Geocentrix

Repute 3

Quick-Start Guide

Acknowledgments

Repute 3.0 was designed and written by Dr Andrew Bond of Geocentrix. Repute 2.5 was developed with the assistance of Ian Spencer of Honor Oak Systems. Repute 2.0 was supported financially by Corus, Atkins, and Stent Foundations.

Repute's boundary element engine is based on PGroupN, which was designed and written by Dr Francesco Basile of Geomarc. Special recognition goes to the late Dr Ken Fleming of Cementation Foundations Skanska for his invaluable advice and support during the development of PGroupN.

The *Repute Quick-Start Guide* was written by Andrew Bond with the assistance of Jenny Bond.

The following people assisted with the production of the program and its documentation: Jenny Bond, Francesco Basile.

Revision history

Last revised 20th August 2026 (for version 3.0.0.9).

Contents

Introducing Repute 3

Repute® provides a rich set of capabilities for engineers to design/analyse:

- various types of single pile, including bored, driven, tubular, precast concrete, and spun piles
- using various design standards, including the second-generation Eurocode 7 (EN 1997-3:2025) and BS 8004:2015
- pile groups under generalized 3-dimensional loading, using linear or non-linear soil models
- pile groups with fully rigid or fully flexible pile caps

Repute considers single pile response using a variety of calculation methods for ultimate and serviceability limit states. Both traditional lumped factors-of-safety and modern partial factors can be applied in these calculations.

Repute analyses pile group behaviour using the boundary element method, employing the leading analytical program PGroupN (developed by and included under exclusive licence from Geomarc). PGroupN provides a complete 3D non-linear boundary element solution of the soil continuum, which overcomes limitations of traditional interaction factor methods and gives more realistic predictions of deformations and the load distribution between piles.

What's new in Repute 3?

New features

- Added analysis of fully-flexible pile caps
- Added 7th tutorial in the Quick-start Guide demonstrating the fully-flexible pile cap options
- Full implementation of the second-generation Eurocode 7 (EN 1997-3:2015)
- Support for octagonal and polygonal (e.g. L-shaped) piles caps
- Added ability to save input and output files in a single RPZ (aka ZIP) file

- Boundary element results automatically saved in RPZ file
- 64-bit build provides faster performance and allows much larger pile groups to be analysed
- Completely redesigned Drawing Board
- Adds standard page sizes (A3/A4/A5 etc) to the Drawing Board
- Project files save last drawing settings separately for each scenario
- Added wide range of skin, palette, and Ribbon options in the user interface
- Added Backstage View similar to Microsoft Word and Excel
- Added templates for many common piled foundations

Improved features

- Simplified the boundary element analysis options
- Configuration files moved to AppData > Roaming folder to ensure User has access without Admin privileges
- Added prompt to delete temporary files if detected at program startup
- Updated Quick-Start Tutorial

Documentation

Repute is supplied with a detailed Quick-start Guide, comprehensive User Manual, and authoritative Reference Manual. The latest versions of these manuals (including any corrections and/or additions since the program's first release) are available in electronic (Adobe® Acrobat®) format from the Geocentrix website. (www.geocentrix.co.uk/repute and follow links to Repute's documentation).

Quick-Start guide (this document)

The *Repute 3 Quick-Start Guide* includes seven tutorials that show you how to use the main features of Repute. Each tutorial provides step-by-step instructions on how to drive the program. There are three tutorials dealing with single pile design and four with pile group design. The tutorials increase in difficulty and are designed to be followed in order.

User manual

The *Repute 3 User Manual* explains how to use Repute. It provides a detailed description of the program's user interface and explains how to employ it to maximum effect.

Reference manual

The *Repute 3 Reference Manual* gives detailed information about the engineering theory that underpins Repute's calculations. The manual assumes you have a working knowledge of the geotechnical design of single piles and pile groups but provides appropriate references for further study if you do not.

Help system

Repute's help system contains detailed information about the program, including most of the content of the *Quick-Start Guide*, *User Manual*, and *Reference Manual* – plus additional information that is not found in any of these documents.

Help appears in a separate window to Repute, allowing you to view the help topics while you continue to work with Repute itself. To open the help system:

- `Press F1`
- `Click the Help button in any dialog box`
- `Click on the Help button on the right-hand side of Repute's Ribbon`

Note

Screenshots in this document were produced on Windows 11. Their appearance may differ on your computer. Not all options are available in every edition of Repute.

