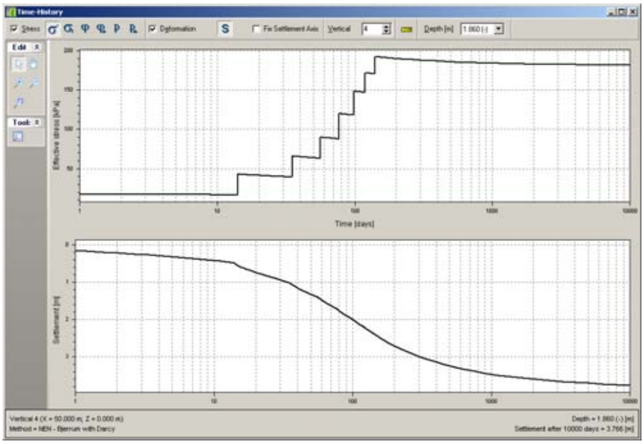
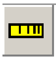


Figure 11-12 – *Time-History* window for Darcy consolidation

Stress	Enable this checkbox and then click one of the buttons to display respectively the effective stress, loading, hydraulic head, excess hydraulic head, pore pressure or excess pore pressure in the top chart.
Deformation	Enable this checkbox to display the graph of settlement in time in the bottom chart.
Fix Settlement Axis	Enable this checkbox to fix the range of the vertical axis of the graph of settlement whatever the selected time step.
Vertical	Type the vertical number that must be displayed or click the arrow-up and arrow-down keys to scroll through the available verticals.
	Click this button to switch from logarithmic to linear scale or vice versa.
Depth	Select a depth from the drop-down list. When typing the first digit of a desired depth, the next available depth starting with that digit is displayed. Use the arrow-down keys to scroll through the available depths.

Use the *Pan* and *Zoom* buttons to select the visible part.

NOTE: Click the right hand mouse button in the *Time-History* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.6 Depth-History

The *Depth-History* window from the *Results* menu displays graphs of settlements and stresses against the depth per vertical. Results displayed depend on the consolidation model:

- [§ 11.6.1] For Terzaghi consolidation model, graphs of settlements and initial and/or final stresses and preconsolidation pressure versus the depth per vertical are displayed;
- [§ 11.6.2] For Darcy consolidation model, graphs of settlements and stresses against the depth per vertical at a particular time are displayed.

11.6.1 Depth-History – Terzaghi

For the Terzaghi consolidation model, the *Depth-History* window displays:

- Graphs of initial or/and final stresses (water, total and effective stresses) and preconsolidation pressure versus the depth per vertical;
- Graph of settlements at a particular time or horizontal displacements against the depth per vertical.

The preconsolidation pressure distribution (red dotted line) corresponds to the initial preconsolidation pressure: maximum between the inputted value [§ 9.2] and the initial effective stress. It is available only for NEN-Koppejan model. Depending on the selected option for *Preconsolidation pressure within a layer* in the *Calculation Options* window [§ 10.1], the preconsolidation pressure distribution can vary: if the *Constant* option was selected, it is a vertical line but if the *Variable* option was selected, the it is parallel to the initial effective stress.

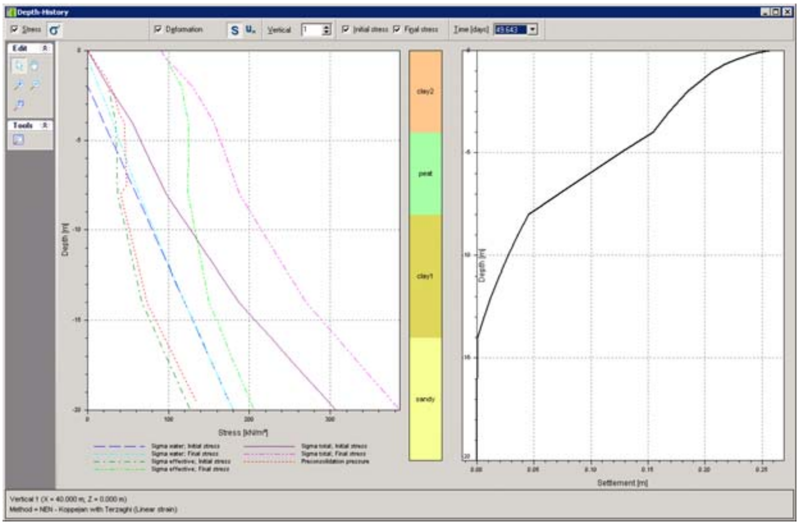


Figure 11-13 – *Depth-History* window for Terzaghi consolidation model

Stress	Enable this checkbox to display the initial and/or final pore pressure, total stress and effective stress in the left-hand chart.
Deformation	Enable this checkbox to display the graph of settlement in time or the graph of horizontal displacements in the right-hand chart.
Vertical	Type the vertical number that must be displayed or click the arrow-up and arrow-down keys to scroll through the available verticals.
Initial stress	Enable this checkbox to display the graphs of the initial stresses (total, effective and water stresses) against the depth.
Final stress	Enable this checkbox to display the graphs of the final stresses (total, effective and water stresses) against the depth.
Time	Select a time from the drop-down list to display the corresponding Depth-Settlement graph. When typing the first digit of a desired time, the next available time starting with that digit is displayed. Use the arrow-down keys to scroll through the available depths.

Use the *Pan* and *Zoom* buttons to select the visible part.

NOTE: Click the right hand mouse button in the *Depth-History* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.6.2 Depth-History – Darcy

For Darcy consolidation model, the *Depth-History* window displays graphs of settlements and stresses against the depth per vertical at a particular time.

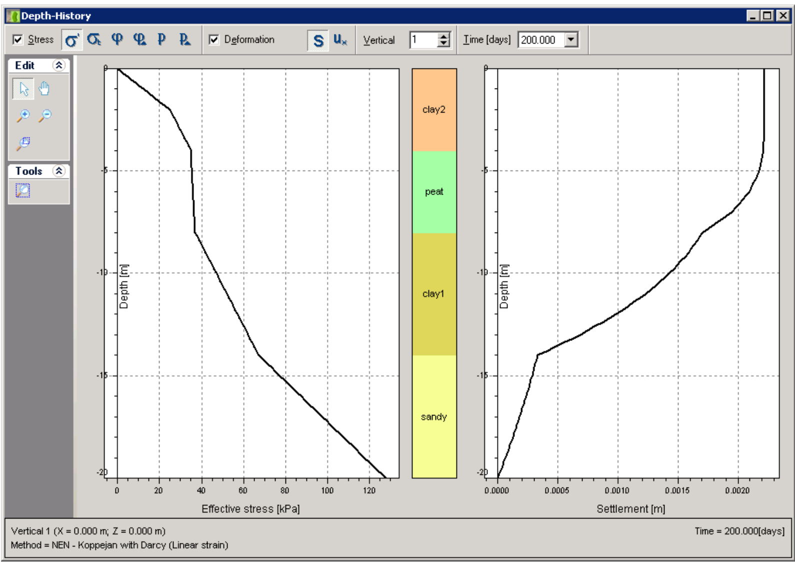


Figure 11-14 – *Depth-History* window for Darcy consolidation model

Stress	Enable this checkbox and then click one of the buttons to display respectively the effective stress, total stress, hydraulic head, excess hydraulic head, pore pressure or excess pore pressure in the left-hand chart.
Deformation	Enable this checkbox to display the graph of settlement in time or the graph of horizontal displacements in the right-hand chart.
Vertical	Type the vertical number that must be displayed or click the arrow-up and arrow-down keys to scroll through the available verticals.
Time	Select a time from the drop-down list. When typing the first digit of a desired time, the next available time starting with that digit is displayed. Use the arrow-down keys to scroll through the available depths.

Use the *Pan* and *Zoom* buttons to select the visible part.

NOTE: Click the right hand mouse button in the *Depth-History* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.7 Residual Settlement

The *Residual Settlement* window shows the residual settlements until the end of calculation. MSettle presents the values for residual settlements starting from different time points. These different points were defined in the *Calculation Times* window [§ 10.2].

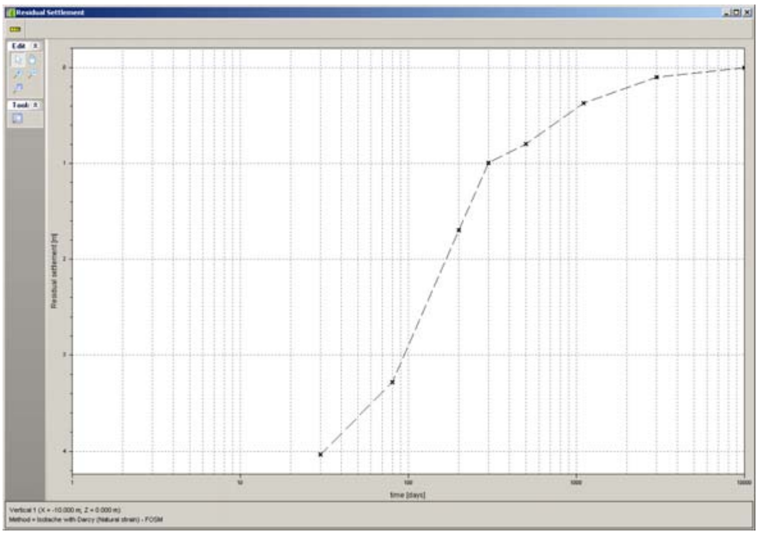


Figure 11-15 – *Residual Settlement* window

Click the button to switch from logarithmic to linear scale or vice versa.

Use the *Pan* and *Zoom* buttons to select the visible part.

NOTE: Click the right hand mouse button in the *Residual Settlement* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.8 Settled Geometry

The *Settled Geometry* option in the *Results* menu displays the settled geometry, drawn in the original geometry. MSettle can only generate a settled geometry if verticals were defined at all geometry points that are used in either a layer boundary or a non-uniform load. The settled geometry can be drawn with an *enlarge factor* that can be defined in the *Settled Geometry* tab of the *Project Properties* window [§ 9.1.3]. The display settings of this window can be modified here. To do this, either choose the *Properties* option in the *Project* menu, or click the right-mouse button anywhere in the drawing and choose *View Preferences* from the pop-up menu.

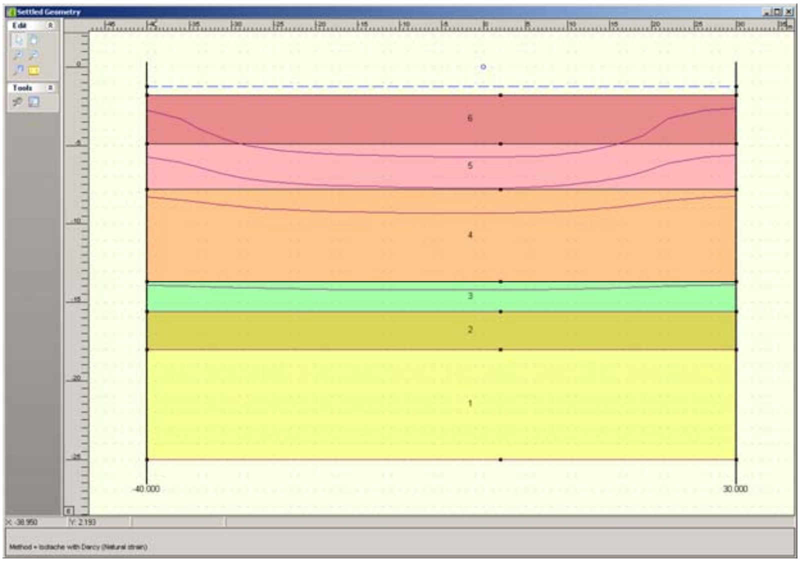


Figure 11-16 – *Settled Geometry* window

11.9 Write Settled Geometry

Once a calculation has been made, the settled geometry can be saved. In that way, a standard M-Series geometry file can be created.

Enable the *Add non-uniform loads as layer boundaries* checkbox to save the inputted non-uniform loads as layer boundaries. This is possible if:

- the volumetric mass of the load is positive;
- the non-uniform load is located above the surface.

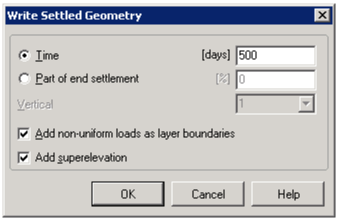


Figure 11-17 – *Write Settled Geometry* window

If the calculation was performed using the *Maintain Profile* option [§ 10.1.2], it is possible to enable the *Add Superelevation* checkbox to adapt the settled geometry with a superelevation load before writing it to file.

MSettle can only generate a settled geometry if verticals were defined at all geometry points that are used in either a layer boundary or a non-uniform load.

11.10 Write MStab Input

Once a calculation has been made, MSettle is able to generate an MStab input filewith settled geometry and with degrees of consolidation. MStab can then perform a slope stability analysis.

The output of the degree of consolidation requires that the *Add dissipation calculation* option in the *Start Calculation* window is enabled [§ 10.4.1].

NOTE: MStab takes only the effect of non-uniform loads on the degree of consolidation into account. The effect of other loading and the effect of underpressure in vertical drains are not included.

The generation of a settled geometry requires the same conditions as for *Write Settled Geometry* [§ 11.9].

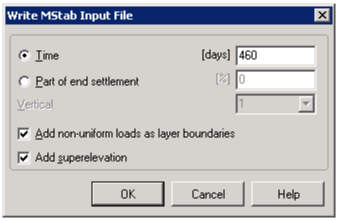


Figure 11-18 – *Write MStab Input* window

Enable the *Add non-uniform loads as layer boundaries* checkbox to save the inputted non-uniform loads as layer boundaries. This is possible if:

- the volumetric mass of the load is positive;
- the non-uniform load is located above the surface.

If the calculation was performed using the *Maintain Profile* option [§ 10.1.2], it is possible to enable the *Add Superelevation* checkbox to adapt the settled geometry with a superelevation load before writing it to file.

MSettle will attach complete soil properties to non-uniform loads and layers, when they are connected to a soil type in the database [§ 9.2.1, § 9.6.1]. While writing the MStab input file, MSettle will compare all materials and non-uniform loads with the materials in the selected database. If a name matches with a material name in the database, the soil properties are compared with the values in the database. If one of them deviates, MSettle prompts if you want to replace the values by the values found in the database.

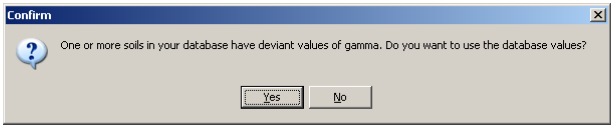


Figure 11-19 – *Confirm* window for replacement of database values

Numbers between parentheses that were added to names of uniform loads while selecting them from the database [§ 9.6.1], are removed before the material names are written to file.

11.11 Time-History (Reliability)

This option is available only if a reliability analysis with the *FOSM* or *Monte Carlo* method was performed [§ 10.4.2].
The *Time-History (Reliability)* window contains a graph of the mean value and the bandwidth of the time dependent settlement, at the surface position of the previously selected vertical. The bandwidth corresponds to a certain confidence interval. This interval can be viewed and modified in the *Confidence interval* at the top of the window.

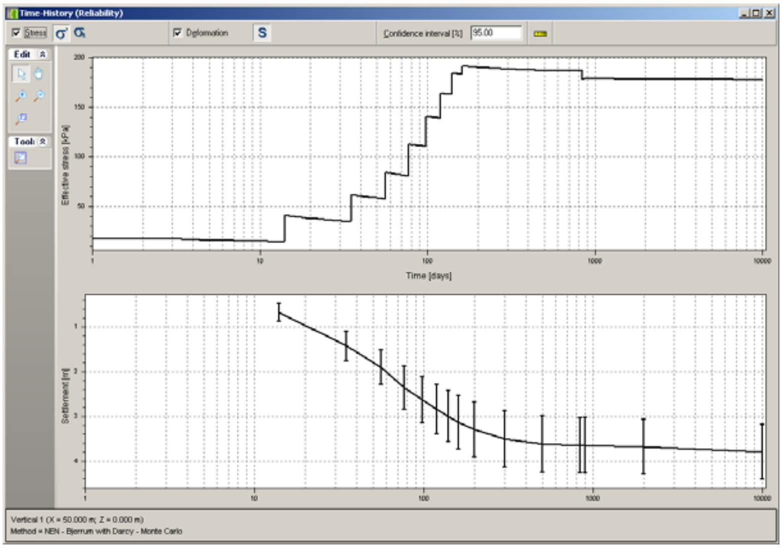


Figure 11-20 – *Time-History (Reliability)* window

See [§ 11.5] for a description of the options that are shared with the regular *Time-History* window.

NOTE: Click the right hand mouse button in the *Time-History (Reliability)* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

11.12 Influencing Factors (Reliability)

This option is available only if a reliability analysis with the *FOSM* or *FORM* method was performed [§ 10.4.2]. The *Influencing Factors (Reliability)* window contains a diagram, showing the relative sensitivity of the total settlement to variations of uncertain parameters. Different diagrams are available for all the different times that were defined in the *Calculation Times* window [§ 10.2].

Use the arrow-down key to scroll between the available time points in the *Time* list, at the top of the *Influencing factors* window.

A reliability analysis with the *FORM* method will yield a similar diagram with influencing factors for residual settlements. Different diagrams are available for residual settlements starting from different time points. These points were defined in the *Calculation Times* window. You can scroll between the available time points in the *Time* list, at the top of the *Influencing factors* window.

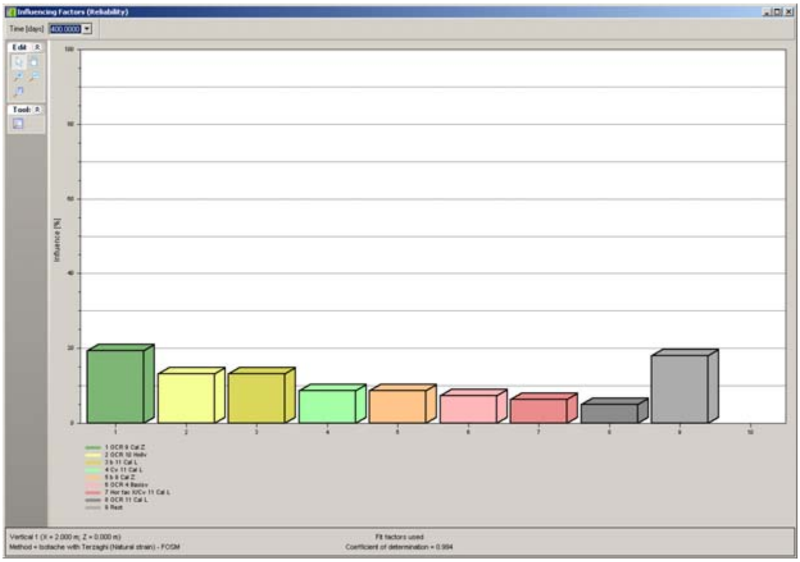


Figure 11-21 – *Influencing Factors (Reliability)* window

11.13 Residual Settlements (Reliability)

This option is available only if a reliability analysis with the *FORM* or *Monte Carlo* method was performed [§ 10.4.2]. The *Residual Settlements (Reliability)* window will contain a graph of the mean value and the bandwidth of the residual settlement, together with a graph of the reliability index (β) or the probability of failure (*P*). MSettle presents these values for residual settlements starting from different time points. These different points were defined in the *Calculation Times* window [§ 10.2].

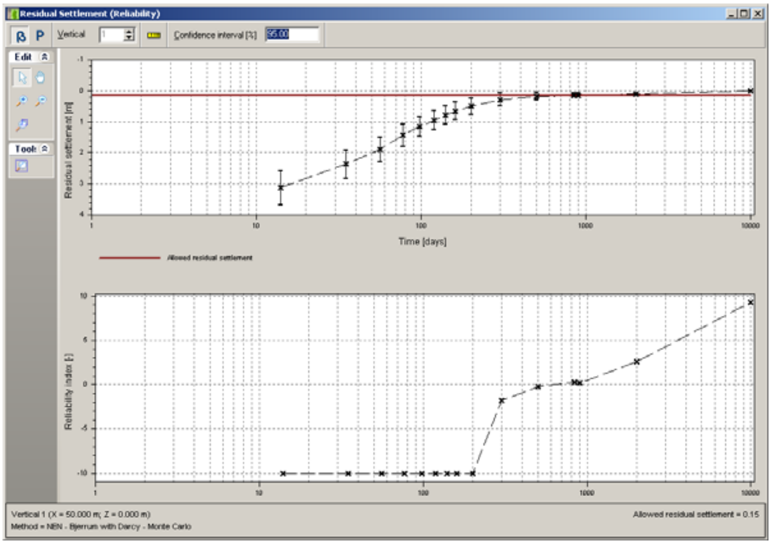


Figure 11-22 – *Residual Settlement (Reliability)* window

NOTE: Click the right hand mouse button in the *Residual Settlement (Reliability)* graph and select the *View Data* option to view all chart data, for convenient export to spread sheets.

12 Graphical Geometry Input

This chapter explains how to define the soil layers in a two-dimensional cross section by drawing, using the shared M-Series options for geometry modelling.

- [§ 12.1] introduces the basic geometrical elements that can be used.
- [§ 12.2] lists the restrictions and assumptions that the program imposes during geometry creation.
- [§ 12.3] gives an overview of the functionality of the *View Input* window.
- [§ 12.4] describes the creation and [§ 12.5] describes the manipulation of general graphical geometry using the *View Input* window.

Besides graphical input, the geometry can also be imported or tabular forms can be used (see [§ 9.3]). See the *MGeobase* manual for a description of special features to create cross-section geometry semi-automatically from CPT and/or boring records.

12.1 Geometrical objects

A M-Series geometry can be built step-by-step through the repetitive use of sketching, geometry creation and geometry manipulation. Each step can be started by using line-shaped construction elements [§ 12.1.2] to add line drawings. After converting these drawings to valid geometry parts, the specific geometry elements created can be manipulated [§ 12.1.1].

12.1.1 Geometry elements

An M-Series geometry can be composed from the following geometry elements:

Points	A point is a basic geometry element defined by its co-ordinates. As stated earlier, the geometry is restricted to two dimensions, allowing to define X and Y co-ordinates only.
Boundary lines	A boundary line is a straight line piece between two points and is part of a boundary.

Boundaries	A boundary is a collection of connected boundary lines that forms the continuous boundary between layers.
PL-lines	A piezometric level line is a collection of connected straight line pieces defining a continuous piezometric level.
Phreatic line	This is a PL-line that acts as phreatic line. The phreatic line (or groundwater level) is used to mark the border between saturated and unsaturated soil.
Layers	A layer is the actual soil layer. Its geometrical shape is defined by its boundaries, and its soil type is defined by its material.
Materials	A material defines the actual soil material (or soil type). It contains the parameters belonging to the soil type, such as its unsaturated weight and its saturated weight. A material can be connected to a layer in order to define the soil type of the layer.
Limits	A limit is a vertical boundary defining the 'end' at either the left or right side of the geometry. It is defined by an X co-ordinate only. **NOTE:** This is the only type of element that cannot be deleted.

Adding, moving and deleting the above-mentioned elements are subject to the conditions for a valid geometry (see [§ 12.2]). For example, while dragging selected geometry elements, the program can perform constant checks on the geometry validity [§ 12.4.4]. Invalid parts will be shown as construction elements (thick blue lines).

12.1.2 Construction elements

Besides the M-Series geometry elements [§ 12.1.1], special construction elements can also be used for sketching the geometry graphically. These elements are not a direct part of the geometry and the restrictions on editing (adding, moving, and deleting); these elements are therefore far less rigid. The only restriction that remains is that these elements cannot be moved and/or defined beyond the limits of the geometry.

Lines	A line consists of a starting point and end point, both defined by a left-hand mouse click in the graphic input screen.
Polylines	A polyline consists of a series of connected lines, all defined by a left-hand mouse click in the graphic input screen.

Construction elements will be displayed as solid blue lines. Valid constructions elements are converted to geometry elements as soon as the geometry is (re-) generated. For more information on adding lines and polylines, see [§ 12.4].

12.2 Assumptions and restrictions

During geometrical modelling, the program uses the following assumptions.

- Boundary number 0 is reserved for the base.

- A soil layer number is equal to the boundary number at the top of the layer.
- The boundary with the highest number defines the soil top surface.
- A material (soil type) must be defined for each layer – except for layer 0 (base). Different layers can use the same material.
- All the boundaries must start and end at the same horizontal co-ordinates.
- Boundaries should not intersect, but they may coincide over a certain length.
- All horizontal co-ordinates on a boundary must be ascending – that is, the equation $X[i+1] \geq X[i]$ must be valid for each following pair of X co-ordinates (vertical parts are allowed).
- PL-lines may intersect and may coincide with each other over a certain length.
- PL-lines and layer boundaries may intersect.
- All PL-lines must start and end at the same horizontal co-ordinate.
- All X co-ordinates on a PL-line must be strictly ascending – that is, the equation $X[i+1] > X[i]$ must be valid for each following pair of X co-ordinates (no vertical parts allowed).

One way for inputting geometry data is through the *Geometry* menu, as explained in the *Reference* section [§ 9.3]. This section describes an other way to create and manipulate geometry graphically using the tool buttons of the *View Input* window.

12.3 View Input Window

12.3.1 General

To use the *View Input* option, click the *Geometry* tab to activate it in the regular *View Input* window or use the menu to select it.

When the *Geometry* tab in the *View Input* window is selected, it displays a graphical representation of only the geometrical data. On the left of the window, the *Edit* and *Tools* buttons are displayed [§ 12.3.2]. On the right, the legend belonging to the geometry is displayed [§ 12.3.3]. At the bottom of the window, the title panel and the info bar are displayed. The title panel displays the project titles defined using the *Properties* option in the *Project* menu. The info bar provides information (from left to right) about the current cursor position, the current mode and the object currently selected. The legend, title panel and info bar are optional and can be controlled using the *Properties* option in the *Project* menu [§ 9.1.3].

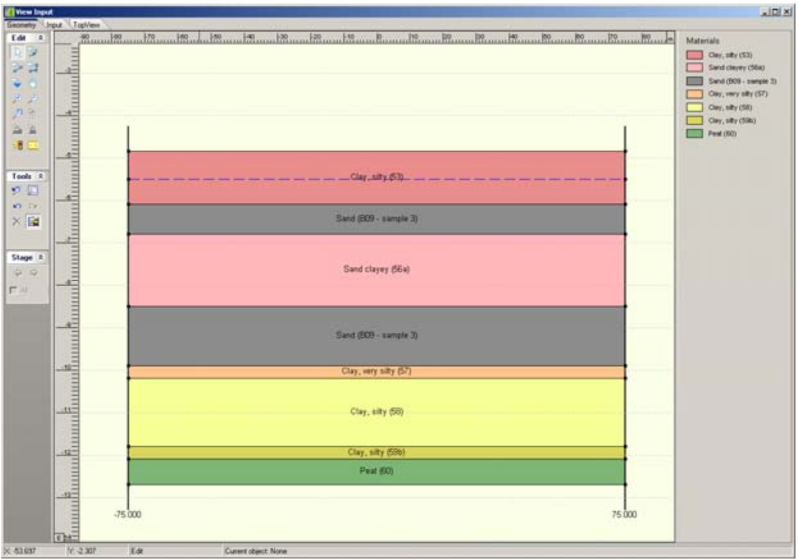


Figure 12-1 – *View Input* window, *Geometry* tab

It is possible to use three different modes when working in the *Geometry* tab of the *View Input* window:

Select	The *Select* mode is the default mode and enables the user to select existing elements in the window.
Add	The *Add* mode allows the addition of elements using one of the *Add* buttons. By selecting one of these buttons, one switches to the *Add* mode. As long as this mode is active, the user can add the type of element which is selected.
Zoom	The *Zoom* mode allows the user to view the input geometry in different sizes. By selecting one of the *Zoom* buttons or the *Pan* button, one activates the *Zoom* mode. While in this mode, the user can repeat the zoom or pan actions without reselecting the buttons.

It is possible to change modes in the following ways. When in *Add* or *Zoom* mode, it is possible to return to the *Select* mode by clicking the right-hand mouse button, or by pressing the *Escape* key, or by clicking the *Select mode* button. To activate the *Add* mode, select one of the *Add* buttons. To activate the *Zoom* mode, select one of the *Zoom* buttons or the *Pan* button.

NOTE: The current mode is displayed on the info bar at the bottom of the *View Input* window.

12.3.2 Buttons

Select and *Edit* mode
In this mode, the left-hand mouse button can be used to graphically select a previously defined grid, load, geotextile or forbidden line. Items can then be deleted or modified by dragging or resizing, or by clicking the right-hand mouse button and choosing an option from the menu displayed. Pressing the *Escape* key will return the user to this *Select* and *Edit* mode.

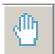
Add point(s) to boundary / PL-line
Click this button to add points to all types of lines (e.g. polylines, boundary lines, PL-lines). By adding a point to a line, the existing line is split into two new lines. This provides more freedom when modifying the geometry.

Add single line(s)
Click this button to add single lines. When this button is selected, the first left-hand mouse click will add the info bar of the new line and a “rubber band” is displayed when the mouse is moved. The second left-hand mouse click defines the end point (and thus the final position) of the line. It is now possible to either go on clicking start and end points to define lines, or stop adding lines by selecting one of the other tool buttons, or by clicking the right-hand mouse button, or by pressing the *Escape* key.

Add polyline(s)
Click this button to add polylines. When this button is selected, the first left-hand mouse click adds the starting point of the new line and a “rubber band” is displayed when the mouse is moved. A second left-hand mouse click defines the end point (and thus the final position) of the first line in the polyline and activates the “rubber band” for the second line in the polyline. Every subsequent left-hand mouse click again defines a new end point of the next line in the polyline. It is possible to end a polyline by selecting one of the other tool buttons, or by clicking the right-hand mouse button, or by pressing the *Escape* key. This also stops adding polylines altogether.
A different way to end a polyline is to double-click the left-hand mouse button. Then the polyline is extended automatically with an ‘end line’. This end line runs horizontally from the position of the double-click to the limit of the geometry in the direction the last line of the polyline was added. Therefore, if the last line added was defined left to right, the ‘end line’ will stop at the right limit.
NOTE: By finishing adding a polyline this way, it is possible to start adding the next polyline straight away.

Add PL-line(s)
Click this button to add a piezometric level line (PL-line). Each PL-line must start at the left limit and end at the right limit. Furthermore, each consecutive point must have a strictly increasing X co-ordinate. Therefore, a PL-line must be defined from left to right, starting at the left limit and ending at the right limit. To enforce this, the program will always relocate the first point clicked (left-hand mouse button) to the left limit by moving it

horizontally to this limit. If trying to define a point to the left of the previous point, the rubber band icon indicates that this is not possible. Subsequently clicking on the left side of the previous point, the new point will be added at the end of the rubber band icon instead of the position clicked.
As with polylines, it is also possible to end a PL-line by double-clicking the left-hand mouse button. In this case, the automatically added 'end line' will always end at the right limit. To stop adding PL-lines, select one of the other tool buttons, or click the right-hand mouse button, or press the *Escape* key.



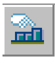
Pan
Click this button to change the visible part of the drawing by clicking and dragging the mouse.

Zoom in
Click this button to enlarge the drawing, then click the part of the drawing which is to be at the centre of the new image. Repeat if necessary.

Zoom out
Click this button, then click on the drawing to reduce the drawing size. Repeat if necessary.

Zoom rectangle
Click this button then click and drag a rectangle over the area to be enlarged. The selected area will be enlarged to fit the window. Repeat if necessary.

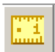
Add vertical
Click this button to graphically define the position of a vertical.



Add non-uniform load
Click this button to display a window in which it is possible to add, modify or delete non-uniform loads.

Add other load
Click this button to display a window in which it is possible to add, modify or delete trapeziform, circular, rectangular or/and uniform loads.

Convert geometry to 1D
Click this button to convert a 2D geometry into 1D.



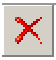
Measure the distance and slope between two points
Click this button, then click the first point on the *View Input* window and place the cross on the second point. The distance and the slope between the two points can be read beside the second point. To turn this option off, click the escape key.

Undo zoom
Click this button to undo the zoom. If necessary, click several times to retrace each consecutive zoom-in step that was made.

Zoom limits
Click this button to display the complete drawing.

Undo
Click this button to undo the last change(s) made to the geometry

Redo
Click this button to redo the previous Undo action

	*Delete* Click this button to delete a selected element. **NOTE:** This button is only available when an element is selected. See [§ 12.5.2] for more information on how using this button.
	*Automatic regeneration of geometry on/off* When selected, the program will automatically try to generate a new valid geometry whenever geometry modifications require this. During generation, (poly)lines (solid blue) are converted to boundaries (solid black), with interjacent layers. New layers receive a default material type. Existing layers keep the materials that were assigned to them. Invalid geometry parts are converted to construction elements. Automatic regeneration may slow down progress during input of complex geometry, because validity will be checked continuously.
	*Previous stage* Click this button to view the previous stage in the sequence of loading.
	*Next stage* Click this button to view the next stage in the sequence of loading.

12.3.3 Legend

At the right side of the *View Input* window (Figure 12-2) the legend belonging to the geometry is shown. This legend is present only if the *Legend* checkbox in the *View Input* tab of the *Project Properties* window is activated (see [§ 9.1.3]).

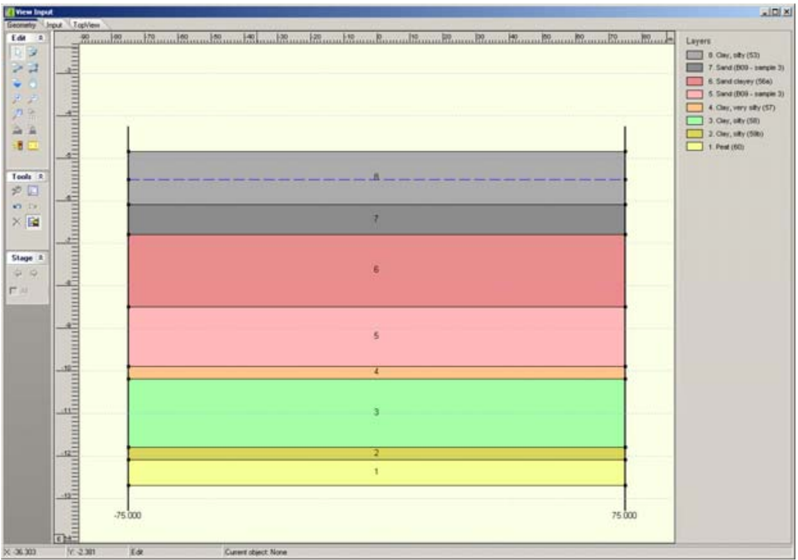


Figure 12-2 – *View Input* window, *Geometry* tab (legend displayed as *Layer Numbers*)

In the *Geometry* tab of the *View Input* window, it is possible to change the type of legend. When a soil type box in the legend is right clicked, the menu from Figure 12-3 is displayed.

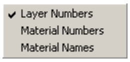


Figure 12-3 – Legend, Context menu

With this menu, there are three ways to display the legend of the layers:

- As *Layer Numbers*: the legend displays one box for each layer. Each layer (and therefore each box) is displayed in a different standard colour. Next to each box, the layer number and the material name are displayed, corresponding to the colour and number of the layer in the adjacent *Geometry* window (see Figure 12-2).
- As *Material Numbers*: the legend displays one box for each material. Each material (and therefore each box) is displayed in a different colour which can be changed by the user (see below). Next to each box, the material number and name are displayed, corresponding to the colour and number of the material in the adjacent *Geometry* window (see Figure 12-4).

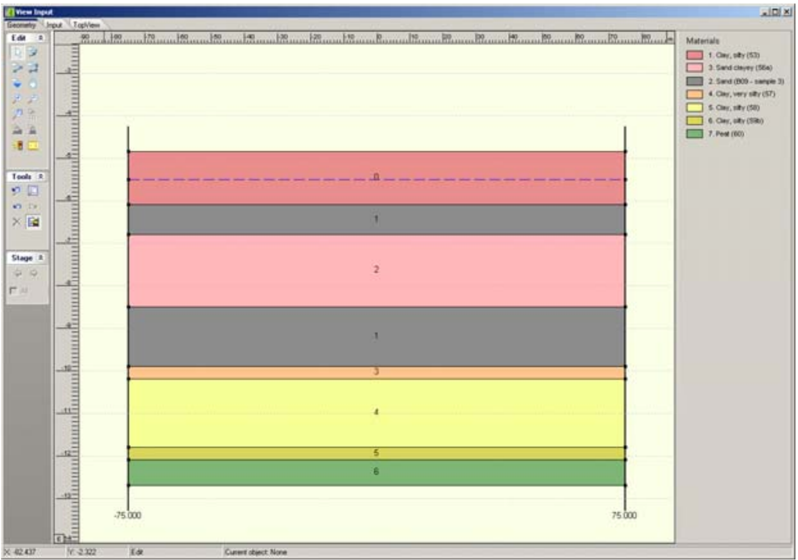


Figure 12-4 – *View Input* window, *Geometry* tab (legend displayed as *Material Numbers*)

- As *Material Names*: the legend displays one box for each material. Each material (and therefore each box) is displayed in a different colour which can be changed by the user (see below). Next to each box, only the material name is displayed, corresponding to the colour and name of the material in the adjacent *Geometry* window (see Figure 12-1).

Unlike the standard colors used to display layers with their layer colors, it is possible to define different colors used when displaying materials. To change the colour assigned to a material, right click the material box. The menu from Figure 12-5 is displayed.

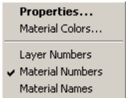


Figure 12-5 – Legend, Context menu (for legend displayed as *Materials*)

When selecting *Material Colors* the *Colour* window appears (Figure 12-6), in which the user can pick a colour or even define customized colors himself (by clicking the *Define Custom Colors* button).

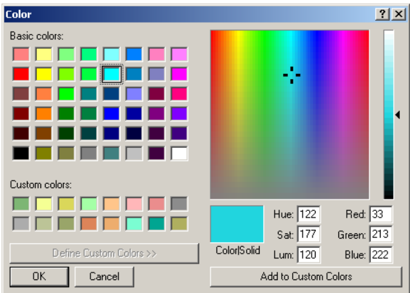


Figure 12-6 – *Colour* window

12.4 Geometry modelling

12.4.1 Create a new geometry

There are two ways to create a new geometry without the wizard:

- Open the *Geometry* menu and choose *New*.
- Open the *File* menu and choose *New*. In the *New File* window displayed, select *New geometry* and click *OK* (see [§ 8.1]).

In both cases, the *Geometry* tab of the *View Input* window is displayed (Figure 12-7) with the default limits of the geometry (from 0 to 100 m).

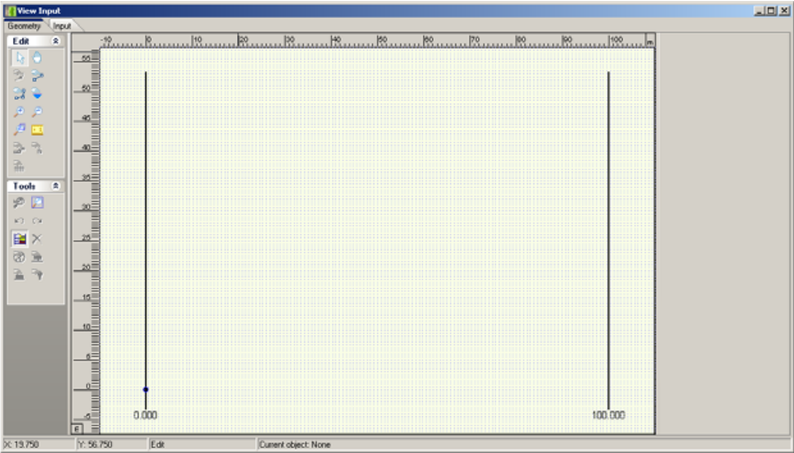


Figure 12-7 – *View Input* window, *Geometry* tab

12.4.2 Set limits

The first thing to do when creating new geometry is to set the model limits. This is possible by selecting and then dragging the limits to their proper place one by one. It is also possible to select a limit and edit its value by clicking the right-hand mouse button after selecting the limit and then choosing the *Properties* option in the pop-up menu. The property window belonging to the selected limit is displayed (Figure 12-8), enabling to define the new X co-ordinate for this limit.

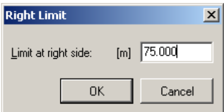
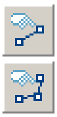


Figure 12-8 – *Right Limit* window

12.4.3 Draw layout

It is possible to use the *Add single line(s)*, *Add polyline(s)* and *Add point(s) to boundary / PL-line* buttons to draw the layout of the geometry. See below for more information on how to use these buttons.



Add single line(s) and Add polyline(s)
Each (poly)line is displayed as a solid blue line, and each point as a small black rectangle (Figure 12-9).

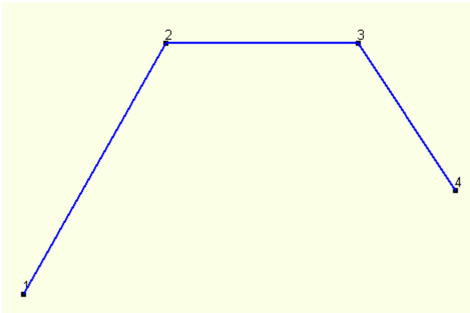
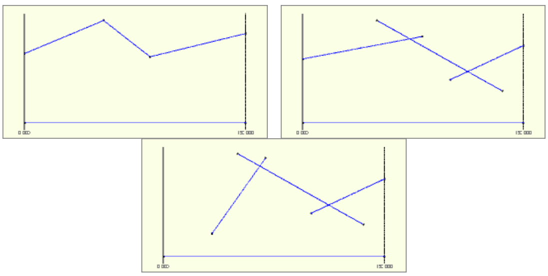


Figure 12-9 – Representation of a polyline

The position of the different points of a (poly)line can be modified by dragging the points as explained in [§ 12.5.4] or by editing the (poly)line. This is done by clicking the right-hand mouse button after selecting the (poly)line and then choosing the *Properties* option in the pop-up menu [§ 12.5.3].

The underlying grid helps the user to add and edit (poly)lines. Use the *Properties* option in the *Project* menu to adjust the grid distance and force the use of the grid by activating *Snap to grid* [§ 9.1.3]. When this option is activated, each point is automatically positioned at the nearest grid point.

The specified line pieces must form a continuous line along the full horizontal width of the model. This does not mean that each line piece has to be connected exactly to its predecessor and/or its successor. Intersecting line pieces are also allowed, as shown in the examples of Figure 12-10.

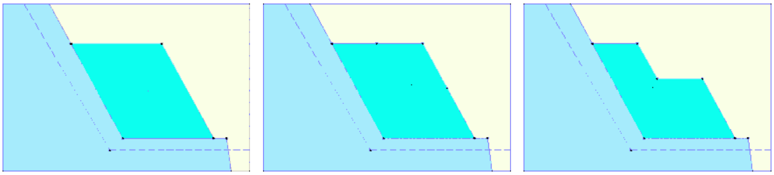


(1) (2) (3)

Figure 12-10 – Examples of configurations of (poly)lines

- Configuration (1) is allowed. The different lines are connected and run from boundary to boundary
- Configuration (2) is also allowed. The different are connected. They are defined as being connected because they intersect. The line construction runs from boundary to boundary.
- Configuration (3) is illegal, as there is no connection with the left boundary.

Add point(s) to boundary / PL-line
Use this button to add extra points to lines (lines, polylines, boundary lines, PL-lines). By adding a point to a line, the existing line is split into two new lines. This provides more freedom when modifying the geometry.
For example, the shape of the berm of Figure 12-11 (1) needs to be modified. Two points are added to the outer lines of the berm as shown in Figure 12-11 (2). Then, the middle point is selected and dragged to the position that completes the new geometry as shown in Figure 12-11 (3).



(1) (2) (3)

Figure 12-11 – Modification of the shape of a berm

NOTE: When the *Add point(s) to boundary / PL-line* button is clicked, each left-hand mouse click adds a new point to the nearest line until one of the other tool buttons is selected, or click the right-hand mouse button, or press the *Escape* key.

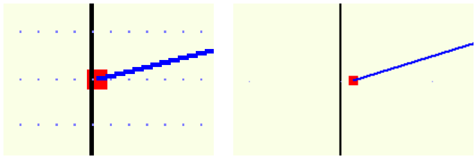
12.4.4 Generate layers

Use the *Automatic regeneration of geometry on/off* button to start or stop the automatic conversion of construction elements to actual boundaries and layers. Valid (poly)lines are converted to boundaries, which are displayed as black lines. Invalid lines remain blue.