In this document, '`[Tutorials]`' refers to the folder where the tutorials that ship with Repute were installed, typically here:

C:\Users\Public\Documents\Geocentrix\Repute\3.0\Tutorials

Software Re-Assurance™

Software Re-Assurance for Repute (including updates, upgrades, and technical support) is available directly from Geocentrix or through your local distributor. To obtain Re-Assurance, contact Geocentrix as follows:

Repute Technical Support
Geocentrix Ltd
Larchwood, 10a Cunningham Road, Banstead, Surrey, SM7 3HG, UK
T | +44 (0)1737 200045
E | support@geocentrix.co.uk
W | www.geocentrix.co.uk/support

Please quote your licence number when contacting technical support

Tutorial 1. H-pile in clay and sand

Introduction

This tutorial demonstrates the basic features of Repute, through a worked example involving the analysis of a single H-pile installed in clay and sand:

- Ground conditions comprise 5 m of clay overlying 20 m of dense sand. The clay has unit weight of 20.5 kN/m^3, angle of shearing resistance of 23 °, and undrained shear strength of 60 kPa. The sand has unit weight of 18 kN/m^3 and angle of shearing resistance of 35°.
- The foundation comprises a 305 x 305 x 95 H-pile of steel grade S275.
- A vertical load of 700 kN is to be applied to the pile.
- You want to determine the minimum length of pile needed to safely carry the applied action according to traditional UK practice.

This tutorial is written for users of the Standard, Enterprise, and Trial Editions of Repute only. Users of the Professional Edition should look at Tutorials 4-7.

Overview

- In Step 1, you will use the Project Wizard to enter project information, select a design standard, and create scenarios to represent short- and long-term conditions.
- In Step 2, you will use the Borehole Wizard to create a borehole containing clay and sand layers.
- In Step 3, you will create an H-pile and specify its cross-section and steel grade.
- In Step 4, you will create the force applied to the pile.
- In Step 5, you will use the Calculation Wizard to create the calculations you want Repute to perform.
- In Step 6, you will perform the calculations and review the results.

- In Step 7, you will produce a report summarising the results of the calculations.
- In Step 8, you will close (and optionally save) the project.

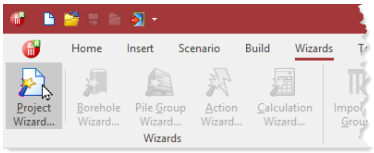
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

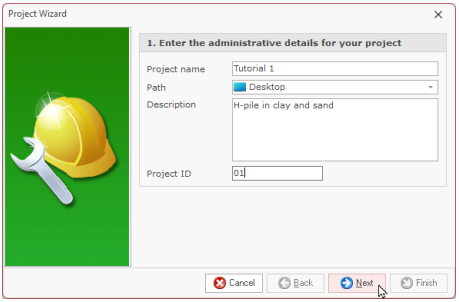
Step 1 – create the project

In Step 1, you will use the Project Wizard to enter project information, select a design standard, and create scenarios to represent short- and long-term conditions.

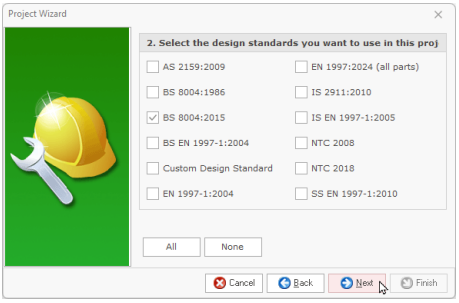
1. Open the **Project Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Project Wizard** button.



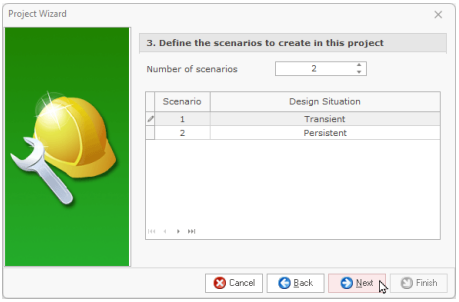
2. When the Wizard appears, type `Tutorial 1` in the **Project name** box. Choose the folder where you want to save this project by using the **Path** control. (If you do not change the setting here, it will be saved on your Desktop.)
3. Enter `H-pile in clay and sand` in the **Description** box and `01` in the **Project ID** box.



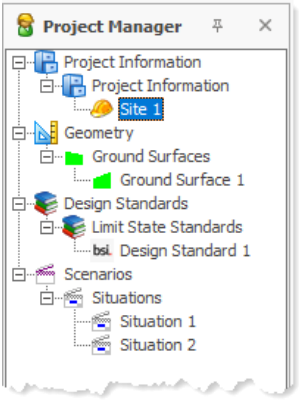
4. Click **Next** to display the next page (the standards that appear depend on which edition of Repute you are running). Select `BS 8004:2015` by clicking on the relevant checkbox (a tick mark appears when a standard is selected).



5. Click **Next** to display the next page. Increase the **Number of scenarios** to 2 and then change the **Design Situation** of Scenario 1 to `Transient` but leave Scenario 2 as `Persistent`.

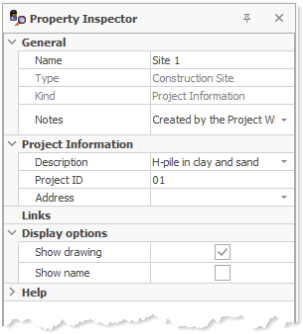


6. Click **Next** to display the final page. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
7. When you are ready, click **Finish** to generate the project. The **Project Wizard** then:
   - creates `Site 1`, `Ground Surface 1`, `Design Standard 1`, and `Situations 1` and `2`
   - adds `Ground Surface 1` to `Situations 1` and `2`
   - creates and saves a new project named `Tutorial 1.rptz` with all these items
8. You can view these items by right-clicking anywhere inside the **Project Manager** to display its context menu and selecting the **Expand All** command. The Project Manager will then look as shown in the picture alongside.



9. To view the properties of any particular item, first select that item in the **Project Manager** (e.g. `Site 1`) and then display its Property Inspector by:
   - right-clicking on the item and choosing the **Properties...** command on its context menu, or
   - double-clicking on the item

10. The program's **Property Inspector** will open and display the properties of the selected item. (For example, for Site 1 the **Description** is shown as `H-pile in clay and sand` and the **Project ID** as `01`.)



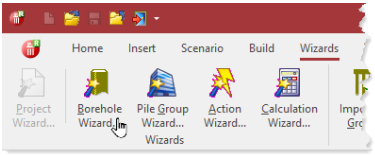
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 1\Step 1.rptz`

Step 2 – create the borehole

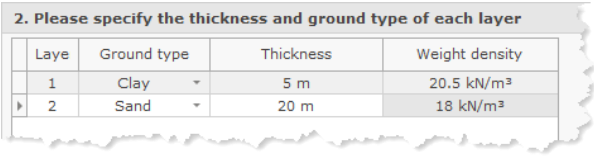
In Step 2, you will use the Borehole Wizard to create a borehole containing clay and sand layers.

1. Open the **Borehole Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Borehole Wizard** button.



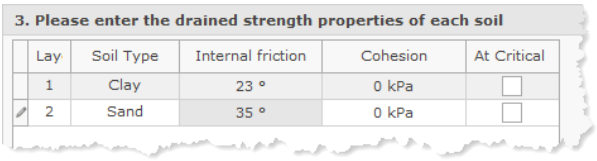
2. When the Wizard appears, increase the number of layers to 2.

3. Click **Next** to display page 2 of the Wizard. Change Layer 1 to `Clay`, its **Thickness** to 5 m, and its **Weight density** to 20.5 kN/m^3. Leave Layer 2 as `Sand` but change its thickness to 20 m and weight density to 18 kN/m^3.



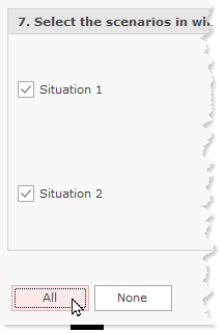
4. Click **Next** to display page 3. Change the **Internal friction** of Layer 1 to 23° and that of Layer 2 to 35°. Leave the **Cohesion**

of both soils unchanged at 0 kPa and the **At Critical State?** boxes unchecked.



5. Click **Next** to display page 4. Change the **Undrained Strength (top)** and **Undrained Strength (bottom)** of Layer 1 to 60 kPa. Note that Soil 2 does not appear on this page, since – being a sand – it does not have undrained strength.