Layers are generated between valid boundaries, and default soil types are assigned.

It is possible to modify the soil type assigned to a layer by first selecting the layer and then clicking the right-hand mouse button and choosing the *Layer Properties* option in the pop-up menu to display the *Layer* window (see Figure 12-20 in [§ 12.5.3]). Once a material has been assigned to a layer, this material will continue to be associated to that layer in subsequent conversions of construction elements as long as the layer is not affected by those conversions.

The most common cause of invalid (poly)lines is that they are not part of a continuous polyline running from limit to limit. Sometimes, lines appear to start/end at a limit without actually being on a limit. Figure 12-12 gives an example: on the left geometry (1), the end of the line seems to coincide with the boundary. However, zooming in on the point (geometry (2) on the right) reveals that it is not connected to the boundary. Therefore the geometry is considered invalid.

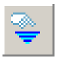


(1) (2)

Figure 12-12 – Example of invalid point not connected to the left limit

It is possible to correct this by dragging the point to the limit while the specific area is zoomed in or by selecting the point, clicking the right-hand mouse button, choosing the *Properties* option in the pop-up menu [§ 12.5.3] and making the X co-ordinate of the point equal to the X co-ordinate of the limit.

12.4.5 Add piezometric level lines



It is possible to use the button *Add PL-line(s)* to add PL-lines. When adding a PL-line, MSettle imposes the limitation that the subsequent points of the PL-line have an increasing X co-ordinate. Furthermore the first point of a PL-line is to be set on the left boundary and the last point on the right boundary.

It is possible to change the position of the different points of a PL-line by dragging the points as explained in [§ 12.5.4] or by editing the PL-line. This is done by selecting the PL-line, clicking the right-hand mouse button and choosing the *Properties* option in the pop-up menu [§ 12.5.3].

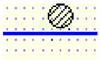
12.5 Graphical manipulation

12.5.1 Selection of elements

After selecting a geometry element it is possible to manipulate it. In order to be able select a geometry element, the select mode should be active. Then it is possible to select an element by clicking the left-hand mouse button. To select a layer, click on the layer number, material number or material name, depending on the option chosen in the *Properties* dialog in the *Project* menu. When successfully selected, the element will be displayed highlighted (for example, a point will be displayed as a large red box instead of a small black box).

The following remarks are relevant to selection accuracy and ambiguity.

Accuracy
The program draws a circular selection area around the mouse pointer. If the element falls within this circle, it will be selected when click the left-hand mouse button is clicked (Figure 12-13).



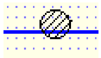


Figure 12-13 – Selection accuracy as area around cursor

The *Selection accuracy* determines the required distance between the mouse pointer and the geometrical element for selection. It is possible to use the *Properties* option in the *Project* menu to modify the accuracy [§ 9.1.3]. This is defined in percentages of the screen size and its default value is 2%. If a larger percentage is defined, this increases the selection area. However, if the percentage is set to a relatively high value, the accuracy required for the selection of certain geometry items may be inaccurate. In other words, it will most likely result in too many 'ambiguous' selections (see the following section), or will make it difficult to perform an intentionally empty selection.

Ambiguous selection

A selection of geometrical elements can be ambiguous. Figure 12-14 gives an example: a user may want to select a point, a boundary line, a boundary or a PL-line. As several elements are in close proximity to each other, MSettle does not automatically select an element.

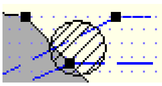


Figure 12-14 – Selection accuracy as area around cursor

In this case MSettle requires the user to assign the element that is to be selected by displaying a pop-up menu (Figure 12-15) with the available types of elements within the range of the selection click. It is possible to select the element from this menu.

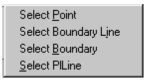
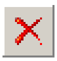


Figure 12-15 – Selection accuracy as area around cursor

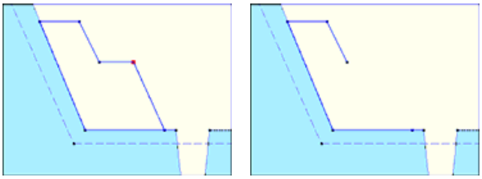
Clear selection

It is possible to clear a selection by clicking in an area without geometry elements in the direct area.

12.5.2 Deletion of elements



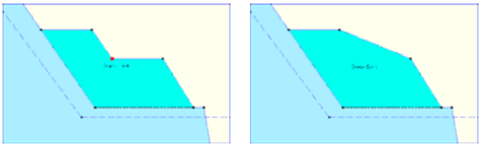
Click the *Delete* button to delete a selected element. This button is only available when an element is selected. When a point is selected and deleted, it and all lines connected to it are deleted as shown in Figure 12-16.



Before After

Figure 12-16 – Example of deletion of a point

When a geometry point (a point used in a boundary or PL-line) is selected and deleted, the program deletes the point and its connected boundary lines as shown in Figure 12-17. It then inserts a new boundary that reconnects the remaining boundary lines to a new boundary.

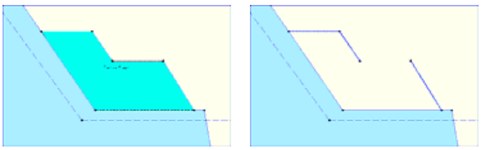


Before After

Figure 12-17 – Example of deletion of a geometry point

Deletion of a geometry element (boundary, boundary line, geometry point, PL-line) can result in automatic regeneration of a new valid geometry, if the *Automatic regeneration* option is switched on.

When a line is selected and then deleted, the line and its connecting points are deleted as shown in Figure 12-18. In addition the layer just beneath that boundary is deleted. All other line parts that are not part of other boundaries will be converted to construction lines.



Before After

Figure 12-18 – Example of deletion of a line

12.5.3 Using the right-hand mouse button

When using the mouse to make geometrical manipulations, the right mouse button enables full functionality in a pop-up menu, while the left button implies the default choice. The options available in the pop-up menu depend on the selected geometrical element and the active mode.

When the *Select* mode is active and the right-hand mouse button is clicked, the pop-up menu of Figure 12-19 is displayed.

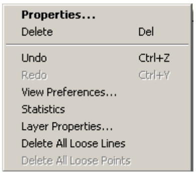


Figure 12-19 – Pop-up menu for right-hand mouse menu (Select mode)

Properties...	When this option is clicked, the property editor for the selected object is displayed. This procedure is performed by first selecting an object by clicking on it with the left-hand mouse button. Then clicking the right-hand mouse button anywhere in the graphic window will display the pop-up menu. It is possible to use the property editor to quickly adapt the values (properties) of the selected object. Each type of element requires its own properties and therefore its own property editor as shown from Figure 12-21 to Figure 12-24 below.
Delete	This option deletes the element that has been selected (see the comments for the *Delete* button in [§ 12.5.2]).
Undo	This option will undo the last change(s) made to the geometry.
Redo	This option will redo the previous *Undo* action.
View Preferences	This option opens the *Properties* dialog in the *Project* menu as displayed in.
Statistics	It is possible to use this option to view a window displaying all the vital statistics of the input data. **NOTE:** In the window construction lines are called free lines.
Layer Properties...	This option is a special feature that edits the material properties of layers. It is possible to click anywhere in a layer and directly choose this option to edit its properties (Figure 12-20). Clicking outside the geometry layers will display the menu with the *Layer Properties* option disabled, as there is no layer for which properties can be displayed.
Delete All Loose Lines	This option will delete all loose lines. Loose lines are actually construction lines that are not part of the boundaries or PL-lines (therefore, all lines displayed as solid blue lines). With this option, it is possible to quickly erase all the “leftover bits” of loose lines that may remain after converting lines to a geometry.
Delete All Loose Points	This option will delete all loose points.

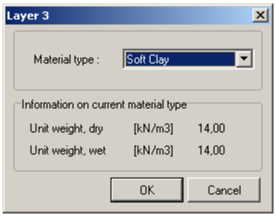


Figure 12-20 – *Layer* window (Property editor of a layer)

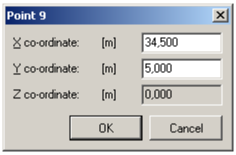


Figure 12-21 – *Point* window (Property editor of a point)

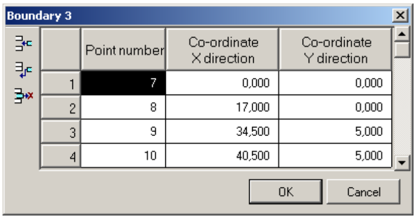


Figure 12-22 – *Boundary* window (Property editor of a polyline)

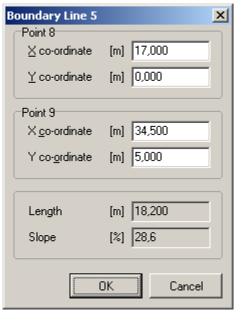


Figure 12-23 – *Boundary* window (Property editor of a line)

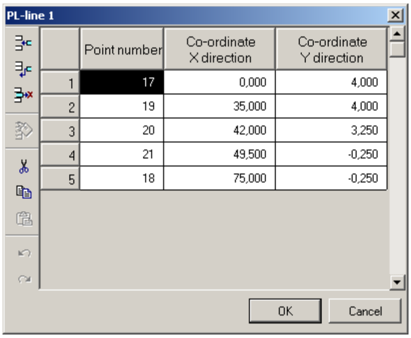
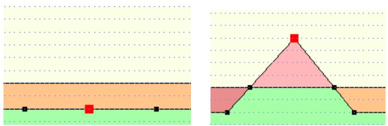


Figure 12-24 – *PL-line* window (Property editor of a PL-line)

NOTE: In the *Boundary* and *PL-line* properties windows, only the point's number can be modified, not the X and Y co-ordinates.

12.5.4 Dragging elements

One way to modify elements is to drag them to other locations. To drag an element, first select it. Once the element has been selected, it is possible to drag it by pressing and holding down the left-hand mouse button while relocating the mouse cursor. Dragging of geometry elements can result in automatic regeneration of geometry, if this option is switched on [§ 12.4.4] as shown in the example of Figure 12-25: when the selected point is moved upwards, a new geometry will be created. MSettle creates new layers according to this new geometry.



Before After

Figure 12-25 – Example of dragging of a point

12.6 Working With 1D Geometries

MSettle is primarily intended for working with 2D geometries. However, a special input window is available for editing 1D geometries, graphically, or by means of a table where levels, material names and a phreatic level can be edited.

12.6.1 Creating a 1D Geometry

MSettle will always start from a new or existing 2D geometry. Therefore, choose the *New* option from the *File* menu to create a new empty geometry, or open an existing

2D geometry and then convert it into a 1D geometry as explained in the paragraph below [§ 12.6.2].

12.6.2 Converting a 2D Geometry into a 1D Geometry

There are three ways of converting 2D geometry into 1D geometry.

The first one is common for new geometries. The first option is to simply change the model from 2D from 1D. In the *Project* menu, open the *Model* dialog and select *1D* for the input option *Dimension* [§ 9.1.1]. After this option is selected, an input window opens that allows entering the x co-ordinate of the location where the 1D geometry should be derived from. Either enter this co-ordinate manually, or select an x co-ordinate by choosing one of the verticals that are listed in the input window. Before the conversion takes place, MSettle prompts if the user really wants to continue.

NOTE: 1D geometry contains less information than a 2D geometry, and therefore conversion nearly always implies a loss of data.

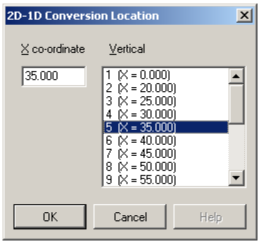


Figure 12-26 – *2D-1D Conversion Location* window

There are two other ways of converting a 2D geometry into 1D geometry. For both of them you need to graphically indicate the location where the conversion must take place.

- One way of indicating this location is by pressing the *Convert geometry to 1D* button in the *View Input* window, and clicking the location in the graphical representation of the geometry.
- The other way is selecting a vertical by mouse and choosing the *Convert geometry to 1D* item from the popup menu that appears when right clicking the input window.

12.6.3 The 1D Geometry Input Window

The *1D Geometry* window enables to edit the 1D geometry, either by dragging lines by mouse, or by editing data from a table.

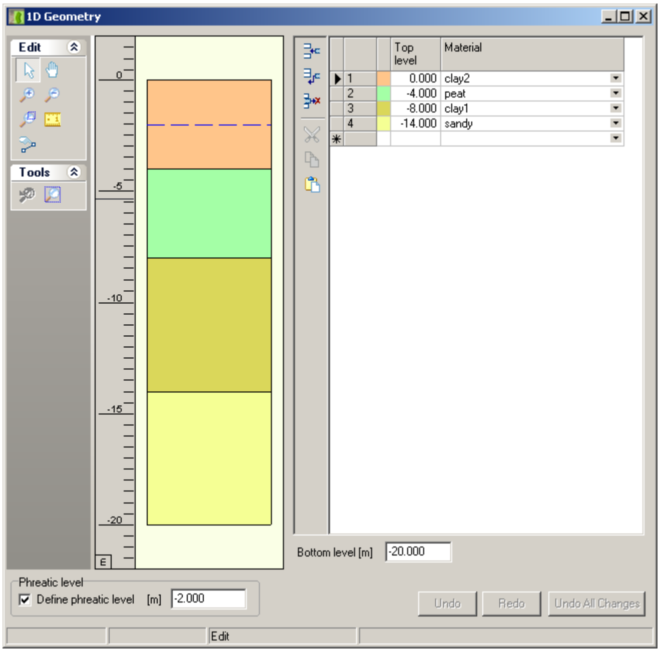
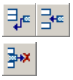


Figure 12-27 – *1D Geometry* window



Add, insert or delete layers by pressing the corresponding buttons on the left side of the table. Top levels can be edited for all layers. For the bottom layer, the bottom level can be edited as well.
Graphically changing the data is possible by dragging layer boundaries and the phreatic level, if present, and by splitting a layer into two layers by clicking on it after you have pressed the Add boundary button on the toolbar.

Introduction

Tutorial

Reference

Background

Verification

Embankment Design and Soil Settlement Prediction

Deltares

Delft GeoSystems

13 Loads

This section includes background information on the following load types:

- Non-uniform loads [§ 13.1]
- Trapeziform loads [§ 13.2]
- Circular loads [§ 13.3]
- Rectangular loads [§ 13.4]
- Uniform loads [§ 13.5]
- Maintain profile [§ 13.6]
- Submerging [§ 13.7]

A negative load will decrease the vertical effective stresses in a vertical. A negative time can be used to indicate that the initial load will only affect the initial effective stress.

See [chapter 14] for background information on calculating stresses by loading.

13.1 Non-uniform loads

The top of a non-uniform load is defined as a layer boundary, and the bottom is equal to the surface level or – when more non-uniform loads have been defined – the top of an underlying non-uniform load. Besides soil raise, you can also use non-uniform loads to model excavations by defining a negative unit weight.

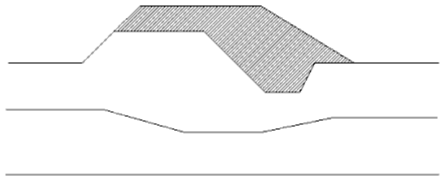


Figure 13-1 – Non-uniform load

Non-uniform loads are subdivided into columns. The weight of these columns depends on the phreatic level in the column.

13.2 Trapeziform loads

The input of trapeziform loads consists of:

P	Unit weight [kN/m³]
XL	Length of the left part of the trapeziform load [m]
XM	Length of the middle part of the trapeziform load [m]
XR	Length of the right part of the trapeziform load [m]
H	Height of the trapeziform load [m]
P(x,y)	Starting point (left side) of the trapeziform load

Trapeziform loads are subdivided into columns.

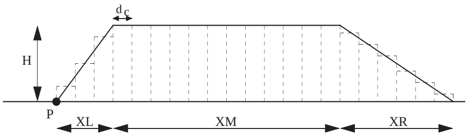


Figure 13-2 – Trapeziform load subdivided into columns

The change of stress at a point on a vertical is calculated for each column using formulas of stress distribution of a load column.
The contact pressure is assumed to be equal to the weight of the column.

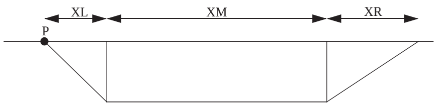


Figure 13-3 – Trapeziform load with a negative height

13.3 Circular loads

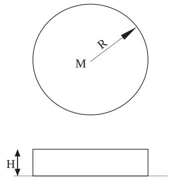


Figure 13-4 – Circular load

The stress due to a circular load is:

(3) $$q(r) = P\left[\alpha + 2(1-\alpha)\left(\frac{r}{R}\right)^2\right]$$

where:

q(r)	Prescribed stress as a function of *r* [kN/m²].
P	Magnitude of the load [kN/m²].
R	Radius of the circular load [m].
r	Distance in R-direction [m].
α	Shape factor to specify the shape of the contact pressure [-].If $\alpha = 1$, the contact pressure is constant (represents flexible footing). If $\alpha = 0$, a parabolic distribution is used with 0 kN/m² in the centre and 2*P* kN/m² at the edge (represents rigid footing).

13.4 Rectangular loads

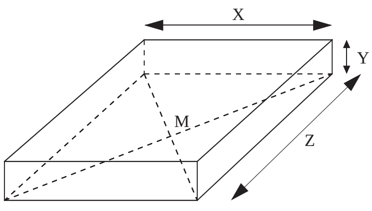


Figure 13-5 – Rectangular load

The stress due to a rectangular load is:

(4) $$q(x,z) = P\left\{\alpha + \frac{12(1-\alpha)}{X+Z}\left[X\left(\frac{z}{Z}\right)^2 + Z\left(\frac{x}{X}\right)^2\right]\right\}$$

where:

$q(x,z)$	Prescribed stress [kN/m²]
$M(x,y,z)$	Co-ordinate of the centre point
P	Magnitude of the load [kN/m²]
X	Width of the load in x-direction [m]
Z	Width of the load in z-direction [m]
α	Shape factor to specify the shape of the contact pressure [-]. If $\alpha = 1$, the contact pressure is constant (represents flexible footing). If $\alpha = 0$, a parabolic distribution is used with 0 kN/m² in the centre and 3P kN/m² at the edge (represents rigid footing).

13.5 Uniform loads

A change of vertical effective stress is calculated at each point on a vertical located below the level of application (y_{app}).

(5) $$d\sigma = q.h$$

where:

q	Unit weight [kN/m³]
h	Height [m]
y_{app}	Y co-ordinate of the level of application [m]

The contact pressure is assumed to be equal to the load of a load column above.

13.6 Maintain profile

MSettle can calculate the settlement caused by a non-uniform load with a fixed position of the top surface. The “Maintain profile” option will iteratively increase the height of all the load columns of which a non-uniform load is composed. The iterative process is stopped when the average difference between the specified and calculated level of the top surface is less than the stop criterion. Swell is neglected, which means that no soil is removed when swell occurs.

13.7 Submerging

Two methods are implemented in MSettle to take submerging into account. The application of each method depends on the consolidation model or the soil model:

- Approximate (Terzaghi or Koppejan): [§ 13.7.1] The approximate method takes submerging of non-uniform loads by deformation into account by an initial load reduction on the basis of final settlements. This method applies either if Terzaghi consolidation model or NEN-Koppejan soil model are selected;

- Accurate [§ 13.7.2] The accurate method takes submerging of non-uniform load and soil layers by deformation into account on the basis of the actual settlement. This method applies for NEN/Bjerrum and Isotache in combination with Darcy.

13.7.1 Submerging – Approximate method (Terzaghi or NEN-Koppejan)

This method applies either if Terzaghi consolidation model or NEN-Koppejan soil model which are selected which corresponds to the four following combinations:

- Isotache soil model with Terzaghi consolidation model;
- NEN-Bjerrum soil model with Terzaghi consolidation model;
- NEN-Koppejan soil model with Terzaghi consolidation model;
- NEN-Koppejan soil model with Darcy consolidation model;

When soil is submerged, the effective unit weight of the (non-uniform) loads decreases:

(6) $$\gamma' = \gamma_{sat} - \gamma_{water}$$

This method determines the submerged weight of non-uniform loads on the basis of final settlements for all load columns. Because of the deformation-dependent weight, these settlements are determined iteratively. The process is stopped when the average settlement increment in a particular iteration is less than the stop criterion.

NOTE: Submerging with the approximate method only works for non-uniform loads. MSettle does not take the submerging of actual soil layers into account.

If a very small stop criterion is defined and a small column width in the *Calculation Options* window [§ 10.1], the calculation can be very time-consuming!

13.7.2 Submerging – Accurate method (Darcy + Isotache/NEN-Bjerrum)

This method applies with two combinations of models:

- NEN/Bjerrum soil model with Darcy consolidation model;
- Isotache soil model with Darcy consolidation model;

When soil is submerged, the effective unit weight of the (non-uniform) loads and the soil layers decreases:

(7) $$\gamma' = \gamma_{sat} - \gamma_{water}$$

MSettle estimates the submerged weight of non-uniform loads and soil layers using an extrapolated settlement based on a linear extrapolation of the two previous time-steps, which writes:

(8) $$s_{extrapolate}\left(t_i\right) = s\left(t_{i-1}\right) + \frac{s\left(t_{i-1}\right) - s\left(t_{i-2}\right)}{t_{i-2} - t_{i-1}}\left(t_{i-1} - t_i\right)$$

A single estimate per time step (without iterations) is usually sufficiently accurate. However, an iteration procedure per time step can be applied in case of large settlement increments per step. Iteration will stop when the average settlement increment in a particular iteration is less than the stop criterion or when the maximum number of iterations is reached.

NOTE: The accurate method takes the submerging of actual soil layers into account oppositely to the approximate method.

If a very small stop criterion is defined and a small column width in the *Calculation Options* window [§ 10.1], the calculation can be very time-consuming!

14 Distribution of stress by loading

Below, the following subjects are discussed:

- General equations for stress distribution [§ 14.1]
- Stress distribution for a strip load [§ 14.2]
- Stress distribution for a circular load [§ 14.3]
- Stress distribution for a rectangular load [§ 14.4]
- Imaginary surface [§ 14.5]

14.1 General equations for stress distribution

14.1.1 Stress increments caused by a surface point force

The basic formula used in MSettle is based on the stress distribution formula for a point load P, where the vertical, horizontal and shear stresses increase in a point at a depth y, and a horizontal distance from the point load of $x = y \times \tan \varphi$ are calculated:

$$\sigma_{yy}(y,\varphi) = \frac{m}{2}\frac{P}{\pi y^2}\cos^{m+2}\varphi$$

(9)
$$\sigma_{xx}(y,\varphi) = \frac{m}{2}\frac{P}{\pi y^2}\sin^{m-1}\varphi\ \cos^{m}\varphi$$

$$\tau_{xy}(y,\varphi) = \frac{m}{2}\frac{P}{\pi y^2}\sin\varphi\cos^{m+1}\varphi$$

where:

σ_{yy} Vertical stress increment [kN/m^2].
σ_{xx} Horizontal stress increment [kN/m^2].
τ_{xy} Shear stress increment [kN/m^2].
P Increment of surface load [kN].
y Depth [m].
φ Angle with the vertical [°].
m Concentration index [-].Boussinesq assumes a concentration index of 3 and Buisman of 4.

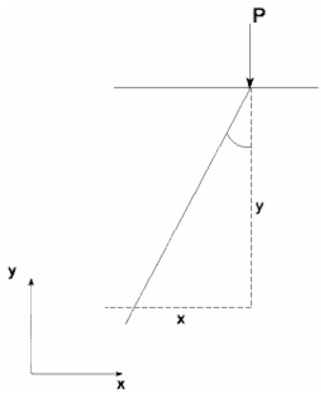


Figure 14-1 – Stress distribution under a point load

NOTE: MSettle automatically calculates the stress distribution according to Buisman. Boussinesq can however be selected in the *Calculation Options* window [§ 10.1], but only for non-uniform and trapeziform loads.

14.1.2 Stress increments caused by a line load

The stress increments due to a line load $Q = P \times h$ can be found by integration of the point load P along the height h of the line load in equation (9):

$$\sigma_{yy} = \frac{2}{\pi}\frac{Q}{z}\cos^4\varphi$$

(10) $$\sigma_{xx} = \frac{2}{\pi}\frac{Q}{z}\cos^2\varphi\sin^2\varphi \qquad \text{for Boussinesq}$$

$$\tau_{xy} = \frac{2}{\pi}\frac{Q}{z}\cos^3\varphi\sin\varphi$$

14.2 Stress distribution for a strip load

The stress increments in a point (x, y, z) due to a strip load can be found by integration of the line load along the width 2 *dx* of the strip load in equation (10):

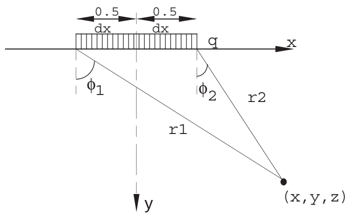


Figure 14-2 – Stress distribution under a load column

(11) for Boussinesq

$$\sigma_{yy} = \frac{q}{\pi}\left[\left(\phi_1 - \phi_2\right) + \sin\phi_1 \cos\phi_1 - \sin\phi_2 \cos\phi_2\right]$$

$$\sigma_{xx} = \frac{q}{\pi}\left[\left(\varphi_1 - \varphi_2\right) - \sin\phi_1 \cos\phi_1 + \sin\phi_2 \cos\phi_2\right]$$

$$\tau_{xy} = \frac{q}{\pi}\left[\sin^2\phi_1 - \sin^2\phi_2\right]$$

(12) for Buisman

$$\sigma_{yy} = \frac{3}{4}q\left[\sin\phi_1 - \sin\phi_2 - \frac{1}{3}\left(\sin^3\phi_1 - \sin^3\phi_2\right)\right]$$

$$\sigma_{xx} = \frac{1}{4}q\left(\sin^3\phi_1 - \sin^3\phi_2\right)$$

$$\tau_{xy} = \frac{1}{4}q\left(\cos^3\phi_2 - \cos^3\phi_1\right)$$

NOTE: Trapeziform and non-uniform loads are subdivided into load columns. The width of these columns and the choice of the stress distribution type (Buisman or Boussinesq) can both be defined in the *Calculation Options* window [§ 10.1].

14.3 Stress distribution for a circular load

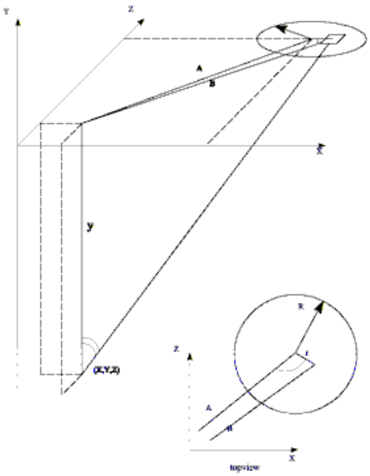


Figure 14-3 – Stress distribution under a circular load

For this figure, the following equation applies:

(13) $$\cos^2\varphi = \frac{y^2}{y^2 + A^2 + r^2 - 2 r A \cos\alpha}$$

The vertical stress increment in a point (x, y, z) due to a circular load can be found by integration in tangential and radial directions of equation (9) (Buisman) :

(14) $$\sigma_{yy}(x, y, z) = \int_0^R \int_0^{2\pi} \frac{\gamma\, y^2\, r\, dr\, d\alpha}{\pi\left(y^2 + A^2 + r^2 - 2 r A \cos\alpha\right)}$$

14.4 Stress distribution for a rectangular load

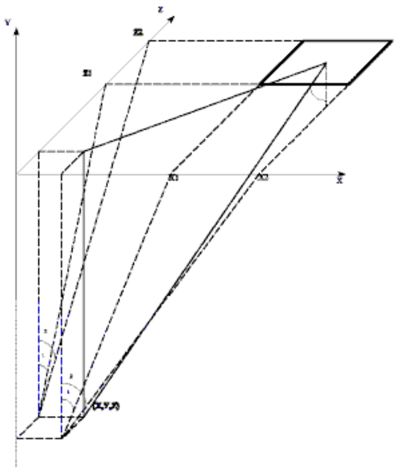


Figure 14-4 – Stress distribution under a rectangular load

For this figure, the following formula applies:

(15) $$\cos^2\varphi = \frac{y^2}{y^2 + (y\tan\alpha)^2 + (y\tan\beta)^2}$$

The vertical stress increment in a point (x, y, z) due to a rectangular load can be found by integration in x and z directions of equation (9) (Buisman) :

(16) $$\sigma_{yy}(x,y,z) = \int_{z_1}^{z_2}\int_{x_1}^{x_2} \frac{\gamma\, y^2\, dx\, dz}{\pi\left[y^2 + (y\tan\alpha)^2 + (y\tan\beta)^2\right]^2}$$

where:

$x_1 = y \tan\alpha_1$

$x_2 = y \tan\alpha_2$

$z_1 = y \tan\beta_1$

$z_2 = y \tan\beta_2$

14.5 Imaginary surface

MSettle will determine the stress distribution in the layers below an imaginary surface caused by the weight of the layers above the surface. This option must be used in the case of an initially non-horizontal surface – for example, for an embankment. Boundary 2 in the following figure is an example of such an imaginary surface.

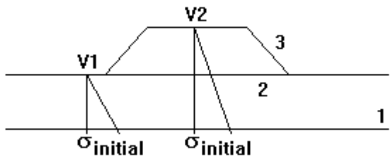


Figure 14-5 – Imaginary surface

The entire soil load above the imaginary surface will only affect the initial stresses. The effect of stress distribution is taken into account. Incorporating stress distribution will result in an increase in the initial stress in vertical v1 near the embankment, and a decrease of initial stress in the vertical v2 below the embankment.

15 Pore pressure

The combination of a static hydraulic pore pressure field with transient excess pore pressures can be modelled with either the approximate Terzaghi model or with the accurate Darcy model. The Terzaghi model uses the theoretical solution for one-dimensional consolidation to modify directly the drained settlement solution. The Darcy model solves the transient development of excess pressures and settlements, using Darcy's general storage equation. Both models use equal input [§ 15.1]:

- The initial hydraulic head distribution from piezometric level lines at each layer boundary.
- The position of the phreatic line.
- The soil weight.
- The consolidation coefficient C_v per layer.

The calculation process and the output results are different:

- Terzaghi [§ 15.2] allows for quick and direct predictions of primary and secondary settlements, including the approximate influence of consolidation.
- Darcy [§ 15.3] enables a more accurate prediction of the transient pore pressure development, by stepwise solution of excess pore pressures. The Darcy model also allows for stepwise determination of the effective stress by submerging of layers and loads.

The influence of vertical drains on pore pressure development can be analyzed with both models [§ 15.4].

15.1 Hydraulic head distribution

15.1.1 Piezometric level lines

A piezometric level line (PL-line) represents the initial and transient hydraulic water head (excluded the excess component). A PL-line can be defined for the top and bottom of each soil layer [§ 9.3.10, § 9.6.2].

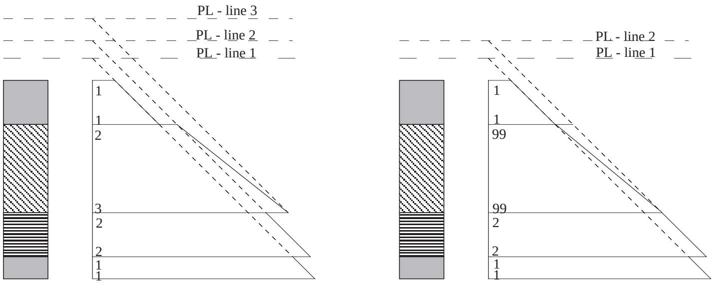


Figure 15-1 – Pore pressure as a result of piezometric level lines

MSettle calculates the hydraulic pore pressure along a vertical in the following way:

- The pore pressure inside a layer is calculated by linear interpolation between the pore pressures at top and bottom.
- The pore pressure at the top or bottom is equal to the vertical distance between this point and the position of the PL-line that belongs to this layer, multiplied by the unit weight of water.
- If PL-line number 99 is specified for the top and/or bottom of any soil layer, MSettle will use at that boundary the PL-line of the nearest soil layer above or below, which has a thickness larger than zero and a PL-line number not equal to 99. If the interpolation point is located above the phreatic line, the pore pressure is assumed to be zero or a capillary pressure, depending on the sign of the PL-line number.

The following options are available, therefore, for giving PL-line numbers:

Positive integer:	Capillary pore pressures are not used – that is, if negative pore pressures are calculated for points above the phreatic line they become zero
Zero:	All points within the layer obtain a pore pressure 0 kN/m²
99:	The pore pressure depends on the first layer above and/or below the point with a PL-line number unequal to 99.

15.1.2 Phreatic line

The phreatic line (or groundwater level) is used to mark the border between dry and wet soil. The phreatic line is treated as if it was a PL-line, and can also be used as such. The PL-line acting as the phreatic line is determined while the geometry is being defined. If no phreatic line is entered, then all the soil is assumed to be dry.

15.1.3 Stress by soil weight

The total stress at depth y due to soil weight is:

$$\sigma_{soil}(y,t) = \begin{cases} \gamma_{unsat}(y_0 - y) & \text{if } y > y_{water} \\ \gamma_{unsat}(y_0 - y_{water}) + \gamma_{sat}(y_{water} - y) & \text{if } y \leq y_{water} \end{cases} \qquad (17)$$

where:

γ_{unsat} Unit weight of soil above phreatic level [kN/m^3].
γ_{sat} Unit weight of soil below phreatic level [kN/m^3].
y Vertical initial co-ordinate [m].
y_0 Initial surface level [m].
y_{water} Phreatic level [m].

15.2 Terzaghi

Terzaghi's one-dimensional consolidation theory is characterized by the consolidation coefficient. Terzaghi's model allows quick determination of final settlements, with approximate effect of consolidation.

Related to MSettle's implementation of the Terzaghi model, the following subjects are discussed hereafter:

- Terzaghi's general consolidation theory [§ 15.2.1]
- Consolidation of multi-layered systems [§ 15.2.2]
- Drainage conditions [§ 15.2.3]
- Effective stress and pore pressure [§ 15.2.4]

15.2.1 Terzaghi – General consolidation theory

Terzaghi's theory on one-dimensional vertical consolidation of a homogeneous elastic layer yields the following expression for the degree of consolidation U:

$$U(t) = 1 - \frac{8}{\pi^2} \sum_{i=1}^{\infty} \frac{1}{(2i-1)^2} \exp\left[-(2i-1)^2 \frac{\pi^2}{4} \frac{c_v t}{d^2} \right] \qquad (18)$$

where:

c_v Consolidation coefficient [m²/sec]
d Drainage length [m].
t Time [sec]
U Degree of consolidation [-]

In case of vertical drains, the expression is more complicated.
MSettle combines the degree of consolidation with the predicted layer deformation under fully drained conditions:

(19) $$\Delta h = \begin{cases} U(t)\ \Delta h_{prim}(\sigma') + \Delta h_{\sec}(\sigma', t) & \text{for NEN - Koppejan} \\ U(t)\ \Delta h_{drained}(\sigma', t) & \text{for Isotache and NEN - Bjerrum} \end{cases}$$

where:
σ' Vertical effective stress [kN/m²]
Δh_{prim} Primary contribution to layer deformation according to Koppejan [m]
Δh_{sec} Secondary contribution to time dependent layer deformation according to Koppejan [m]
$\Delta h_{drained}$ Theoretical time dependent layer deformation under fully drained conditions according to Isotache/NEN-Bjerrum [m]
Δh Total layer deformation with approximate influence of consolidation [m]

15.2.2 Terzaghi – Consolidation of multi-layered systems

MSettle considers clusters of consolidating layers, between drained layers or drained dispersion boundaries. MSettle models these multi-layered clusters by introducing a fictitious homogeneous layer with equivalent consolidation coefficient. MSettle scales the vertical co-ordinate z in layer i with the vertical consolidation coefficient $c_{v.i}$. The following cases show the expressions used, including the contributions of optional vertical drains [§ 15.4].

(20) $$\frac{d^2\overline{\varphi}}{d\zeta^2} = \frac{d\overline{\varphi}}{dt} + \frac{c_v}{\lambda^2}\left(\overline{\varphi} - \varphi_{drain}\right) \quad \text{with} \quad \zeta = \frac{z_i}{\sqrt{c_{v.i}}}$$

(21) $$\frac{\sum_{i=1}^{n} h_i}{\sqrt{c_{v.eq}}} = \sum_{i=1}^{n} \frac{h_i}{\sqrt{c_{v.i}}} \quad \text{and} \quad \frac{\sum_{i=1}^{n} h_i}{\sqrt{c_{h.eq}}} = \sum_{i=1}^{n} \frac{h_i}{\sqrt{c_{h.i}}}$$

where n is the number of layers and h_i the thickness of layer i.

15.2.3 Terzaghi – Drainage conditions

The theoretical Terzaghi solution is based on drained conditions at just one side. MSettle will halve the drainage length in case of drainage at both sides. Drainage at the boundary of a cluster of consolidation layers can be specified via the dispersion condition at the top or bottom of the geometry (see *Calculation Options* window [§ 10.1.1]), or via a drained property of certain soil layers (see *Materials* window [§ 9.2.2]).

MSettle sets the degree of consolidation in drained layers directly to 100%.

15.2.4 Terzaghi – Effective stress and pore pressure

Terzaghi determines the effective stress at time t and initial vertical position y, disregarding excess pore pressures, using:

$$\sigma'(y,t) = \sigma_{soil}(y,t) + \Delta\sigma_{load}(y,t) + p_{hydr}(y,t) \tag{22}$$

$$p_{hydr}(y,t) = \sigma_{water}(y,t) - \max\left(\varphi_{hydr}(y,t) - y, 0\right)\gamma_w \tag{23}$$

where:

σ_{soil}	[kPa]	Stress due to soil weight [§ 15.1.3].
$\Delta\sigma_{load}$	[kPa]	Incremental stress due to loads [chapter 13].
p_{hydr}	[kPa]	Hydraulic component of pore pressure.
σ_{water}	[kPa]	Stress due to a water level above the soil surface: $\sigma_{water}(y,t) = \max\left[\left(y_{water}(t) - y_{surface}(t)\right)\gamma_w ; 0\right]$
y	[m]	Initial vertical co-ordinate.
φ_{hydr}	[m]	The user-defined hydraulic head at time t. It can either be defined in the *Pl-lines per Layer* window [§ 9.3.13] for the initial state or in the *Water Loads* window [§ 9.6.2] for a specific time. **NOTE:** The influence of excess pore pressures during consolidation is therefore neglected.

Only for postprocessing purposes in graphs and the report, Terzaghi will use the final position y_{final} for the calculation of the values of final pore pressure and effective stress along the depth.

NOTE: The Terzaghi's model doesn't calculate a pore pressure distribution, but applies directly a degree of consolidation on settlements. Output of pore pressure distribution is only available at the initial and final state, without influence of excess pore pressure. In Darcy's model, pore pressures are calculated at each time step by means of the storage equation given in [§ 15.3.1].

15.3 Darcy

Darcy's model can be applied to find the pore pressure development in clusters of compressible (creeping) layers. Application of Darcy enables accurate 1D solution of the full hydraulic head, and allows combination with vertical drains modelling. The implemented Darcy model is designed for saturated soil only.

Related to MSettle's implementation of the Darcy model, the following subjects are discussed hereafter:

- Darcy's consolidation theory [§ 15.3.1]
- Drainage conditions [§ 15.3.2]
- Effective stress and pore pressure [§ 15.3.3]
- Numerical solution [§ 15.3.4]

15.3.1 Darcy – Consolidation theory

Darcy's consolidation model is based on the storage equation (24).

$$(24) \qquad k_y \frac{d^2\varphi}{dy^2} + \frac{d\varepsilon}{dt} - \gamma_w \frac{n}{K_w} \frac{d\varphi}{dt} = 0$$

where

φ	Hydraulic water head [m]
k_y	Darcy permeability [m/day]
K_w	Bulk modulus of water [kPa]
γ_w	Unit weight of water [kN/m^3]
n	Porosity of the soil [-]

The implemented equation is based on excess heads, and assumes full saturation below the phreatic line, even when the calculated pore pressure becomes negative. Saturation dependent phreatic storage and permeability changes are therefore neglected.

The real permeability of soil is a function of void ratio. MSettle offers therefore a strain dependent model according to equation (25).

$$(25) \qquad k = k_0 \, 10^{-\frac{1+e_0}{C_k}\varepsilon}$$

where:

k_0	Initial permeability at undeformed state [m/sec].
$\frac{C_k}{1+e_0}$	Permeability strain modulus [-].
ε	Strain [-]
C_k	Permeability strain factor [-].
e_0	Initial void ratio [-].

This type of strain dependency follows also from the assumption of a constant value for the consolidation coefficient, in combination with MSettle's stress dependent compressibility models.

MSettle can derive the values for the permeability strain modulus and the initial permeability at different locations from the input of a consolidation coefficient, in combination with the compression parameters (primary consolidation parameters), the preconsolidation stress and the overconsolidation ratio, using equation (26):

(26) $$k_0 = \gamma_w \frac{CR}{\sigma_p} c_v \exp\left(\frac{RR}{CR} \ln OCR\right) \quad \text{with} \quad CR = \frac{C_k}{1+e_0}$$

Equation (26) is expressed in NEN-Bjerrum parameters. It can be changed to Isotache or Koppejan parameters by using:

(27) $$a \cong \frac{RR}{\ln 10} \cong \frac{1}{C_p} \quad \text{and} \quad b \cong \frac{CR}{\ln 10} \cong \frac{1}{C'_p}$$

15.3.2 Darcy – Drainage conditions

Darcy assumes drainage at the surface and the bottom of the geometry. Additionally, intermediate drained layers can be defined between clusters of consolidating layers.

15.3.3 Darcy – Effective stress and pore pressure

Darcy determines the effective stress at time *t* and current vertical position y_t, including the influence of the excess head, using:

(28) $$\sigma'(y_t,t) = \sigma_{soil}(y,t) + \Delta\sigma_{load}(y,t) + p(y_t,t)$$

(29) $$p(y_t,t) = \sigma_{water}(y,t) + \gamma_w\left[\varphi_{hydr}(y,t) + \varphi_{excess}(y,t) - y_t\right]$$

where:

γ_w	Unit weight of water [kN/m³].
γ_{unsat}	Unit weight of soil above phreatic level [kN/m³].
γ_{sat}	Unit weight of soil below phreatic level [kN/m³].

y Initial vertical initial co-ordinate [m].

y_t Current vertical initial co-ordinate [m].

σ_{water} [kPa] Stress due to a water level above the soil surface:

$$\sigma_{water}\left(y,t\right)=\max\left[\left(y_{water}\left(t\right)-y_{surface}\left(t\right)\right)\gamma_w;0\right]$$

φ_{hydr} The user-defined hydraulic head, defined in the *PL-lines per Layer* window [§ 9.3.13] for the initial state.

φ_{excess} The excess head at time t

15.3.4 Darcy – Numerical solution

The transient pore pressure distribution is solved numerically with an automatic time stepping scheme, using an efficiently integrated spatial Fourier interpolation along sections of the verticals. Within each time step, the settlements at the section interfaces are solved iteratively. MSettle determines the time step sizes such that a stable solution is achieved under all practical circumstances.

15.4 Vertical drains

Three types of vertical drains can be modelled in MSettle:

- Strip drains
- Column drains
- Sand wall

NOTE: The initial and final head distributions can be different when using vertical drains. The reason is that the vertical drains contribution [§ 15.4] is not included during the initial head determination.

15.4.1 Modified storage equation

In case of vertical drains, MSettle solves the average head between the drains along each vertical. MSettle uses the modified storage equation (30) for Darcy, and the modified consolidation equation (31) for Terzaghi. The Terzaghi solution can be considered as an extension of the classic solutions by Barron [Lit 4]and Carillo [Lit 5].

(30) $$\frac{d\bar{\varepsilon}}{dt}+k_y\frac{d^2\bar{\varphi}}{dy^2}+k_y\frac{\bar{\varphi}-\varphi_{drain}}{\lambda^2}-\gamma_w\frac{n}{K_w}\frac{d\bar{\varphi}}{dt}=0$$ for Darcy consolidation model

(31) $$\frac{d^2\bar{\varphi}}{dy^2}=\frac{1}{C_V}\frac{d\bar{\varphi}}{dt}+\frac{\bar{\varphi}-\varphi_{drain}}{\lambda^2}$$ for Terzaghi consolidation model

where:

$\overline{\varphi}$	The average value of the head between the drains [m].
φ_{drain}	The head in the drain [m]. See [§ 15.4.2] for line-shaped drains (strip or column) and [§ 15.4.3] for plane-shaped drains (granular wall).
λ	The so-called leakage length [m]. See [§ 15.4.2] for line-shaped drains (strip or column) and [§ 15.4.3] for plane-shaped drains (granular wall).
γ_w	The unit weight of water [kN/m^3].
K_w	The bulk modulus of water [kPa].
n	The porosity of the soil layer [-].

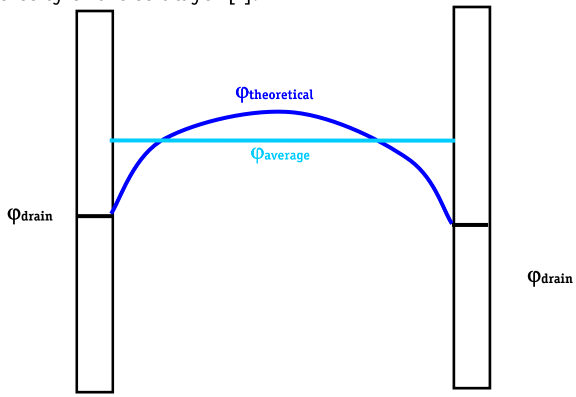


Figure 15-2 – Theoretical and average pressure distribution between two drains

15.4.2 Line-shaped vertical drains (strip/column drains)

In case of line-shaped drainage strips (i.e. Strip or Column), water will flow radially out on top of the drains. Sometimes a combination with an enforced underpressure on top is applied, via a drained layer with impermeable cover.

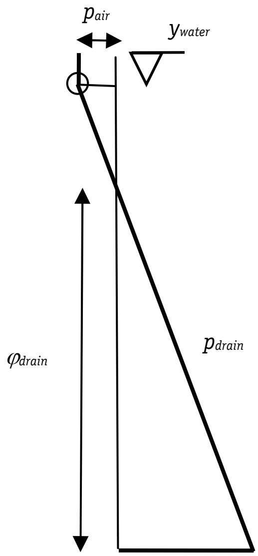


Figure 15-3 – Pressure distribution along a line-shaped drain (radial flow)

MSettle assumes that φ_{drain} is equal to a certain water level in the drain, with an optional reduction by underpressure.

(32) $$\varphi_{drain} = \max(Y; Y_{water}) - \frac{P_{air}}{\gamma_w}$$

where:

Y_{water} The water level in the drain [m]. If underpressure is applied, this water level is equal to the position where the underpressure is applied. Otherwise, the water level simply equals the phreatic level.

P_{air} The enforced underpressure [kPa].

The leakage length for radial flow is equal to:

(33) $$\lambda^2 = \frac{D_{eq}^{\ 2}}{8}\frac{k_y}{k_x}\left[\frac{D_{eq}^{\ 2}}{D_{eq}^{\ 2} - d_{eq}^{\ 2}}\ln\left(\frac{D_{eq}}{d_{eq}}\right) - \frac{1}{2} - \frac{D_{eq}^{\ 2} - d_{eq}^{\ 2}}{4D_{eq}^{\ 2}}\right]$$

where:

k_x/k_y The ratio horizontal/vertical permeability [-].

D_{eq} The equivalent distance between the drains depending on the position of the calculated vertical and the type of grid (triangular of rectangular):

$$D_{eq} = f_{grid} \times \begin{cases} D & \text{inside the drainage range} \\ \max\left(2\ D;\ 4\ \left|x - X_{\text{limit}}\right|\right) & \text{outside the drainage range} \end{cases}$$

D The actual distance between the drains [m].

f_{grid} Factor depending on the grid type [-]: 1.05 for a triangular grid and 1.13 for a rectangular grid.

X_{limit} The (horizontal) X co-ordinate of the limit of the drained area [m].

d_{eq} The equivalent diameter of the drain cross-section [m].
For strip drain, this value is the circumferential distance of the rectangular cross section (width × thickness) divided by π.
For column drains, this value equals the actual diameter d of the drain cross-section

15.4.3 Plane-shaped vertical drains (plane flow)

In case of plane-shaped drains (trenches filled with granular material), water will flow out via drainage tubes, located downwards in the drain. Sometimes an additional air underpressure is enforced at the top of the drains.

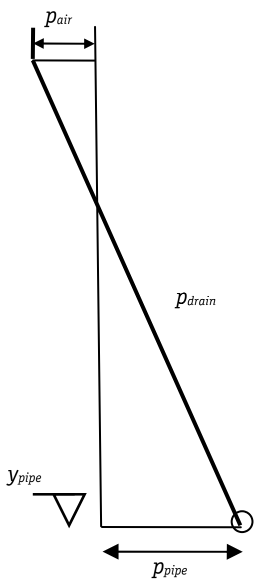


Figure 15-4 – Pressure distribution along a plane-shaped drain (plane flow)

MSettle assumes that the negative pore pressures in the drain above the water level are equal to the air underpressure while the head under the water level is equal the water level minus the air underpressure.

(34) $$\varphi_{drain} = \max\left(Y; Y_{water}\right) - \frac{P_{air}}{\gamma_w} \quad \text{with} \quad Y_{water} = Y_{pipe} + \frac{P_{pipe}}{\gamma_w} + \frac{P_{air}}{\gamma_w}$$

where:

Y_{water} The water level in the drain [m].
Y_{pipe} The vertical location of the drainage tube [m].
P_{pipe} The pressure in the drainage tube [kPa].
Y The vertical location of a point on the plane-shaped drain [m].
P_{air} The enforced air underpressure at the top of the vertical drain [kPa].

The leakage length for sand wall (plane flow) is equal to:

(35) $$\lambda^2 = \frac{1}{12}\frac{k_y}{k_x}\left(D_{eq} - w\right)^2$$

where:

k_x/k_y The ratio horizontal/vertical permeability [-].
D_{eq} The equivalent distance between the drains depending on the position of the calculated vertical and the type of grid (triangular of rectangular):

$$D_{eq} = \begin{cases} D & \text{inside the drainage range} \\ \max\left(2\,D;\, 4\left|x - X_{\text{limit}}\right|\right) & \text{outside the drainage range} \end{cases}$$

D The actual distance between the drains [m].
X_{limit} The (horizontal) X co-ordinate of the limit of the drained area [m].
w The width of the granular wall [m].

16 Soil and strain models

MSettle calculates the transient settlement of all layers along user-defined verticals, using one of the following soil models:

- NEN-Bjerrum [§ 16.1]. The NEN-Bjerrum model is suited for cases with un- and reloading, by using a rate-type visco-plastic isotache formulation (all plastic compression results from creep). The NEN-Bjerrum model is based on linear strain and supports the common linear strain parameters C_r, C_c and C_α.
- Isotache [§ 16.2]. The Isotache a/b/c model is suited for cases with large strains and/or un-/reloading. The model uses a rate-type visco-plastic formulation (all plastic compression results from creep) and is based intrinsically on natural strain. The model uses the objective natural strain parameters *a, b, c*.
- NEN-Koppejan [§ 16.3]. The classic Dutch soil model for many years. The model makes a distinction between primary and secondary settlement. Major differences with NEN-Bjerrum are the less realistic stress-dependency of the secondary creep and the poor description of un-/reloading. Usage of Koppejan for cases with load removal is therefore not recommended.

16.1 NEN-Bjerrum

The NEN-Bjerrum model is based on the same theory as the a/b/c/ isotache model. The only difference is that the NEN-Bjerrum model supports the common linear strain parameters C_r, C_c and C_α instead of the natural strain parameters a/b/c. The shared isotache formulation implies that all inelastic compression results from visco-plastic creep. The NEN-Bjerrum model therefore assumes that creep rate will reduce with increasing overconsolidation and that overconsolidation will grow by unloading and by ageing. Bjerrum's name is attached to this model, because he was the first to notice that creep rate depends on both overconsolidation ratio and age. Den Haan [Lit 7] has developed the full mathematical formulation.

Parameters for the NEN-Bjerrum model are easily determined from common oedometer tests [§ 17.3], especially when you use the M-Series program MCompress.

NOTE: Practice proves that the methods for determination of NEN-Bjerrum parameters can differ from laboratory to laboratory. Therefore please read the description of the expected parameter determination method [§ 17], in order to assure that it is compliant with the actual parameter determination is compliant with the actual determination method.

Hereafter is a global description of the following aspects of MSettle's NEN-Bjerrum implementation.

- Idealized behaviour [§ 16.1.1]
- Mathematical formulation [§ 16.1.2]

16.1.1 NEN-Bjerrum – Idealized behaviour

Figure 16-1 and Figure 16-2 show that the behaviour of drained soil according to the NEN-Bjerrum model can be schematized to an idealized primary and secondary contribution, with different stiffness below and above preconsolidation. This schematized behaviour is also known from popular textbooks, from standards like NEN 6744 [Lit 8] and from recommendations like ISSMGE-ETC5 [Lit 10].

NOTE: The true isotache behavior differs from the idealized behavior, especially in combination with consolidation. The final settlement after consolidation will however be the same.

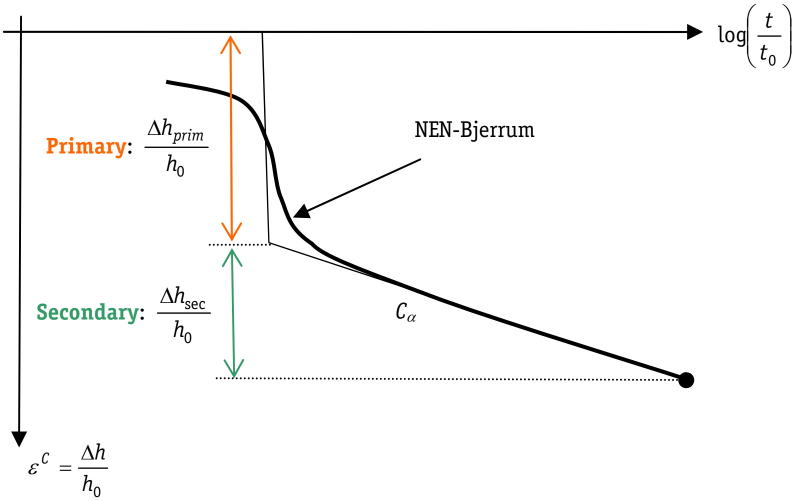


Figure 16-1 – NEN-Bjerrum: Idealized primary and secondary settlement during time (drained conditions)

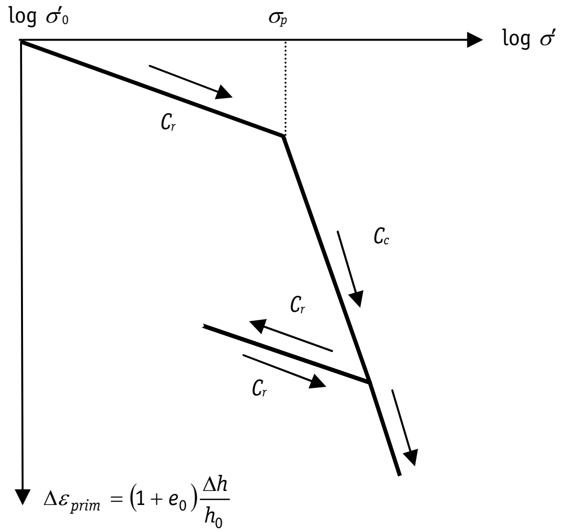


Figure 16-2 – NEN-Bjerrum: Idealized primary settlement during loading (drained conditions)

For the idealized drained NEN-Bjerrum behaviour, three contributions exist.

- If the vertical effective stress after loading is smaller than the preconsolidation pressure σ_p, the primary settlement contribution according to the idealized behaviour can be calculated from:

(36) $$\frac{\Delta h_{\text{prim}}}{h_0} = RR \log\left(\frac{\sigma'}{\sigma_0}\right), \qquad \sigma_0 < \sigma' < \sigma_p$$

where:

$$RR = \frac{C_r}{1+e_0}$$

C_r	Reloading/swelling index below preconsolidation pressure [-]
Δh_{prim}	Primary settlement contribution of a layer [m]
h_0	Initial layer thickness [m]
e_0	Initial void ratio [-]

- If the vertical effective stress after loading is larger than the preconsolidation pressure σ_p, the primary settlement according to the idealized behaviour can be calculated from:

(37) $$\frac{\Delta h_{prim}}{h_0} = RR \log\left(\frac{\sigma_p}{\sigma_0}\right) + CR \log\left(\frac{\sigma'}{\sigma_p}\right), \quad \sigma_p < \sigma'$$

where:

$$CR = \frac{C_c}{1+e_0}$$

C_c Compression index above preconsolidation pressure [-]

- If the vertical effective stress after loading is larger than the preconsolidation pressure σ_p, the secondary settlement according to the idealized behaviour can be calculated from:

(38) $$\frac{\Delta h_{sec}}{h_0} = C_\alpha \log\left(\frac{t}{\tau_0}\right), \qquad \sigma_p < \sigma'$$

where:

C_α Coefficient of secondary compression above preconsolidation pressure [-]

16.1.2 NEN-Bjerrum – Mathematical Formulation

A full description of the mathematical formulation of the NEN-Bjerrum model can directly be derived from the a/b/c Isotache description [§ 16.2], by application of the following small strain limits:

If $\varepsilon^H \rightarrow \varepsilon^C$ (small strains) then:

(39) $$\begin{aligned} a &\rightarrow \frac{RR}{\ln(10)}, \quad RR = \frac{C_r}{1+e_0} \\ b &\rightarrow \frac{CR}{\ln(10)}, \quad CR = \frac{C_c}{1+e_0} \\ c &\rightarrow \frac{C_\alpha}{\ln(10)} \end{aligned}$$

The basic ingredients of the formulation are summarized below.

- **Strain decomposition.** The total strain consists of a direct elastic contribution and a transient viscous contribution.

(40) $$\varepsilon^C = \varepsilon_s^C + \varepsilon_d^C$$

- **Elastic (direct) contribution.** The elastic contribution is determined by parameter *RR*.

(41) $$\varepsilon_d^C = RR \log\frac{\sigma'}{\sigma'_0}$$

- **Visco-plastic (creep) contribution.** The viscous creep rate $\dot{\varepsilon}_s^C$ depends on the stress rate, the already reached creep strain at a certain time and the current overconsolidation ratio σ_p / σ'.

(42) $$\varepsilon_s^C = C_\alpha \log\left[1 + \int_0^t \left(\frac{\sigma'}{\sigma_p}\right)^{\frac{CR-RR}{C_\alpha}} \frac{d\tau}{\tau_0}\right]$$

The graphical illustration in Figure 16-3 shows that creep will also grow below preconsolidation stress (un-/reloading), but that the rate will rapidly decrease at larger values of overconsolidation (stress more below preconsolidation stress).

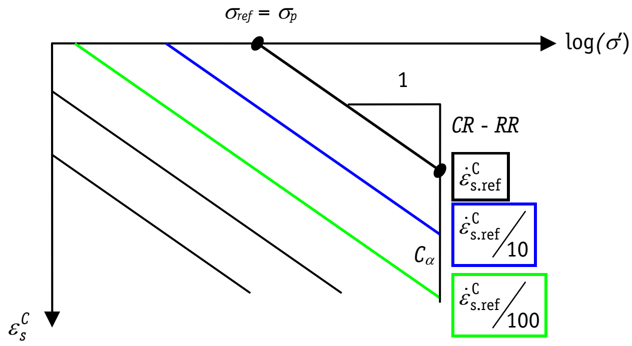


Figure 16-3 – NEN-Bjerrum: Creep rate depending on overconsolidation

In case of several loading and un/reloading steps, the drained solution of equation (42) becomes:

(43) $$\varepsilon^C(t) = RR \log\left(\frac{\sigma_p}{\sigma'_0}\right) + CR \log\left(\frac{\sigma'_n}{\sigma_p}\right) + C_\alpha \log\left(\frac{t - t_n + \theta_n}{\tau_0}\right)$$

where the equivalent age θ_n is calculated as follows:

$$\theta_n = \left(\frac{\sigma'_{n-1}}{\sigma'_n}\right)^{\frac{CR-RR}{C_\alpha}} \cdot \left(\theta_{n-1} + t_n - t_{n-1}\right) \quad \text{with} \quad \theta_0 = \tau_0 \cdot \left(\frac{\sigma_p}{\sigma'_1}\right)^{\frac{CR-RR}{C_\alpha}}$$

$$\sigma_p = \begin{cases} \sigma'_0 + POP & \text{for POP compression} \\ \sigma'_0 \cdot OCR & \text{for OCR compression} \\ \sigma'_0 \cdot \left(t_{age}/t_0\right)^{\frac{c}{b-a}} & \text{for equivalent age compression} \end{cases}$$

t_n Begin time of step n [days].

n Number of the load steps [-].

16.2 Isotache a/b/c

MSettle's a/b/c Isotache model is based on natural strain, and uses a rate type formulation. Natural strain is referred to the deformed state. A rate formulation means that all inelastic compression is assumed to result from visco-plastic creep. The a/b/c model might be advantageous to the NEN-Bjerrum model if large strains are involved.

Hereafter you can find a global description of the following aspects of MSettle's Isotache a/b/c implementation.

- Natural strain [§ 16.2.1]
- Creep [§ 16.2.2]

See Den Haan [Lit 7] for more information on the Isotache model. For a basic description of the a/b/c parameter determination see [§ 17.4]. These natural strain parameters can also be derived from linear strain parameters at given stress levels [§ 17.7].

16.2.1 Isotache – Natural strain

The Isotache model intrinsically uses natural strain, whereas the NEN-Bjerrum model uses linear strain by default.
Natural (or logarithmic) strain is advantageous when compressions are large. When strains are small, the two strain measures become equivalent. The Isotache model obtains the natural strain by defining the increment of strain relative to the present, actual thickness, and by integrating the increments:

(44) $$d\varepsilon^H = -\frac{dh}{h}, \qquad \varepsilon^H = -\int_{h_0}^{h} \frac{dh}{h} = -\ln\left(\frac{h}{h_0}\right)$$

where:

h Actual layer thickness [m]

h_0 Initial layer thickness [m]

The linear strain, given by:

(45) $$d\varepsilon^C = -\frac{dh}{h}, \qquad \varepsilon^C = 1 - \frac{h}{h_0}$$

is related to natural strain by:

(46) $$\varepsilon^H = -\ln\left(1 - \varepsilon^C\right)$$

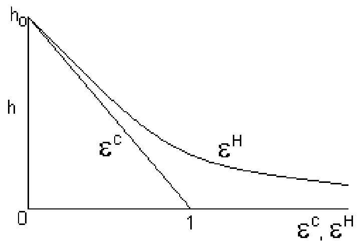


Figure 16-4 – Height related to linear and natural strain

The superscripts C and H refer to Cauchy and Hencky, respectively, to whom the respective measures of strain are ascribed. The figure above relates ε^C and ε^H to compression. ε^C can numerically exceed 100%, and compressions larger than the initial layer thickness are indeed found from conventional models – for example, by using a small initial stress and a large stress increase. This is impossible using natural strain. Natural strain also allows a better fit for oedometer tests, when compression is large (see the figure below).

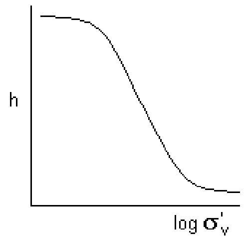


Figure 16-5 – Compressed height compression as a function of effective stress

16.2.2 Isotache – Creep

The Isotache model assumes that the creep rate will reduce with increasing overconsolidation and that overconsolidation can grow by unloading and by ageing. This concept is encapsulated by means of creep Isotaches.

Creep Isotaches are lines of equal rate (speed, velocity) of secular (visco-plastic) strain ε_S^H in a plot of (natural) strain versus (natural) logarithm of vertical effective stress. These are displayed in the figure below.

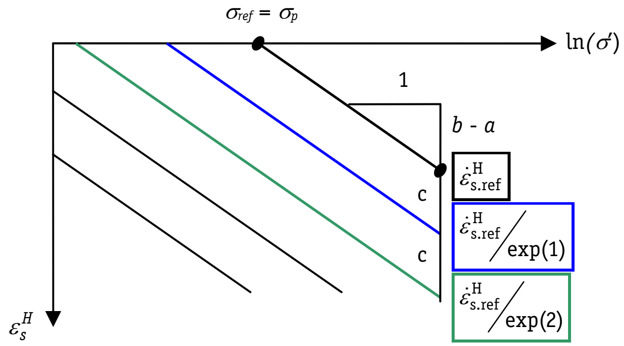


Figure 16-6 – Creep Isotache pattern

The Isotaches are all parallel with slope *b-a*. The Isotache *a* parameter determines the direct (elastic) strain component ε_d^H. The *b* and *c* parameters determine the secular (visco-plastic) creep component ε_s^H.

(47) $$b - a = \frac{d\varepsilon_s^H}{d \ln \sigma'}$$

(48) $$c = -\frac{d\varepsilon_s^H}{d \ln\left(\dot{\varepsilon}_s^H\right)}$$

(49) $$a = \frac{d\varepsilon_d^H}{d \ln \sigma'}$$

(50) $$\varepsilon^H = \varepsilon_s^H + \varepsilon_d^H$$

The reference Isotache starts at preconsolidation stress $\sigma_{ref} = \sigma_p$ and is characterized by a reference creep strain rate $\dot{\varepsilon}^H_{s.ref}$.

The secular creep rate is given by:

(51) $$\dot{\varepsilon}_s^H = \dot{\varepsilon}_{s.ref}^H \exp\left(\frac{(b-a)}{c} \ln\left(\frac{\sigma'}{\sigma_p}\right) - \varepsilon_s^H\right)$$

This equation assumes in fact that the secular creep rate is related to a so-called intrinsic time τ, which is related to the common time *t* by an equivalent age t_{age}.

(52) $\dot{\varepsilon}_s^H = \frac{c}{\tau}, \qquad \tau = t + t_{age}$

The initial equivalent age represents the theoretical age of the soil since the end of virgin loading, if the current overconsolidation ratio would have been caused by ageing only.

(53) $t_{age} = \tau_0 \; OCR^{\left(\frac{b-a}{c}\right)}$

The total rate of strain is the sum of the elastic and secular rates:

(54) $\dot{\varepsilon}^H = \dot{\varepsilon}_s^H + \dot{\varepsilon}_d^H$

Time integration of equation (51) finally yields equation (55).

(55) $\varepsilon^H = a \ln\left(\frac{\sigma'}{\sigma'_0}\right) + c \ln\left[1 + \int_0^t \left(\frac{\sigma'}{\sigma_p}\right)^{\frac{b-a}{c}} \frac{d\tau}{\tau_0}\right]$

MSettle sets the reference time τ_0 by default to 1 day.

(56) $\tau_0 = 1 \text{ day}$

During a constant stress period after virgin loading, equation (55) simplifies to:

(57) $\varepsilon^H(t) = a \ln\frac{\sigma_p}{\sigma'_0} + b \ln\frac{\sigma'}{\sigma_p} + c \ln\frac{\tau}{\tau_0}$

This equation applies to the creep tail when σ' has become constant, and this is the familiar relation for one-dimensional creep, with strain depending on logarithm of time.

In case of several loading and un/reloading steps, the drained solution of equation (55) becomes:

(58) $\varepsilon^H(t) = a \ln\left(\frac{\sigma_p}{\sigma'_0}\right) + b \ln\left(\frac{\sigma'_n}{\sigma_p}\right) + c \ln\left(\frac{t - t_n + \theta_n}{\tau_0}\right)$

where the equivalent age θ_n is calculated as follows:

$$\theta_n = \left(\frac{\sigma'_{n-1}}{\sigma'_n}\right)^{\frac{b-a}{c}} \cdot \left(\theta_{n-1} + t_n - t_{n-1}\right), \quad \text{with} \quad \theta_0 = \tau_0 \cdot \left(\frac{\sigma_p}{\sigma'_1}\right)^{\frac{b-a}{c}}$$

$$\sigma_p = \begin{cases} \sigma'_0 + POP & \text{for POP compression} \\ \sigma'_0 \cdot OCR & \text{for OCR compression} \\ \sigma'_0 \cdot \left(\frac{t_{age}}{\tau_0}\right)^{\frac{c}{b-a}} & \text{for equivalent age compression} \end{cases}$$

t_n Begin time of step n [days].

n Number of the load step [-].

16.3 NEN-Koppejan

NEN-Koppejan's model is based on separate primary (instantaneous) and secondary (creep) contributions to the settlement. Compared to the NEN-Bjerrum model, the NEN-Koppejan model assumes that direct deformation under drained conditions occurs instantaneously, and that secondary settlement is the result of superposition of separate contributions from loading and/or unloading steps.

Hereafter can be found a short description of the following aspects of MSettle's NEN-Koppejan implementation:

- Settlement calculation [§ 16.3.1]
- Swelling calculation [§ 16.3.2]
- Natural strain calculation [§ 16.3.3]

See [Lit 2] for more information on the NEN-Koppejan model. See [§ 17.5] for a basic description of the NEN-Koppejan parameter determination.

16.3.1 NEN-Koppejan – Settlement

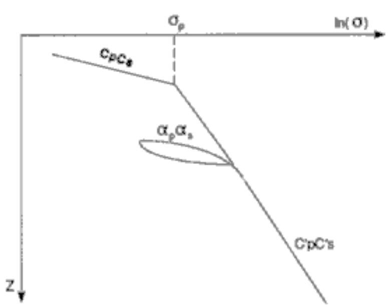


Figure 16-7 – Koppejan settlement

Four different situations can be distinguished for NEN-Koppejan:

- If the vertical effective stress is smaller than the preconsolidation pressure, the primary settlement can be calculated from:

(59) $$\frac{\Delta h_{prim}}{h_0} = \frac{1}{C_p} \ln\left(\frac{\sigma'}{\sigma_0}\right), \quad \sigma_0 < \sigma' < \sigma_p$$

- If the vertical effective stress is larger than the preconsolidation pressure, the primary settlement can be calculated from:

(60) $$\frac{\Delta h_{prim}}{h_0} = \frac{1}{C_p} \ln\left(\frac{\sigma_p}{\sigma_0}\right) + \frac{1}{C'_p} \ln\left(\frac{\sigma'}{\sigma_0}\right), \quad \sigma_0 < \sigma_p < \sigma'$$

- If vertical effective stress is smaller than the preconsolidation pressure, the secondary settlement for one loading can be calculated from:

(61) $$\frac{\Delta h_{sec}}{h_0} = \frac{1}{C_s} \log\left(1 + \frac{t}{\tau_0}\right) \ln\left(\frac{\sigma'}{\sigma_0}\right), \quad \sigma_0 < \sigma' < \sigma_p$$

- If the vertical stress is larger than the preconsolidation pressure, the secondary settlement for one loading can be calculated using the following equation:

(62) $$\frac{\Delta h_{sec}}{h_0} = \frac{1}{C_s} \log\left(1 + \frac{t}{\tau_0}\right) \ln\left(\frac{\sigma_p}{\sigma_0}\right) + \frac{1}{C'_s} \log\left(1 + \frac{t}{\tau_0}\right) \ln\left(\frac{\sigma'}{\sigma_p}\right), \quad \sigma_0 < \sigma_p < \sigma'$$

where:

C_p	Primary compression coefficient below preconsolidation pressure [-]
C_p'	Primary compression coefficient above preconsolidation pressure [-]
C_s	Secular compression coefficient below preconsolidation pressure [-]
C_s'	Secular compression coefficient above preconsolidation pressure [-]
Δh_{prim}	Primary settlement contribution of a layer [m]
Δh_{sec}	Secondary settlement contribution of a layer [m]
h_0	Initial layer thickness [m]
σ_0	Initial vertical effective stress [kN/m²]
σ_p	Preconsolidation pressure [kN/m²]
t	Time [days].
τ_0	Reference time [days]

16.3.2 NEN-Koppejan – Swelling

For NEN-Koppejan, the swelling can be formulated as:

(63) $$\frac{\Delta h_{prim}}{h_0} = \frac{1}{A_p} \ln\left(\frac{\sigma'}{\sigma_0}\right) + \frac{1}{A_s} \log\left(\frac{t}{\tau_0}\right) \ln\left(\frac{\sigma'}{\sigma_0}\right), \ \sigma_p < \sigma_0$$

where:

A_p Primary swelling coefficient [-]

A_s Secondary swelling coefficient [-]

NOTE: The A_s parameter will also be used in case of load removal. A large value of A_s implies that there will be no effect of load removal on creep. A large value is therefore only valid for cases with initial unloading.

16.3.3 NEN-Koppejan – Natural strain

MSettle's NEN-Koppejan model uses the following equation to describe the optional deformation reduction of each layer by natural strain:

(64) $$\Delta h_{nat} = h_0 \left[1 - \exp\left(-\frac{\Delta h_{koppejan}}{h_0}\right)\right]$$

where:

Δh_{nat} The settlement contribution of a certain layer, based on natural strain.

$\Delta h_{Koppejan}$ The original Koppejan settlement contribution, based on linear strain.

NOTE: Application of natural strain strictly speaking requires that soil parameters are also determined on the basis of natural strain.

17 Determining soil parameters

In order to determine proper parameters for MSettle's soil models, the usage of the M-Series program MCompress is recommended. MCompress can interpret results from both oedometer tests and the modern *Constant Rate of Strain* tests (K_0 –CRS) in order to generate consistent parameters for MSettle's models. In this paragraph, just some basic ingredients for parameter determination are discussed, based on oedometer test results and simplified conversion formulas.

- Oedometer tests [§ 17.1]
- Overconsolidation [§ 17.2]
- NEN-Bjerrum parameters [§ 17.3]
- Isotache parameters [§ 17.4]
- Koppejan parameters [§ 17.5]
- Conversion of NEN-Bjerrum parameters from Koppejan parameters [§ 17.6]
- Conversion of Isotache parameters [§ 17.7]

An overview of important parameter definitions can be found in the first chapter of this manual [§ 1.2].

17.1 Oedometer tests

17.1.1 Description

Oedometer tests are also called 'confined compression tests' or 'consolidation tests'. In these tests, the vertical settlement Δh of a sample with initial height h_0 and initial void ratio e_0 is determined during step-wise loading, with intermediate consolidation and creep. Lateral deformation is prevented. It is common to double the load every 24 hours. Occasionally, unloading steps are also applied. Complete information on practical oedometer test interpretation can be found for example in the NEN 5118 standard [Lit 9] (in Dutch).

The MSerie software called MCompress interprets oedometer test data's according to NEN-Bjerrum, NEN-Koppejan and Isotache models. For more information on this software, contact our sales department: sales@delftgeosystems.nl.

17.1.2 Simulating an oedometer test with MSettle

MSettle uses a minimum time step of 1 day by default. To simulate a short term oedometer test with typical loading stages of just 1 day, a smaller unit of time can be applied by using a trick:

- Enter a multiplication factor for the *Creep rate reference time* in the *Calculation Options* window [§ 10.1.1]. For example a value of 24 × 60 = 1440 for a time unit of minutes.
- Enter all input of time in the new unit:
  - The end of calculation time in the *Calculation Options* window [§ 10.1.1]
  - The times of applying changes in loading or water pressures
  - The times in the measurement file, when using the *Fit for Settlement Plate* option [§ 4.9.14]. The fit option enables you in fact to perform advanced parameter determination.
- Divide all values of permeability or consolidation coefficient in the *Materials* window with the same factor (1440 for minutes).
- Interpret time values in the results in the modified unit of time, when inspecting graphs and reports.

17.2 Overconsolidation

A sample can be over-consolidated, either by geological history (undisturbed) or artificially. This overconsolidation can result from ageing and/or pre-overburden pressure. The overconsolidation is characterized via the preconsolidation stress σ_p. This value marks the transition point between the reloading branch and the virgin loading branch in the strain versus ln(σ) diagram (Figure 17-1). Soil will behave differently below and above the preconsolidation pressure.
The preconsolidation stress varies however along the depth. Therefore, the pre-consolidation stress must be transformed into a stress-independent soil parameter. The Koppejan model can calculate the preconsolidation stress from the Over-Consolidation Ratio (*OCR*), or from the gradient in the initial stress. The NEN-Bjerrum and Isotache models can calculate the preconsolidation stress from the *OCR* or the pre-overburden pressure (*POP*).

- The *OCR* is defined as the preconsolidation stress divided by the actual in-situ vertical stress.
- The *POP* is defined as the difference between the preconsolidation stress and the actual in-situ vertical stress. This means that the gradient along the depth is equal to the gradient of the initial stress.

See Figure 17-1 for a graphical representation. In general, *OCR* is considered more appropriate if the preconsolidation stress results predominantly from ageing. *POP* (or using the same gradient as the initial stress) is considered more appropriate if the cause is predominantly a large overburden pressure in the past.

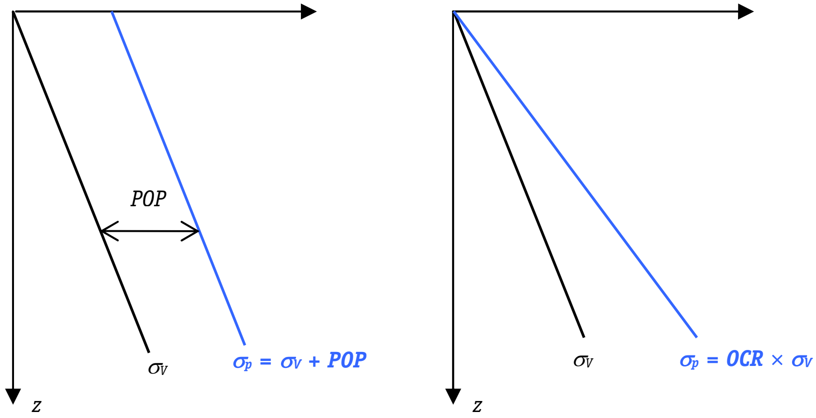


Figure 17-1 – Over-consolidation: *POP* and *OCR*

17.3 NEN-Bjerrum parameter determination

MSettle's NEN-Bjerrum model [§ 16.1] uses parameters that correspond to today's international de-facto standard. The reloading/swelling index C_r describes the elastic stiffness during unloading and reloading (below preconsolidation pressure). The primary compression index C_c and the coefficient of secondary compression C_α describe respectively the idealized elasto-plastic deformation and the viscous creep rate during virgin loading.
All these parameters are traditionally determined using a linear strain assumption instead of natural strain [§ 16.2.1].

NOTE: With regard to the NEN-Bjerrum parameter definition, please note the following important attention points:

- Linear strain parameters are determined with reference to the initial height. However, some standards and recommendations for interpretation of oedometer tests prescribe that parameters (especially C_α) are determined with reference to the height before the next loading step. Therefore you should always check if your parameters have been determined in the way that MSettle expects.
- Linear strain parameters are not objective if strains become large. In cases with large strains, you must therefore determine linear strain parameters from tests that use the same initial and final stress levels as experienced in the field.
- The parameters C_r and C_c are in fact related to changes in void ratio. C_α is however directly related to changes in linear strain. Please note that this

definition of the C_α complies with common practice, but differs from the original definition by Mesri [Lit 6].

Assuming drained conditions, the NEN-Bjerrum model defines the idealized linear strain increment by one virgin load step (above preconsolidation pressure) by the following equation.

(65) $$\frac{\Delta h(t-t_n)}{h_0} = \varepsilon^C(t) - \varepsilon^C(t_n) = \frac{C_{c.n}}{1+e_0}\log\left(\frac{\sigma_n}{\sigma_{n-1}}\right) + C_{\alpha.n}\max\left(0;\log\left(\frac{t-t_n}{\tau_0}\right)\right)$$

where:

n The subscript denoting the load step number.
t_n The start time of load step n [days].
τ_0 The reference time (1 day).

Assuming again that pore pressures are dissipated before the following load increment, $C_{\alpha.n}$ can be determined from the tangent of the tail of the strain increment during one virgin load step. This is illustrated in Figure 17-2.

(66) $$C_{\alpha.n} = \frac{\mathrm{d}\Delta\varepsilon^C(t_{n+1}-t_n)}{\mathrm{d}\log(t_{n+1}-t_n)}, \quad \sigma' > \sigma_p$$

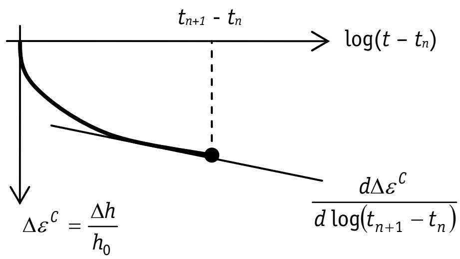


Figure 17-2 – Determining the common coefficient of secondary compression

The compression ratio for the virgin load step n follows by substitution of $C_{\alpha.n}$ into equation (65).

(67) $$CR_n = \frac{C_{c.n}}{1+e_0} = \frac{\Delta\varepsilon^C(t_{n+1}-t_n) - \max\left(C_{\alpha.n}\log(t_{n+1}-t_n);0\right)}{\log\left(\frac{\sigma_n}{\sigma_{n-1}}\right)}, \quad \sigma' > \sigma_p$$

The reloading/swell index (un-/reloading below preconsolidation) is determined in complete analogy.

$$RR_n = \frac{C_{r.n}}{1+e_0} = \frac{\Delta\varepsilon^C\left(t_{n+1}-t_n\right)}{\log\left(\dfrac{\sigma_n}{\sigma_{n-1}}\right)}, \quad \sigma' \le \sigma_p \tag{68}$$

The parameter *RR* is preferably determined from unloading curves. Determination from loading before the initial preconsolidation stress will usually result in values that are too low, because of the sample disturbance.

17.4 Isotache parameters determination

Hereafter is explained how Isotache natural strain parameters are determined from oedometer test results. These parameters are: the Isotache natural primary compression index *a*, the Isotache natural swelling index *b* and the Isotache natural secondary compression constant *c*. See [§ 17.7] for conversion from existing soil parameters for other models. The simplified treatment is based on the assumption that a common oedometer test is used, with doubling of load each step, and a limited duration of each step.

Assuming drained conditions, the natural strain increment at the end of one virgin load step (above preconsolidation pressure) can be defined approximately by equation (69).

$$\begin{aligned} \Delta\varepsilon^H\left(t_{n+1}-t_n\right) &= \varepsilon^H\left(t_{n+1}\right)-\varepsilon^H\left(t_n\right) \\ &\approx b\ln\left(\frac{\sigma_n}{\sigma_{n-1}}\right)+\max\left(0;c\ln\left(\frac{t_{n+1}-t_n+t_{shift.n}}{\tau_0}\right)\right) \end{aligned} \tag{69}$$

where:

n The subscript denoting the load step number [-].

t_n The start time of load step n [days].

τ_0 The reference time (1 day).

NOTE: The expression for the final natural strain increment at the end of the load step is similar to equation (65) for the NEN-Bjerrum model [§ 17.3]. The actual behavior of both the NEN-Bjerrum model and the Isotache model during the first part of the load step will however be quite different, due to the rate type formulation.

The value of t_{shift} determines the influence of creep from previous load steps, and can be determined by curve fitting. For interpretation of common oedometer tests (doubling of load each step) however, the assumption is justified that t_{shift} is close to zero.

Assuming that pore pressures are dissipated before the following load increment, and assuming t_{shift} = 0, c can be determined from the tangent of the tail of the natural strain increment by one virgin load increment.

(70) $$c_n = \frac{d\Delta\varepsilon^H\left(t_{n+1}-t_n\right)}{d\ln\left(t_{n+1}-t_n\right)}, \quad \sigma' > \sigma_p$$

This is illustrated in Figure 17-3.

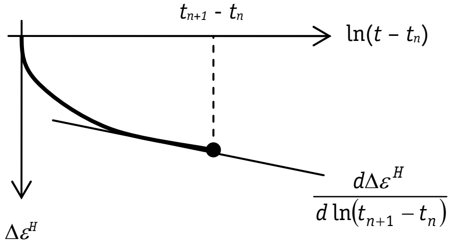


Figure 17-3 – Determining the Isotache natural secondary compression index c

The Isotache natural compression index b for the virgin load step n follows by substitution of c_n into equation (69).

(71) $$b_n \approx \frac{\Delta\varepsilon\left(t_{n+1}-t_n\right)-c_n\ln\left(t_{n+1}-t_n\right)}{\ln\left(\dfrac{\sigma_n}{\sigma_{n-1}}\right)}, \quad \sigma' > \sigma_p$$

A more refined estimate of b can be found if the reference creep rate is known (the strain rate after one day loading at the initial preconsolidation stress). The strain increment $\Delta\varepsilon$ should then be determined exactly at the moment where the strain rate is equal to the reference strain rate after one day of loading.

(72) $$b_n = \frac{\Delta\varepsilon\left(t-t_n\right)}{\ln\left(\dfrac{\sigma_n}{\sigma_{n-1}}\right)}, \quad \dot{\varepsilon}\left(t-t_n\right)=\dot{\varepsilon}_{ref}\left(\tau_0 = 1\ day\right), \sigma' > \sigma_p$$

The parameter a is preferably determined from unloading curves, where creep rates are low.

(73) $$a_n = \frac{\Delta\varepsilon\left(t_{n+1}-t_n\right)}{\ln\left(\dfrac{\sigma_n}{\sigma_{n-1}}\right)}, \sigma' < \sigma_p$$

Determination of a from loading before the initial preconsolidation stress will usually result in too low values, because of the sample disturbance.

Rough estimates of parameter values can be derived from correlation formulas. Usage of these formulas is at own risk, as accurate parameters can only determined by soil testing.

Equation (74) gives a rough correlation between the b parameter and the saturated unit weight in undeformed state.

$$(74) \qquad b \approx 0.326 \left(\frac{\gamma_{sat.0}}{\gamma_w} \right)^{-2.11}$$

Table 17-1 gives rough estimates of b/a and b/c for different soft soil types.

Table 17-1 – Rough Isotache parameter correlation for soft soil types

	$\gamma_{sat.0}$ [kN/m^3]	b/a [-]	b/c [-]
Peat	11	7	12
Organic soft clay	12	8	13
Organic clay	14	12	20
Silty clay	16	12	25

17.5 NEN-Koppejan parameter determination

The NEN-Koppejan model [§ 16.3] distinguishes primary and secondary settlements. The elasto-plastic primary compression is a function of only the effective stress. The viscous secondary compression (creep) is a function of both the effective stress and the time. The values of the primary and secondary coefficients are different below and above the preconsolidation stress. Traditionally, NEN-Koppejan parameters are determined using a linear strain assumption instead of natural strain [§ 16.3.3]. This means that applicability of linear NEN-Koppejan parameters for soft soil is limited to stress levels in the field that are comparable to the stress levels used for parameter determination.

17.5.1 Primary and secular compression coefficients

To determine the compression coefficients from the measured strains in the interval between load step n and $n+1$, you must first subtract the approximate settlement/swelling contributions from all preceding load steps $i=1,n$-1.

(75)
$$\Delta\varepsilon'(t-t_n)=\varepsilon(t)-\sum_{i=1}^{n-1}\ln\left(\frac{\sigma_i}{\sigma_{i-1}}\right)\left[\frac{1}{C_{prim.i}}+\frac{1}{C_{\sec.i}}\log\left(\frac{t-t_i}{\tau_0}\right)\right]$$
$$=\left[\frac{1}{C_{prim.n}}+\frac{1}{C_{\sec.n}}\log\left(\frac{t-t_n}{\tau_0}\right)\right]\ln\left(\frac{\sigma_n}{\sigma_{n-1}}\right)$$

where:

n The subscript denoting the load step number [-].

t_n The start time of load step n [days].

τ_0 The reference time (1 day).

The parameters $C_{prim;i}$ and $C_{sec;i}$ in interval i possess either the value below or above the preconsolidation pressure.

- $\sigma' < \sigma_p$: $C_{prim} = C_p$ and $C_{\sec} = C_s$
- $\sigma' \geq \sigma_p$: $C_{prim} = C'_p$ and $C_{\sec} = C'_s$

Each load step that passes preconsolidation must be split into one sub-step before preconsolidation stress, and one sub-step after preconsolidation stress.
If it is assumed that pore pressures are dissipated before the following load increment, then C_{sec} can be estimated from the tangent of the tail of $\Delta\varepsilon'$, according to Figure 17-4 and equation (76).