6. Click **Next** to display page 5. Since the ground profile does not include rock, there is nothing to set on this page.
7. Click **Next** to display page 6. Leave the stiffness properties (i.e. **Shear modulus**) of both layers unchanged.
8. Click **Next** to display page 7. Click **All** to select both scenarios.



9. Click **Next** to display the final page. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
10. When you are ready, click **Finish** to generate the borehole. The **Borehole Wizard** then:
    - creates `Borehole 1, Layers 1 and 2`, and `Soils 1 and 2`
    - links `Soil 1` to `Layer 1 and Soil 2` to `Layer 2`
    - adds `Layers 1 and 2` to `Borehole 1`

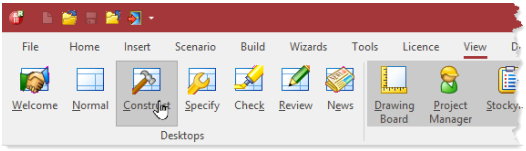
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 1\Step 2.rptz`

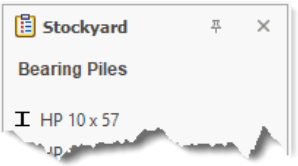
Step 3 – create the pile

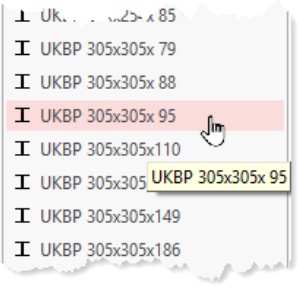
In Step 3, you will create an H-pile and specify its cross-section and steel grade.

1. Open the **Stockyard** by selecting the **View** tab on Repute's ribbon and then clicking on the **Construct** button.



2. When the program has switched to its **Construction Desktop** (in which the **Drawing Board**, **Project Manager**, and **Stockyard** are displayed), right-click anywhere inside the **Stockyard** to display its context menu and select the **Sections > Bearing Piles** command. The Bearing Piles group will open.
3. Create the section by selecting the item labelled `UKBP 305x305x95`, right-clicking to display its context menu, and then selecting the command **Create 'UKBP 305x305x95'**. The newly created hot-rolled section will appear in the **Project Manager** (under Sections > Bearing Piles) as `Section 1`.

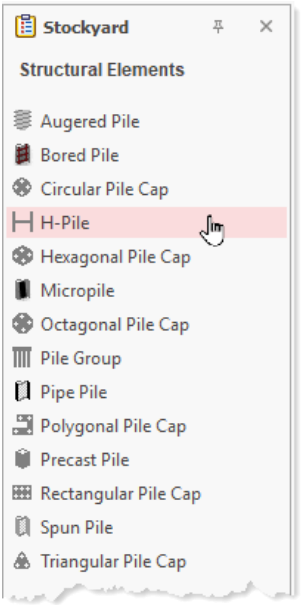




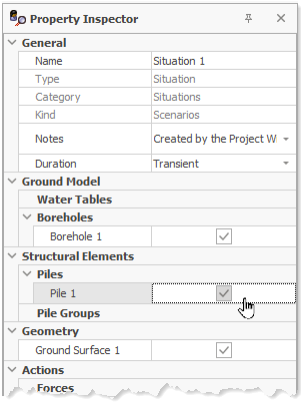
4. Next, right-click anywhere inside the **Stockyard** to display its context menu and select the **Materials > Steels** command. The Steels group will open.
5. Create the steel by holding the Ctrl key down and clicking on the item labelled `S275`. (When the Ctrl key is pressed, Repute automatically creates any item that you select in the Stockyard. This saves you the effort of displaying the popup menu each time you want to create a new item.) The newly

created steel will appear in the **Project Manager** (under Materials > Steels) as `Steel 1`.

6. Finally, click on the **Stockyard**'s **Structural Elements** caption (near the bottom of the Stockyard). The Structural Elements panel will open.



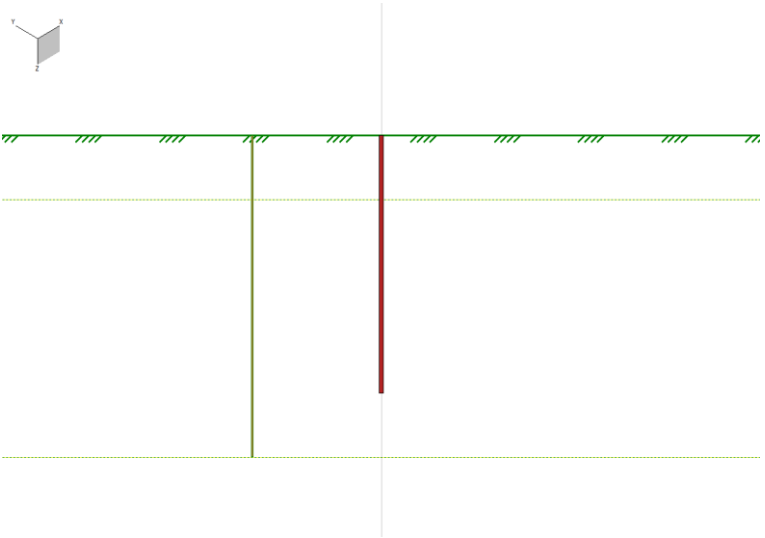
7. Create the pile by holding the Ctrl key down and clicking on the item labelled `H-pile`. The newly created pile will appear in the **Project Manager** (under Structural Elements > Piles) as `Pile 1`.
8. In the **Project Manager**, right-click on `Pile 1` to display its context menu and select the **Properties...** command. The Property Inspector will appear.
9. In the **Property Inspector**, change the **Material Name** (under Material Properties) from `Not specified` to `Steel 1`. Then change the **Section Name** (under Section Properties) from `Not specified` to `Section 1`. Leave all other properties of the pile unchanged.



10. Returning to the **Project Manager**, select `Situation 1` (under Scenarios > Situations) and, in its **Property Inspector**, place a tick next to `Pile 1` (under Structural Elements > Piles) to add the pile to this scenario. The **Drawing Board** will refresh.
11. Repeat the previous instruction for `Situation 2`.
12. In this step, you have:
    - created `Steel 1`, `Section 1`, and `Pile 1`

- linked `Section 1` and `Steel 1` to `Pile 1`
- added `Pile 1` to `Situations 1` and `2`

13. With `Situation 2` selected in the **Property Inspector**, under Drawing Board, change the **Page size** to A4, **Page orientation** to Landscape, **Viewing plane** to Cross-section, and **Scale** to 1:200.
14. The **Drawing Board** will now look something like this:



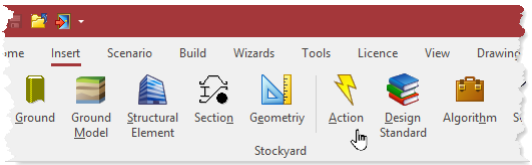
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 1\Step 3.rptz

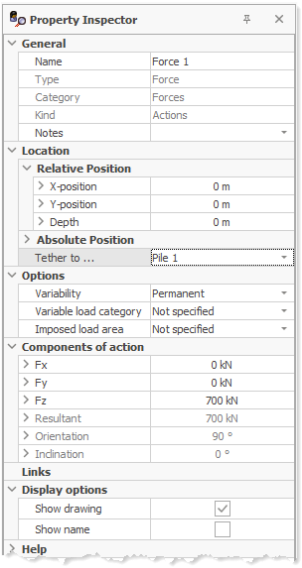
Step 4 – create the force

In Step 4, you will create the force applied to the pile.

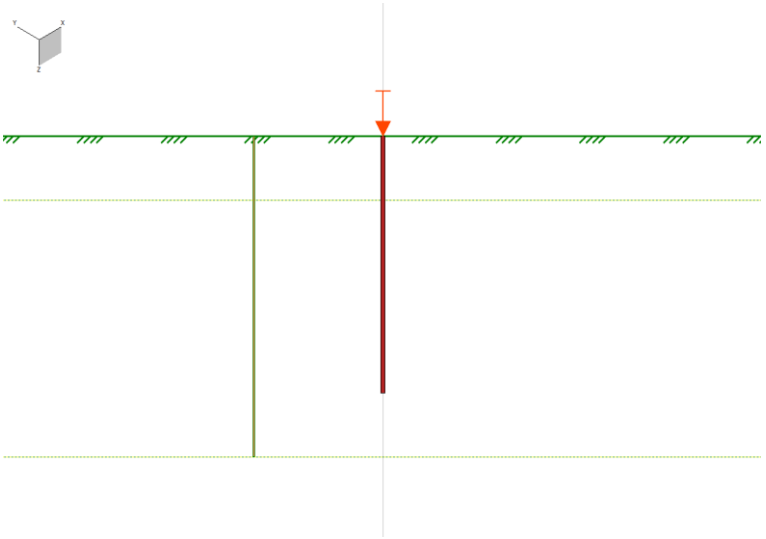
1. Open the **Stockyard**'s Actions panel by selecting the **Insert** tab on Repute's ribbon and then clicking on the **Action** button.