(76)
$$C_{\sec.n}=\ln\left(\frac{\sigma_n}{\sigma_{n-1}}\right)\frac{d\log(t_{n+1}-t_n)}{d\Delta\varepsilon'}$$

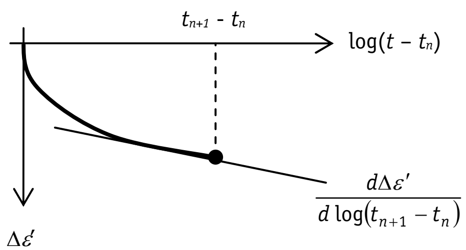


Figure 17-4 – Determining Koppejan's secondary compression index

The primary compression index for the current step follows then by substitution of $C_{sec.n}$ into equation (75).

(77)
$$\frac{1}{C_{prim.n}}=\frac{\Delta\varepsilon'(t_{n+1}-t_n)}{\ln\left(\frac{\sigma_n}{\sigma_{n-1}}\right)}-\frac{\log\left(\frac{t_{n+1}-t_n}{\tau_0}\right)}{C_{\sec.n}}$$

17.5.2 Primary and Secondary swelling coefficients

Theoretically, the primary and secondary swelling indices can be determined from unloading steps, analogous to determining the compression coefficients.
In practice, the primary swelling index is mostly set equal to the value of the primary compression index below preconsolidation, and the secondary swelling coefficient is set to a large value.

(78) $A_p = C_p$ and $A_s \to \infty$

NOTE: A_s will also be used by the NEN-Koppejan model in case of load removal. A large value of A_s implies that there will be no effect of load removal on creep. Therefore, the swelling part of the Koppejan model with large A_s value is only valid for cases with initial unloading.

17.6 NEN-Bjerrum parameters from Koppejan parameters

17.6.1 For a single load

In case of single load $\Delta\sigma$, conversion of existing NEN-Koppejan parameters to NEN-Bjerrum parameters is performed easily, using the following formulas.

(79) $$RR = \frac{\ln(10)}{C_p}$$

(80) $$CR = \frac{\ln(10)}{C'_p}$$

(81) $$C_\alpha = \frac{1}{C_s}\ln\left(\frac{\sigma_p}{\sigma'_0}\right) + \frac{1}{C'_s}\ln\left(\frac{\sigma'_0 + \Delta\sigma}{\sigma_p}\right)$$

17.6.2 From oedometer test results

The NEN-Bjerrum parameters (RR, CR, C_α) can be calculated from the NEN-Koppejan parameters using the results of an oedometer test (C_p, C_p', C_s') and as additional information the preconsolidation stress σ_p and the stresses σ'_i at the different virgin loading steps. It is assumed that creep before preconsolidation stress can be neglected.

The calculation of RR is still straightforward, as long as the creep before preconsolidation stress is neglected.

(82) $$RR = \frac{\ln(10)}{C_p}$$

For the calculation of C_α, the theoretical slope of the creep tail according to C_s' at a certain time has been calculated for each of the virgin loading steps, and C_α is then determined from these slopes by averaging. The creep before preconsolidation stress is again neglected. The resulting formula is:

(83) $$C_\alpha = \frac{1}{n\,C_s'} \sum_{i=1}^{n} \ln\left(\frac{\sigma_i'}{\sigma_p}\right)$$

where:

n Number of load steps above pre-consolidation pressure (i.e. virgin loading steps).

The calculation of CR is most complicated, because the C_p' parameter has been determined from a primary strain increment after a certain load step, after subtracting the theoretical creep contributions caused by the preceding load steps, according to C_s'. Simplifications are possible by: (a) neglecting the creep before the first virgin loading step; (b) assuming a doubling of loading after each load step; (c) assuming a duration of 1 day for each load step. The resulting approximate conversion formula is given below.

(84) $$CR = \ln(10)\left[\frac{1}{C_p'} + \frac{1}{n\,C_s'} \sum_{i=2}^{n} (n+1-i)\log(i)\right]$$

17.7 Isotache a/b/c parameter conversion

Existing soil parameter collections often consist of NEN-Bjerrum and NEN-Koppejan parameters, determined using a linear strain assumption. Alternatively, also Cam-Clay based parameters for finite element analysis might be available. The following equations show how you can convert these parameters to natural Isotache parameters and vice versa.

The formulas were derived, by equaling the separate deformation contributions by reloading to preconsolidation stress, virgin loading and creep. Equation (57) was used for the Isotache model. Equations (36) and (37) were used for the NEN-Bjerrum model. Equations (59) and (60) were used for the NEN-Koppejan model.

NOTE: Using the conversion formulas, the user should realize that settlement prediction with linear parameters and natural parameters will only yield approximately equal settlements at one specific stress level and at one specific time.

Due to the different nature of the formulations, equal settlements at any stress and any time can never be expected.

The following assumptions have been used during derivation:

- The conversion is based on the condition that the linear strain contributions are set equal at a given effective stress σ' and time t.
- The consolidation is finished at time t, so that the effective stress rate has become approximately zero.
- The parameters for primary swelling and primary reloading below preconsolidation stress are equal for both the NEN-Bjerrum model and the NEN-Koppejan model.
- The secondary settlement contribution in the NEN-Bjerrum and NEN-Koppejan model for loading below preconsolidation stress is neglected.

17.7.1 Linear NEN-Bjerrum parameters

(85) $$a = -\frac{\ln\left(1-\varepsilon_p^C\right)}{\ln\left(\frac{\sigma_p}{\sigma_0'}\right)} \qquad RR = \frac{C_r}{1+e_0} = \frac{1-\left(\frac{\sigma_p}{\sigma_0'}\right)^{-a}}{\log\left(\frac{\sigma_p}{\sigma_0'}\right)}$$

(86) $$b = \frac{\ln\left(1-\varepsilon_p^C\right)-\ln\left[1-\varepsilon_{prim}^C\right]}{\ln\left(\frac{\sigma'}{\sigma_p}\right)} \qquad CR = \frac{C_c}{1+e_0} = \frac{\left(1-\varepsilon_p^C\right)\left(1-\left(\frac{\sigma'}{\sigma_p}\right)^{-b}\right)}{\log\left(\frac{\sigma'}{\sigma_p}\right)}$$

(87) $$c = \frac{\ln\left(1-\varepsilon_{prim}^C\right)-\ln\left[1-\varepsilon_{prim}^C - C_\alpha \log\left(\frac{t}{\tau_0}\right)\right]}{\ln\left(\frac{t}{\tau_0}\right)}$$

$$C_\alpha = \frac{\left(1-\varepsilon_{prim}^C\right)\left(1-\left(\frac{t}{\tau_0}\right)^{-c}\right)}{\log\left(\frac{t}{\tau_0}\right)}$$

where:

ε^C_p Primary linear deformation below preconsolidation:

$$\varepsilon_p^C = RR \log\left(\frac{\sigma_p}{\sigma_0'}\right)$$

ε^C_{prim} Total primary linear deformation (at reference stress σ'):

$$\varepsilon_{prim}^C = RR \log\left(\frac{\sigma_p}{\sigma_0'}\right) + CR \log\left(\frac{\sigma'}{\sigma_p}\right)$$

σ' Reference stress level for which the conversion is made. The stress level used should be representative for the final stresses after embankment construction.

NOTE: For small strains ($\varepsilon^H \rightarrow \varepsilon^C$) the following limits apply:

$$a \rightarrow \frac{C_r}{\ln(10)(1+e_0)}, \quad b \rightarrow \frac{C_c}{\ln(10)(1+e_0)}, \quad c \rightarrow \frac{C_\alpha}{\ln(10)}$$

17.7.2 Linear NEN-Koppejan parameters

The conversion of NEN-Koppejan parameters into Isotache parameters can be performed in 2 steps:

- NEN-Koppejan parameters are first converted into NEN-Bjerrum parameters using equations given in [§ 17.6.1] for a single load or in [§ 17.6.2] for several load steps (i.e. oedometer test).
- Then Isotache parameters are deduced from NEN-Bjerrum parameters using equations given above [§ 17.7.1].

17.7.3 Natural and linear Cam-Clay-creep parameters

A Cam Clay based visco-plastic model is available in many finite element programs to describe the two-dimensional or three-dimensional soft soil behaviour. A well-known example is the Plaxis soft-soil-creep model. The strain based soft-soil-creep parameters are expressed in the classic void-ratio based Cam Clay parameters using:

(88) $$\lambda^* = \frac{\lambda}{1+e_0} \quad \text{and} \quad \kappa^* = \frac{\kappa}{1+e_0}$$

Cam Clay parameters relate volumetric strain to isotropic stress, whereas Isotache parameters relate vertical strain to vertical stress. The optional Updated Mesh method (or Updated Lagrange method) in finite element programs is completely equivalent with Isotache's natural strain method. Cam-Clay-creep parameters are in practice however often determined and used with a linearized strain assumption.

Cam-Clay-creep parameters that were determined on a natural strain basis are hereafter indicated by the addition (ε^H), while the parameters on linear strain basis are indicated by the addition (ε^C).

The a parameter can be expressed in the soft-soil-creep parameter κ^*, using the normally consolidated earth pressure coefficient K_{NC} and the Poisson's ratio ν.

$$(89) \qquad a = \kappa^*\left(\varepsilon^H\right)\frac{\ln\left(\dfrac{\left(1+2K_{\mathrm{NC}}\right)\sigma_{\mathrm{p}}}{\left(1+2K_{\mathrm{NC}}\right)\sigma_{\mathrm{p}} - \dfrac{1+\nu}{1-\nu}\left(\sigma_{\mathrm{p}}-\sigma_0\right)}\right)}{\ln\left(\dfrac{\sigma_{\mathrm{p}}}{\sigma_0}\right)}$$

where:

$$\kappa^*\left(\varepsilon^{\mathrm{H}}\right) = -\frac{\ln\left[1-\kappa^*\left(\varepsilon^{\mathrm{C}}\right)\ln\left(\dfrac{\sigma_{\mathrm{p}}}{\sigma_0}\right)\right]}{\ln\left(\dfrac{\sigma_{\mathrm{p}}}{\sigma_0}\right)}$$

$$\kappa^*\left(\varepsilon^{\mathrm{C}}\right) = \frac{1-\left(\dfrac{\sigma_{\mathrm{p}}}{\sigma_0}\right)^{-\kappa^*\left(\varepsilon^{\mathrm{H}}\right)}}{\ln\left(\dfrac{\sigma_{\mathrm{p}}}{\sigma_0}\right)}$$

ν Poisson's ratio for elastic unloading and reloading.

K_{NC} Earth pressure coefficient in normally consolidated state (virgin loading).

Parameter b is directly equal to natural soft-soil-creep parameter $\lambda^*(\varepsilon^H)$, on the condition that the yield cap of the constitutive model has been constructed in such a way that the earth pressure coefficient during virgin loading is preserved.

$$(90) \qquad b = \lambda^*\left(\varepsilon^{\mathrm{H}}\right) \;\text{ if }\; \frac{d\sigma_h'}{d\sigma_v'} = \mathrm{K}_{NC}$$

where:

$$\lambda^*\left(\varepsilon^H\right) = \frac{\ln\left(1-\varepsilon_{\mathrm{p}}^{\mathrm{C}}\right) - \ln\left[1-\varepsilon_{\mathrm{p}}^{\mathrm{C}} - \lambda^*\left(\varepsilon^C\right)\ln\left(\dfrac{\sigma'}{\sigma_p}\right)\right]}{\ln\left(\dfrac{\sigma'}{\sigma_p}\right)}$$

$$\varepsilon_{\mathrm{p}}^{\mathrm{C}} = \kappa^*\left(\varepsilon^C\right)\ln\left(\frac{\sigma_{\mathrm{p}}}{\sigma_0}\right)$$

$$\lambda^*\left(\varepsilon^C\right) = \frac{\left(1-\varepsilon_p^C\right)\left[1-\left(\frac{\sigma'}{\sigma_p}\right)^{-\lambda^*\left(\varepsilon^H\right)}\right]}{\ln\left(\frac{\sigma'}{\sigma_p}\right)}$$

Parameter c is directly equal to the natural soft-soil-creep parameter $\mu^*(\varepsilon^H)$, as vertical strain equals volumetric strain under confined compression conditions.

$$c = \mu^*\left(\varepsilon^H\right) \tag{91}$$

where:

$$\mu^*\left(\varepsilon^H\right) = \frac{\ln\left(1-\varepsilon_{prim}^C\right) - \ln\left[1-\varepsilon_{prim}^C - \mu^*\left(\varepsilon^C\right)\ln\left(\frac{t}{\tau_0}\right)\right]}{\ln\left(\frac{t}{\tau_0}\right)}$$

$$\varepsilon_{prim}^C = \kappa^*\left(\varepsilon^C\right)\ln\left(\frac{\sigma_p}{\sigma_0}\right) + \lambda^*\left(\varepsilon^C\right)\ln\left(\frac{\sigma'}{\sigma_p}\right)$$

$$\mu^*\left(\varepsilon^C\right) = \frac{\left(1-\varepsilon_{prim}^C\right)\left(1-\left(\frac{t}{\tau_0}\right)^{-\mu^*\left(\varepsilon^H\right)}\right)}{\ln\left(\frac{t}{\tau_0}\right)}$$

18 Special Calculations

The following sections contain a short theoretical background on three special calculation types:

- Fit for settlement plate [§ 18.1]
- Reliability Analysis [§ 18.2]
- Horizontal displacements [§ 18.3]

18.1 Fit for Settlement Plate

MSettle can iteratively improve the match between measured and predicted settlements in a single vertical, by using a special *Weighted Least Squares* (WLS) method, also known as *Maximum A-Posteriori estimate* (MAP). This method will update the values of fit parameters, by minimizing not only the difference between measurements and predictions, but also the difference between the initial value and the updated value of the fit parameters. Separate weights to each of the differences can be attached. Such a weight determines the relative importance of each difference. A large weight implies a more certain value of a measurement or parameter; a small weight implies a more uncertain value.

The weighted least squares method minimizes the following expression.

(92) $$S = \left(z_m - z_p\right)^T W_z \left(z_m - z_p\right) + \left(x - x_0\right)^T W_x \left(x - x_0\right)$$

where:

z_p The vector with predicted settlements.
z_m The vector with measured settlements.
W_z A diagonal matrix, containing the weights for the measurements. In a probabilistic framework, this matrix can be considered as the inverse of the covariance matrix of the imperfections: $W_z = C_\varepsilon^{-1}$, see [§ 18.2]. The imperfections represent the inaccuracies in the measuring method and in the model assumptions.
x The vector with updated fit parameters. MSettle uses 5 special fit parameters, to scale the values of the corresponding parameters for all the different soil layers.
x_0 The vector with initial values of the fit parameters.
W_x A diagonal matrix, with the weights for the fit parameters. In a probabilistic framework, this matrix is equal to the inverse of the covariance matrix of the fit parameters: $W_x = C_x^{-1}$.

Equation (93) shows the iterative solution scheme, in case of a nonlinear relationship between the fit parameters and the predicted settlements.

(93) $$x^{(i+1)} = x^{(i)} + \left(J^{(i)^T} W_z^{-1} J^{(i)} + W_x\right)^{-1} \left(J^{(i)^T} W_z \left(z_m - z_p^{(i)}\right) + W_x \left(x^{(0)} - x^{(i)}\right)\right)$$

where:
i The number of the iteration.
J The Jacobian, containing derivatives of z_p for variations of x:

$$J_{ij} = \frac{\partial z_{p.i}}{\partial x_j}.$$

MSettle approximates the coefficients of J for each iteration numerically, by using small parameter variations (perturbation method).

(94) $$J_{ij} \approx \frac{\Delta z_{p.i}}{\Delta x_j}$$

MSettle will temporary increase the diagonal terms of the matrix $J^{(i)^T} W_z J^{(i)} + W_x$ according to the *Levenberg-Marquardt* algorithm, whenever this is required for further convergence during the iteration process.

MSettle indicates the goodness of fit by a so-called imperfection and a coefficient of determination.

(95)

$$\text{Imperfection}: \quad \varepsilon^{(i)} = \sqrt{\frac{r^T r}{n-1}}, r = z_m - z_p^{(i)}$$

$$\text{Coefficient of determination}: \quad 1 - \frac{\left(\varepsilon^{(i)}\right)^2}{\left(\varepsilon^{(0)}\right)^2}$$

where:

n is the number of measurements.

18.2 Reliability Analysis

The bandwidth and the parameter sensitivity for total and residual settlements in a single vertical can be determined by using a *reliability analysis*. The bandwidth and sensitivity of the settlements depend on the assumed uncertainty in the input parameters, expressed in standard deviations. MSettle can update (and thereby reduce) the initial parameter uncertainty, by using settlement measurements. The following sections will present the basic background on:

- Stochastic distributions and parameters [§ 18.2.1]
- Initial and updated parameter covariance [§ 18.2.2]
- Sensitivity analysis with MSettle [§ 18.2.3]
- The probabilistic methods in MSettle [§ 18.2.4].

18.2.1 Stochastic distributions and parameters

MSettle can apply a standard normal probability distribution for all stochastic (uncertain) parameters and all probabilistic methods. The alternative lognormal distribution is currently only available for testing purposes. Both distribution types are characterized by a mean μ and a standard deviation σ for a standard normal distribution.

Normal
The probability that a value x is smaller than the value $x_{characteristic}$ is for a normal distribution expressed by:

(96) $$P\left(x < x_{characteristic}\right) = \Phi_N\left(u_{characteristic}\right)$$

where:

u	is the parameter of a standard normal distribution: $u = \frac{x - \mu[x]}{\sigma[x]}$
$\Phi_N(u_{characteristic})$	integral of the standard normal probability density: $\Phi_N(u_{characteristic}) = \int_{-\infty}^{u_{characteristic}} \varphi_N(u) \cdot du$
$\varphi_N(u)$	Standard normal probability density: $\varphi_N(u) = \frac{\exp(-u^2/2)}{\sqrt{2\pi}\sigma[x]}$

Lognormal

If parameter $y = \ln(x)$ has a normal distribution, then parameter x has a lognormal distribution. A lognormal distribution always yields positive values. For small ratio's between standard deviation and mean, the two distribution types will become equivalent. The normal and lognormal distributions are similar for small ratios between the standard deviation and the mean. MSettle uses the following two equations to calculate $\mu[y]$ and $\sigma[y]$ from the user input of $\mu[x]$ and $\sigma[x]$:

(97) $$\sigma[y] = \sqrt{\ln\left(1 + \left(\frac{\sigma[x]}{\mu[x]}\right)^2\right)}$$

(98) $$\mu[y] = \ln\left(\mu[x] - \tfrac{1}{2}\sigma^2[x]\right)$$

Mean

the mean value of parameter x can be calculated straightforwardly from equation (99):

(99) $$\mu[x] = \frac{1}{n}\sum_{i=1}^{n} x_i$$

where n is the number of samples.

Standard deviation

The standard deviation quantifies the initial uncertainty in a parameter. MSettle supplies defaults via the variation coefficient V_x:

(100) $$V_x = \frac{\sigma[x]}{\mu[x]}$$

The default values for the coefficient of variation are mainly based on the Dutch NEN standard [Lit 8]. The input value of the standard deviation should be somewhere between the standard deviation of a local value and the standard deviation of the mean value, depending on the thickness of the layers and the scale of horizontal and vertical variability.

(101) local: $$\sigma_{\text{total}} = \sqrt{(\mu V_{sys})^2 + \left(\frac{1}{n}+1\right)\left(\frac{t}{u}\sigma_{\text{statistical}}\right)^2}$$

(102) mean: $$\sigma_{\text{total}} = \sqrt{(\mu V_{\text{sys}})^2 + \frac{1}{n}\left(\frac{t}{u}\sigma_{\text{statistical}}\right)^2}$$

where:

(103) $$\sigma^2_{statistical} = \frac{1}{n-1}\sum_{i=1}^{n}(x_i - \mu)^2$$

and where:

t is the parameter from a Student distribution, which depends on the number of samples n. The parameter becomes equal to u for large values of n.

V_{sys} is the coefficient of variation that quantifies the systematic uncertainty by soil testing and by the transformation from measurements to parameters. A usual value for soil compression parameters is 0.1.

18.2.2 Initial and updated parameter covariance

MSettle determines the bandwidth in an initial design analysis from the input values of the parameter standard deviations. MSettle stores the square values of these standard deviations in the diagonal terms of the initial parameter covariance matrix $C_{x.0}$.

(104) $$C_{\text{x.0}.ii} = \sigma^2(x_{0.i})$$

MSettle can update the mean parameter values via a fit on measured settlements [§ 18.1]. If you use these updated mean values in a reliability analysis, then MSettle will apply *Bayesian Updating* of the parameter covariance matrix, according to equation (105). This update will introduce correlations between the different uncertain (stochastic) parameters, which finally yield a reduced bandwidth for the updated mean values of the settlement prediction.

(105) $$C_{\text{x}.fit} = \left(J_{fit}^T\, C_\varepsilon^{-1}\, J_{fit} + C_{\text{x.0}}^{-1}\right)^{-1}, \; C_\varepsilon = I\varepsilon^2$$

The jacobian matrix J contains the derivatives of the settlements to the different parameters :

(106) $$J_{ij} = \frac{\partial z_{\mathrm{p},i}}{\partial x_j}$$

MSettle approximates the coefficients of J numerically, by using small parameter variations (perturbation method). MSettle updates the derivatives after a fit, by using the updated mean values of the parameters.

The input value of the imperfection ε defines the diagonal covariance matrix C_ε. This imperfection represents in fact the combined inaccuracy of the measurements and the prediction model. Equation (105) shows that the effect of measurements on the update of the parameter covariance will increase if the value of the imperfection ε becomes smaller, and if parameter variations show more influence on the measured part of the settlement curve.

Finding a proper value for the imperfection is therefore important. One might consider using:

(107) $$\varepsilon^2 = \max\left(\frac{r^T r}{n-p}, \varepsilon^2_{\mathrm{measurement}}\right), \; r = z_m - z_p$$

where n is the number of measurements, p is the number of fit parameters, $\varepsilon_{measurement}$ is the size of the inaccuracy in the measurements, z_m is the vector with measurements and z_p is the vector with predictions after a fit.

18.2.3 Sensitivity analysis with influencing factors

Influencing factors show the relative influence of uncertain parameters on total and residual settlements at different time points. The value of the influencing factor increases if the parameter is more uncertain, and if the effect of parameter variation on the considered part of the settlement curve is larger. MSettle calculates the influencing factors by using:

(108) $$\alpha^2_{kj} = \frac{J_{kj} \sum_i C_{\mathrm{x}.ji} J_{ki}}{\sum_j J_{kj} \sum_i C_{\mathrm{x}.ji} J_{ki}}$$

where the index k is related to the time t_k and the index j is related to parameter x_j. MSettle determines the initial parameter covariances from the input values of the parameter standard deviations, see equation (104). MSettle updates the parameter covariances after a fit on measurement data, see equation (105). The jacobian matrix J contains the linearized derivatives of the settlements to the different parameters. MSettle updates the derivatives after a fit, by using the updated mean values of the parameters.

18.2.4 Probabilistic methods

MSettle offers a choice between three different probabilistic methods. The Monte Carlo method is the most accurate method (level I), but also the most time-consuming. The quick linearized FOSM method and the iterative FORM method are approximate methods (level II) for respectively total and residual settlements. Output of influencing factors for sensitivity analysis is only available for the FOSM and FORM methods.

Linearized First Order Second Moment method (FOSM)
This method can be selected for a quick and approximate determination of the bandwidth and sensitivity factors for total settlements. MSettle determines the standard deviation of the settlements from the diagonal terms of the covariance matrix of the settlements.

(109) $$\sigma^2(z_i) = C_{z..ii} \ , \ C_z = J\, C_x J^T$$

MSettle linearizes the derivatives in the Jacobian matrix at the mean values of the uncertain parameters. The derivatives are updated after a fit, by using the updated mean values of the parameters. MSettle will also update the parameter covariance matrix after a fit, by using equation (105).

Iterative First Order Reliability method (FORM) for bandwidth and sensitivity factors of residual settlements
This method can be selected for an approximate determination of the bandwidth and sensitivity factors for residual settlements.
This method will give the approximate probability that the residual settlement exceeds an allowed value. The *limit state function Z* equals the predicted residual settlement minus the allowed residual settlement.

(110) $$Z = F_{\text{allowed}} - F \ , \ F = z_{\text{end}} - z_{\text{t}}$$

F is the residual settlement starting from time t, z_t is the settlement at time t and z_{end} is the final settlement at the end of the calculation. Each different input value for the time t will yield a different limit state function.

All combinations of parameter values where the residual settlement equals the allowed value are together called the *Limit State Surface*.

The FORM procedure determines for each limit state function the most likely parameter combination on this surface (the *design point*), by iteratively calculating the probability of failure, using a linearization of Z.

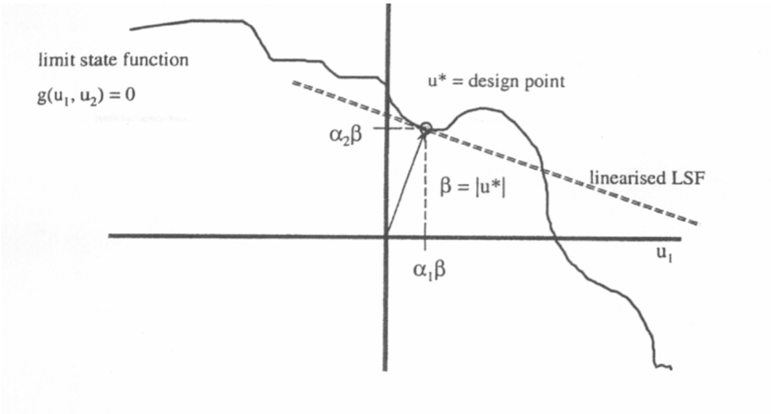


Figure 18-1 – FORM method

Output of a FORM analysis is the standard deviation of the residual settlement in the design point, together with the *reliability index* β.

(111) $$\beta = \frac{F_{\text{allowed}} - \mu[F]}{\sigma[F]}$$

where $\mu[F]$ defines the expected mean value and $\sigma[F]$ the standard deviation of the residual settlement. A large value of β implies a large probability that the allowed residual settlement will not be exceeded.

Crude Monte Carlo method for bandwidth of total and residual settlements
The Monto Carlo method is based on the execution of a large number of settlement predictions, using different parameter values that are generated from the initial or updated parameter distributions. These distributions are derived from the mean value and the matrix of covariances. The integration of all individual results yields the probability distribution of the settlements.

18.3 Horizontal Displacements

18.3.1 Principles of De Leeuw method

The De Leeuw method [Lit 24] estimates the horizontal displacements based on an elastic solution for a single elastic incompressible layer, characterized by the Young's modulus *E,* and loaded by a uniform load with a certain width. The solution assumes that the horizontal deformations of the elastic layer are always constrained at the bottom by a stiff foundation layer. Optionally the deformations can also be constrained by a stiff layer at the top.

The method considers the following two situations (Figure 18-2):
- I: elastic layer on a rigid base;
- II: elastic layer on a rigid base with a stiff layer on top.

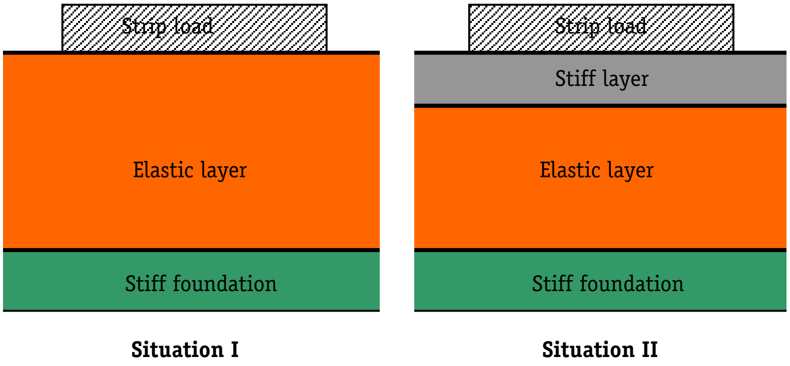


Figure 18-2 – Situations considered by De Leeuw method

NOTE: In case of an inputted embankment load, MSettle schematizes it as an equivalent uniform load with a certain width as illustrated in Figure 18-3.

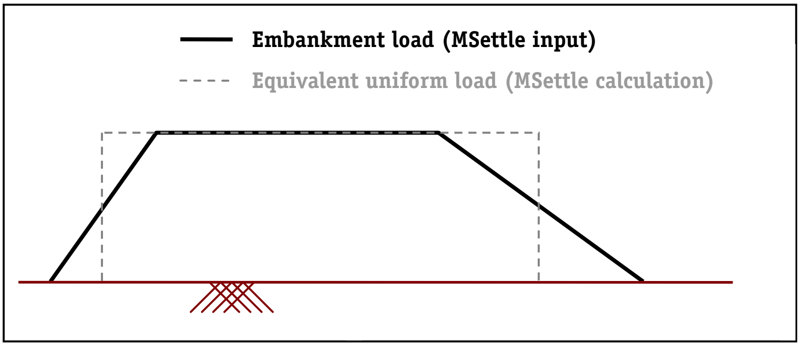


Figure 18-3 – Non-uniform load schematized as a uniform load

18.3.2 Limitations

The method has the following limitations:
- As Poisson ratio ν = 0.5 is used (i.e. incompressible layer), this gives the elastic response of the soil in an undrained situation, so in fact directly after applying

the load; additional horizontal deformations due to consolidation are not accounted for;

- The thickness of the stiff top layer is not taken into account.
- The horizontal distance of the considered vertical to the boundaries of the surcharge load is limited to 6 times the thickness of the elastic layer.

18.3.3 E-Modulus

The Young's modulus of the elastic layer can either be directly prescribed by the user or automatically estimated by MSettle from the average unit weight γ of the soft layers. MSettle determines the average unit weight γ_{avg} of several soft layers using the following formula:

(112) $$\gamma_{avg} = \frac{\sum_{i=1}^{n} \gamma_i \cdot h_i}{H}$$

where:

γ_i	Unit weight of elastic layer i
n	Number of elastic layers
h_i	Thickness of elastic layer i
H	Total thickness of the elastic layers

The elasticity modulus is then derived from the unit weight by linear interpolation in the table below, according to De Leeuw & Timmermans.

Table 18-1 – E-modulus vs. unit weight (De Leeuw & Timmermans)

γ [kN/m³]	E [kN/m²]
10	575
13	1000
18	1500
19	2800

The E-modulus can also be determined from compression parameters like C_p' and C_s', in combination with an assumption for the Poisson's ratio ν:

(113) $$E = \frac{\Delta\sigma'}{\left(\frac{1}{C_p'} + \frac{1}{C_s'}\log(t)\right)\ln\left(\frac{\sigma_0' + \Delta\sigma}{\sigma'}\right)} \cdot \frac{(1+\nu)(1-2\nu)}{1-\nu}$$

Introduction
Tutorial
Reference
Background
Verification

Embankment Design
and Soil Settlement
Prediction

Deltares

Delft
GeoSystems

19 Benchmarks introduction

Delft GeoSystems commitment to quality control and quality assurance has leaded them to develop a formal and extensive procedure to verify the correct working of all of their geotechnical engineering tools. An extensive range of benchmark checks have been developed to check the correct functioning of each tool. During product development these checks are run on a regular basis to verify the improved product. These benchmark checks are provided in the following sections, to allow the users to overview the checking procedure and verify for themselves the correct functioning of MSettle.

The benchmarks for Delft GeoSystems are subdivided into five separate groups as described below.

- **Group 1 [chapter 20] – Benchmarks from literature (exact solution)**
  Simple benchmarks for which an exact analytical result is available from literature.
- **Group 2 [chapter 21] – Benchmarks from literature (approximate solution)**
  More complex benchmarks described in literature for which an approximate solution is known.
- **Group 3 [chapter 22] – Benchmarks from spread sheets**
  Benchmarks which test program features specific to MSettle.
- **Group 4 [chapter 23] – Benchmarks generated by MSettle**
  Benchmarks for which the reference results are generated using MSettle.
- **Group 5 [chapter 24] – Benchmarks compared with other programs**
  Benchmarks for which the results of MSettle are compared with the results of other programs.

The number of benchmarks in group 1 will probably remain the same in the future. The reason for this is that they are very simple, using only the most basic features of MSettle.

The number of benchmarks in group 2 may grow in the future. The benchmarks in this chapter are well documented in literature. There are no exact solutions for these available problems; however in the literature estimated results are available. When verifying MSettle, the results should be close to the results found in the literature.

Groups 3, 4 and 5 of benchmarks will grow as new versions of MSettle are released. These benchmarks are designed in such a way that (new) features specific to MSettle can be verified. The benchmarks are kept as simple as possible so that, per benchmark, only one specific feature is verified.

As much as software developers would wish they could, it is impossible to prove the correctness of any non-trivial program. Re-calculating all the benchmarks in this report, and making sure the results are as they should be, will prove to some degree that the program works as it should. Nevertheless there will always be combinations of input values that will cause the program to crash or produce wrong results. Hopefully by using the verification procedure the number of times this occurs will be limited.

The benchmarks will all be described to such detail that reproduction is possible at any time. In some cases, when the geometry is too complex to describe, the input file of the benchmark is needed. The results are presented in text format with each benchmark description.

The input files belonging to the benchmarks can be found on CD-ROM or can be downloaded from our website www.delftgeosystems.com.

20 Benchmarks from literature (exact solution)

This chapter describes a number of benchmarks for which an exact analytical solution can be found in the literature.

20.1 Stress distribution acc. Buisman

Description

The load distribution in an elastic half space with a stiffness which increases with depth is calculated by Fröhlich in [Lit 21].

Benchmark

According to [Lit 21] page 426, a point load (4π kN) on an elastic half space leads to a stress increase at 2 m depth under the load of 2 kPa.

MSettle result

The point load is modeled as a circular load with radius R = 0.01 m and magnitude P = 40000 kPa. This leads to a total force $F = \pi R^2 P = 4\pi$ kN.

Table 20-1 – Results of benchmark 1-1 – Increase of stress distribution under point load acc. to Buisman/Fröhlich

Co-ordinates [m]	Benchmark [kPa]	MSettle [kPa]	Relative error [%]
X = 0.0, Y = -2.0	2.0	2.0	0.00

Use MSettle input file bm1-1.sli to run this benchmark.

20.2 Strip-load at surface (acc. to Flamant)

Description
The load distribution in an elastic half space (with a constant stiffness with depth) is calculated by Flamant in [Lit 21].

Benchmark
According to [Lit 21] page 426, a loaded strip (width $2a$ = 2 m, load = 1 kPa) on an elastic half space leads to a stress distribution in x-direction at 1 m depth.

MSettle result
The point load is modeled as a trapezoidal load with width X_m = 2.0 m. The left and right parts have zero length. The magnitude is defined by unit weight P = 1 kN/m^3 and height H = 1 m. The calculation method is chosen to be according to Boussinesq.

Table 20-2 – Results of benchmark 1-2 – Increase of stress distribution under line-load acc. to Boussinesq/Flamant

Co-ordinates [m]	Benchmark [kPa]	MSettle [kPa]	Relative error [%]
X = 0.00, Y = -1.00	0.818	0.818	0.00
X = 1.00, Y = -1.00	0.480	0.480	0.00

Use MSettle input file bm1-2.sli to run this benchmark.

20.3 Settlement acc. to Terzaghi (no secondary compression)

Description
The final settlement of a cubic element of soil is calculated in [Lit 21]. The deformation behavior of the soil is according to NEN-Koppejan. No secondary compression occurs. Due to the loading of the soil and its initial state the pre-consolidation stress must be taken into account.

Benchmark
In [Lit 21] page 427, the settlements for loading under the pre-consolidation stress and above the pre-consolidation stress are calculated. Since NEN-Koppejan rule is not consistent for the number of layers, the number of layers is prescribed to be 10.

MSettle result
The secondary compression cannot be switched off. The influence of secondary compression is reduced by choosing very large secular compression coefficients. The results are not influenced by secondary compression any more.

Table 20-3 – Results of benchmark 1-3 – Settlement according to NEN-Koppejan without secondary compression

	Benchmark [mm]	MSettle [mm]	Relative error [%]
Total settlement	2.6	2.7	3.70

Use MSettle input file bm1-3.sli to run this benchmark.

20.4 Settlement acc. to NEN-Koppejan (with secondary compression)

Description
The time dependant settlement of a cubic element of soil is calculated in [Lit 21]. The deformation behavior of the soil is according to NEN-Koppejan. Secondary compression occurs. Due to the loading of the soil and its initial state the pre-consolidation stress must be taken into account.

Benchmark
In [Lit 21] page 429, the settlements for loading under the pre-consolidation stress and above the pre-consolidation stress are calculated. Since NEN-Koppejan rule is not consistent for the number of layers, the number of layers is prescribed to be 10.

MSettle result
MSettle results are found in the *Report* window.

Table 20-4 – Results of benchmark 1-4 – Settlement according to NEN-Koppejan, with secondary compression

		Benchmark [mm]	MSettle [mm]	Relative error [%]
10 days	primary	2.6	2.7	3.70
	secondary	0.7	0.7	0.00
10^4 days	total	5.4	5.5	1.82

Use MSettle input file bm1-4.sli to run this benchmark.

20.5 One-dimensional consolidation

Description
A cubic soil element is loaded and the one-dimensional consolidation is calculated in [Lit 21]. The outflow of water is possible at both the top and the bottom of the sample. The soil stiffness is independent of the effective stress.

Benchmark
In [Lit 21] page 429, the consolidation is expressed as the degree of consolidation as a function of time. The degree of consolidation is the actual settlement divided by the settlement which will be reached after infinite time.

MSettle result
The time dependency in the material behavior according to NEN-Koppejan is switched off by choosing high numbers for secondary compression. The pre-consolidation stress is also chosen above the maximum stress in the soil.
MSettle results are found in the *Part of final settlement* column of the *Residual Times* table in the *Report* window.

Table 20-5 – Results of benchmark 1-5 – Degree of consolidation

Time [days]	Benchmark [%]	MSettle [%]	Relative error [%]
1	46.89	46.80	0.19
10	98.86	98.77	0.09

Use MSettle input file bm1-5.sli to run this benchmark.

20.6 Stress distribution under the corner of a rectangular load (acc. to Buisman)

Description
A layer is loaded by a rectangular load (magnitude: q = 35 kN/m²; length L = 6 m; width B = 3 m). The change in vertical stress due to this rectangular load is calculated using an equation from literature.

Benchmark
The integration of the stress distribution equation under a uniformly loaded rectangular area according to Buisman has been solved in [Lit 22]. The change in vertical stress is given by the following equation:

$$\Delta\sigma_y = \frac{q}{4\pi}\left[\frac{B\ L\ y^2\left(B^2+L^2+2y^2\right)}{\left(B^2+y^2\right)\left(L^2+y^2\right)\left(B^2+L^2+y^2\right)} + \frac{B\left(2B^2+3y^2\right)}{\left(B^2+y^2\right)^{3/2}}\arctan\left(\frac{L}{\sqrt{B^2+y^2}}\right) + \frac{L\left(2L^2+3y^2\right)}{\left(L^2+y^2\right)^{3/2}}\arctan\left(\frac{B}{\sqrt{L^2+y^2}}\right)\right] \tag{114}$$

The change in vertical stress is calculated at different depths (see results in Table 20-6).

MSettle result

The changes in vertical stress are compared with the benchmark results in Table 20-6.

Table 20-6 – Results of benchmark 1-6 – Change in vertical effective stress under the corner of a rectangular load acc. to Buisman

Depth [m…GL]	Benchmark [kPa]	MSettle [kPa]			Relative error [%]
	$\Delta\sigma'$	$\sigma'_{initial}$	σ'_{final}	$\Delta\sigma'$	$\Delta\sigma'$
0	25.00	0.00	25.00	25.00	0.00
-5	16.70	28.75	45.45	16.70	0.00
-10	7.93	53.75	61.68	7.93	0.00
-12	6.08	63.75	69.83	6.08	0.00
-14	4.76	73.75	78.51	4.76	0.00
-16	3.81	83.75	87.56	3.81	0.00
-18	3.11	93.75	96.86	3.11	0.00
-20	2.58	103.75	106.33	2.58	0.00

Use MSettle input file bm1-6.sli to run this benchmark.

20.7 Stress distribution due to a triangular strip load (acc. to Boussinesq)

Description

A layer is loaded by a triangular load (unit weight: $\gamma = 20$ kN/m^3; maximal height $H = 4$ m; width $B = 40$ m). The change in vertical stress due to this triangular load is checked using an equation from literature that integrates Boussinesq theory.

Benchmark

The integration of the stress distribution equation under a vertical loading increasing linearly according to Boussinesq has been solved in [Lit 22]. The change in vertical stress is given by equation 3.4a page 38 of [Lit 22]:

$$(115) \qquad \Delta\sigma_z = \frac{p}{2\pi}\left[\frac{x}{b}\,\alpha - \sin(2\delta)\right]$$

The definition of parameters b, p, α, δ, x and z is given in Figure 20-1.
Parameter p is the maximal load magnitude: $p = \gamma \times H = 20 \times 4 = 80$ kN/m^2.
Parameter b is half the load width: $b = B / 2 = 20$ m.

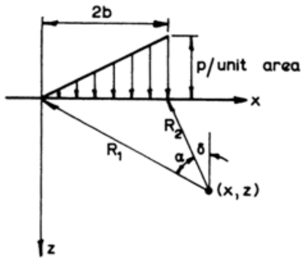


Figure 20-1 – Definition of parameters b, p, α, δ, x and z (Fig. 3.4 of [Lit 22])

The change in vertical stress at 25 m depth is calculated at 7 locations (see co-ordinates and results in Table 20-7).

MSettle result

The Boussinesq soil stress distribution in the *Calculation Option* window must be chosen. The triangular load is inputted in MSettle using a trapeziform load (bm1-7a) or a non-uniform load (bm1-7b). The changes in vertical stress are compared with the benchmark results in Table 20-7.

Table 20-7 – Results of benchmark 1-7 – Change in vertical effective stress at 25 m depth acc. to Boussinesq

X co-ordinate [m]	Benchmark [kPa] $\Delta\sigma'$	MSettle [kPa] $\sigma'_{initial}$	σ'_{final}	$\Delta\sigma'$	Relative error [%] $\Delta\sigma'$
-10	5.56	128.75	134.31	5.56	0.00
0	11.44	128.75	140.19	11.44	0.00
10	20.52	128.75	149.27	20.52	0.00
20	29.60	128.75	158.35	29.60	0.00
30	32.78	128.75	161.53	32.78	0.00
40	25.78	128.75	154.53	25.78	0.00
50	14.35	128.75	143.10	14.35	0.00

Use MSettle input files bm1-7a.sli and bm1-7b.sli to run this benchmark.

20.8 Stress distribution due to asymmetrical triangular strip load (acc. to Boussinesq)

Description

A layer is loaded by an asymmetrical triangular load (unit weight: $\gamma = 20$ kN/m^3; maximal height $H = 4$ m; width left side $B_1 = 30$ m; width right side $B_2 = 10$ m). The change in vertical stress due to this asymmetrical triangular load is checked using an equation from literature that integrates Boussinesq theory.

Benchmark

The integration of the stress distribution equation under a asymmetrical vertical triangular loading according to Boussinesq has been solved in [Lit 22]. The change in vertical stress is given by equation 3.8a page 40 of [Lit 22]:

(116) $$\Delta\sigma_z = \frac{p}{\pi}\left[\frac{x}{a}\alpha + \frac{a+b-x}{b}\beta\right]$$

The definition of parameters a, b, p, α, β, x and z is given in Figure 20-2. Parameter p is the maximal load magnitude: $p = \gamma \times H = 20 \times 4 = 80$ kN/m^2. Parameters a and b are indeed B_1 and B_2 respectively (i.e. 30 m and 10 m).

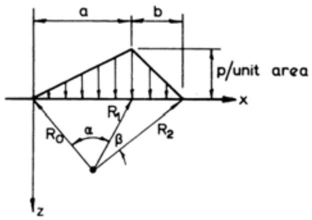


Figure 20-2 – Definition of parameters a, b, p, α, β, x and z

The change in vertical stress at 25 m depth is calculated at 7 locations; see the co-ordinates and the results in Table 20-8.

MSettle result

The Boussinesq soil stress distribution in the *Calculation Option* window must be chosen. The triangular load is inputted in MSettle using the *Other Loads* window (trapeziform) (i.e. bm1-8a) or the *Non-Uniform Loads* window (i.e. bm1-8b). The changes in vertical stress are compared with the benchmark results in Table 20-8.