2. The Actions group will open in the **Stockyard**. Hold the Ctrl key down and click on the item labelled `Force`.
3. The newly created force will appear in the **Project Manager** (under Actions > Forces).
4. In the **Property Inspector**, change the **Variability** of `Force 1` (under Options) to `Permanent` and the value of **Fz** (under Components of Action) to 700 kN. The **Resultant** will automatically change to 700 kN.



5. Change the **Tether to ...** item to `Pile 1`. This will ensure that the force's plan position will always match the pile's plan position, even if the pile is moved.
6. In the **Project Manager**, select `Situation 1` (under Scenarios > Situations) and then, in its **Property Inspector**, place a tick next to `Force 1` (under Actions > Forces) to add the action to this scenario.
7. Repeat the previous instruction for `Situation 2`.
8. In this step, you have:
   - created `Force 1`
   - added `Force 1` to `Situations 1` and `2`
9. With `Situation 2` selected in the **Property Inspector**, under Drawing Board, change the **Force scale (kN/m)** to 200.
10. The **Drawing Board** will now look something like this:



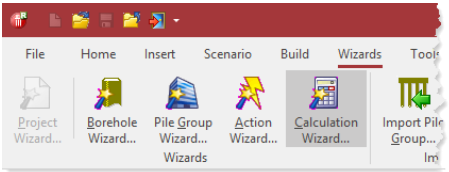
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 1\Step 4.rptz

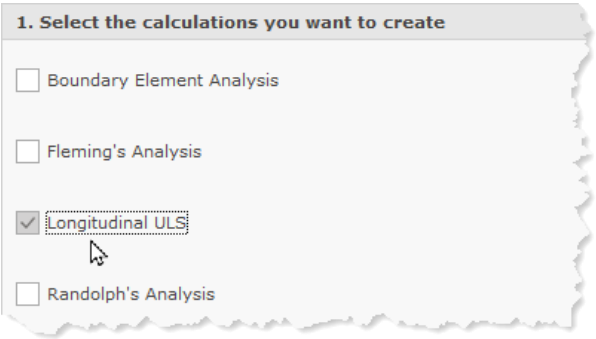
Step 5 – create the calculations

In Step 5, you will use the Calculation Wizard to create the calculations you want to perform.

1. Open the **Calculation Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Calculation Wizard** button.



2. When the Wizard appears, select `Longitudinal ULS`. (The calculations that appear here depend on which edition of Repute you are running.)



3. Click **Next** to display page 2 of the Wizard. Select `Design Standard 1` (this is the BS 8004:2015 design standard created in Step 1 of this tutorial).
4. Click **Next** to display page 3. Click **All** to select both situations.
5. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
6. When you are ready, click **Finish** to generate the calculations.
7. The Calculation Wizard then:
   - creates `Calculations 1` and `2`
   - links `Situation 1` to `Calculation 1` and `Situation 2` to `Calculation 2`
   - links `Design Standard 1` to `Calculations 1` and `2`

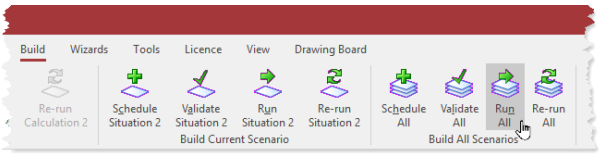
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 1\Step 5.rptz

Step 6 – perform and review the calculations

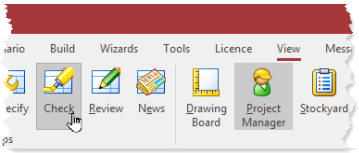
In Step 6, you will perform the calculations and review the results.