Table 20-8 – Results of benchmark 1-8 – Change in vertical effective stress at 25 m depth acc. to Boussinesq

X co-ordinate [m]	Benchmark [kPa]	MSettle [kPa]			Relative error [%]
	$\Delta\sigma'$	$\sigma'_{initial}$	σ'_{final}	$\Delta\sigma'$	$\Delta\sigma'$
-10	6.73	128.75	135.48	6.73	0.00
0	13.87	128.75	142.62	13.87	0.00
10	24.34	128.75	153.09	24.34	0.00
20	32.90	128.75	161.65	32.90	0.00
30	32.00	128.75	160.75	32.00	0.00
40	21.45	128.75	150.20	21.45	0.00
50	10.86	128.75	139.61	10.86	0.00

Use MSettle input files bm1-8a.sli and bm1-8b.sli to run this benchmark.

20.9 Stress distribution due to an "embankment" loading (acc. to Boussinesq)

Description

A layer is loaded by an "embankment" loading (unit weight: $\gamma = 20$ kN/m^3; maximal height H = 4 m; width left side B_1 = 10 m; width right side B_2 = 30 m). The change in vertical stress due to this asymmetrical triangular load is checked using an equation from literature that integrates Boussinesq theory.

Benchmark

The integration of the stress distribution equation under a vertical "embankment" loading according to Boussinesq has been solved in [Lit 22]. The change in vertical stress is given by equation 3.9a page 40 of [Lit 22]:

(117) $$\Delta\sigma_z = \frac{p}{\pi}\left[\beta + \frac{x}{a}\alpha - \frac{z}{R_2^{\,2}}(x-b)\right]$$

The definition of parameters a, b, p, α, β, x and z is given in Figure 20-3. Parameter p is the maximal load magnitude: $p = \gamma \times H = 20 \times 4 = 80$ kN/m^2. Parameters a and b are indeed B_1 and $B_1 + B_2$ respectively (i.e. 10 m and 40 m).

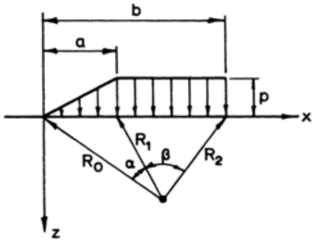


Figure 20-3 – Definition of parameters a, b, p, α, β, x and z

The change in vertical stress at 25 m depth is calculated at 7 locations; see the co-ordinates and the results in Table 20-9.

MSettle result

The Boussinesq soil stress distribution in the *Calculation Option* window must be chosen. The triangular load is inputted in MSettle using the *Other Loads* window (trapeziform) (i.e. bm1-9a) or the *Non-Uniform Loads* window (i.e. bm1-9b). The changes in vertical stress are compared with the benchmark results in Table 20-9.

Table 20-9 – Results of benchmark 1-9 – Change in vertical effective stress at 25 m depth acc. to Boussinesq

X co-ordinate [m]	Benchmark [kPa]	MSettle [kPa]			Relative error [%]
	$\Delta\sigma'$	$\sigma'_{initial}$	σ'_{final}	$\Delta\sigma'$	$\Delta\sigma'$
-10	13.70	128.75	142.45	13.70	0.00
0	27.53	128.75	156.28	27.53	0.00
10	44.52	128.75	173.27	44.52	0.00
20	54.28	128.75	183.03	54.28	0.00
30	51.03	128.75	179.78	51.03	0.00
40	36.18	128.75	164.93	36.18	0.00
50	19.39	128.75	148.14	19.39	0.00

Use MSettle input files bm1-9a.sli and bm1-9b.sli to run this benchmark.

20.10 Stress distribution due to circular load (acc. to Buisman)

Description

A layer is loaded by a uniform circular loading (magnitude: q = 20 kN/m^2; radius R = 10 m). The change in vertical stress under the center of this circular load is checked using equation from literature.

Benchmark

The integration of the stress distribution equation under the center of a circular load according to Buisman has been solved in [Lit 22]. The change in vertical stress is given by the following equation:

(118) $$\Delta\sigma_y = q\left(1 - \frac{y^4}{R^2 + y^2}\right)$$

The change in vertical stress is calculated at different depths. Results are given in Table 20-10.

MSettle result

The changes in vertical stress are compared with the benchmark results in Table 20-10.

Table 20-10 – Results of benchmark 1-10 – Change in vertical effective stress under the center of a circular load acc. to Buisman

Depth [m]	Benchmark [kPa]	MSettle [kPa]			Relative error [%]
	$\Delta\sigma'$	$\sigma'_{initial}$	σ'_{final}	$\Delta\sigma'$	$\Delta\sigma'$
0	20.00	0.00	20.00	20.00	0.00
-5	19.93	28.75	48.68	19.93	0.00
-10	19.20	53.75	72.95	19.20	0.00
-12	18.60	63.75	82.35	18.60	0.00
-14	17.84	73.75	91.59	17.84	0.00
-16	16.95	83.75	100.70	16.95	0.00
-18	15.99	93.75	109.74	15.99	0.00
-20	15.00	103.75	118.75	15.00	0.00

Use MSettle input files bm1-10.sli to run this benchmark.

21 Benchmarks from literature (approximate solution)

The benchmarks in this chapter have no exact analytical solution, but are documented in literature and therefore approximate solutions are available.

21.1 Stress distribution due to uniform strip load acc. to Boussinesq

Description

A layered half space is loaded by a strip-load (width 20 m, load 35 kPa). The stress distribution in the half space is calculated using the model of Boussinesq with a column width of 0.5 m.

Benchmark

In [Lit 21] page 443, the vertical stress at 20 m depth is calculated at 7 locations (see the co-ordinates in Table 21-1).

MSettle result

The Boussinesq soil stress distribution in the *Calculation Option* window must be chosen. The strip-load is inputted in MSettle using a trapeziform load (bm2-1a) or a non-uniform load (bm2-1b). The final effective stresses are compared with the benchmark results in Table 21-1. These are independent of the consolidation coefficient.

Table 21-1 – Results of benchmark 2-1 – Distribution of vertical effective stress at 20 m depth acc. to Boussinesq

X co-ordinate [m]	Benchmark [kPa]	MSettle [kPa]	Relative error [%]
0	115.990	115.990	0.00
10	116.217	116.217	0.00
20	116.761	116.761	0.00
30	118.220	118.220	0.00
40	122.219	122.219	0.00
50	130.070	130.070	0.00
60	134.994	134.994	0.00

Use MSettle input files bm2-1a.sli and bm2-1b.sli to run this benchmark.

21.2 Stress distribution due to uniform strip load acc. to Buisman

Description
A layered half space is loaded by a strip-load (width 20 m, load 35 kPa). The stress distribution in the half space is calculated using the model of Buisman. This problem is identical to the problem discussed in the previous section, only the stress distribution is according to Buisman (instead of Boussinesq).

Benchmark
In [Lit 21] page 443, the vertical stress at 20 m depth is calculated at 7 locations (see the co-ordinates in Table 21-2).

MSettle result
The Buisman soil stress distribution in the *Calculation Options* window must be chosen. The strip-load is inputted in MSettle using a trapeziform load (bm2-2a) or a non-uniform load (bm2-2b). The final effective stresses are compared with the benchmark results in Table 21-2. These are independent of the consolidation coefficient.

Table 21-2 – Results of benchmark 2-2 – Distribution of vertical effective stress at 20 m depth acc. to Buisman

X co-ordinate [m]	Benchmark [kPa]	MSettle [kPa]	Relative error [%]
0	115.843	115.843	0.00
10	115.964	115.964	0.00
20	116.318	116.318	0.00
30	117.500	117.500	0.00
40	121.594	121.594	0.00
50	131.217	131.218	0.00
60	137.663	137.663	0.00

Use MSettle input files bm2-2a.sli and bm2-2b.sli to run this benchmark.

21.3 Settlement acc. to NEN-Koppejan (creep)

Description
A layered half space is loaded by a uniform load of 35 kPa. The time dependant settlement of this (one-dimensional) problem is calculated. Full consolidation is assumed. The settlement due to primary and secondary compression is calculated.

Benchmark
In [Lit 21] page 444, the settlement of the surface is calculated after 1, 10, 100, 1000 and 10000 days. The settlements due to loading under and above the pre-consolidation stress are distinguished. The settlements due to primary and secondary compression are distinguished.

MSettle result
The primary settlement at 1 day, the primary and secondary settlements after 10 days and the total settlement after 10000 days are printed by MSettle. The settlements at 100 and 1000 days are calculated using the *Calculation Times* window.

Table 21-3 – Results of benchmark 2-3 – Surface settlement acc. to NEN-Koppejan

Time [day]		Benchmark [m]	MSettle [m]	Relative error [%]
1	total	1.198	1.197	0.08
10	primary	1.198	1.197	0.08
	secondary	0.195	0.195	0.00
	total	1.393	1.392	0.07
100	total	1.588	1.588	0.00
1000	total	1.783	1.783	0.00
10000	total	1.979	1.978	0.05

Use MSettle input file bm2-3.sli to run this benchmark.

21.4 One-dimensional consolidation

Description
A layered half space is loaded by a uniform load of 35 kPa. The time dependant settlement of this (one-dimensional) problem is calculated. The settlement due to primary compression, secondary compression and consolidation is calculated.

Benchmark
In [Lit 21] page 444, the settlement of the surface is calculated after 10, 100, 1000 and 10000 days.

MSettle result
The total settlement after 10, 100, 1000 and 10000 days, consolidation included, are determined in MSettle using the *Calculation Times* window.

Table 21-4 – Results of benchmark 2-4 – Settlement with consolidation

Time [days]	Benchmark [m]	MSettle [m]	Relative error [%]
10	0.232	0.240	3.33
100	0.508	0.509	0.20
1000	0.958	0.959	0.10
10000	1.831	1.830	0.05

Use MSettle input file bm2-4.sli to run this benchmark.

21.5 Total settlement acc. to NEN-Koppejan

Description
A layered half space is loaded by a non-uniform load with a dry weight of 17.5 kN/m^3 and a wet weight of 20 kN/m^3. The height of the load is 2 m. The total settlement of this (one-dimensional) problem is calculated with and without submerging taken into account.

Benchmark
In [Lit 21] page 443, the total settlement of the surface is calculated (10000 days, 100% consolidation) with and without submerging taken into account.

MSettle result
The total settlements are compared with the benchmark results in Table 21-5.

Table 21-5 – Results of benchmark 2-5 – Total settlement (100% consolidation) after 10000 days

Submerging	File name	Benchmark [m]	MSettle [m]	Relative error [%]
OFF	bm2-5a	1.951	1.951	0.00
ON	bm2-5b	1.409	1.408	0.07

Use MSettle input files bm2-5a.sli and bm2-5b.sli to run this benchmark.

22 Benchmarks from spread sheets

The benchmarks in this chapter test program features specific to MSettle using spread sheets as the solution is often complex.

22.1 Settlements acc. to NEN-Koppejan model during loading and un/re-loading steps (drained layer)

Description

An oedometer test with loading and unloading steps is performed for both Terzaghi and Darcy consolidation models in combination with NEN-Koppejan parameters. The layer is drained to avoid any consolidation process. MSettle results are compared to an analytical solution (without consolidation) worked out in an Excel spreadsheet.

Benchmark

A saturated clay layer (H_0 = 20 mm and γ_{sat} = 18 kN/m^3) is loaded with the loading steps given in Table 22-1. The compression and swelling coefficients are: C_p = 50, C_p' = 12.5, C_s = 300, C_s' = 75, A_p = 30 and A_s = 150. The creep rate reference time is t_0 = 4 days.

Three types of variables are used to simulate the initial pre-consolidation process:

- pre-consolidation pressure: σ_p = 8 kPa (bm3-1a, bm3-1b, bm3-1e and bm3-1f)
- over-consolidation ratio: OCR = 1.2 (bm3-1c and bm3-1g)
- pre-overburden pressure: POP = 5 kPa (bm3-1d and bm3-1h)

The pre-consolidation process is set variable within the layer and corrected at every step which writes:

$$(119) \qquad P_{c;i} = \max\left(P_{c;i-1}; \sigma'_{i-1}\right) \quad \text{with} \quad P_{c;1} = \begin{cases} \sigma_p & \text{(for benchmarks a, b, e and f)} \\ OCR\, \sigma'_0 & \text{(for benchmarks c and g)} \\ POP + \sigma'_0 & \text{(for benchmarks d and h)} \end{cases}$$

The phreatic/piezometric line is situated 20 mm above the layer.

Table 22-1 – Loading-steps (bm3-1)

Load-step i	Application time t_i [days]	Loading/Unloading $\Delta\sigma_i$ [kPa]	Cumulative load σ_i [kPa]
0	Initial state	2	Initial load
1	0	5	5
2	1	-5	0
3	2	5	5
4	3	5	10
5	4	-5	5
6	5	5	10
7	6	10	20
8	7	20	40

An initial load of 2 kPa and a layer thickness of only 20 mm permit to assume a constant initial effective stress distribution along the layer (σ_0' = 0.28 kPa).
The calculation without consolidation yields the analytical solution given by:

- equations (59) to (62) page 305 [§ 16.3.1] for loading steps;
- equation (63) page 306 [§ 16.3.2] for unloading steps.

Settlement calculations are performed using both linear and natural strains. For natural strain, equation (64) page 306 [§ 16.3.3] applies.

MSettle result

The settlements calculated by MSettle are exported to the spread sheet using the *View Data* option in *Time-History* window for comparison (see figures below). The settlements after 4 and 8 days are given in Table 22-2.

Table 22-2 – Results of benchmark 3-1 – Settlements acc. to NEN-Koppejan model (for different cases)

Case	Model	Type	Strain	Time	Benchmark	MSettle		Error
				[days]	[mm]	File	[mm]	[%]
A	Terzaghi	Pc	Linear	4	0.97	bm3-1a	0.97	0.00
				8	3.03		3.03	0.00
B		Pc	Natural	4	0.94	bm3-1b	0.94	0.00
				8	2.81		2.81	0.00
C		OCR	Linear	4	4.94	bm3-1c	4.96	0.40
				8	7.13		7.14	0.14
D		POP	Linear	4	1.47	bm3-1d	1.47	0.00
				8	3.56		3.56	0.00
E	Darcy	Pc	Linear	4	0.97	bm3-1e	0.97	0.00
				8	3.03		3.03	0.00
F		Pc	Natural	4	0.94	bm3-1f	0.94	0.00
				8	2.81		2.81	0.00
G		OCR	Linear	4	4.94	bm3-1g	4.96	0.40
				8	7.13		7.14	0.14
H		POP	Linear	4	1.47	bm3-1h	1.47	0.00
				8	3.56		3.56	0.00

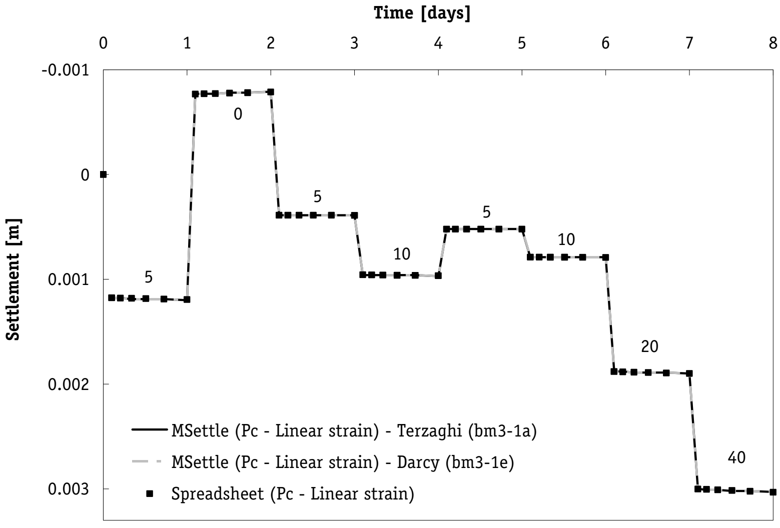


Figure 22-1 – Comparison between MSettle and the spreadsheet results Pc compression and linear strain

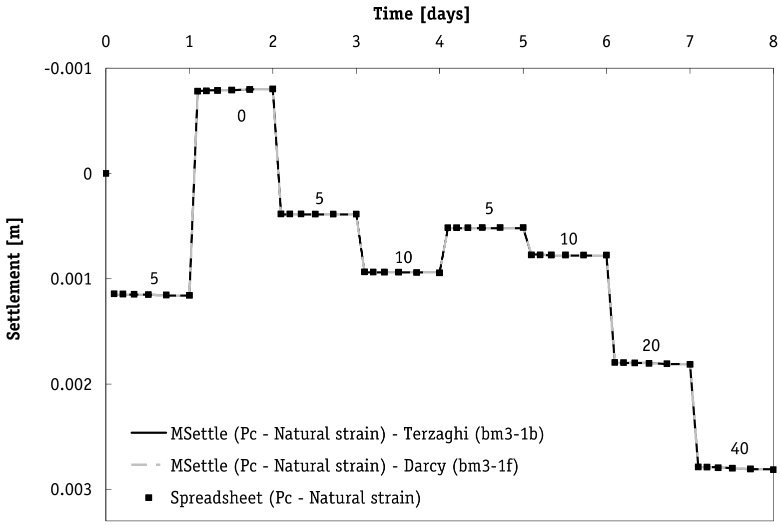


Figure 22-2 – Comparison between MSettle and the spreadsheet results Pc compression and natural strain

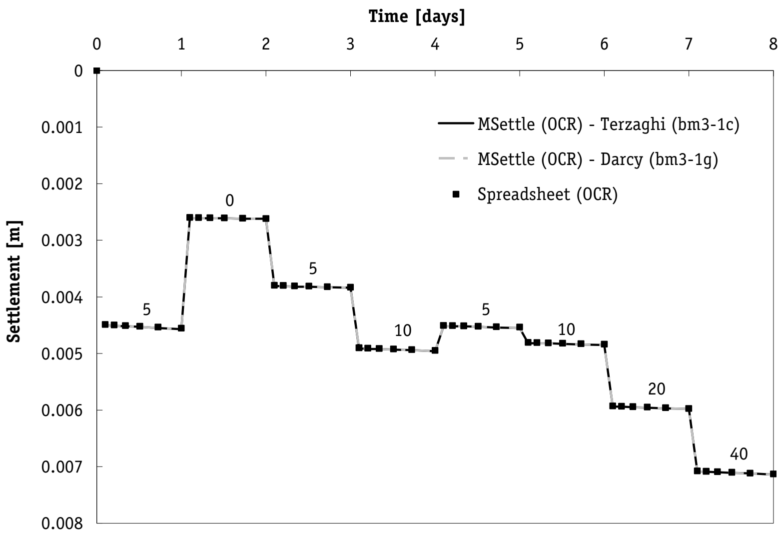


Figure 22-3 – Comparison between MSettle and the spreadsheet results for OCR compression

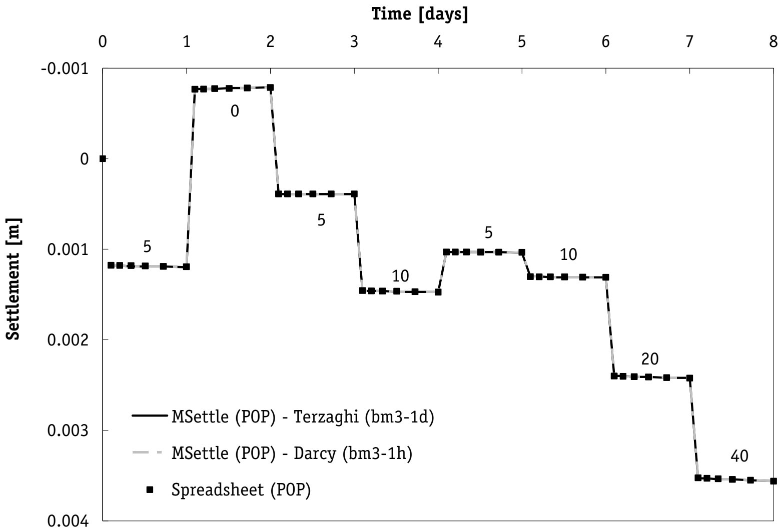


Figure 22-4 – Comparison between MSettle and the spreadsheet results for POP compression

Use MSettle input files bm3-1a.sli till bm3-1h.sli to run this benchmark.

22.2 Settlements acc. to Isotache model during loading and un/re-loading steps (drained layer)

Description

The same oedometer test as benchmark 3-1 [§ 22.1] is performed using the Isotache model instead of the NEN-Koppejan model.

Benchmark

The same input values as benchmark 3-1 [§ 22.1] are used except for the Isotache parameters which are: a = 0.01, b = 0.1 and c = 0.04. Four types of variables are used to simulate the pre-consolidation process:

- preconsolidation pressure: σ_p = 8 kPa (bm3-2a and e)
- pre-overburden pressure: *POP* = 5 kPa (bm3-2b and f)
- over-consolidation ratio: *OCR* = 1.2 (bm3-2c and g)
- equivalent age: t_{age} = 10 days (bm3-2d and h).

The calculation without consolidation yields the analytical solution given by equation (58) page 303 [§ 16.2.2]. Settlements deduced from natural strain are equal to: $s(t) = h_0 \cdot [1 - \exp(-\varepsilon(t))]$.

MSettle result

The settlements calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison. The settlements after 3 and 8 days are given in Table 22-3.

Table 22-3 – Results of benchmark 3-2 – Settlements acc. to Isotache model (for different cases)

Case	Model	Type	Time	Benchmark	MSettle		Error
			[days]	[mm]	File	[mm]	[%]
A	Terzaghi	Pc	3	0.72	bm3-2a	0.72	0.00
			8	2.86		2.86	0.00
B		POP	3	0.89	bm3-2b	0.89	0.00
			8	3.46		3.46	0.00
C		OCR	3	4.42	bm3-2c	4.43	0.23
			8	7.08		7.08	0.00
D		Eq.	3	4.10	bm3-2e	4.12	0.49
		age	8	6.81		6.82	0.15
E	Darcy	Pc	3	0.72	bm3-2f	0.72	0.00
			8	2.86		2.86	0.00
F		POP	3	0.89	bm3-2g	0.89	0.00
			8	3.46		3.46	0.00
G		OCR	3	4.42	bm3-2h	4.43	0.23
			8	7.08		7.08	0.00
H		Eq.	3	4.10	bm3-2j	4.12	0.49
		age	8	6.81		6.82	0.15

Use MSettle input files bm3-2a.sli till bm3-2h to run this benchmark.

22.3 Settlements acc. to NEN-Bjerrum model during loading and un/re-loading steps (drained layer)

Description

The same oedometer test as benchmark 3-1 [§ 22.1] is performed using the NEN-Bjerrum model instead of the NEN-Koppejan model.

Benchmark

The same input values as benchmark 3-1 [§ 22.1] are used except for the NEN-Bjerrum parameters which are:

- Ratio: RR = 0.022, CR = 0.22 and C_α = 0.01 (cases b, d, f and h)
- Index: C_r = 0.008, C_c = 0.12, C_α = 0.01 and e_0 = 0.15 (cases a, c, e and g).

Four types of variables are used to simulate the pre-consolidation process:

- preconsolidation pressure: σ_p = 8 kPa (bm3-3a and e)
- pre-overburden pressure: POP = 5 kPa (bm3-3b and f)

- over-consolidation ratio: OCR = 1.2 (bm3-3c and g)
- equivalent age: t_{age} = 10 days (bm3-3d and h).

The calculation without consolidation yields the analytical solution given by equation (43) page 299 [§ 16.1.2]. Settlements deduced from linear strain are equal to: $s(t) = H_0 \cdot \varepsilon(t)$.

MSettle result
The settlements calculated by MSettle are exported to the spread sheet using the *View Data* option in *Time-History* window for comparison. The settlements after 3 and 8 days are given in Table 22-4.

Table 22-4 – Results of benchmark 3-3 – Settlements acc. to NEN-Bjerrum model (for different cases)

Case	Model	Type	Time	Benchmark	MSettle		Error
			[days]	[mm]	File	[mm]	[%]
A	Terzaghi	Pc	3	0.18	bm3-3a	0.18	0.00
			8	1.55		1.55	0.00
B		POP	3	0.60	bm3-3b	0.60	0.00
			8	4.32		4.32	0.00
C		OCR	3	2.45	bm3-3c	2.46	0.41
			8	4.23		4.24	0.24
D		Eq. age	3	5.47	bm3-3e	5.48	0.18
			8	9.29		9.31	0.21
E	Darcy	Pc	3	0.18	bm3-3f	0.18	0.00
			8	1.55		1.55	0.00
F		POP	3	0.60	bm3-3g	0.60	0.00
			8	4.32		4.32	0.00
G		OCR	3	2.45	bm3-3h	2.46	0.41
			8	4.23		4.24	0.24
H		Eq. age	3	5.47	bm3-3j	5.48	0.18
			8	9.29		9.31	0.21

Use MSettle input files bm3-3a.sli till bm3-3h to run this benchmark.

22.4 Settlements using submerging option

Description
The submerging modeling in MSettle depends on the consolidation model:

- For Terzaghi consolidation model and for the combination Darcy/NEN-Koppejan, MSettle determines the submerged weight of non-uniform loads only on the basis of final settlements for all load columns [§ 13.7.1].

- For Darcy consolidation in combination with Isotache or NEN-Bjerrum soil model, MSettle determines the submerged weight of non-unifrom loads and soils on the basis of the settled surface level extrapolated from the two previous time-steps [§ 13.7.2].

Therefore, the submerging option is checked for six cases, A to F (i.e. six combinations) of soil and consolidation models), as shown in Table 22-6.
For both consolidation models, the stop criterion is set to 0.01 m. For Darcy model, the number of iteration steps is set to 1.
A layered half space with a phreatic line at –0.1 m is loaded by an initial load, and then 2 loading steps and finally an unloading step (see details in Table 22-5). A high initial load of 0.2 × 100 = 20 kPa permits to assume a constant initial effective stress distribution (σ_0' = 21.375 kPa).

Table 22-5 – Non-uniform loads (bm3-4)

Load-step i	Time t_i [days]	Height h_i [m]	Level Y_i [m NAP]	Unit weight [kN/m³] Unsaturated $\gamma_{unsat;i}$	Saturated $\gamma_{sat;i}$
0	initial	0.2	0	100	30
1	0	0.2	0.2	100	80
2	100	0.3	0.4	70	50
3	2000	-0.3	0.7	70	50

Benchmark
For accurate submerging model (cases D and F), each time-step is considered as a new load-step with an effective unit weight for non-uniform loads and soil layers that decreases according to equation (7) page 275 [§ 13.7.2].
The submerging effect can be seen in Figure 22-5 depending on the settlement $\Delta s(t)$:

- Part A: $\Delta s < Y_0 - Y_W$ (= 0.1 m)
  The initial load and the first load are dry:
  $\sigma_A = h_0 \times \gamma_{unsat;0} + h_1 \times \gamma_{unsat;1} = 0.2 \times 100 + 0.2 \times 100 = 40$ kPa
- Part B: $Y_0 - Y_W$ (0.1 m) $\leq \Delta s < Y_0 - Y_W + h_0$ (0.3 m)
  The initial load is partly submerged and the first load is dry:
  $\sigma_B = (Z_0 - Z_W - \Delta s + h_0) \times \gamma_{unsat;0} + (Y_0 - Y_W - \Delta s) \times (\gamma_{sat;0} - \gamma_w) + h_1 \times \gamma_{unsat;1}$
- Part C: $Y_0 - Y_W + h_0$ (0.3 m) $\leq \Delta s < Y_0 - Y_W + h_0 + h_1$ (0.5 m)
  The initial load is completely saturated and the first load is partly submerged:
  $\sigma_C = h_0 \times (\gamma_{sat;0} - \gamma_w) + (Y_0 - Y_W - \Delta s + h_0 + h_1) \times \gamma_{unsat;1} + (Y_0 - Y_W - \Delta s + h_0) \times (\gamma_{sat;1} - \gamma_w)$
- Part D: $Y_0 - Y_W + h_0 + h_1$ (0.5 m) $\leq \Delta s$ and $t \leq 100$ days
  Both initial load and first load are completely submerged:
  $\sigma_D = h_0 \times (\gamma_{sat;0} - \gamma_w) + h_1 \times (\gamma_{sat;1} - \gamma_w) = 0.2 \times (30 - 10) + 0.2 \times (80 - 10) = 18$ kPa
- Part E: $\Delta s < Y_0 - Y_W + h_0 + h_1 + h_2$ (0.8 m) and $100 < t \leq 2000$ days
  Both initial load and first load are completely submerged and the second load is partly submerged:
  $\sigma_E = \sigma_D + (Y_0 - Y_W - \Delta s + h_0 + h_1 + h_2) \times \gamma_{unsat;2} + (Y_0 - Y_W - \Delta s + h_0 + h_1) \times (\gamma_{sat;2} - \gamma_w)$

- Part F: $Y_0 - Y_W + h_0 + h_1 + h_2$ (0.8 m) $\leq \Delta s$ and $100 < t \leq 2000$ days
  All loads are completely submerged:
  $\sigma_F = \sigma_D + h_2 \times (\gamma_{sat;2} - \gamma_w) = 18 + 0.3 \times (50 - 10) = 30$ kPa
- Part G: $t > 2000$ days
  The second load is removed (i.e. part D):
  $\sigma_G = \sigma_D = 18$ kPa.

For approximate submerging model (cases A, B, C and E), the submerged weight of non-uniform loads is determined on the basis of final settlements for all load columns. Because of the deformation-dependent weight, these settlements are determined iteratively. The process is stopped when the average settlement increment in a particular iteration is less than the stop criterion.

Whatever the submerging model, the settlements are given by:
- equation (58) page 303 [§ 16.2.2] for Isoatche model;
- equation (43) page 299 [§ 16.1.2] for NEN-Bjerrum model;
- equations (59) to (63) page 305 [§ 16.3.1, § 16.3.2] for NEN-Koppejan model.

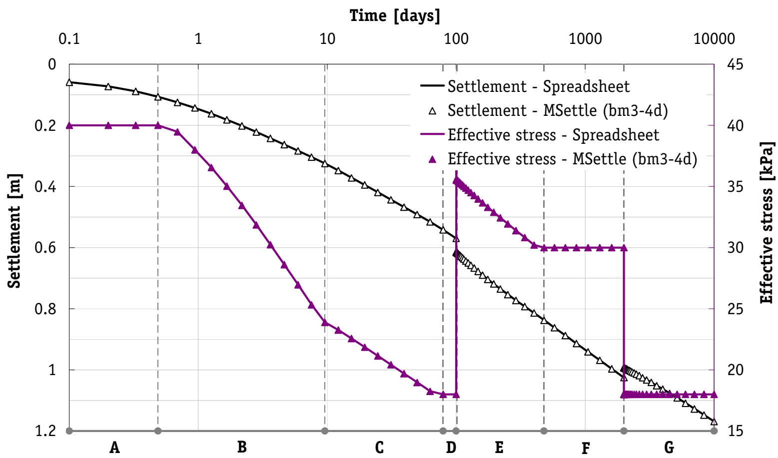


Figure 22-5 – Settlement and loading curves vs. time (NEN-Bjerrum model) with representation of the different submerging phases

MSettle result
The settlements calculated by MSettle are exported to the spread sheet using the *View Data* option in *Time-History* window for comparison (see figures below). The settlements and effective stress at times 100, 2000 and 10000 days are in Table 22-6 and Table 22-7.

Table 22-6 – Results of benchmark 3-4 – Settlements for different cases

Case	Soil model	Cons. model	Subm. method	Time [days]	Benchmark [m]	MSettle File	[m]	Error [%]
A	NEN-Koppejan	Terzaghi	Approx.	100	0.168	bm3-4a	0.166	1.20
				2000	0.454		0.453	0.22
				10000	0.425		0.423	0.47
B		Darcy	Approx.	100	0.168	bm3-4b	0.166	1.20
				2000	0.454		0.453	0.22
				10000	0.425		0.423	0.47
C	NEN-Bjerrum	Terzaghi	Approx.	100	0.661	bm3-4c	0.661	0.00
				2000	1.093		1.093	0.00
				10000	1.265		1.265	0.00
D		Darcy	Accurate	100	0.570	bm3-4d	0.570	0.00
				2000	1.025		1.025	0.00
				10000	1.169		1.169	0.00
E	Isotache	Terzaghi	Approx.	100	0.486	bm3-4e	0.486	0.00
				2000	0.676		0.676	0.00
				10000	0.709		0.709	0.00
F		Darcy	Accurate	100	0.412	bm3-4f	0.413	0.24
				2000	0.641		0.642	0.16
				10000	0.654		0.654	0.00

Table 22-7 – Results of benchmark 3-4 – Effective stress at the surface for different cases

Case	Soil model	Cons. model	Subm. method	Time [days]	Benchmark [kPa]	MSettle File	[kPa]	Error [%]
A	NEN-Koppejan	Terzaghi	Approx.	100	36.240	bm3-4a	36.269	0.08
				2000	57.240		57.269	0.05
				10000	36.240		36.269	0.08
B		Darcy	Approx.	100	36.240	bm3-4b	36.269	0.08
				2000	57.240		57.269	0.05
				10000	36.240		36.269	0.08
C	NEN-Bjerrum	Terzaghi	Approx.	100	34.000	bm3-4c	34.000	0.00
				2000	46.000		46.000	0.00
				10000	34.000		34.000	0.00
D		Darcy	Accurate	100	18.000	bm3-4d	18.000	0.00
				2000	30.000		30.000	0.00
				10000	18.000		18.000	0.00
E	Isotache	Terzaghi	Approx.	100	34.000	bm3-4e	34.000	0.00
				2000	48.734		48.730	0.01
				10000	34.000		34.000	0.00
F		Darcy	Accurate	100	21.021	bm3-4f	21.009	0.06
				2000	35.028		35.023	0.01
				10000	18.000		18.000	0.00

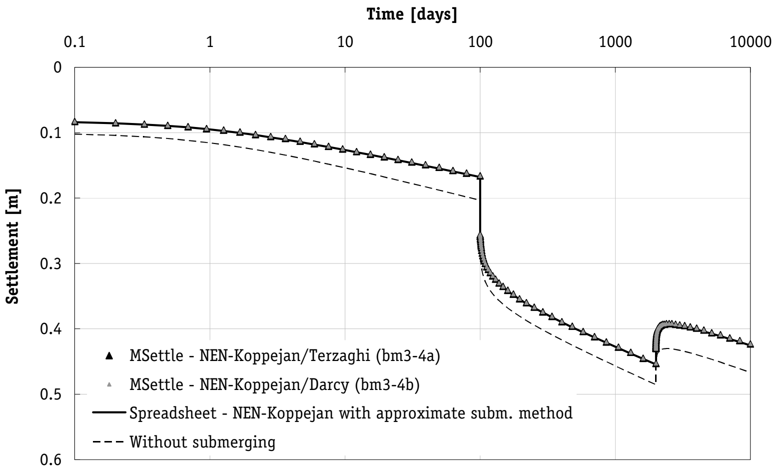


Figure 22-6 – Results of benchmark 3-4 – Comparison between MSettle and the spreadsheet results for NEN-Koppejan model

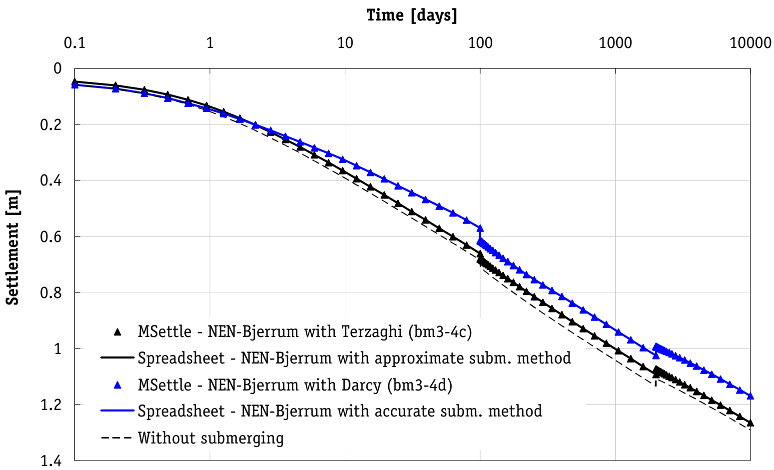


Figure 22-7 – Results of benchmark 3-4 – Comparison between MSettle and the spreadsheet results for NEN-Bjerrum model

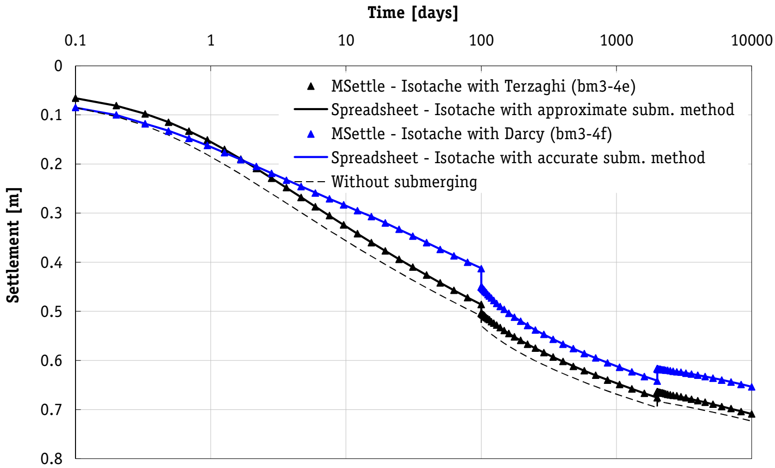


Figure 22-8 – Results of benchmark 3-4 – Comparison between MSettle and the spreadsheet results for Isotache model

NOTE: In this benchmark, some cases lead to a settlement with submerging larger than without submerging. This is not commun but due to the unrealistic saturated and unsatured weights used.

Use MSettle input files bm3-4a.sli till bm3-4f.sli to run this benchmark.

22.5 Initial and final stresses distribution of a multi-layered system

Description

This benchmark checks the initial and final stresses distributions of a multi-layered system for both Darcy and Terzaghi consolidation models. The input data's for each layers are given in Table 22-8. PL-lines nr. 1, 2, 3 and 4 are respectively at depths 1 m, 2 m, 3 m and -6.5 m. Two cases are checked:

- Case 1: the phreatic line is above the ground surface (i.e. PL-line nr. 1)
- Case 2: the phreatic line is below the ground surface (i.e. PL-line nr. 4)

Table 22-8 – Geometry and properties of the different layers

Layer nr.	Top level	Thickness	Drained	PL-line nr.		γ_{unsat}	γ_{sat}
[-]	[m]	[m]		top	bottom	[kN/m^3]	[kN/m^3]
1	0.5	0.5	No	2	3	12.5	15
2	0	0.5	Yes	1	1	17	20
3	-1	1	No	4	99	12.5	15
4	-2	3	No	99	3	12.5	15
5	-5	1	Yes	0	0	17	20
6	-6	1.5	No	4	2	12.5	15
7	-7.5	1.5	No	0	0	12	18
8	-8	2	No	3	4	12.5	15

Benchmark

The initial hydraulic head at the top and bottom of each layer corresponds with the inputted piezometric level (see Table 22-8) on condition that $\varphi \geq z$ to avoid negative pore pressures. The hydraulic head inside a layer is calculated by linear interpolation between the top and the bottom.

The pore pressure is $p(y,t) = \gamma_w \left[\varphi(y,t) - y\right] + p_a(y,t)$.

The total stress is at the bottom of layer i is $\sigma_i(t) = \sigma_{i-1}(t) + h_i \times \gamma$ where $\gamma = \gamma_{unsat}$ if layer i is unsaturated and $\gamma = \gamma_{sat}$ if layer i is saturated.

The effective stress is $\sigma'(y,t) = \sigma(y,t) - p(y,t)$.

The initial excess pore pressure and hydraulic head are nil as the consolidation process has not yet started. The final excess hydraulic head is nil the consolidation process is finished (high permeability of the layers) but the excess pore pressure is: $p_a(y,t) = \gamma_w \, \Delta s(y,t)$.

Calculations are performed in an Excel spreadsheet using the formulas given above and lead to the results given in Table 22-9 to Table 22-11 and also presented in the figures below.

Table 22-9 – Initial and final stresses for case 1 (phreatic line above ground surface)

Depth	Initial state				Final state					
	σ′	σ	φ	p	σ′	σ	φ	p	p_a	Δz[(1)]
[m]	[kPa]	[kPa]	[m]	[kPa]	[kPa]	[kPa]	[m]	[kPa]	[kPa]	[m]
0.5	90	105	2	15	290	339.56	2	49.56	34.56	3.456
0	82.5	112.5	3	30	282.5	344.85	3	62.35	32.35	3.235
0	102.5	112.5	1	10	302.5	344.85	1	42.35	32.35	3.235
-1	112.5	132.5	1	20	312.5	360.88	1	48.38	28.38	2.838
-1	132.5	132.5	-1	0	332.5	360.88	-1	28.38	28.38	2.838
-2	147.5	147.5	-2	0	347.5	372.42	-2	24.92	24.92	2.492
-5	112.5	192.5	3	80	312.5	406.92	3	94.42	14.42	1.442
-5	192.5	192.5	-5	0	392.5	406.92	-5	14.42	14.42	1.442
-6	212.5	212.5	-6	0	412.5	424.15	-6	11.65	11.65	1.165
-7.5	140	235	2	95	340	442.12	2	102.12	7.12	0.712
-7.5	235	235	-7.5	0	435	442.12	-7.5	7.12	7.12	0.712
-8	244	244	-8	0	444	449.88	-8	5.88	5.88	0.588
-8	134	244	3	110	334	449.88	3	115.88	5.88	0.588
-10	239	274	-6.5	35	439	474	-6.5	35	0	0

(1) MSettle results (bm3-5a and bm3-5b)

Table 22-10 – Initial stresses for case 2 (phreatic line below ground surface)

Depth	Layer nr.	σ′	σ	φ	p
[m]	[-]	[kPa]	[kPa]	[m]	[kPa]
0.5	1	100	100	0.5	0
0	1	106.25	106.25	0	0
0	2	106.25	106.25	0	0
-1	2	123.25	123.25	-1	0
-1	3	123.25	123.25	-1	0
-2	3 and 4	135.75	135.75	-2	0
-5	4	173.25	173.25	-5	0
-5	5	173.25	173.25	-5	0
-6	5 and 6	190.25	190.25	-6	0
-7.5	6	116.5	211.5	2	95
-7.5	7	211.5	211.5	-7.5	0
-8	7	220.5	220.5	-8	0
-8	8	110.5	220.5	3	110
-10	8	215.5	250.5	-6.5	35

Table 22-11 – Final stresses for case 2 (phreatic line below ground surface)

Depth	σ′	σ		φ	p		p_a		Δz[2]
		Darcy	Terza.		Darcy	Terza.	Darcy	Terza.	
[m]	[kPa]	[kPa]	[kPa]	[m]	[kPa]	[kPa]	[kPa]	[kPa]	[m]
0.5	300	334.69	300	0.5	34.69	0	34.69	0	3.469
0	306.25	338.92	306.25	0	32.67	0	32.67	0	3.267
0	306.25	338.92	306.25	0	32.67	0	32.67	0	3.267
-1	323.25	352.07	323.25	-1	28.82	0	28.82	0	2.882
-1	323.25	352.07	323.25	-1	28.82	0	28.82	0	2.882
-2	335.75	360.96	335.75	-2	25.21	0	25.21	0	2.521
-5	373.25	388.63	373.63	-5	15.38	0.38	15.38	0.38	1.538
-5	373.25	388.63	373.63	-5	15.38	0.38	15.38	0.38	1.538
-6	390.25	402.66	397.66	-6	12.41	7.41	12.41	7.41	1.241
-7.5	316.5	419.23	419.23	2	102.73	102.73	7.73	7.73	0.773
-7.5	411.5	419.23	419.23	-7.5	7.73	7.73	7.73	7.73	0.773
-8	420.5	426.90	426.90	-8	6.40	6.40	6.40	6.40	0.640
-8	310.5	426.90	426.90	3	116.40	116.40	6.40	6.40	0.640
-10	415.5	450.5	450.5	-6.5	35	35	0	0	0

(2) MSettle results (bm3-5c and bm3-5d)

MSettle result

MSettle results are found using the *View Data* option in the *Depth-History* window of the *Results* menu. Comparison with the spreadsheet results gives exactly the same results as in Table 22-9 for case 1 and Table 22-10 and Table 22-11 for case 2 as illustrated by Figure 22-9 and Figure 22-10.

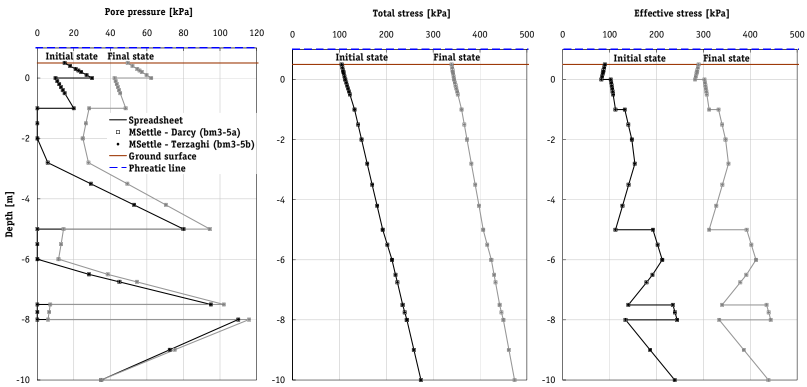


Figure 22-9 – Case 1: Initial and final stresses distributions – Comparison between MSettle and the spreadsheet results

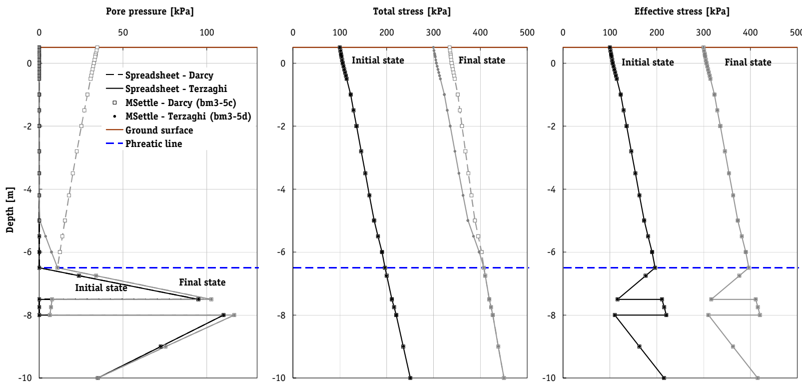


Figure 22-10 – Case 2: Initial and final stresses distributions – Comparison between MSettle and the spreadsheet results

Use MSettle input files bm3-5a.sli to bm3-5d.sli to run this benchmark.

22.6 Effect of water load

Description
This benchmark checks the stresses and settlements distributions of a multi-layered system for both consolidation model. The inputs are the same as benchmark 3-5b [§ 22.5] except that two water loads are added respectively after 10 and 100 days.

Benchmark
The same formulas as benchmark 3-5b [§ 22.5] are used except that the piezometric levels from the water loads are used for the stresses calculation at 10 and 100 days. Calculations are performed in a Excel spreadsheet and lead to the results given in Table 22-12.

MSettle result

Table 22-12 – Results for benchmark 3-6 – Settlements vs. Depth for different times

y	Spreadsheet			MSettle			Error		
Time	10	100	10000	10	100	10000	10	100	10000
[m]	[m]	[m]	[m]	[m]	[m]	[m]	[m]	[m]	[m]
0.5	2.336	2.625	2.595	2.336	2.625	2.595	0.00	0.00	0.00
0	2.184	2.459	2.430	2.184	2.459	2.430	0.00	0.00	0.00
-1	1.914	2.166	2.138	1.914	2.166	2.138	0.00	0.00	0.00
-2	1.680	1.919	1.894	1.680	1.919	1.894	0.00	0.00	0.00
-5	0.970	1.114	1.095	0.970	1.114	1.095	0.00	0.00	0.00
-6	0.784	0.916	0.899	0.784	0.916	0.899	0.00	0.00	0.00
-7.5	0.479	0.557	0.548	0.479	0.557	0.548	0.00	0.00	0.00
-8	0.396	0.472	0.465	0.396	0.472	0.465	0.00	0.00	0.00
-10	0.000	0.000	0.000	0.000	0.000	0.000	0.00	0.00	0.00

Use MSettle input files bm3-6a.sli and bm3-6b.sli to run this benchmark.

22.7 NEN-Koppejan settlements using different types of pre-consolidation pressure (within the layer and in time)

Description

This benchmark checks the functioning of the option *Preconsolidation pressure within a layer* in the *Calculation Options* window [§ 10.1.2] available for NEN-Koppejan model. The same oedometer test that the one used for benchmark 3-1 [§ 22.1] is performed for NEN-Koppejan model with Terzaghi consolidation using different types of pre-consolidation pressure as shown in Table 22-13. The initial effective stress distribution is also different (not assumed constant) to check the influence of a variable preconsolidation stress distribution: at the top, middle and bottom of the layer, effective stresses are respectively equal to 5, 10.4 and 15.8 kPa by means of initial loads.

Table 22-13 – Pre-consolidation types for benchmark 3-7

	Constant within the layer			Variable (parallel to effective stress) within the layer		
	Constt in time	Correct. at t=0 day	Correct. at every step	Constt in time	Correct. at t=0 day	Correct. at every step
σ_p = 8 kPa	bm3-7a	bm3-7b	bm3-7c	bm3-7d	bm3-7e	bm3-7f
OCR = 1.2	bm3-7g	bm3-7h	bm3-7i	bm3-7j	bm3-7k	bm3-7l
POP = 5 kPa	bm3-7m	bm3-7n	bm3-7o	bm3-7p	bm3-7q	bm3-7r

Benchmark

The analytical formulas are the same as benchmark 3-1 [§ 22.1] except the value of the pre-consolidation pressure which depends on the selected options:

- for constant in time $P_{c;i} = \max\left(P_{c;0}; \sigma'_0\right) \;\; \forall i \;\; 1 \le i \le 8$
- for correction at time t = 0 day $P_{c;i} = \max\left(P_{c;0}; \sigma'_0\right) \;\; \forall i \;\; 1 \le i \le 8$
- for correction at every time step $P_{c;i} = \max\left(P_{c;i-1}; \sigma'_{i-1}\right) \;\; \forall i \;\; 1 \le i \le 8$

where:

$$P_{c;0} = \begin{cases} \sigma_p & \text{for bm3-7a until bm3-7c (constant within the layer)} \\ \sigma'_0 - (\sigma'_{0;middle} - \sigma_p) & \text{for bm3-7d until bm3-7f (variable within the layer)} \\ OCR\,\sigma'_0 & \text{for bm3-7g until bm3-7l} \\ POP + \sigma'_0 & \text{for bm3-7m until bm3-7r} \end{cases}$$

MSettle results are compared to an analytical solution worked out in an Excel spreadsheet.

MSettle result

In the *Calculation Options* window, the *Preconsolidation pressure within a layer* is adapted for each benchmark according to Table 22-13. The settlements calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison (see Figure 22-11). The final settlements and initial pre-consolidation are respectively given in Table 22-14 and Table 22-15.

Table 22-14 – Results of benchmark 3-7 – Final settlements for different pre-consolidation types

Type	Type within the layer	Type in time	Benchmark [mm]	MSettle File	[mm]	Error [%]
Pc	Constant	Constant	16.60	bm3-7a	16.60	0.00
		Correction t=0	16.60	bm3-7b	16.60	0.00
		Corr. every step	12.29	bm3-7c	12.29	0.00
	Variable (parallel to effective stress)	Constant	16.85	bm3-7d	16.85	0.00
		Correction t=0	16.85	bm3-7e	16.85	0.00
		Corr. every step	12.42	bm3-7f	12.42	0.00
OCR	Constant	Constant	14.50	bm3-7g	14.50	0.00
		Correction t=0	14.50	bm3-7h	14.50	0.00
		Corr. every step	11.24	bm3-7i	11.24	0.00
	Variable (parallel to effective stress)	Constant	14.50	bm3-7j	14.50	0.00
		Correction t=0	14.50	bm3-7k	14.50	0.00
		Corr. every step	11.24	bm3-7l	11.24	0.00
POP	Constant	Constant	11.55	bm3-7m	11.55	0.00
		Correction t=0	11.55	bm3-7n	11.55	0.00
		Corr. every step	9.75	bm3-7o	9.75	0.00
	Variable (parallel to effective stress)	Constant	11.55	bm3-7p	11.55	0.00
		Correction t=0	11.55	bm3-7q	11.55	0.00
		Corr. every step	9.75	bm3-7r	9.75	0.00

Table 22-15 – Results of benchmark 3-7 – Initial pre-consolidation pressure distribution for different pre-consolidation types

Type	Type within the layer	Depth [m NAP]	Benchmark [kPa]	MSettle File	[kPa]	Error [%]
Pc	Constant	-0.025	8.00	bm3-7a to c	8.00	0.00
		-0.075	13.10		13.10	0.00
	Variable	-0.025	7.70	bm3-7d to f	7.70	0.00
		-0.075	13.10		13.10	0.00
OCR	Constant	-0.025	9.24	bm3-7g to l	9.24	0.00
		-0.075	15.72		15.72	0.00
POP	Constant	-0.025	12.70	bm3-7m to r	12.70	0.00
		-0.075	18.10		18.10	0.00

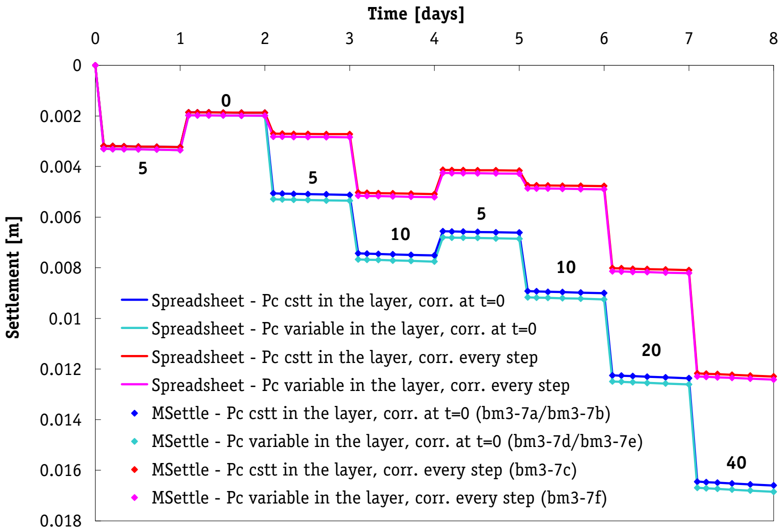


Figure 22-11 – Comparison between MSettle and the spreadsheet results for Pc compression

Use MSettle input files bm3-7a.sli to bm3-7r to run this benchmark.

22.8 Settlements and dissipations during Terzaghi consolidation process (loading/un-reloading steps)

Description

The same oedometer tests as benchmarks bm3-1a [§ 22.1], bm3-2a [§ 22.2] and bm3-3a [§ 22.3] are performed for respectively NEN-Koppejan, Isotache and NEN-Bjerrum models except that the layer is not *Drained* anymore but has a coefficient of consolidation of $C_v = 10^{-10}$ m^2/s which leads to a slow consolidation process (contrary of benchmark 3-2). In MSettle, two types of calculation are performed:

- Benchmarks bm3-8a, b and c use the Terzaghi consolidation model;
- Benchmarks bm3-8d, e and f use the Darcy consolidation model with C_v as storage parameter.

Benchmark

The analytical solution for a calculation with consolidation Terzaghi consolidation model for load-step *i* depends on the soil model:

(120) $$s_i(t) = s_{i-1}(t) + \Delta s_{i;prim}(t) \cdot U(t - t_i) + \Delta s_{i;sec}(t)$$ for NEN-Koppejan

(121) $$s_i(t) = s_{i-1}(t) + \Delta s_i(t) \cdot U(t - t_i)$$ for Isotache and NEN-Bjerrum

where:

$s_0(t) = 0$

$\Delta s_{i;prim}$ Primary settlement acc. to Koppejan theory due to load-step i (see equation (60) page 305).

$\Delta s_{i;sec}$ Secondary settlement acc. to Koppejan theory due to load-step i (see equation (62) page 305).

Δs_i Relative settlement at time t due to load-step i:

- for Isotache: $\Delta s_i(t) = H_0 \left[\exp\left(-\varepsilon_{i-1}(t)\right) - \exp\left(-\varepsilon_i(t)\right)\right]$
- for NEN-Bjerrum: $\Delta s_i(t) = H_0 \left[\varepsilon_i(t) - \varepsilon_{i-1}(t)\right]$

$\varepsilon_i(t)$ Total deformation at time t:

- for Isotache model, see equation (58) page 303;
- for NEN-Bjerrum, see equation (43) page 299.

t_i Start time of load-step i.

$U(t)$ Degree of consolidation at time t:

$$U(t) = 1 - \frac{8}{\pi^2}\sum_{n=1}^{\infty}\frac{1}{(2n-1)^2}\exp\left[-\frac{\pi^2(2n-1)^2 c_v}{4h^2}\frac{t}{t_0}\right]$$

h Drainage height. As the sample is drained at both sides, $h = H_0 / 2 = 10$ mm.

H_0 Initial height of the sample (20 mm).

t_0 Creep rate reference time (4 days).

MSettle result

The settlements and dissipations calculated by MSettle are exported to the spreadsheet for comparison using the *View Data* option in the *Time-History* and *Dissipations* windows respectively (see figures below). The final settlement and the dissipations are respectively given in Table 22-16 and Table 22-17.

Figures below show that results for Darcy consolidation with Cv are largely different from Terzaghi results because Terzaghi theorie assumes time dependent dissipations whereas Darcy theorie assumes strain dependent dissipations. Therefore, to compare Terzaghi and Darcy dissipations in a proper way, the deformation must be almost zero or the consolidation coefficient for Terzaghi must be adapted, see [§ 23.1, 23.2].

Table 22-16 – Results of benchmark 3-8 – Final settlements

Consolidation model	Soil model	Spreadsheet [mm]	MSettle File	[mm]	Relative error [%]
Terzaghi	NEN-Koppejan	6.98	bm3-8a	6.96	0.29
	Isotache	7.66	bm3-8b	7.66	0.00
	NEN-Bjerrum	8.16	bm3-8c	8.18	0.24
Darcy (with Cv storage)	NEN-Koppejan	6.98	bm3-8d	6.14	13.68
	Isotache	7.66	bm3-8e	7.47	2.54
	NEN-Bjerrum	8.16	bm3-8f	7.53	8.37

Table 22-17 – Results of benchmarks 3-8a/b/c (Terzaghi model) – Dissipations in

time			
Time	Spreadsheet	MSettle	Relative error
[days]	[%]	[%]	[%]
0.1	5.37	5.25	2.29
0.95	16.16	16.14	0.12
9.66	51.47	51.50	0.06
80	98.86	98.86	0.00

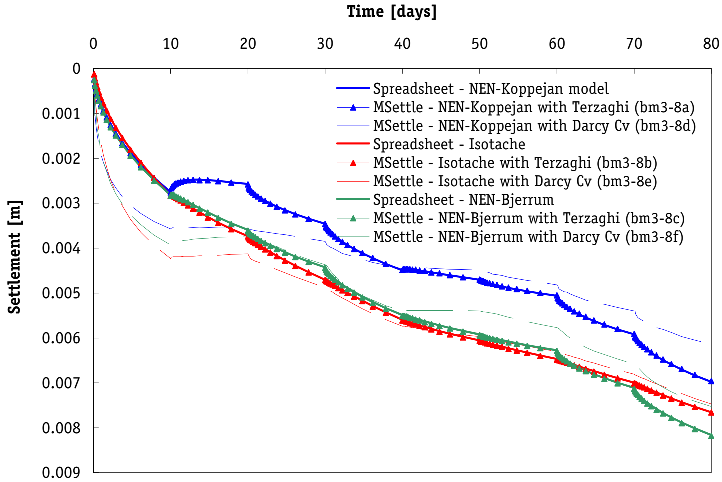


Figure 22-12 – Benchmark 3-8 – Comparison between MSettle and the spreadsheet settlement curves

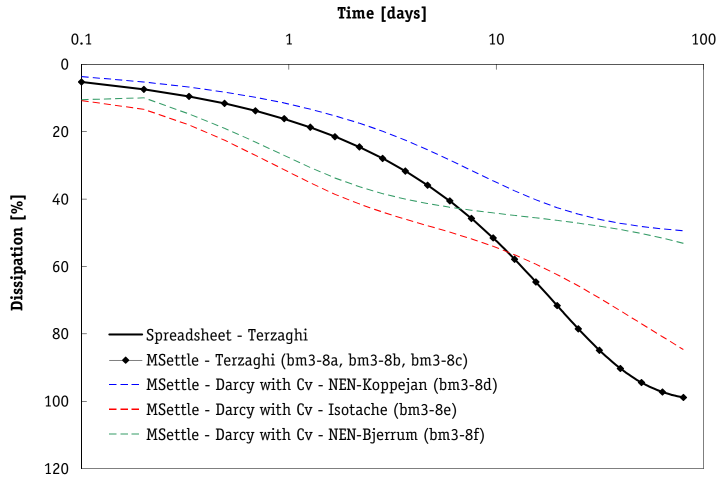


Figure 22-13 – Benchmark 3-8 – Comparison between MSettle and the spreadsheet dissipation curves

Use MSettle input files bm3-8a.sli to bm3-8f.sli to run this benchmark.

22.9 Hydraulic head during Darcy consolidation process

Description

This benchmark tests the Darcy consolidation model, for Isotache and NEN-Koppejan soil models, by calculating the excess pore pressure variation of a clay layer (height H = 20 m) during its consolidation. The layer is first loaded with an initial load of $\sigma_{initial}$ = 1000 kPa and then with a uniform load of σ = 100 kPa. The initial hydraulic head distribution is constant along the layer with φ_0 = 10 m. For the storage, three kinds of inputs are tested:

- a consolidation coefficient C_v = 0.0002 m^2/s
- a constant permeability k_v = 0.001 m/day
- a strain dependent permeability, with an initial permeability of $k_{v;0}$ = 0.001 m/day and permeability strain modulus of E_k = 0.01.

Benchmark

The analytical solution is a solution for linear elastic storage. The effect of creep is not involved. The resolution of the storage equation (see equation (24) page 288) leads to the following expression of the hydraulic head at depth z ant time t:

$$(122) \qquad \varphi(z,t) = \varphi_0(z) + \frac{\sigma}{\gamma_w} \sum_{n=1}^{\infty} \frac{4}{m} \exp\left(-m^2 \frac{c_v\, t}{4\, d^2} \right) \sin\left(\frac{m}{2} \frac{z}{d} \right) \quad \text{with } m = (2n-1)\,\pi$$

where:

$$c_v = \frac{k_v}{\gamma_w \cdot (m_v + n / K_w)}$$

n	0.4	Porosity
d	10 m	Drainage length
K_w	2000 MPa	Bulk modulus of water

In case of strain dependent permeability, the permeability is expressed as:

$$(123) \qquad k_{v;\varepsilon}(t) = k_{v;0}\; 10^{\left(\frac{\varepsilon^C(t)}{E_k} \right)}$$

As the initial effective stress distribution is quite constant within the layer (top 1000 kPa, bottom 1000.2 kPa) therefore stress variation against strain is quite linear for the small second load-step. So the soil stiffness is constant:

$$(124) \qquad m_v = \frac{\varepsilon^C}{\sigma} \quad \text{where } \varepsilon^C = \begin{cases} a \ln\left(\frac{\sigma_0' + \sigma}{\sigma_0'} \right) & \text{for Isotache model (with } a = 0.01) \\ \frac{1}{C_p} \ln\left(\frac{\sigma_0' + \sigma}{\sigma_0'} \right) & \text{for Koppejan model (with } C_p = 100) \end{cases}$$

The solutions are worked out in an Excel spreadsheet.

MSettle result

The hydraulic heads calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison (see Figure 22-14). The maximum relative errors are given in Table 22-18.

Table 22-18 – Results of benchmark 3-9 – Hydraulic head at the middle of the layer for different cases

Case	Soil model	Storage	Time [days]	Benchmark [m]	MSettle File	[m]	Error [%]
A	Isotache	C_v	0.944	18.40	bm3-9a	18.21	1.04
			2.820	13.83		14.00	1.21
			4.650	11.75		12.06	2.57
B	NEN-Koppejan	C_v	0.944	18.40	bm3-9b	18.22	0.99
			2.820	13.83		14.01	1.28
			4.650	11.75		12.06	2.57
C	Isotache	k_V	0.944	19.54	bm3-9c	19.35	0.98
			2.820	16.22		16.12	0.62
			4.650	13.92		13.94	0.14
D	NEN-Koppejan	k_V	0.944	19.54	bm3-9d	19.35	0.98
			2.820	16.22		16.12	0.62
			4.650	13.92		13.94	0.14
E	Isotache	k_V strain dep.	0.944	19.63	bm3-9e	19.38	1.29
			2.820	16.79		16.31	2.94
			4.650	14.67		14.18	3.46
F	NEN-Koppejan	k_V strain dep.	0.944	19.63	bm3-9f	19.38	1.29
			2.820	16.79		16.31	2.94
			4.650	14.67		14.19	3.38

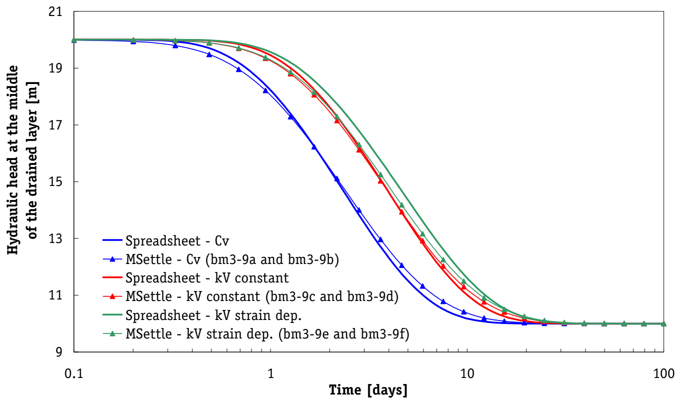


Figure 22-14 – Benchmark 3-9 – Comparison between MSettle and the spreadsheet results for different types of storage

Use MSettle input files bm3-9a.sli till bm3-9f.sli to run this benchmark.

22.10 Hydraulic head distribution in stationary phase using vertical drainage (Darcy consolidation)

Description

A layer (height h = 20 m) with a constant initial piezometric level of φ_0 = 1 m is consolidated by means of vertical drains. At the end of drainage, the hydraulic head distribution along the layer is stabilized. Results are compared with the analytical solution given in [Lit 11] in which the storage equation is written for a stationary phase (after consolidation). Verifications are performed for the three types of drain (sand wall, column drain and strip drain) in combination with three types of dewatering (off, simple or detailed input). Therefore, nine cases are checked as shown in Table 22-19.

Table 22-19 – Cases overview for benchmark 3-10

Case	Drain type	MSettle file	Soil model	Input dewatering	Grid
A	Sand wall	bm3-10a	Isotache	Off	-
B		bm3-10b	NEN-Bjerrum	Simple	-
C		bm3-10c	NEN-Koppejan	Detailed	-
D	Column	bm3-10d	NEN-Koppejan	Off	Undetermined
E		bm3-10e	Isotache	Simple	Rectangular
F		bm3-10f	NEN-Bjerrum	Detailed	Triangular
G	Strip	bm3-10g	NEN-Bjerrum	Off	Rectangular
H		bm3-10h	NEN-Koppejan	Simple	Triangular
I		bm3-10i	Isotache	Detailed	Undetermined

The drain characteristics and the dewatering data's are given in the table below. The hydraulic head distribution is calculated for two verticals:

- Vertical 1 is situated within the drainage range (at the right limit);
- Vertical 2 is situated 10 m at the right of the drainage right limit.

The unit weight of water is set to γ_w = 9.81 kN/m^3 and the ratio hor./vert. permeability is k_H / k_V = 1.3.

Table 22-20 – Vertical drains characteristics (benchmark 3-10)

Drain type			Strip	Column	Sand wall
Bottom position	[m NAP]	y_{bot}	-19	-18	-17
Distance between 2 drains	[m]	D	3	2.5	2
Diameter/width	[m]	d		0.25	
Width	[m]	w	0.3		0.2
Thickness	[m]	t	0.05		

Table 22-21 – Enforced dewatering data's (benchmark 3-10)

Drain type			Strip	Column	Sand wall
Dewatering Off					
Start of drainage	[days]		0	0	0
Phreatic level in drain	[m NAP]		1[(1)]	1[(1)]	-8
Dewatering with Simple Input					
Start of drainage	[days]		0	0	0
Phreatic level in drain	[m NAP]	y_w	1[(1)]	1[(1)]	-7
Begin time dewatering	[days]		300	300	300
End time dewatering	[days]		600	600	600
Underpressure	[kPa]	P_{air}	5	15	25
Water head during dewat.	[m NAP]	y_w	-4	-5.5	-6.34[(2)]
Tube pressure during dewat.	[kPa]	P_{tube}			60
Position of the drain pipe	[m NAP]	y_{pipe}			-15
Dewatering with Detailed Input					
Times	[days]	t_1	0	0	0
	[days]	t_2	400	400	400
Underpressure	[kPa]	$P_{air;1}$	30	20	25
	[kPa]	$P_{air;2}$	10	35	15
Tube pressure	[kPa]	$P_{tube;1}$			10
	[kPa]	$P_{tube;2}$			5
Water head	[m NAP]	$y_{w;1}$	-7.5	-5	-10.43[(2)]
	[m NAP]	$y_{w;2}$	-6	-2.5	-11.96[(2)]
Position of the drain pipe	[m NAP]	y_{pipe}			-14

(1) Not a user input. MSettle uses the inputted phreatic level.

(2) Not a user input, deduced from equation (34) page 294.

Benchmark

Along the drain, the average hydraulic head is given by the differential equation (30) page 290 [§ 15.4.1] and below the drain, the hydraulic head has a linear distribution. Therefore, the hydraulic head distribution is:

$$\text{(125)} \qquad \varphi(y) = \begin{cases} -P_{air} + y + C_1 \exp(-y/\lambda) + C_2 \exp(y/\lambda) & \text{if } y \geq y_{water} \\ -P_{air} + y_{water} + C_3 \exp(-y/\lambda) + C_4 \exp(y/\lambda) & \text{if } y_{water} > y \geq y_{bot} \\ C_5\ y + C_6 & \text{if } y < y_{bot} \end{cases}$$

where constants C_1 to C_6 are unknown. The conditions at the top and bottom and the continuity of the head along the layer lead to the six following equations:

$$\begin{aligned}
&\varphi_1(0) = \varphi_0 && \Rightarrow -P_{air}/\gamma_w + C_1 + C_2 - \varphi_0 = 0 \\
&\varphi_3(-h) = \varphi_0 && \Rightarrow -C_5\ h + C_6 - \varphi_0 = 0 \\
&\varphi_1 = \varphi_2\big|_{y_w} && \Rightarrow C_1 - C_3 + (C_2 - C_4)\exp(2y_w/\lambda) = 0 \\
&\varphi_1' = \varphi_2'\Big|_{y_w} && \Rightarrow \lambda + (C_3 - C_1)\exp(-y_w/\lambda) + (C_2 - C_4)\exp(y_w/\lambda) = 0 \\
&\varphi_2 = \varphi_3\big|_{y_{bot}} && \Rightarrow -P_{air}/\gamma_w + y_w + C_3\exp(-y_{bot}/\lambda) + C_4\exp(y_{bot}/\lambda) - C_5\ y_{bot} - C_6 = 0 \\
&\varphi_2' = \varphi_3'\Big|_{y_{bot}} && \Rightarrow -C_3\exp(-y_{bot}/\lambda) + C_4\exp(y_{bot}/\lambda) - \lambda\ C_5 = 0
\end{aligned}$$

The resolution of this system leads to the following constants:

$$C_5 = \frac{2\exp\left(\frac{y_{bot}}{\lambda}\right)\left[\lambda\sinh\left(\frac{y_w}{\lambda}\right) - \frac{P_{air}}{\gamma_w} - \varphi_0\right] + \left[1 + \exp\left(\frac{2y_{bot}}{\lambda}\right)\right]\left(\frac{P_{air}}{\gamma_w} + \varphi_0 - y_w\right)}{\exp\left(\frac{2y_{bot}}{\lambda}\right)(\lambda - h - y_{bot}) - \lambda - h - y_{bot}}$$

$$C_6 = \varphi_0 + h\ C_5$$

$$C_4 = \frac{1}{2}\exp(-y_{bot}/\lambda)\left[P_{air}/\gamma_w - y_w + \varphi_0 + C_5(\lambda + y_{bot} + h)\right]$$

$$C_3 = C_4\exp(2y_{bot}/\lambda) - \lambda\ C_5\exp(y_{bot}/\lambda)$$

$$C_2 = \left[C_4\exp(2y_w/\lambda) + C_3 - P_{air}/\gamma_w - \varphi_0\right] / \left[\exp(2y_w/\lambda) - 1\right]$$

$$C_1 = P_{air}/\gamma_w + \varphi_0 - C_2$$

Calculations are worked out in an Excel spreadsheet using the parameters given in Table 22-22 deduced from the formulas given in [§ 15.4]. The analytical results for hydraulic head are given in Table 22-23 to Table 22-25.

Table 22-22 – Parameters used for each case of benchmark 3-10

Case	Vertical	Time	y_w	P_{air}	D	d	λ
	[-]	[days]	[m]	[kPa]	[m]	[m]	[m]
A	1	1000	-8	0	2	0.2	0.456
	2	1000	-8	0	40	0.2	10.077
B	1	300/1000	-7	0	2	0.2	0.456
	1	600	-6.34	25	2	0.2	0.456
	2	300/1000	-7	0	40	0.2	10.077
	2	600	-6.34	25	40	0.2	10.077
C	1	400	-10.43	25	2	0.2	0.456
	1	1000	-11.96	15	2	0.2	0.456
	2	400	-10.43	25	40	0.2	10.077
	2	1000	-11.96	15	40	0.2	10.077
D	1	1000	1	0	2.5	0.25	0.974
	2	1000	1	0	40	0.25	25.796
E	1	300/1000	1	0	2.825	0.25	1.141
	1	600	-5.5	15	2.825	0.25	1.141
	2	300/1000	1	0	45.2	0.25	29.559
	2	600	-5.5	15	45.2	0.25	29.559
F	1	400	-5	20	2.625	0.25	1.038
	1	1000	-2.5	35	2.625	0.25	1.038
	2	400	-5	20	42	0.25	27.238
	2	1000	-2.5	35	42	0.25	27.238
G	1	1000	1	0	3.39	0.223	1.481
	2	1000	1	0	45.2	0.223	29.938
H	1	300/1000	1	0	3.15	0.223	1.351
	1	600	-4	5	3.15	0.223	1.351
	2	300/1000	1	0	42	0.223	27.594
	2	600	-4	5	42	0.223	27.594
I	1	400	-7.5	30	3	0.223	1.271
	1	1000	-6	10	3	0.223	1.271
	2	400	-7.5	30	40	0.223	26.137
	2	1000	-6	10	40	0.223	26.137

MSettle result

In order to compare the MSettle output to the analytical result in a proper way, the creep must be set to nought (i.e. $c = 0$ for Isotache, $C_\alpha = 0$ for NEN-Bjerrum and $C_s = C_s' = 10^{30}$ for NEN-Koppejan).

The stationary hydraulic head distribution along the layer calculated by MSettle can be found using the *View Data* option in the *Depth-History* window. MSettle results are compared to the spreadsheet results in Table 22-23 to Table 22-25.

Table 22-23 – Results of benchmark 3-10 for sand wall – Hydraulic head distribution

Case	Time [days]	Depth [m]	Spreadsheet [m]		MSettle [m]		Relative error [%]	
			Vert. 1	Vert. 2	Vert. 1	Vert. 2	Vert. 1	Vert. 2
A	1000	-4	-4.00	-0.56	-4.00	-0.55	0.00	1.82
		-8	-7.77	-1.57	-7.77	-1.56	0.00	0.64
		-12	-8.00	-1.65	-8.00	-1.65	0.00	0.00
		-16	-7.87	-0.73	-7.87	-0.73	0.00	0.00
B	300 and 1000	-4	-4.00	-0.46	-4.00	-0.45	0.00	2.22
		-8	-6.97	-1.36	-6.97	-1.36	0.00	0.00
		-12	-7.00	-1.41	-7.00	-1.40	0.00	0.71
		-16	-6.88	-0.56	-6.88	-0.56	0.00	0.00
	600	-4	-6.55	-0.96	-6.55	-0.95	0.00	1.05
		-8	-8.88	-2.04	-8.87	-2.03	0.11	0.49
		-12	-8.88	-2.04	-8.88	-2.04	0.00	0.00
		-16	-8.74	-0.96	-8.74	-0.96	0.00	0.00
C	400	-4	-6.55	-1.29	-6.55	-1.28	0.00	0.78
		-8	-10.55	-2.75	-10.55	-2.74	0.00	0.36
		-12	-12.97	-2.97	-12.97	-2.96	0.00	0.34
		-16	-12.78	-1.61	-12.78	-1.61	0.00	0.00
	1000	-4	-5.53	-1.13	-5.53	-1.11	0.00	1.80
		-8	-9.53	-2.56	-9.53	-2.55	0.00	0.39
		-12	-13.28	-2.87	-13.27	-2.87	0.08	0.00
		-16	-13.28	-1.59	-13.28	-1.59	0.00	0.00

Table 22-24 – Results of benchmark 3-10 for column drain – Hydraulic head distribution

Case	Time [days]	Depth [m]	Spreadsheet [m]		MSettle [m]		Relative error [%]	
			Vert. 1	Vert. 2	Vert. 1	Vert. 2	Vert. 1	Vert. 2
D	1000	All	1.00	1.00	1.00	1.00	0.00	0.00
E	300/1000	All	1.00	1.00	1.00	1.00	0.00	0.00
	600	-4	-5.30	0.75	-5.29	0.75	0.19	0.00
		-8	-6.96	0.60	-6.96	0.60	0.00	0.00
		-12	-7.01	0.60	-7.01	0.60	0.00	0.00
		-16	-6.52	0.74	-6.52	0.74	0.00	0.00
F	400	-4	-5.78	0.70	-5.78	0.70	0.00	0.00
		-8	-7.01	0.53	-7.01	0.53	0.00	0.00
		-12	-7.03	0.53	-7.03	0.53	0.00	0.00
		-16	-6.64	0.69	-6.64	0.69	0.00	0.00
	1000	-4	-5.85	0.72	-5.84	0.72	0.17	0.00
		-8	-6.06	0.58	-6.06	0.58	0.00	0.00
		-12	-6.06	0.58	-6.06	0.58	0.00	0.00
		-16	-5.72	0.73	-5.72	0.73	0.00	0.00

Table 22-25 – Results of benchmark 3-10 for strip drain – Hydraulic head distribution

Case	Time	Depth	Spreadsheet [m]		MSettle [m]		Relative error [%]	
	[days]	[m]	Vert. 1	Vert. 2	Vert. 1	Vert. 2	Vert. 1	Vert. 2
G	1000	All	1.00	1.00	1.00	1.00	0.00	0.00
H	300/1000	All	1.00	1.00	1.00	1.00	0.00	0.00
	600	-4	-3.76	0.79	-3.76	0.79	0.00	0.00
		-8	-4.47	0.68	-4.47	0.68	0.00	0.00
		-12	-4.49	0.68	-4.49	0.68	0.00	0.00
		-16	-4.17	0.78	-4.17	0.79	0.00	1.27
I	400	-4	-6.84	0.56	-6.84	0.56	0.00	0.00
		-8	-10.12	0.30	-10.09	0.30	0.30	0.00
		-12	-10.51	0.28	-10.51	0.28	0.00	0.00
		-16	-9.95	0.51	-9.95	0.51	0.00	0.00
	1000	-4	-4.80	0.68	-4.80	0.68	0.00	0.00
		-8	-6.88	0.50	-6.88	0.50	0.00	0.00
		-12	-7.00	0.49	-7.00	0.49	0.00	0.00
		-16	-6.60	0.66	-6.60	0.66	0.00	0.00

Use MSettle input files bm3-10a.sli till bm3-10i.sli to run this benchmark.

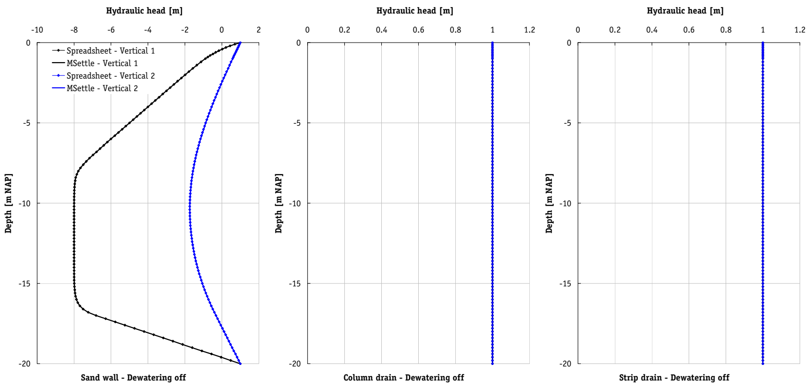


Figure 22-15 – Comparison between MSettle and the spreadsheet hydraulic head distribution for *Enforced Dewatering Off*

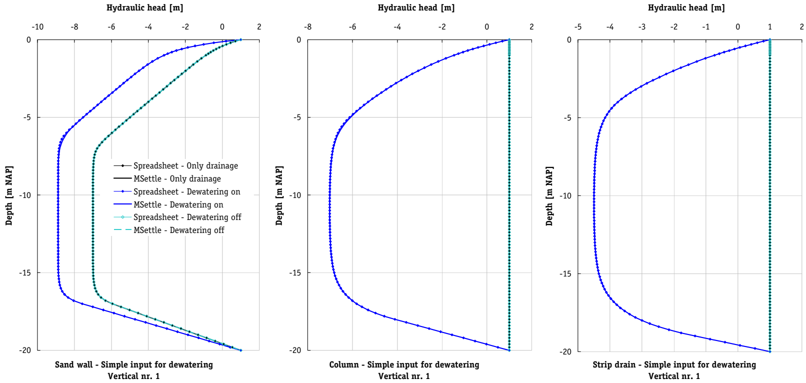


Figure 22-16 – Comparison between MSettle and the spreadsheet hydraulic head distribution for *Simple Enforced Dewatering*

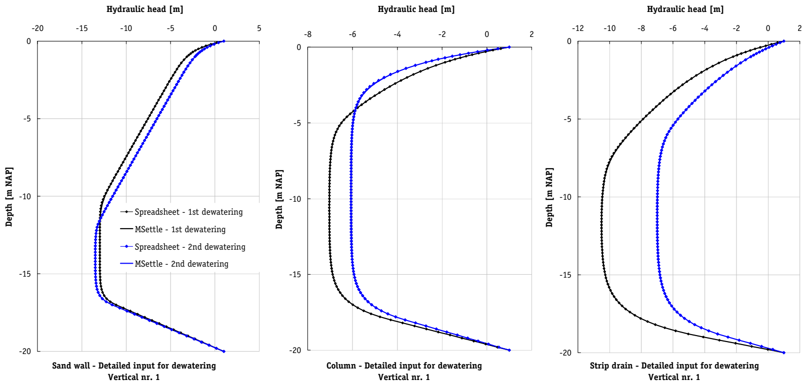


Figure 22-17 – Comparison between MSettle and the spreadsheet hydraulic head distribution for *Detailed Enforced Dewatering*

22.11 Settlements during the Terzaghi consolidation process with vertical drainage

Description

A two-layers sytem (Table 22-27) with initials piezometric levels of φ_{top} = 9 m and φ_{bottom} = 3 m respectively at the top and bottom is consolidated by means of vertical drains. A uniform load of σ_{load} = 200 kPa is applied. Verifications are performed for the three types of drain (sand wall, column drain and strip drain) in combination with three types of dewatering (off, simple or detailed input). Therefore, nine cases are checked as shown in the following table.

Table 22-26 – Cases overview for benchmark 3-11

Case	Drain type	MSettle file	Soil model	Input dewater.	Grid
A	Sand wall	bm3-11a	Isotache	Off	-
B		bm3-11b	NEN-Bjerrum	Simple	-
C		bm3-11c	NEN-Koppejan	Detailed	-
D	Column	bm3-11d	NEN-Koppejan	Off	Undetermined
E		bm3-11e	Isotache	Simple	Rectangular
F		bm3-11f	NEN-Bjerrum	Detailed	Triangular
G	Strip	bm3-11g	NEN-Bjerrum	Off	Rectangular
H		bm3-11h	NEN-Koppejan	Simple	Triangular
I		bm3-11i	Isotache	Detailed	Undetermined
J	No drain	bm3-11j	Isotache	-	-
K		bm3-11k	NEN-Bjerrum	-	-
L		bm3-11l	NEN-Koppejan	-	-

Table 22-27 – Materials properties (bm3-11)

			Layer 1	Layer 2
Top level	$y_{top;i}$	[m NAP]	10	6
Thickness	h_i	[m]	4	16
Coefficient of consolidation	$c_{v;i}$	[m^2/s]	4×10^{-7}	6.4×10^{-6}
Permeability ratio	k_{Hi}/k_{Vi}	[-]	1	0.3
Saturated unit weight	γ_{sat}	[kN/m^3]	17	17
Unsaturated unit weight	γ_{unsat}	[kN/m^3]	15	15
NEN-Koppejan parameters		[-]	$C_p = C_p' = 25$	$C_s = C_s' = 100$
NEN-Bjerrum parameters		[-]	$RR = CR = 0.1$	$C_s = 0.01$
Isotache parameters		[-]	$a = b = 0.04$	$c = 0.006$

Table 22-28 – Vertical drains characteristics (benchmark 3-11)

Drain type			Strip	Column	Sand wall
Bottom position	[m NAP]	y_{drain}	-16	-17	-18
Distance between 2 drains	[m]	D	3	2.5	2
Diameter	[m]	d		0.25	
Width	[m]	w	0.3		0.2
Thickness	[m]	t	0.05		

Table 22-29 – Enforced dewatering data's (benchmark 3-11)

Drain type		Strip	Column	Sand wall
Dewatering Off				
Start of drainage	[days]	200	200	200
Phreatic level in drain	[m NAP]	-1(1)	-1(1)	-2

Drain type			Strip	Column	Sand wall
Dewatering with Simple Input					
Start of drainage	[days]		50	50	50
Phreatic level in drain	[m NAP]	y_w	-1[(1)]	-1[(1)]	-10
Begin time dewatering	[days]		200	200	200
End time dewatering	[days]		400	400	400
Underpressure	[kPa]	P_{air}	5	2.5	10
Water head during dewat.	[m NAP]	y_w	-3	-2.5	-10.5[(2)]
Tube pressure during dewat.	[kPa]	P_{tube}			5
Position of the drain pipe	[m NAP]	y_{pipe}			-12
Dewatering with Detailed Input					
Times	[days]	t_1	50	50	50
	[days]	t_2	200	200	200
Underpressure	[kPa]	$P_{air;1}$	15	30	10
	[kPa]	$P_{air;2}$	0	0	5
Tube pressure	[kPa]	$P_{tube;1}$			20
	[kPa]	$P_{tube;2}$			45
Water head	[m NAP]	$y_{w;1}$	-1.5	-5	-3.5[(2)]
	[m NAP]	$y_{w;2}$	-3	-2	-1.5[(2)]
Position of the drain pipe	[m NAP]	y_{pipe}			-6.5

(1) Not a user input. MSettle uses the inputted phreatic level.

(2) Not a user input, deduced from equation (34) page 294.