1. Run the calculations by selecting the **Build** tab on Repute's ribbon and then clicking on the **Run All** button.

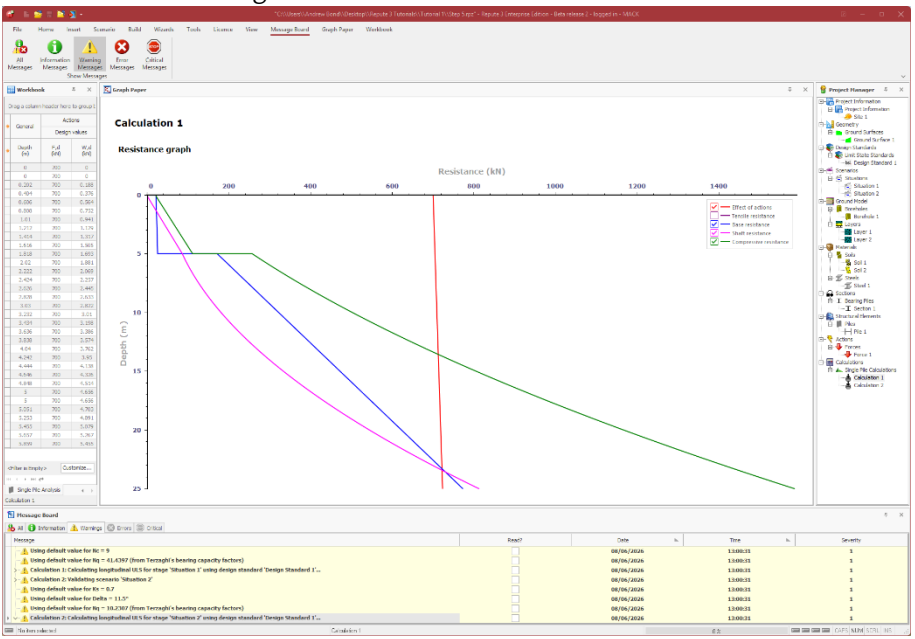


2. Repute will perform all the calculations that you have specified (i.e. `Calculations 1` and 2) and then change to its **Checking Desktop** (which displays the Workbook and Graph Paper).

3. You can switch to this display at any time by clicking on the **Check** button on the **View** tab of Repute's ribbon.

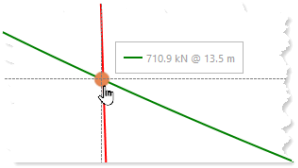


4. Your screen will now look something like this:

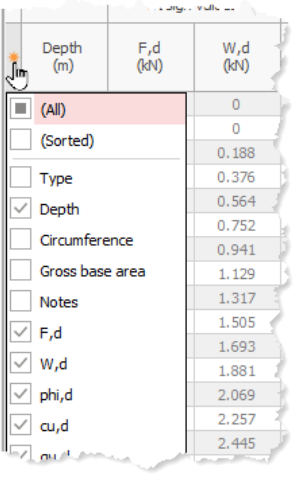


5. The **Graph Paper** (central panel) shows:
   - effects-of-actions (i.e. internal forces in the pile) increasing slightly with depth (the red line)
   - separate components of shaft and base resistances (the blue and pink lines)
   - the total compressive resistance (the green line)

6. If you move the cursor over the lines on the graph it will display spot values at key depths – in this way you can see that the compressive resistance equals the effect of actions at a depth of 13.5 m



7. The **Workbook** (top-left panel) shows the same information, but in tabular format. Drag the right hand boundary of the Workbook to the right to display its contents.
8. The **Workbook** contains a lot more information than is initially shown. To display this additional information, click on the button labelled * in the top-left-hand corner of the Workbook (to the left of the heading **Depth**) and select the data you want to see.
9. The **Message Board** (bottom panel) shows any warning or error messages that were generated during the calculation. You should review these messages to ensure that the calculations have performed as you expected.
10. To view the results of the second calculation, select `Calculation 2` in the **Project Manager**. Repute will automatically update the Workbook and Graph Paper with this calculation's data.



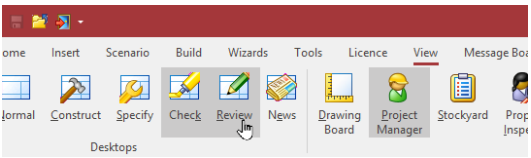
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 1\Step 6.rptz