Benchmark

Settlements during the Terzaghi consolidation process with vertical drains are calculated with the same formulas as for benchmark 3-8 [§ 22.8]: equations (120) and (121) for respectively NEN-Koppejan and Isotache/NEN-Bjerrum models. The degree of consolidation *U(t)* should includes the effect of vertical drainage:

$$U(t) = 1 - \sum_{n=1}^{\infty} \frac{8}{\pi^2 (2n-1)^2} \exp\left\{ -\left(\frac{\pi^2}{4}(2n-1)^2 + \frac{h'^2}{\lambda^2} \right) \frac{c_v \cdot t/t_0}{h^2} \right\} \quad (126)$$

where:

h Drainage height (equal to the half-thickness of the layer-system because both sides are drained): $h = h_1 + h_2 = 10$ m

h' Drainage height along the drain: $h' = (z_{top} - z_{drain})/2 = 7$ m

λ Leakage length [m]. See equations (33) and (35) respectively in [§ 15.4.2] for strip/column and [§ 15.4.3] for sand wall.

k_V/k_H Global permeability ratio along the drain:

$$\frac{k_H}{k_V} = \frac{1}{y_{top;1} - y_{drain}} \cdot \left(\frac{k_{H1}}{k_{V1}} \left(y_{top;1} - y_{top;2} \right) + \frac{k_{H2}}{k_{V2}} \left(y_{top;2} - y_{drain} \right) \right)$$

c_V Global coefficient of consolidation along the drained layers:

$$c_V = \left(\frac{h}{h_1/\sqrt{c_{V1}} + h_2/\sqrt{c_{V2}}} \right)^2 = 0.216 \text{ m}^2/\text{day}$$

MSettle will model the effect of vertical drainage by automatically adding a water load with an adapted hydraulic head distribution:

$$\varphi(y) = \begin{cases} y - P_{air}/\gamma_w & \text{for } y \geq y_w \\ y_w - P_{air}/\gamma_w & \text{for } y_{bottom} < y < y_w \\ \left[\varphi_1 (y - y_{bot}) - \varphi_2 (y - y_{top})\right] / H & \text{for } y \leq y_{bottom} \end{cases} \quad (127)$$

The average hydraulic head along the drained layers is:

$$\tilde{\varphi}_d = \frac{1}{y_{top} - y_{bottom}} \int_{y_{bottom}}^{y_{top}} \varphi_d \cdot dy \quad (128)$$

Figure 22-18 illustrates the average hydraulic head for case H at time 200 days.

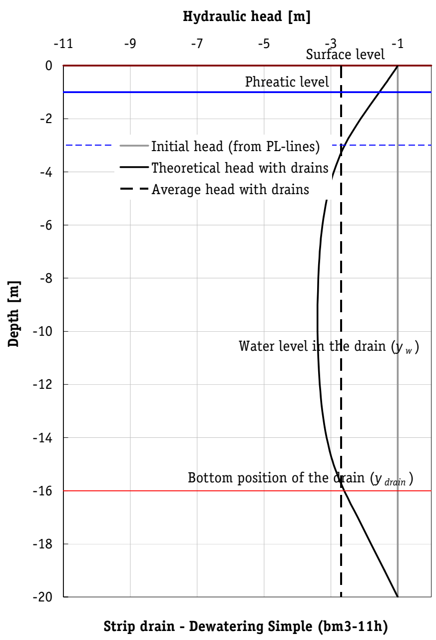


Figure 22-18 – Distribution of the hydraulic head along the layer for case H

Calculations are worked out in an Excel spreadsheet using the parameters given in Table 22-30 deduced from the formulas given in [§ 15.4]..

Table 22-30 – Parameters used for each case of benchmark 3-11

Case	Vert.	Time	y_w	P_{air}	D	d	λ	k_H/k_V	φ_{avg}
	[-]	[days]	[m]	[kPa]	[m]	[m]	[m]	[-]	[m]
A	1	200	-2	0	2	0.2	0.770	0.456	-1.833
	2	200	-2	0	40	0.2	17.022	0.456	-1.096
B	1	50/400	-2	0	2	0.2	0.770	0.456	-1.833
	1	200	-10.5	10	2	0.2	0.770	0.456	-7.911
	2	50/400	-2	0	40	0.2	17.022	0.456	-1.096
	2	200	-10.5	10	40	0.2	17.022	0.456	-1.814
C	1	50	-3.5	10	2	0.2	0.770	0.456	-3.894
	1	200	-1.5	5	2	0.2	0.770	0.456	-1.859
	2	50	-3.5	10	40	0.2	17.022	0.456	-1.333
	2	200	-1.5	5	40	0.2	17.022	0.456	-1.097
D	1	200	-1	0	2.5	0.25	1.629	0.465	-1.000
	2	200	-1	0	40	0.25	43.146	0.465	-1.000
E	1	50/400	-1	0	2.825	0.25	1.908	0.465	-1.000
	1	200	-2.5	2.5	2.825	0.25	1.908	0.465	-2.282
	2	50/400	-1	0	45.2	0.25	49.438	0.465	-1.000
	2	200	-2.5	2.5	45.2	0.25	49.438	0.465	-1.022
F	1	50	-5	30	2.625	0.25	1.736	0.465	-6.107
	1	200	-2	0	2.625	0.25	1.736	0.465	-1.747
	2	50	-5	30	42	0.25	45.558	0.465	-1.099
	2	200	-2	0	42	0.25	45.558	0.465	-1.014
G	1	200	-1	0	3.39	0.223	2.450	0.475	-1.000
	2	200	-1	0	45.2	0.223	49.528	0.475	-1.000
H	1	50/400	-1	0	3.15	0.223	2.235	0.475	-1.000
	1	200	-3	5	3.15	0.223	2.235	0.475	-2.703
	2	50/400	-1	0	42	0.223	45.650	0.475	-1.000
	2	200	-3	5	42	0.223	45.650	0.475	-1.034
I	1	50	-1.5	15	3	0.223	2.102	0.475	-2.433
	1	200	-3	0	3	0.223	2.102	0.475	-2.368
	2	50	-1.5	15	40	0.223	43.240	0.475	-1.031
	2	200	-3	0	40	0.223	43.240	0.475	-1.030

MSettle models the effect of vertical drainage by automatically adding a water load. Therefore, a second check has been made in benchmark 4-10 by performing MSettle calculations without vertical drainage but using water loads (in the *Water Loads* window) with the average hydraulic head distribution given in Table 22-30.

MSettle result

Table 22-31 – Results of benchmark 3-11 for sand wall – Settlements

Case	Time	Spreadsheet [m]		MSettle [m]			Relative error [%]	
	[days]	Vert. 1	Vert. 2	File	Vert. 1	Vert. 2	Vert. 1	Vert. 2
A	50	0.300	0.300	bm3-11a	0.302	0.302	0.66	0.66
	200	0.694	0.694		0.695	0.695	0.14	0.14
	400	2.016	1.074		2.016	1.075	0.00	0.09
	10000	2.585	2.568		2.584	2.568	0.04	0.00
B	50	0.280	0.280	bm3-11b	0.281	0.281	0.36	0.36
	200	1.654	0.628		1.654	0.629	0.00	0.16
	400	1.869	0.936		1.860	0.934	0.48	0.21
	10000	1.993	1.972		1.992	1.972	0.05	0.00
C	50	0.556	0.556	bm3-11c	0.556	0.556	0.00	0.00
	200	1.798	0.971		1.794	0.970	0.22	0.10
	400	1.820	1.250		1.818	1.250	0.11	0.00
	10000	2.239	2.202		2.237	2.201	0.09	0.05

Table 22-32 – Results of benchmark 3-11 for column drain – Settlements

Case	Time	Spreadsheet [m]		MSettle [m]			Relative error [%]	
	[days]	Vert. 1	Vert. 2	File	Vert. 1	Vert. 2	Vert. 1	Vert. 2
D	50	0.556	0.556	bm3-11d	0.556	0.556	0.00	0.00
	200	0.948	0.948		0.948	0.948	0.00	0.00
	400	1.754	1.220		1.753	1.220	0.06	0.00
	10000	2.198	2.198		2.197	2.198	0.05	0.00
E	50	0.300	0.300	bm3-11e	0.302	0.302	0.66	0.66
	200	1.635	0.697		1.637	0.698	0.12	0.14
	400	2.001	1.050		1.999	1.051	0.10	0.10
	10000	2.566	2.566		2.566	2.566	0.00	0.00
F	50	0.280	0.280	bm3-11f	0.281	0.281	0.36	0.36
	200	1.606	0.608		1.593	0.609	0.82	0.16
	400	1.708	0.891		1.705	0.891	0.18	0.00
	10000	1.991	1.970		1.988	1.970	0.15	0.00

Table 22-33 – Results of benchmark 3-11 for strip drain – Settlements

Case	Time	Spreadsheet [m]		MSettle [m]			Relative error [%]	
	[days]	Vert. 1	Vert. 2	File	Vert. 1	Vert. 2	Vert. 1	Vert. 2
G	50	0.280	0.280	bm3-11g	0.281	0.281	0.36	0.36
	200	0.605	0.605		0.605	0.605	0.00	0.00
	400	1.436	0.887		1.436	0.888	0.00	0.11
	10000	1.970	1.970		1.969	1.970	0.05	0.00
H	50	0.556	0.556	bm3-11h	0.556	0.556	0.00	0.00
	200	1.431	0.950		1.430	0.950	0.07	0.00
	400	1.789	1.222		1.778	1.221	0.62	0.08
	10000	2.198	2.198		2.197	2.198	0.05	0.00
I	50	0.300	0.300	bm3-11i	0.302	0.302	0.66	0.66
	200	1.533	0.697		1.531	0.698	0.13	0.14
	400	1.965	1.051		1.961	1.052	0.20	0.10
	10000	2.597	2.566		2.591	2.566	0.23	0.00

Table 22-34 – Results of benchmark 3-11 without drains – Settlements

Case	Time	Spreadsheet	MSettle		Relative error
	[days]	[m]	File	[m]	[%]
J	50	0.300	bm3-11j	0.302	0.66
	200	0.694		0.695	0.14
	400	1.045		1.046	0.10
	10000	2.566		2.566	0.00
K	50	0.280	bm3-11k	0.281	0.36
	200	0.605		0.605	0.00
	400	0.885		0.885	0.00
	10000	1.970		1.970	0.00
L	50	0.556	bm3-11l	0.556	0.00
	200	0.948		0.948	0.00
	400	1.218		1.218	0.00
	10000	2.198		2.198	0.00

Use MSettle input files bm3-11a.sli till bm3-11l.sli to run this benchmark.

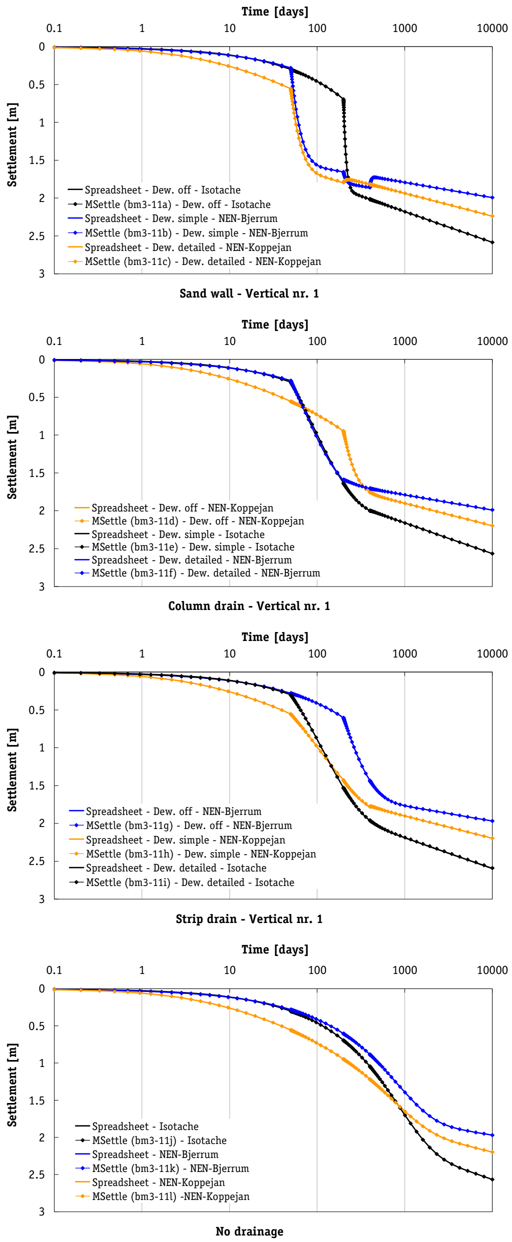


Figure 22-19 – Comparison between MSettle and the spreadsheet settlement curve for vertical 1

22.12 Dissipations for coupling with MStab

Description

A 3-layers system (see Figure 22-20) drained at both sides is loaded with two non-uniform loads with different time application: t_1 = 0 day for the first one and t_2 = 20 days for the second one. The option "Maintain profile" is used by adding a material called "Super-elevation" at time t_{sup} = 30 days. Vertical drainage is used with plane flow. An MStab input file is created by adding non-uniform loads as layer boundaries which become layers number 4 and 5 and by adding the "Super-elevation" material as layer 6.

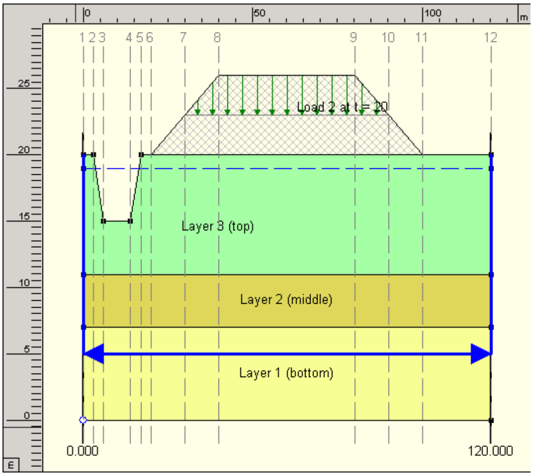


Figure 22-20 – Configuration of benchmark 3-12

Two calculations are performed with two different geometries: for benchmark 3-12a the height of layer 3 is 9 m whereas for benchmark 3-12b it is 4 m. The other characteristics of the layers are given in Table 22-35.

Table 22-35 – Characteristics of the layers

	Layer 1	Layer 2	Layer 3
Coefficient of consolidation c_v [m²/s]	1.96×10^{-6}	6.4×10^{-7}	7.29×10^{-6}
Permeability ratio k_H/k_V	0.1	0.7	0.4

Benchmark

A fictive vertical scale is introduced, called ζ with $0 \leq \zeta \leq H$, in which the pore pressure distribution of the global layer-system is parabolic as shown in Figure 22-21. In this fictive scale, the co-ordinate at the top of each layer i is:

$$\zeta_i = \sum_{k=1}^{i} H_k \cdot \frac{\sqrt{c_v}}{\sqrt{c_{vk}}} \quad \text{for } 1 \leq i \leq 3 \qquad \text{and} \qquad \zeta_0 = 0 \tag{129}$$

and the degree of consolidation of layer i is equal to:

$$U_i(t) = 1 - \frac{h}{\zeta_i - \zeta_{i-1}} \sum_{n=1}^{\infty} \frac{2}{m^2} \cdot e^{-\left(m^2 + \frac{h'^2}{\lambda^2}\right)\frac{c_v t/t_0}{h^2}} \cdot \left[\cos\left(\frac{m\zeta_{i-1}}{h}\right) - \cos\left(\frac{m\zeta_{i-1}}{h}\right)\right] \tag{130}$$

where:

$m = \frac{\pi}{2}(2n-1)$

c_v Global coefficient of consolidation along the drained layers:

$$c_v = \left(H \Big/ \left(\sum_{k=1}^{3} H_k \Big/ \sqrt{c_{vk}}\right)\right)^2$$

h Drainage height of the global system-layers (equal to the half-thickness of the layer-system because both sides are drained).

H Height of the global system-layers:

$$H = \sum_{k=1}^{3} H_k = \begin{cases} 20\text{ m} & \text{for benchmark 3 - 12a} \\ 15\text{ m} & \text{for benchmark 3 - 12b} \end{cases}$$

h' Drainage height along the drain:

$$h' = \frac{y_{surface} - y_{drain}}{2} = \begin{cases} 15\text{ m} & \text{for benchmark 3-12a} \\ 10\text{ m} & \text{for benchmark 3-12b} \end{cases}$$

y_{drain} Bottom position of the drain: y_{drain} = 5 m.

λ Leakage length. For sand wall:

$$\lambda = \sqrt{\frac{1}{12}\frac{k_V}{k_H}}(D - d)$$

D Distance between two drains: D = 6 m.

d Diameter of the drain: d = 0.2 m.

t_0 Creep rate reference time: t_0 = 1 day.

k_V/k_H Equivalent permeability ratio along the drain:

$$\frac{k_H}{k_V} = \frac{\frac{k_{H1}}{k_{V1}}\left(Y_{surface} - H_1 - H_2 - Y_{drain}\right) + \frac{k_{H2}}{k_{V2}} H_2 + \frac{k_{H3}}{k_{V3}} H_3}{Y_{surface} - Y_{drain}}$$

(i.e. 0.44 and 0.46 respectively for benchmarks 3-12a and 3-12b).

NOTE: In MSettle, during the calculation of the degree of consolidation for coupling with other MSeries program, the time application of the vertical drainage is set equal to 0 instead of its inputted time (for this benchmark t_{drains} = 2days).

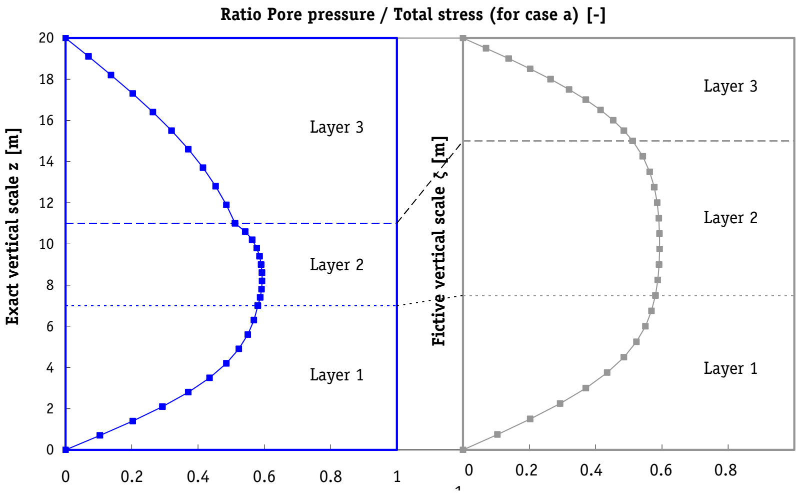


Figure 22-21 – Distribution of the pore pressure dissipation along the layers

Calculations are performed at different time in an Excel spreadsheet and compared to the MSettle results in the three tables below.
An MStab input file can be created from the MSettle file. The non-uniform loads 1 and 2 and the Super-elevation load become material layers (respectively layers 4, 5 and 6). The effect of those three loads on the material layers (layers 1, 2 and 3) is calculated at time t = 35days, see Table 22-36.

Table 22-36 – Degree of consolidation of each layer (bm3-12)

Description		Case a	Case b
Effect of layer 4 on layer 1	$U_1(t - t_1) = U_1(35\text{ days})$	68.92 %	59.22 %
Effect of layer 4 on layer 2	$U_2(t - t_1) = U_2(35\text{ days})$	51.86 %	44.98 %
Effect of layer 4 on layer 3	$U_3(t - t_1) = U_3(35\text{ days})$	76.20 %	84.50 %
Effect of layer 5 on layer 1	$U_1(t - t_2) = U_1(15\text{ days})$	44.53 %	37.63 %
Effect of layer 5 on layer 2	$U_2(t - t_2) = U_2(15\text{ days})$	25.71 %	21.33 %
Effect of layer 5 on layer 3	$U_3(t - t_2) = U_3(15\text{ days})$	53.62 %	71.20 %
Effect of layer 6 on layer 1	$U_1(t - t_{super}) = U_1(5\text{ days})$	22.75 %	19.67 %
Effect of layer 6 on layer 2	$U_2(t - t_{super}) = U_2(5\text{ days})$	9.29 %	6.50 %
Effect of layer 6 on layer 3	$U_3(t - t_{super}) = U_3(5\text{ days})$	29.47 %	50.11 %

The effect of load layers on themselves is nil (i.e. 100 %) as well as the effect of material layers on themselves.

MSettle result

Two calculations are performed with MSettle using two different verticals for the dissipation calculation: for benchmark 3-12a, vertical 1 (X = 0 m) is used (i.e. h_3 = 9 m) whereas for benchmark 3-12b vertical 3 (X = 6 m) is used (i.e. h_3 = 4 m).

The values of the dissipation ratio are found using the *View Data* option in *Dissipations* window. In order to check the coupling with MStab, an input file is created using the *Write MStab input* option in the *Results* menu at time t = 35 days. In MStab, the values of the degree of consolidation in the *Water* menu are checked.

Table 22-37 – Results of benchmark 3-12a – Dissipations

	Time [days]	MSettle [%]	Benchmark [%]	Relative error [%]
Layer 1	2	12.439	12.439	0.00
	5	21.837	21.837	0.00
	10	33.439	33.439	0.00
	20	50.043	50.043	0.00
	30	61.688	61.688	0.00
	80	88.300	88.300	0.00
Layer 2	2	3.374	3.374	0.00
	5	8.225	8.225	0.00
	10	15.851	15.851	0.00
	20	29.879	29.879	0.00
	30	42.241	42.241	0.00
	80	79.828	79.828	0.00
Layer 3	2	16.971	16.971	0.00
	5	28.641	28.641	0.00
	10	42.155	42.155	0.00
	20	59.475	59.475	0.00
	30	70.215	70.215	0.00
	80	91.558	91.558	0.00

Table 22-38 – Degree of consolidation in MStab (bm3-12aAt35.sti)

	MStab [%]	Benchmark [%]	Relative error [%]
Effect of superelevation load on layer 3	29	29	0.00
Effect of superelevation load on layer 2	8	8	0.00
Effect of superelevation load on layer 1	22	22	0.00
Effect of load 2 on layer 3	52	52	0.00
Effect of load 2 on layer 2	23	23	0.00
Effect of load 2 on layer 1	43	43	0.00
Effect of load 1 on layer 3	74	74	0.00
Effect of load 1 on layer 2	48	48	0.00
Effect of load 1 on layer 1	66	66	0.00

Table 22-39 – Results of benchmark 3-12b – Dissipations

	Time [days]	MSettle [%]	Benchmark [%]	Relative error [%]
Layer 1	2	11.310	11.310	0.00
	5	19.292	19.292	0.00
	10	29.034	29.034	0.00
	20	43.216	43.216	0.00
	30	53.607	53.607	0.00
	80	81.195	81.195	0.00
Layer 2	2	2.171	2.171	0.00
	5	6.066	6.066	0.00
	10	13.268	13.268	0.00
	20	26.664	26.664	0.00
	30	38.096	38.096	0.00
	80	73.725	73.725	0.00
Layer 3	2	32.969	32.969	0.00
	5	49.871	49.871	0.00
	10	63.410	63.410	0.00
	20	75.636	75.636	0.00
	30	81.837	81.837	0.00
	80	93.491	93.491	0.00

Table 22-40 – Degree of consolidation in MStab (bm3-12bAt35.sti)

	MStab [%]	Benchmark [%]	Relative error [%]
Effect of superelevation load on layer 3	50	50	0.00
Effect of superelevation load on layer 2	6	6	0.00
Effect of superelevation load on layer 1	19	19	0.00
Effect of load 2 on layer 3	71	71	0.00
Effect of load 2 on layer 2	20	20	0.00
Effect of load 2 on layer 1	37	37	0.00
Effect of load 1 on layer 3	84	84	0.00
Effect of load 1 on layer 2	43	43	0.00
Effect of load 1 on layer 1	58	58	0.00

Use MSettle input files bm3-12a.sli and bm3-12b.sli to run this benchmark.

22.13 Effect of the stress distribution simulated inside non-uniform loads

Description

This benchmark checks the functioning of the option *Simulate Stress distribution in Loads* in the *Calculation Options* window [§ 10.1.2] available for non-uniform loads. A single layer (height of 20 m) is loaded with a trapezoidal load (unit weight: γ = 18 kN/m^3; maximal height: H = 4 m; width left side: x_{left} = 20 m; width middle: x_{middle} = 20 m; width right side: x_{right} = 20 m). The stress distribution is calculated according to Boussinesq theory. Three calculations are performed with MSettle:

- bm3-13a: Option Simulate Stress distribution in Loads is ON;
- bm3-13b: Option Simulate Stress distribution in Loads is OFF.

Benchmark

The change in vertical stress due to this trapezoidal load is checked by dividing the load into parts of 1 meter height, as done by MSettle. Equation (11) page 279 is used. The final vertical effective stress at -10 m depth is calculated at 5 location s, see Table 22-41.

MSettle result

The Boussinesq soil stress distribution in the *Calculation Option* window must be chosen. The final effective stresses are compared with the benchmark results in Table 22-41.

Table 22-41 – Results of benchmark 3-13 – Vertical effective stress at -10 m depth

Effect of stress distribution in load	X co-ordinate [m]	Benchmark [kPa]	MSettle [kPa]	Relative error [%]
ON	20	72.56	72.54	0.03
	30	97.80	97.87	0.07
	40	120.54	120.54	0.00
	50	129.30	129.28	0.02
	60	130.83	130.81	0.02
OFF	20	71.39	71.39	0.00
	30	96.73	96.73	0.00
	40	122.04	122.04	0.00
	50	130.49	130.49	0.00
	60	131.59	131.59	0.00

Use MSettle input files bm3-13a.sli and bm3-13b.sli to run this benchmark.

22.14 Effect of the dispersion conditions at layer boundaries (Terzaghi consolidation)

Description
This benchmark checks the functioning of the option *Dispersion conditions layer boundaries* in the *Calculation Options* window [§ 10.1.2] available for Terzaghi consolidation model. The same oedometer test that the one used for benchmark 3-8b [§ 22.8] is performed for Isotache model with Terzaghi consolidation using two different types of dispersion conditions:

- Case a: one of the sample side is drained and the other is undrained (bm3-14a);
- Case b: both sample sides are undrained (bm3-14b).

The condition where both sample sides are drained was already checked is benchmark 3-8b [§ 22.8].

Benchmark
The analytical formula is the same as benchmark 3-8b [§ 22.8] except the value of the drainage height which is now equal to the total height of the sample instead of half of it ($H_{drainage}$ = H_{sample} = 20 mm for both cases). MSettle results are compared to an analytical solution worked out in an Excel spreadsheet.

MSettle result
The settlements calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison (see figures below). The settlements and the dissipations in time are respectively given in Table 22-42 and Table 22-43.

Table 22-42 – Results of benchmark 3-14 – Settlements in time

Dispersion	Time	Benchmark	MSettle		Relative error
conditions	[days]	[mm]	File	[mm]	[%]
Drained at only one side	10	1.41	bm3-14a	1.42	0.70
	40	3.21		3.21	0.00
	80	5.31		5.32	0.19
Undrained at both sides	10	1.41	bm3-14b	1.42	0.70
	40	3.21		3.21	0.00
	80	5.31		5.32	0.19

Table 22-43 – Results of benchmark 3-14 – Dissipations in time

Dispersion conditions	Time [days]	Benchmark [%]	MSettle File	[%]	Relative error [%]
Drained at only one side	0.1	2.62	bm3-14a	2.62	0.00
	0.95	8.08		8.07	0.12
	9.66	25.77		25.79	0.08
	80	72.08		72.11	0.04
Undrained at both sides	0.1	2.62	bm3-14b	2.62	0.00
	0.95	8.08		8.07	0.12
	9.66	25.77		25.79	0.08
	80	72.08		72.11	0.04

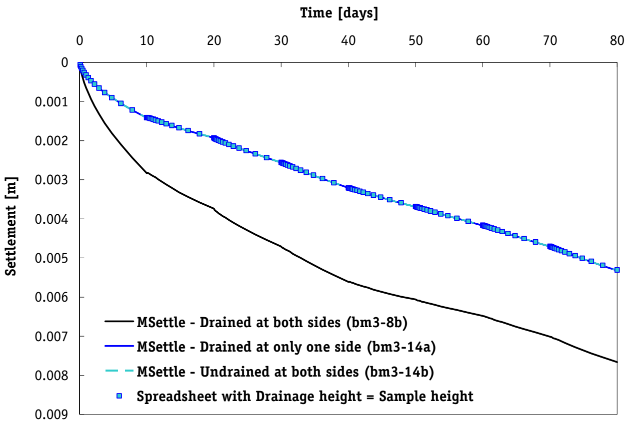


Figure 22-22 – Results of benchmark 3-14 – Comparison between MSettle and the spreadsheet settlement results

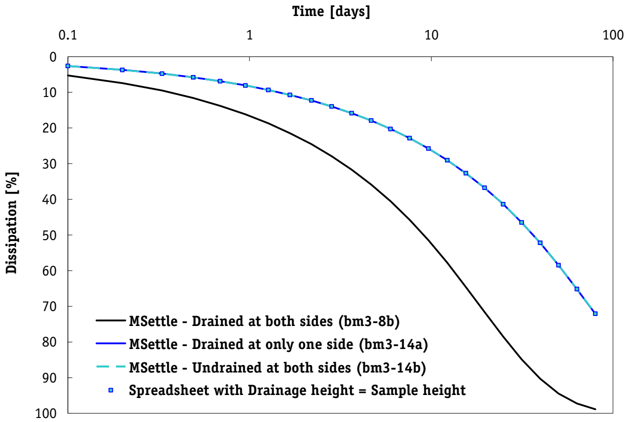


Figure 22-23 – Results of benchmark 3-14 – Comparison between MSettle and the spreadsheet dissipation results

Use MSettle input files bm3-14a.sli and bm3-14b to run this benchmark.

22.15 Reliability analysis using FOSM method

Description

A probabilistic calculation using the FOSM method is performed for several combinations of soil model, consolidation model, storage type, compression type (*POP*, *OCR*, σ_p or equivalent age) variable and probabilistic parameter types as shown in Table 22-44.

For a detailed description of the geometry, loading and soil parameters used for each benchmark, refer to [Lit 25].

Table 22-44 – Cases overview for benchmark bm3-15

Case	Soil model	Consolid. model	Storage	Geom.	Load	Variables	Distrib.
A	Koppejan	Darcy	Drained	1 layer	Load Unload	γ_{dry} γ_{wet} C_p C_p' C_s C_s' A_p A_s OCR	Normal
B	Koppejan	Terzaghi	Drained	1 layer	Load	C_p C_p' C_s C_s' P_c	Normal
C	Bjerrum	Darcy	Drained	1 layer	Load	γ_{dry} γ_{wet} C_α RR CR OCR	Normal
D	Bjerrum	Terzaghi	Cv	1 layer	Load	C_v C_α RR CR POP	Normal
E	Bjerrum	Terzaghi	Drained	2 layers	Load	RR_1 RR_2 Z_{bound}	Normal
F	Isotache	Darcy	Drained	1 layer	Load	a b c P_c	Normal

Benchmark
The analytical solution has been solved in [Lit 25]. Calculations are performed at four different times (10, 100, 1000 and 1000 days) in an Excel spreadsheet and results are given in Table 22-45.

MSettle result
The band width results for a confidence interval of 95% can be found using the *View Data* option in the *Time-History (Reliability)* window.

Table 22-45 – Results of benchmark bm3-15

Case	Time	Spreadsheet		MSettle		Relative error	
		Mean settl.	Band width 95%	Mean settl.	Band width 95%	Mean settl.	Band width 95%
	[days]	[m]	[m]	[m]	[m]	[%]	[%]
A	10	0.0777	0.0186	0.0778	0.0182	0.13	2.20
	100	0.0899	0.0216	0.0900	0.0212	0.11	1.89
	1000	0.0302	0.0238	0.0302	0.0235	0.00	1.28
	10000	0.0308	0.0281	0.0309	0.0278	0.32	1.08
B	10	0.0515	0.0183	0.0515	0.0182	0.00	0.55
	100	0.0606	0.0202	0.0606	0.0200	0.00	1.00
	1000	0.0701	0.0228	0.0701	0.0227	0.00	0.44
	10000	0.0795	0.0259	0.0795	0.0257	0.00	0.78
C	10	0.2440	0.0631	0.2440	0.0625	0.00	0.96
	100	0.2890	0.0686	0.2891	0.0680	0.03	0.88
	1000	0.3340	0.0759	0.3341	0.0755	0.03	0.53
	10000	0.3790	0.0848	0.3791	0.0843	0.03	0.59
D	10	0.0087	0.0018	0.0087	0.0017	0.00	5.88
	100	0.0360	0.0065	0.0360	0.0065	0.00	0.00
	1000	0.1398	0.0241	0.1398	0.0241	0.00	0.00
	10000	0.2817	0.0403	0.2817	0.0402	0.00	0.25
E	10	0.2110	0.0044	0.2110	0.0043	0.00	2.33
	100	0.2110	0.0044	0.2110	0.0043	0.00	2.33
	1000	0.2110	0.0044	0.2110	0.0043	0.00	2.33
	10000	0.2110	0.0044	0.2110	0.0043	0.00	2.33
F	10	0.1352	0.0395	0.1352	0.0396	0.00	0.25
	100	0.1516	0.0404	0.1516	0.0404	0.00	0.00
	1000	0.1673	0.0424	0.1673	0.0423	0.00	0.24
	10000	0.1823	0.0449	0.1823	0.0449	0.00	0.00

Use MSettle input files bm3-15a.sli till bm3-15f.sli to run this benchmark.

23 Benchmarks generated by MSettle

These benchmarks are intended to verify specific features of MSettle using reference results generated with MSettle itself.

23.1 Settlements curve during consolidation process – Comparison between Darcy and Terzaghi models in a simple case

Description

This benchmark tests the Terzaghi consolidation model by comparing Terzaghi settlement curve with Darcy settlement curve from benchmark 3-9 [§ 22.9]. The hydraulic head curves calculated by Darcy model with a consolidation coefficient of C_v = 0.0002 m^2/s have been checked in benchmarks 3-9a (Isotache model) and 3-9b (NEN-Koppejan model). In this benchmark, the settlement curves of those two benchmarks are compared to the settlement curves calculated by MSettle with the Terzaghi consolidation model and an identical consolidation coefficient of C_v = 0.0002 m^2/s.

MSettle result

The settlements calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison (see Figure 23-1). The maximum relative errors are given in Table 22-18. Results are very close.

Table 23-1 – Results of benchmark 4-1 – Settlements calculated by MSettle for Darcy and Terzaghi models

Soil model	Time [days]	MSettle with Darcy (Cv)		MSettle with Terzaghi		Relative error [%]
		File name	Settlement	File name	Settlement	
Isotache	0.94	bm4-1a	0.0087	bm3-9a	0.0088	1.14
	4.65		0.0169		0.0166	1.81
	31.21		0.0191		0.0190	0.53
	100		0.0191		0.0191	0.00
NEN-Koppejan	0.94	bm4-1b	0.0087	bm3-9b	0.0088	1.14
	4.65		0.0169		0.0166	1.81
	31.21		0.0191		0.0191	0.00
	100		0.0191		0.0191	0.00

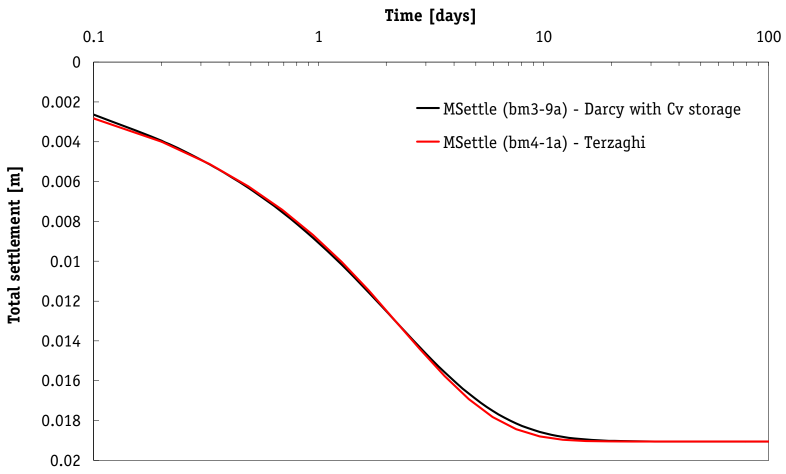


Figure 23-1 – MSettle settlement curves – Comparison between Darcy (with consolidation coefficient) and Terzaghi consolidation models

Use MSettle input files bm4-1a.sli and bm4-1b.sli to run this benchmark.

23.2 Settlements curve during consolidation process – Comparison between Darcy and Terzaghi models in a complex case

Description

This benchmark compares the settlements curve calculated by MSettle for both Terzaghi and Darcy consolidation models.

A first test consists in comparing both consolidation models in case layers are drained (see files bm4-2a and bm4-2b for Terzaghi and Darcy respectively). Results

are expected to be exactly the same as drained layers are not influenced by the consolidation model.
A second test consists in comparing both consolidation models in case consolidation process is present (see files bm4-2c and bm4-2d for Terzaghi and Darcy respectively). Therefore the Darcy model uses the same coefficients of consolidation as Terzaghi model. Results are expected to be different during the consolidation process but final settlements (end of consolidation) should be the same.

MSettle result
The settlements calculated by MSettle are exported to the spreadsheet using the *View Data* option in *Time-History* window for comparison (see Figure 23-2). Table 23-2 shows that the final settlement (i.e. end of consolidation) is the same in all cases.

Table 23-2 – Results of benchmark 4-2 – Comparison of the final settlements using Darcy and Terzaghi consolidation models

Drainage	MSettle - Terzaghi consolidation		MSettle - Darcy (Cv) consolidation		Relative error
	File name	[m]	File name	[m]	[%]
Drained	bm4-2a	3.50	bm4-2b	3.50	0.00
Consolidated	bm4-2c	3.50	bm4-2d	2.97	17.85

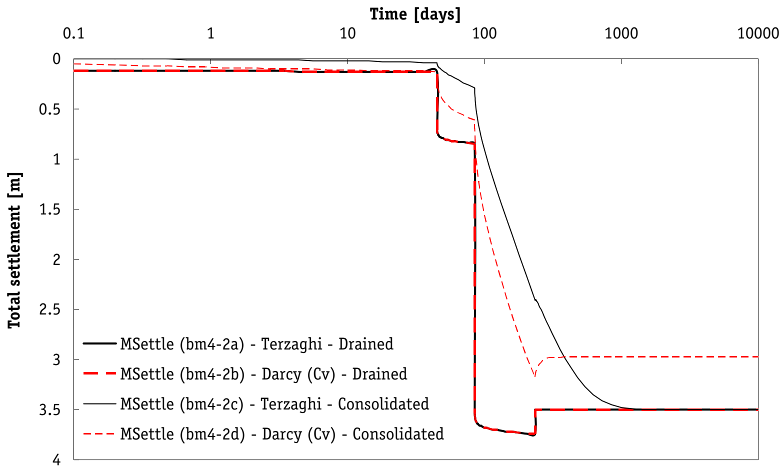


Figure 23-2 – MSettle results – Comparison between Darcy and Terzaghi consolidation models (Isotache model)

Use MSettle input files bm4-2a.sli and bm4-2d.sli to run this benchmark.

23.3 Settlement using the Maintain Profile option

Description

A 4-layers system is loaded with a non-uniform load (height H_{load} = 2 m, dry weight γ_{unsat} = 17.5 kN/m^3 and wet weight γ_{sat} = 20 kN/m^3).
On one hand, a calculation with the *Maintain Profile* option is performed for the three models (NEN-Koppejan, NEN-Bjerrum and Isotache) in combination with the two consolidation models (Terzaghi and Darcy) in six different files (bm4-3a till bm4-3f). The *Maintain Profile* option starts at time *t* = 60 days and uses a *Sand* filling material with a dry weight of γ_{unsat} = 17.5 kN/m^3 and a wet weight of γ_{sat} = 20 kN/m^3.
On the other hand, a second calculation is performed (for the six combinations of models; bm3-g till bm3-l) without the *Maintain Profile* option, but using a "compensation" non-uniform load with the following characteristics:

- A height equal to the final settlement calculated with the *Maintain Profile* option, for each vertical
- A unit weight equal to the unit weight of the *Sand* filling material (see above).

The extra amount of soil to be added to maintain the original profile for both type of calculation are compared for each model (see Table 23-3) and expected to be the same.

MSettle result

The accuracy for the *Maintain Profile* option is set to its minimum (0.01 m) in the *Calculation Options* window of MSettle.
The settlements of the different verticals calculated with MSettle using the *Maintain Profile* option (bm4-3a to bm4-3f) are given in Table 23-3 and used as input values for the height of the compensation load (bm4-3g to bm4-3l). Due to symmetry, only half of the vertical results are given.

Table 23-3 – Results of benchmark 4-3 – Settlements using the *Maintain Profile* option

X co-ordiante	bm4-3a	bm4-3b	bm4-3c	bm4-3d	bm4-3e	bm4-3f
0	0.014	0.145	1.095	1.128	0.438	0.502
20	1.188	1.340	1.374	1.400	0.900	0.973
25	2.252	2.351	1.588	1.602	1.378	1.422
30	2.778	2.847	1.690	1.702	1.631	1.664
35	3.112	3.165	1.764	1.775	1.805	1.832
40	3.321	3.363	1.815	1.825	1.919	1.942
45	3.415	3.457	1.835	1.845	1.963	1.986
50	3.451	3.493	1.840	1.851	1.978	2.002
55	3.465	3.507	1.842	1.853	1.983	2.007
60	3.469	3.511	1.843	1.853	1.985	2.009

Settlement results and shape of the loads are represented in Figure 23-3 for NEN-Koopejan model with Terzaghi consolidation. Note that the original shape of the load coincide with the shape of the compensation load after settlement which means that the original profile has been maintained thanks to the compensation load.

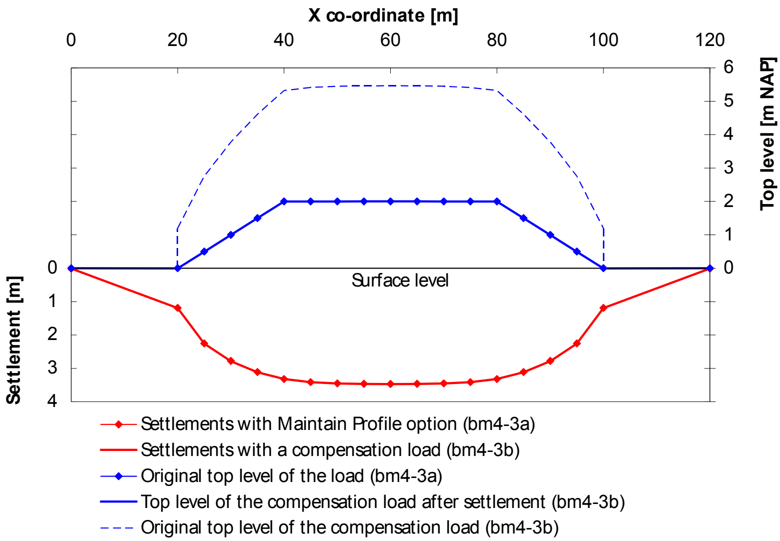


Figure 23-3 – MSettle results – Comparison of the final settlements and the load shape according to the *Maintain Profile* option (bm4-3a) and the compensation load (bm4-3b)

The extra amount of soil to be added to maintain the original profile is given in Table 23-4 for the six combinations of models.

Table 23-4 – Results of benchmark 4-3 – Extra amount of soil to be added to maintain the original profile

Soil model	Consolidation model	MSettle with *Maintain Profile* option		MSettle with a compensation load		Relative error [%]
		File name	Volume [m^3/m]	File name	Volume [m^3/m]	[%]
NEN-Koppejan	Terzaghi	bm4-3a	240.304	bm4-3g	241.225	0.38
	Darcy	bm4-3b	245.275	bm4-3h	246.085	0.33
NEN-Bjerrum	Terzaghi	bm4-3c	139.526	bm4-3i	139.825	0.21
	Darcy	bm4-3d	140.508	bm4-3j	140.795	0.20
Isotache	Terzaghi	bm4-3e	140.410	bm4-3k	140.995	0.41
	Darcy	bm4-3f	142.920	bm4-3l	143.460	0.38

Use MSettle input files bm4-3a.sli till bm4-3l.sli to run this benchmark.

23.4 Fit factors from a Fit for Settlement Plate calculation

Description

A measurement file (*.slm file) needed for the fitting is generated with MSettle by multiplying the different parameters by a known fit-factor. Verifications are performed for NEN-Koppejan, NEN-Bjerrum and Isotache models in combination with Terzaghi and Darcy consolidation models.

An embankment with a 100 kN/m^3 unit weight material and a ½ slope is constructed on a two layers system using the following load-steps (see Figure 23-4):

- at t_1 = 35 day, top level of the embankment at 2 m height above surface level;
- at t_2 = 45 day, top level of the embankment at 5 m height above surface level;
- at t_3 = 85 days, top level of the embankment at 7.5 m height above surface level;
- at t_4 = 235 days, embankment removed.

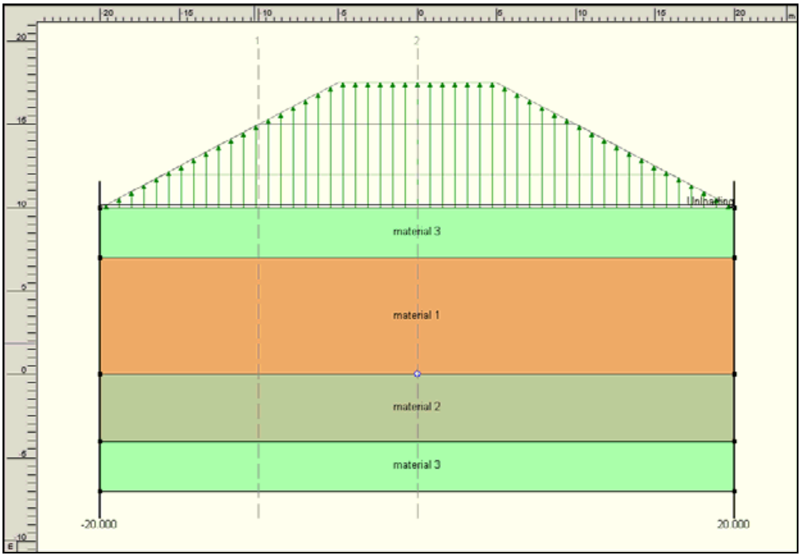


Figure 23-4 – Geometry of benchmark 4-4

The material properties are given in Table 23-5. A shift time of 35 days and a shift settlement of -0.3 m are used.

Table 23-5 – Materials properties (bm4-4)

Parameters	Unit	Material 1 (bottom)	Material 2 (top)
C_v	[m^2/s]	5×10^{-7}	1×10^{-6}
k_v	[m/d]	3×10^{-4}	6×10^{-4}
OCR	[-]	1.4	1.8
NEN-Bjerrum soil model:			
RR	[-]	0.05	0.03
CR	[-]	0.5	0.3
C_α	[-]	0.05	0.03
NEN-Koppejan soil model:			
C_p	[-]	50	25
C_p'	[-]	10	5
C_s	[-]	300	400
C_s'	[-]	80	100
A_p	[-]	50	25
A_s	[-]	300	400
Isotache soil model:			
a	[-]	0.05	0.03
b	[-]	0.5	0.3
c	[-]	0.05	0.03

Measurement files (.slm) generated with MSettle*

The measurement files are created using MSettle settlement curve results for the same geometry, but using material parameters multiplied by known fit-factors (see values in Table 23-6 to Table 23-11).

In order to take into account the shift settlement, a settlement of 0.3 m is added to the output settlements. In order to take into account the shift time, the loading steps are shifted by 35 days which means that the time-steps are chronologiquely t_1 = 0 day, t_2 = 10 days, t_3 = 50 days and t_4 = 200 days.

MSettle result

In the *Fit for Settlement Plate* window, the fit is performed using a required iteration accuracy of 0 and, a required coefficient of determination of 1 and a number of iterations of 20.

Two fits are performed for each case in order to check the effect of the weight: "fit 1" uses default weight values (found by clicking the *Reset* button) in the *Fit for Settlement Plate* window whereas "fit 2" optimizes the weight to get the expected convergence for the fit factors. That's why results for fit 2 are better than fit 1.

Table 23-6 – Results of benchmark 4-4g – NEN-Koppejan model with Terzaghi consolidation

Used fit factors in SLM file [-]		Fit 1 (default weight) MSettle [-]	Weight [-]	Error [%]	Fit 2 MSettle [-]	Weight [-]	Error [%]
C_p'/C_p	1	1.015	10	1.48	1.002	100	0.20
$1/C_p'$	2	1.872	4	6.84	2.003	4	0.15
C_p'/C_s'	1.25	1.077	10	16.06	1.245	9	0.40
OCR	1.8	0.001	3	79900.00	0.806	20	0.74
C_v	5	4.732	1	5.66	4.992	1	0.16
r^2	1	1.000	-	0.00	1.000	-	0.00

Table 23-7 – Results of benchmark 4-4h – NEN-Koppejan model with Darcy consolidation

Used fit factors in SLM file [-]		Fit 1 (default weight) MSettle [-]	Weight [-]	Error [%]	Fit 2 MSettle [-]	Weight [-]	Error [%]
C_p'/C_p	1	1.085	10	7.83	1.014	100	1.38
$1/C_p'$	2	1.602	4	24.84	1.990	4	0.50
C_p'/C_s'	1.25	1.948	10	35.83	1.262	2	0.95
OCR	1.8	0.280	3	185.71	0.809	25	1.11
k_v	5	4.666	1	7.16	5.009	1	0.18
r^2	1	0.997	-	0.30	1.000	-	0.00

Table 23-8 – Results of benchmark 4-4i – NEN-Bjerrum model with Terzaghi consolidation

Used fit factors in SLM file		Fit 1 (default weight)			Fit 2		
		MSettle	Weight	Error	MSettle	Weight	Error
[-]		[-]	[-]	[%]	[-]	[-]	[%]
RR/CR	1	1.066	10	6.19	1.007	100	0.70
CR	1.5	1.437	4	4.38	1.498	1	0.13
C_α/CR	0.9	1.020	10	11.76	0.904	1	0.44
OCR	1.8	1.813	3	0.72	1.803	3	0.17
C_v	3	2.992	1	0.27	2.998	1	0.07
r^2	1	1.000	-	0.00	1.000	-	0.00

Table 23-9 – Results of benchmark 4-4j – NEN-Bjerrum model with Darcy consolidation

Used fit factors in SLM file		Fit 1 (default weight)			Fit 2		
		MSettle	Weight	Error	MSettle	Weight	Error
[-]		[-]	[-]	[%]	[-]	[-]	[%]
RR/CR	1	1.069	10	6.45	1.004	100	0.40
CR	1.5	1.023	4	46.63	1.454	1	3.16
C_α/CR	0.9	0.885	10	1.69	0.920	10	2.17
OCR	1.8	1.428	3	26.05	1.817	1	0.94
k_v	2	2.605	1	23.22	2.078	1	3.75
r^2	1	1.000	-	0.00	1.000	-	0.00

Table 23-10 – Results of benchmark 4-4k – Isotache model with Terzaghi consolidation

Used fit factors in SLM file		Fit 1 (default weight)			Fit 2		
		MSettle	Weight	Error	MSettle	Weight	Error
[-]		[-]	[-]	[%]	[-]	[-]	[%]
a/b	1	1.018	10	1.77	1.003	100	0.30
b	1.5	1.442	4	4.02	1.504	0.1	0.27
c/b	0.9	1.012	10	11.07	0.892	3	0.90
OCR	2	2.030	3	1.48	1.998	4	0.10
C_v	3	3.018	1	0.60	2.999	2	0.03
r^2	1	1.000	-	0.00	1.000	-	0.00

Table 23-11 – Results of benchmark 4-4l – Isotache model with Darcy consolidation

Used fit factors in SLM file [-]		Fit 1 (default weight) MSettle [-]	Weight [-]	Error [%]	Fit 2 MSettle [-]	Weight [-]	Error [%]
a/b	1	1.089	10	8.17	1.001	100	0.10
b	1.5	1.185	4	26.58	1.512	1	0.79
c/b	0.9	0.784	10	14.80	0.894	9	0.67
OCR	2	1.686	3	18.62	2.015	0.5	0.74
k_v	2	2.455	1	18.53	1.997	1	0.15
r^2	1	1.000	-	0.00	1.000	-	0.00

Use MSettle input files bm4-4a.sli till bm4-4l.sli to run this benchmark.

23.5 Initial stresses using Imaginary Surface option

Description

The initial stress distribution at verticals X = 0 and X = 10 m is calculated for a 2-layers system composed of a bottom layer of 5 m height (γ_{unsat} = 17 kN/m^3 and γ_{sat} = 20 kN/m^3) and a top layer (γ_{unsat} = 14 kN/m^3 and γ_{sat} = 16 kN/m^3) with a trapezoidal form (slope of ½ and maximal height of 4 m). The imaginary surface is assumed to be the top of the bottom layer (i.e. level 0 m NAP). The phreatic line is at level 4 m NAP.

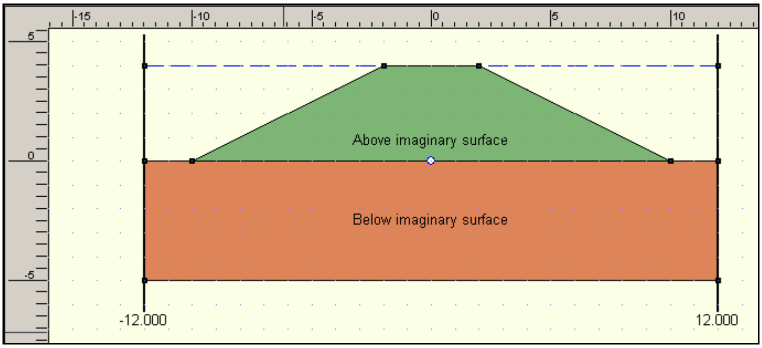


Figure 23-5 – Geometry of benchmark 4-5

The initial stress distribution of this 2-layers system is calculated with MSettle using the *Imaginary Surface* option. Results are compared to the final stress distribution calculated by MSettle without the *Imaginary Surface* option but by modeling the top layer as a trapeziform load with the same properties. Results are expected to be the same.

MSettle result
For cases without *Imaginary Surface* option, the final stress distribution is calculated with MSettle (see bm4-5g.sli) for a 1-layer system (γ_{unsat} = 17 kN/m^3 and γ_{sat} = 20 kN/m^3) loaded with a trapeziform load which has the same form and weight that the previous top layer. Final effective stress distribution calculated by MSettle is given in Table 23-12 (see column bm4-5).

For case with *Imaginary Surface* option, the initial effective stress distribution calculated by MSettle using the *Imaginary Surface* option are found in the *Report* window and written in Table 23-12.

The verification is perfomed for the six combinations of models and results are identical:

- bm4-5a: NEN-Koppejan soil model with Terzaghi consolidation model
- bm4-5b: NEN-Koppejan soil model with Darcy consolidation model
- bm4-5c: NEN-Bjerrum soil model with Terzaghi consolidation model
- bm4-5d: NEN- Bjerrum soil model with Darcy consolidation model
- bm4-5e: Isotache soil model with Terzaghi consolidation model
- bm4-5f: Isotache soil model with Darcy consolidation model

Table 23-12 – Results of benchmark 4-5 – Effective stress distribution using the *Imaginary Surface* option

Vertical X [m]	Depth [m]	MSettle (bm4-5g) Final stresses [kPa]	MSettle (bm4-5) Initial stresses [kPa]	Relative error [%]
0 m	0	24.00	24.00	0.00
	-2.5	48.29	48.29	0.00
	-5	70.94	70.94	0.00
10 m	0	0.75	0.75	0.00
	-2.5	26.93	26.93	0.00
	-5	53.58	53.58	0.00

Use MSettle input files bm4-5a.sli till bm4-5g.sli to run this benchmark.

23.6 Initial stresses due to an Initial Load

Description
The same geometry as benchmark 4-5g [§ 23.5] is used. The initial stress distribution at verticals X = 0 m and X = 10 m is calculated for a layer load with an initial trapeziform load. Results are compared to the final stress distribution calculated by MSettle using the same trapeziform load applied at time 0 day instead of as an initial load. Results are expected to be the same.

MSettle result

For cases without Initial Load, the final stress distribution is calculated with MSettle (see bm4-5g.sli) for a 1-layer system (γ_{unsat} = 17 kN/m^3 and γ_{sat} = 20 kN/m^3) loaded with a trapeziform load (slope of ½ and maximal height of 4 m). Final effective stress distribution calculated by MSettle is given in Table 23-13 (third column).

For cases with Initial Load, the initial effective stress distributions calculated by MSettle using an initial load are found in the *Depth-History* window and written in Table 23-13.

The verification is perfomed for the six combinations of models and results are identical:

- bm4-6a: NEN-Koppejan soil model with Terzaghi consolidation model
- bm4-6b: NEN-Koppejan soil model with Darcy consolidation model
- bm4-6c: NEN-Bjerrum soil model with Terzaghi consolidation model
- bm4-6d: NEN- Bjerrum soil model with Darcy consolidation model
- bm4-6e: Isotache soil model with Terzaghi consolidation model
- bm4-6f: Isotache soil model with Darcy consolidation model

Table 23-13 – Results of benchmark 4-6 – Effective stress distribution using a trapeziform initial load

Vertical X [m]	Depth [m NAP]	MSettle (bm4-5g) Final stresses [kPa]	MSettle (bm4-6) Initial stresses [kPa]	Relative error [%]
0 m	0	24.00	24.00	0.00
	-2.5	48.29	48.29	0.00
	-5	70.94	70.94	0.00
10 m	0	0.75	0.75	0.00
	-2.5	26.93	26.93	0.00
	-5	53.58	53.58	0.00

Use MSettle input files bm4-6a.sli till bm4-6f.sli to run this benchmark.

23.7 Comparison of Isotache, NEN-Bjerrum and NEN-Koppejan settlements using conversion formulas

Description

A clay layer is loaded with an initial load of 1 kPa and a uniform load of σ_{load} = 10 kPa in case of single loading (bm4-7a to c) and 8 load-steps starting with 1 kPa and double every year 10 days in case of oedometer test (bm4-7d to f). The same geometry as benchmark 3-1 [§ 22.1] is used. Settlements are calculated for the three soil models using Terzaghi consolidation. Parameters of Isotache and NEN-Bjerrum models are deduced from NEN-Koppejan parameters (C_p = 30, C_p' = 10, C_s = 60, C_s' = 30, σ_p = 10 kPa, $C_v = 6 \times 10^{-8}$ m²/s) using the conversion formulas, see [§ 17.7].

MSettle input

As the height of the clay layer (γ_{sat} = 14 kN/m³) is only 20 mm, the initial effective stress distribution is set constant (σ_0' = 1.04 kPa).

The conversion is based on the condition that the strain contributions are set equal at the final time t = 10000 days with an effective stress of $\sigma' = \sigma_0' + \sigma_{load}$ = 11.08 kPa. The NEN-Koppejan and NEN-Bjerrum linear parameters are deduced from the Isotache natural parameters using the conversion formulas given in [§ 17.1]. This leads to the parameters given in the following table.

Table 23-14 – Isotache and NEN-Bjerrum parameters deduced from conversion

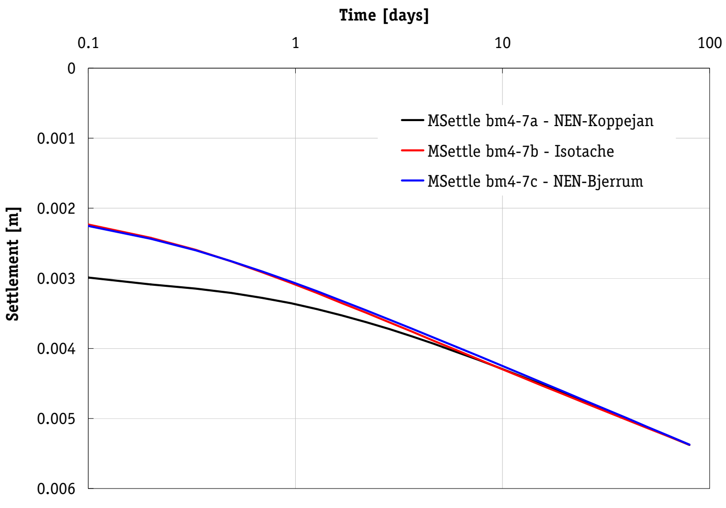
		Single load step	Oedometer test
RR	[-]	0.0767528	0.1097234
CR	[-]	0.2302585	0.3054891
C_α	[-]	0.0624900	0.0769798
a	[-]	3.466E-02	5.042E-02
b	[-]	1.128E-01	2.030E-01
c	[-]	3.439E-02	8.704E-02
$\varepsilon_{p;prim}^{C}$	[-]	0.07538	0.10777
ε_{prim}^{C}	[-]	0.14978	0.53799

MSettle result

The settlements calculated by MSettle are exported to the spread sheet using the *View Data* option in *Time-History* window for comparison (see Figure 23-6). The relative error is given in Table 23-14.

Table 23-15 – Results of benchmark 4-7 – Settlements at 0.1 and 100000 days

	Time	MSettle			Relative error	
		Koppejan	Isotache	Bjerrum	Isotache	NEN-Bjerrum
	[days]	[mm]	[mm]	[mm]	[%]	[%]
Single load-step		(bm4-7a)	(bm4-7b)	(bm4-7c)		
	0.1	2.99	2.23	2.25	34.08	32.89
	9.66	4.28	4.28	4.23	0.00	1.18
	39.74	5.01	5.02	4.99	0.20	0.40
	80	5.38	5.37	5.37	0.19	0.19
Oedometer test		(bm4-7d)	(bm4-7e)	(bm4-7f)		
	10	0.68	1.46	0.75	53.42	9.33
	20	1.44	3.31	1.81	56.50	20.44
	30	2.24	5.29	3.33	57.66	32.73
	40	3.86	7.16	5.09	46.09	24.17
	50	5.92	8.82	6.91	32.88	14.33
	60	8.04	10.28	8.74	21.79	8.01
	70	10.22	11.55	10.58	11.52	3.40
	80	12.42	12.66	12.41	1.90	0.08



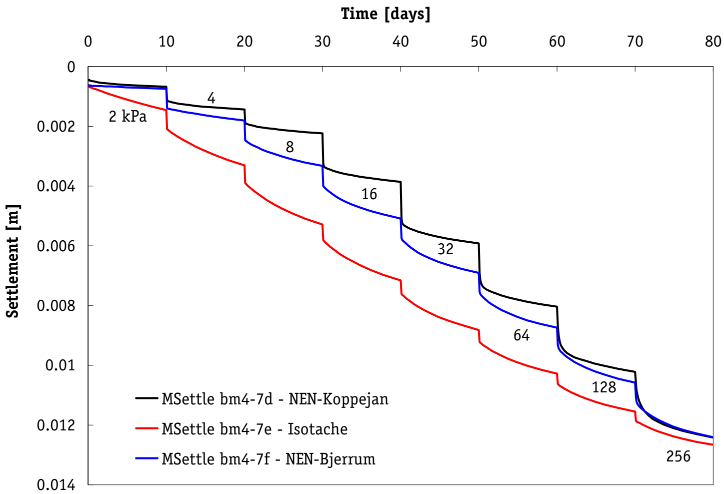


Figure 23-6 – Comparison of the settlement curve for the three models

Use MSettle input files bm4-7a.sli to bm4-7f.sli to run this benchmark.

23.8 Settlement curve during consolidation process with vertical drainage – Comparison between Darcy and Terzaghi models

Description

Settlements calculated by MSettle during the Darcy (Cv) and Terzaghi consolidation processes with vertical drainage are compared in this benchmark, using the NEN-Bjerrum model and a coefficient of consolidation of $C_v = 2 \times 10^{-6}$ m²/s. A clay layer is pre-loaded with $\sigma_{pre\text{-}load}$ = 1000 kPa and loaded with a uniform load of σ_{load} = 200 kPa. The piezometric level is at the surface level.

Terzaghi and Darcy consolidation models don't model the hydraulic head distribution along vertical drains in the same way: for Terzaghi model, the effect of vertical drains is simulated with an extra water load with a linear distribution whereas for Darcy model the resolution of the hydraulic equation leads to an exact solution with a non-linear distribution, as shown in Figure 23-8.

Consequence is that for Terzaghi the PL-line at the top will be different at the end of the consolidation but not for Darcy. Therefore, the total stress distribution will be different for both models.

MSettle result

Table 23-16 – Results of benchmark 4-8 – Comparison of settlement curves for Darcy and Terzaghi consolidation models

Time	MSettle - Darcy Cv (bm4-8a)	MSettle - Terzaghi (bm4-8b)	Relative error
[days]	[m]	[m]	[%]
1.26	0.084	0.096	12.50
9.51	0.351	0.361	2.77
30.8	0.743	0.744	0.13
49.04	0.936	0.936	0.00
98.35	1.187	1.183	0.34
394	1.327	1.308	1.45

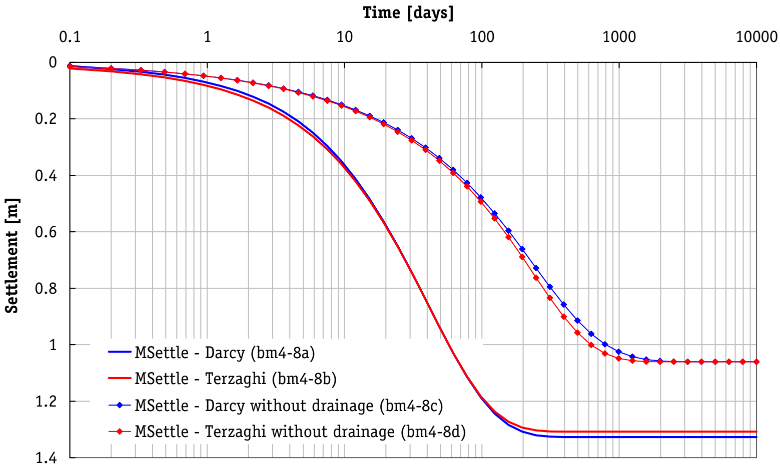


Figure 23-7 – Settlements during the consolidation process with vertical drainage – Comparison between Darcy and Terzaghi models

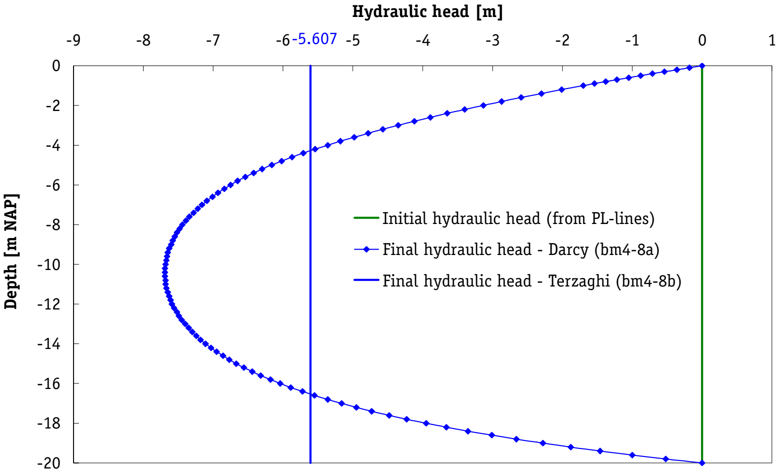


Figure 23-8 – Hydraulic head distributions for Darcy and Terzaghi models

Use MSettle input files bm4-8a.sli to bm4-8d.sli to run this benchmark.

23.9 Terzaghi with vertical drainage - Modeling dewatering off and simple using equivalent detailed input

Description

The same inpus as benchmark 3-11 [§ 22.11] is used except that in case of dewatering off and simple an equivalent detailed input is used in the *Vertical Drains* window [§ 9.4.2]. Six cases are checked as shown in Table 23-16.

Table 23-17 – Cases overview for benchmark 4-9

Case	Drain type	MSettle		MSettle using equiv. detailed dewatering
		File name	Dewatering	File name
1	Sand wall	bm3-11a	Off	bm4-9a
2	Sand wall	bm3-11b	Simple	bm4-9b
3	Column	bm3-11d	Off	bm4-9c
4	Column	bm3-11e	Simple	bm4-9d
5	Strip	bm3-11g	Off	bm4-9e
6	Strip	bm3-11h	Simple	bm4-9f

MSettle results (with dewatering Off and Simple)

Settlements calculated by MSettle are the same as benchmark 3-11 [§ 22.11] and are given in Table 23-17.

MSettle results (with equivalent Detailed dewatering)

Settlements calculated by MSettle are given in Table 23-17.

Table 23-18 – Results of benchmark 4-9 – Settlements

Case	Time	MSettle using dewatering off or simple		MSettle using equivalent detailed dewatering		Relative error
	[days]	File name	[m]	File name	[m]	[%]
1	50	bm3-11a	0.302	bm4-9a	0.302	0.00
	200		0.695		0.695	0.00
	400		2.016		2.016	0.00
	10000		2.584		2.584	0.00
2	50	bm3-11b	0.281	bm4-9b	0.281	0.00
	200		1.654		1.654	0.00
	400		1.860		1.860	0.00
	10000		1.992		1.992	0.00
3	50	bm3-11d	0.556	bm4-9c	0.556	0.00
	200		0.948		0.948	0.00
	400		1.753		1.753	0.00
	10000		2.197		2.197	0.00
4	50	bm3-11e	0.302	bm4-9d	0.302	0.00
	200		1.637		1.637	0.00
	400		1.999		1.999	0.00
	10000		2.566		2.566	0.00
5	50	bm3-11g	0.281	bm4-9e	0.281	0.00
	200		0.605		0.605	0.00
	400		1.436		1.436	0.00
	10000		1.969		1.969	0.00
6	50	bm3-11h	0.556	bm4-9f	0.556	0.00
	200		1.430		1.430	0.00
	400		1.778		1.778	0.00
	10000		2.197		2.197	0.00

Use MSettle input files bm4-9a.sli to bm4-9f.sli to run this benchmark.

23.10 Final settlement using water loads to simulate drains (Terzaghi)

Description

The same inpus as benchmark 3-11 [§ 22.11] is used except that the different dewatering steps of the vertical drainage are replaced by water loads with an equivalent piezometric level equals to the average stationary hydraulic head calculated by the Terzaghi model. Values are given in Table 23-18 for the nine checked cases.

Table 23-19 – Cases overview for benchmark 4-10

Case	Drain type	Soil model	Input dewat.	Grid	Time [days]	PL-line [m]
A	Sand wall	Isotache	Off		200	-1.833
B		NEN-Bjerrum	Simple		50 and 400	-1.833
					200	-7.911
C		NEN-Koppejan	Detailed		50	-3.894
					200	-1.859
D	Column	NEN-Koppejan	Off	Undet.	200	-1.000
E		Isotache	Simple	Rectang.	50 and 400	-1.000
					200	-2.282
F		NEN-Bjerrum	Detailed	Triang.	50	-6.107
					200	-1.747
G	Strip	NEN-Bjerrum	Off	Rectang.	200	-1.000
H		NEN-Koppejan	Simple	Triang.	50 and 400	-1.000
					200	-2.703
I		Isotache	Detailed	Undet.	50	-2.433
					200	-2.368

MSettle results (with Vertical Drains)

Final settlements calculated by MSettle for vertical 1 (situated in the drainage range) are the same as benchmark 3-11 [§ 22.11] and are given in Table 23-19.

MSettle results (with Water Loads)

Final settlements calculated by MSettle using Water Loads are given in Table 23-19.

Table 23-20 – Results of benchmark 4-10 – Final settlements

Case	MSettle with Vertical Drains		MSettle with Water Loads		Relative error
	File name	[m]	File name	[m]	[%]
A	bm3-11a	2.584	bm4-10a	2.584	0.00
B	bm3-11b	1.992	bm4-10b	1.994	0.10
C	bm3-11c	2.237	bm4-10c	2.240	0.13
D	bm3-11d	2.197	bm4-10d	2.198	0.05
E	bm3-11e	2.566	bm4-10e	2.565	0.04
F	bm3-11f	1.988	bm4-10f	1.991	0.15
G	bm3-11g	1.969	bm4-10g	1.970	0.05
H	bm3-11h	2.197	bm4-10h	2.198	0.05
I	bm3-11i	2.591	bm4-10i	2.596	0.19

Use MSettle input files bm4-10a.sli to bm4-10i.sli to run this benchmark.

23.11 Settlement acc. to approximate submerging model

Description

This benchmark checks the approximate submerging model by adapting the weight of the loads (saturated or unsaturated) depending on their final position after settlement (below or above phreatic level). The same input as benchmark 3-4 [§ 22.4] is used except that the submerging option is off and the unit weight of the loads is adapted according to the final settlement calculated by benchmark 3-4 (with submerging on). MSettle settlement results of benchmarks 3-4 and 4-11 should be the same. Four cases are checked as shown in Table 23-20.

Table 23-21 – Cases overview for benchmark 4-11

Case	Soil model	Consolidation model	MSettle file (Submerging ON)	MSettle file (Submerging OFF)
1	NEN-Koppejan	Terzaghi	bm3-4a	bm4-11a
2	NEN-Koppejan	Darcy	bm3-4b	bm4-11b
3	NEN-Bjerrum	Terzaghi	bm3-4c	bm4-11c
4	Isotache	Terzaghi	bm3-4e	bm4-11d

Figure 23-9 illustrates the position of the loads at final state of benchmark 3-9 compare to the phreatic line. There are used as input in benchmark 4-11.

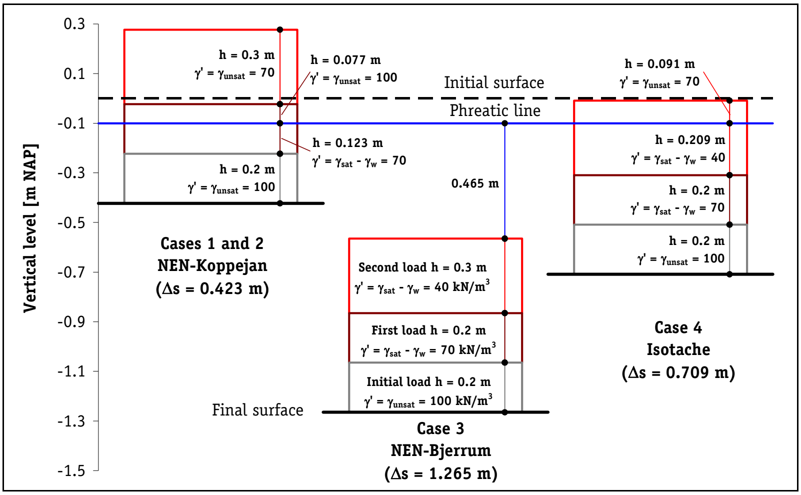


Figure 23-9 – Position of the loads at final state compare to the phreatic line for different cases

MSettle results (with Submerging ON)
Settlements calculated by MSettle with the Submerging option are the same as benchmark 3-4 [§ 22.4] and are given in Table 23-21.

MSettle results (with Submerging OFF and adapted loads)
Final settlements calculated by MSettle using those adapted loads are given in Table 23-21.

Table 23-22 – Results of benchmark 4-11 – Settlements

Case	Time	MSettle with Submerging		MSettle with adapted loads		Relative error
	[days]	File	[m]	File	[m]	[%]
1	100	bm3-4a	0.166	bm4-11a	0.166	0.00
	2000		0.453		0.453	0.00
	10000		0.423		0.424	0.24
2	100	bm3-4b	0.166	bm4-11b	0.166	0.00
	2000		0.453		0.453	0.00
	10000		0.423		0.424	0.24
3	100	bm3-4c	0.661	bm4-11c	0.661	0.00
	2000		1.093		1.093	0.00
	10000		1.265		1.265	0.00
4	100	bm3-4e	0.486	bm4-11d	0.486	0.00
	2000		0.676		0.676	0.00
	10000		0.709		0.709	0.00

Use MSettle input files bm4-11a.sli to bm4-11d.sli to run this benchmark.

23.12 Effect of the creep rate reference time on the simulation of a short term oedometer test

Description

MSettle uses a minimum time step of 1 day by default. To simulate a short term oedometer test with typical loading stages of just 1 day, a smaller unit of time can be applied by increasing the *Creep rate reference time* in the *Calculation Options* window [§ 10.1.1]. In this benchmark, a value of 24 × 60 = 1440 is used to change the time unit from days to minutes. Then all parameters using a time unit must be multiplied by this value. Two oedometer tests are simulated with MSettle and compared: case A uses a unit weight of 1 day whereas case B uses 1440 day as creep rate reference time. The load is double at each load-step starting with 1 kPa. Eight load-steps are applied on a 20 mm height sample. Input parameters are given in Table 23-22.

Table 23-23 – Input parameters for benchmark 4-12

Case			A	B
MSettle file			bm4-12a	bm4-12b
Reloading/Swelling constant	a	[-]	0.02	
Primary compression constant	b	[-]	0.4	
Secondary compression constant	c	[-]	0.05	
Creep rate reference time	t_0	[days]	1	1440
Consolidation coeff.	C_V	[m^2/s]	1.44E-06	1.00E-09
Equivalent age	t_{age}	[days]	3000	4320000
Last of a load-step	Δt	[days]	1	1440
End of calculation time	t_{final}	[days]	8	11520

MSettle results

Comparison of the settlement curve is given in Table 23-23 and in Figure 23-10. Note that case B uses more time steps than case A leading to a more accurate modeling of the consolidation process. This can explain the few differences in comparison especially for the first load-steps.

Table 23-24 – Results of benchmark 4-12 – Settlements in time

Load step	MSettle (bm4-12a) Time unit in Days		MSettle (bm4-12b) Time unit in Minutes		Error
	Time	Settlement	Time	Settlement	Settlement
	[days]	[mm]	[Minutes]	[mm]	[%]
1	1	0.32	1440	0.32	0.00
2	2	2.73	2880	2.39	14.23
3	3	6.82	4320	6.48	5.25
4	4	10.00	5760	9.86	1.42
5	5	12.41	7200	12.37	0.32
6	6	14.25	8640	14.23	0.14
7	7	15.64	10080	15.63	0.06
8	8	16.69	11520	16.69	0.00

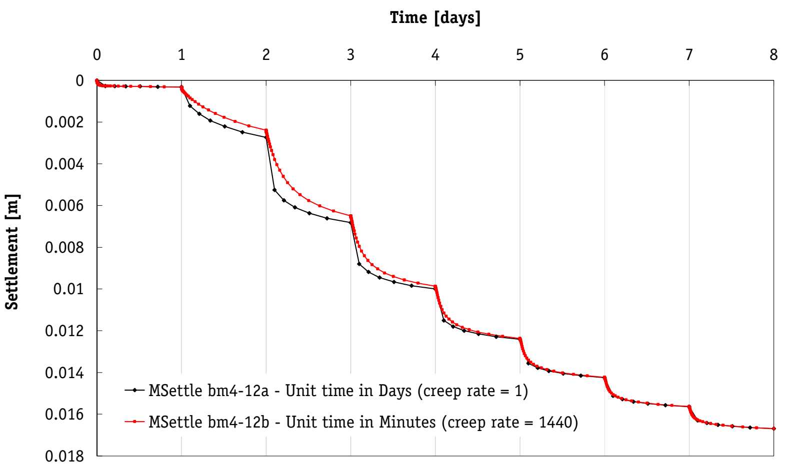


Figure 23-10 – Results of benchmark 4-12 – Comparison of the settlement curve in time for cases A and B

Use MSettle input files bm4-12a.sli and bm4-12b.sli to run this benchmark.

24 Benchmarks compared with other programs

These benchmarks are intended to verify specific features of MSettle comparing MSettle results with those from an other program.

24.1 Calculation of the horizontal displacements

Description

In this benchmark, horizontal displacements calculated by MSettle are compared to the results of the program LEEUWIN.EXE based on the Tables of De Leeuw [Lit 24].

The following parameters are used in each calculation:

- Thickness elastic layer: 5 m
- Thickness stiff top layer: 0 m and 1 m
- Young's modulus elastic layer: 1500 kN/m^2 (i.e. γ_{unsat} = 18 kN/m^3)
- Surcharge load: 10 kPa
- Width of surcharge load: 10 m

Three situations are checked:

- Situation A (bm5-1a): Situation with a stiff top layer of 1 m thickness.
- Situation B (bm5-1b): Situation without stiff top layer.
- Situation C (bm4-10c): Situation without stiff top layer and with a layered elastic layer: top layer of 1 m thick with E = 1500 kN/m^2 (i.e. γ_{unsat} = 18 kN/m^3) and bottom layer of 4 m thick with E = 575 kN/m^2 (i.e. γ_{unsat} = 10 kN/m^3). The average Young's modulus thus becomes
  $E_{avg} = (1 \times 1500 + 4 \times 575)/5 = 760$ kN/m^2.

Resulting horizontal displacements are calculated for verticals at 2 m and 10 m from the edge of the surcharge load.

Program LEEUWIN.EXE

The three situations described above are modeled with the program LEEUWIN.EXE and results are shown below.

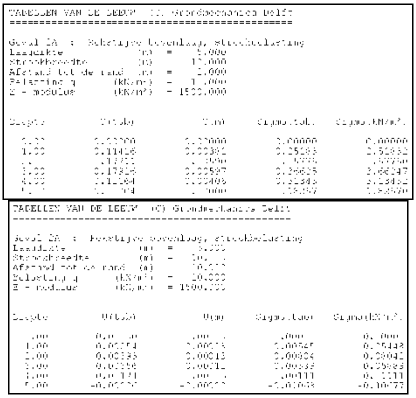


Figure 24-1 – Horizontal displacements acc. to LEEUWIN.EXE program (situation A)

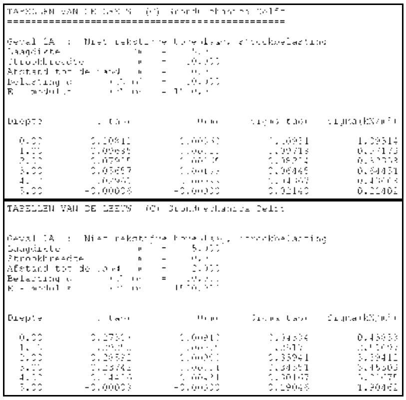


Figure 24-2 – Horizontal displacements acc. to LEEUWIN.EXE program (situation B)

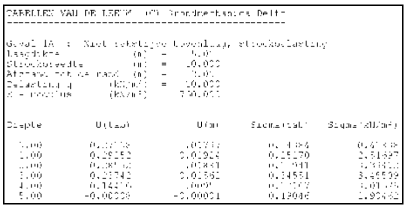


Figure 24-3 – Horizontal displacements acc. to LEEUWIN.EXE program (situation C)

MSettle

Results show that the horizontal displacements calculated by MSettle are in agreement with the horizontal displacements from the program LEEUWIN.EXE based on the “Tables of De Leeuw” [Lit 24].

Table 24-1 – Results of benchmark 5-1 – Horizontal displacements at 2 m from the edge of the surcharge load for different situations

Situation	Depth along elastic layer	Benchmark [mm]	MSettle [mm]	Relative error [%]
A	0 m	0.00	0.00	0.00
	-1 m	3.81	3.81	0.00
	-2 m	5.90	5.90	0.00
	-3 m	5.97	5.97	0.00
	-4 m	4.05	4.05	0.00
	-5 m	0.00	0.00	0.00
B	0 m	9.10	9.13	0.33
	-1 m	9.75	9.75	0.00
	-2 m	9.53	9.53	0.00
	-3 m	7.91	7.91	0.00
	-4 m	4.81	4.81	0.00
	-5 m	0.00	0.00	0.00
C	0 m	17.97	18.02	0.28
	-1 m	19.24	19.24	0.00
	-2 m	18.81	18.81	0.00
	-3 m	15.62	15.62	0.00
	-4 m	9.50	9.50	0.00
	-5 m	0.01	0.00	-

Table 24-2 – Results of benchmark 5-1 – Horizontal displacements at 10 m from the edge of the surcharge load for different situations

Situation	Depth along elastic layer	Benchmark [mm]	MSettle [mm]	Relative error [%]
A	0 m	0.00	0.00	0.00
	-1 m	0.08	0.08	0.00
	-2 m	0.13	0.13	0.00
	-3 m	0.12	0.12	0.00
	-4 m	0.06	0.06	0.00
	-5 m	0.00	0.00	0.00
B	0 m	3.60	3.59	0.28
	-1 m	3.23	3.23	0.00
	-2 m	2.65	2.65	0.00
	-3 m	1.89	1.89	0.00
	-4 m	0.99	0.99	0.00
	-5 m	0.00	0.00	0.00

Use MSettle input files bm5-1a.sli, bm5-1b.sli and bm5-1c.sli to run this benchmark.

Literature

[Lit 1] Bjerrum, L., Engineering geology of Norwegian normally consolidated marine clays as related to settlements of buildings, 1972, Géotechnique Vol. 17(2), pp. 81-118.

[Lit 2] Koppejan, A.W. A formula combining the Terzaghi load compression relationship and the Buisman secular time effect, Proc. 2nd Int. Conf. Soil Mech. and Fnd. Eng. Rotterdam, 1948, pp. 32-37.

[Lit 3] Terzaghi, K. & Peck, R.B., Soil Mechanics in Engineering Practice, 1967.

[Lit 4] Barron, R.A. Consolidation of fine-grained soils by drainwells, Trans ASCE 113, pp. 718-742, 1948.

[Lit 5] Carillo, N., Simple two and three-dimensional cases in the theory of consolidation of soils, Journal of Math. Phys., Vol 21, pp. 1-5, 1942.

[Lit 6] Mesri, A.M., Coefficient of Secondary Compression, Journal of Soil Mechanics and Foundations Division, January 1973, pp. 123-137.

[Lit 7] Den Haan, E. J. Vertical Compression of Soil, Ph.D. Thesis Delft University, 1994.

[Lit 8] NEN 6744-1991, Geotechnics - Calculation Method for shallow foundations (in Dutch), Nederlands Normalisatie Instituut (Dutch Normalisation Institute).

[Lit 9] NEN 5118-1991, Geotechnics - Determination of the one-dimensional consolidation properties of soil (in Dutch), Nederlands Normalisatie Instituut.

[Lit 10] ISSMGE/DIN, 1998, Recommendations of the ISSMGE for Geotechnical Labatory Testing ETC5-D1.97.

[Lit 11] Sellmeijer, J.B., Vertical Drains simulated as Leakage, Learned and Applied Soil Mechanics out of Delft 75-80, 2002.

[Lit 12] Den Haan, E.J. & Sellmeijer, J.B., Calculation of soft ground settlement with an isotache model, "Soft Ground Technology", ASCE Geotech, Special Publication nr. 112, pp. 94-104, 2000.

[Lit 13] Den Haan, E.J., Het a,b,c-isotachenmodel: hoeksteen van een nieuwe aanpak voor zettingsberekeningen (in Dutch), Geotechniek 2003, Vol. 4, pp 28-35, 2003.

[Lit 14] Den Haan, E.J., Van Essen, H.M., Visschedijk, M.A.T. & Maccabiani, J., Isotachenmodellen: Help, hoe kom ik aan de parameters (in Dutch), Geotechniek 2004, Vol. 1, pp 62-69, 2004.

[Lit 15] H. Den Adel & V. Trompille & J.B. Sellmeijer & M. Van, Geforceerde drainage 5^{e} Schipholbaan (in Dutch), Geotechniek 2004, Vol. 2, pp 58-64, 2004.

[Lit 16] H. Den Adel, Uitwerking K_0-CRS proef, bepaling a/b/c parameters (in Dutch), Delft Cluster report, 01.04.02, March 2002.

[Lit 17] Sellmeijer, J.B., Visschedijk, M.A.T. & Weinberg, M.J.M. Rekenen met verticale drains (in Dutch), Geotechniek, 2004, Vol. 4, pp 36-41, 2004.

[Lit 18] Calle, E.O.F., Sellmeijer, J.B. & Visschedijk, M.A.T., Reliability of settlement prediction based on monitoring, Proc. 16th Int. Conf. Soil Mechanics Geotechnical Engineering, Osaka, September 2005, Rotterdam, Millpress, Vol. 3, pp 1681-1684.

[Lit 19] Beacher, G.B. & Christian, J.T., Reliability and Statistics in Geotechnical Engineering, 2003.

[Lit 20] CUR publicatie 2005-1, Geforceerde consolidatie door het afpompen van water (in Dutch).

[Lit 21] Building on Soft soil, Balkema, 1996 (translation of CUR Publicatie, Construeren met Grond, 1992, in Dutch)

[Lit 22] Poulos, H.G. & Davis, E.H., Elastic Solutions for Soil and Rock Mechanics", John Wiley & Sons, New York, 1974.

[Lit 23] Ahlvin, R. G. & Ulery, H., Tabulated values for determining the complete pattern of stresses, strains, and deflections beneath a uniform circular load on a homogeneous half space, Highw. Res. Board, Bull, Vol. 342, pp 1–13, 1962.

[Lit 24] De Leeuw, Ir. E. H. Tabellen ter bepaling van horizontale spanningen en verplaatsingen in een homogene elastische laag van eindige dikte, 1963, Laboratorium voor Grondmechanica, Delft (The Netherlands).

[Lit 25] Deltares Report CO-432110-850, Verification of the FOSM method in MSettle – Analytical solutions, Nov. 2008.

Index