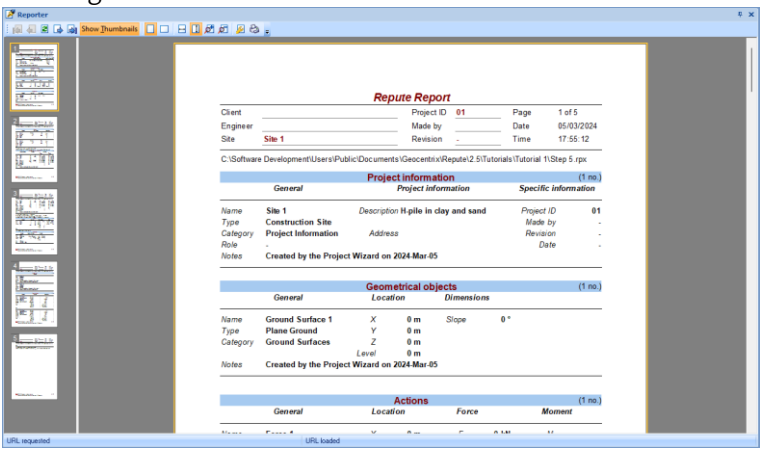
Step 7 – produce a report

In Step 7, you will produce a report summarising your calculations.

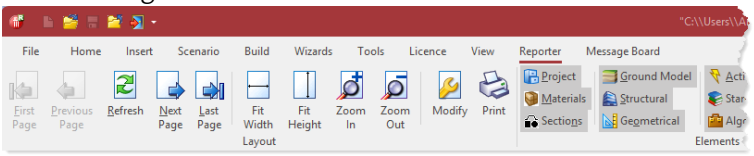
1. Display Repute's **Reporter** by selecting the **View** tab on the ribbon and then clicking on the **Review** button.



2. The Reporter will appear and automatically generate a report together with a set of thumbnails.



3. You can navigate around the report using the controls on the Reporter tab on the ribbon: **First Page**, **Previous Page**, **Next Page**, and **Last Page**. You can jump to specific pages by clicking on the thumbnails on the left-hand side.



4. You can also re-size the report using the **Fit Width**, **Fit Height**, **Zoom In**, and **Zoom Out** controls.
5. You can choose what to include in your report by selecting or deselecting individual items in the **Elements** group, then clicking the **Refresh** button.

6. Finally, you can output the report onto paper using the **Print** command and choosing from the options provided in the dialog box that appears on your screen.

Step 8 – close the project

In Step 8, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the File menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before proceeding. Click the appropriate button to answer **Yes** or **No**.
3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

[Tutorials]\Tutorial 1\Tutorial 1.rptz

What next?

Tutorial 2 shows you how to perform Fleming's hyperbolic analysis.

Tutorial 2. Fleming's hyperbolic analysis

Introduction

Tutorial 2 shows you how to construct a load vs displacement curve for a single pile, using Fleming's hyperbolic analysis, described in his 1992 Géotechnique paper *A new method for single pile settlement prediction and analysis* (see vol. 42, no. 3, pp 411-425). This tutorial demonstrates how to setup a calculation in Repute without using the program's built-in wizards, thereby showing you how versatile the program's user interface is.

The worked example is taken from Figure 6 in Fleming's paper, which is based upon from tests carried out at Wembley by Whitaker and Cooke. We are interested in replicating the load vs displacement curve given by Fleming's analysis.

- Ground conditions at the site are not given in the paper, so we will assume 25 m of London Clay with an undrained strength of 100 kPa and internal friction angle of 23°.
- The pile studied is a 12.2 m long, 775 mm diameter bored pile made of concrete with Young's modulus equal to 19.5 GPa.
- A vertical load of 200 tonnes (approximately 2000 kN) is applied at the centre of the pile.
- The ultimate load that the pile can carry has been calculated (separately) as 1994 kN on the shaft and 1002 kN from the base. The soil modulus below the pile base is 33.125 MPa. Other parameters used by Fleming are the shaft flexibility factor (0.0017 or 0.17%) and effective column length factor (0.45).

This tutorial is written for users of the Standard, Enterprise, and Trial Editions of Repute only. Users of the Professional Edition should look at Tutorials 4-7.

Overview

- In Step 1, you will enter project information about the site and the Engineer.

- In Step 2, you will define the site's ground conditions.
- In Step 3, you will create a bored pile and specify its cross-section and concrete grade.
- In Step 4, you will create the force applied to the pile.
- In Step 5, you will sleeve the pile through the made ground.
- In Step 6, you will create the scenario and calculation and link various items together.
- In Step 7, you will perform the calculation and produce a report showing the relationship between load and displacement.
- In Step 8, you will close (and optionally save) the project.

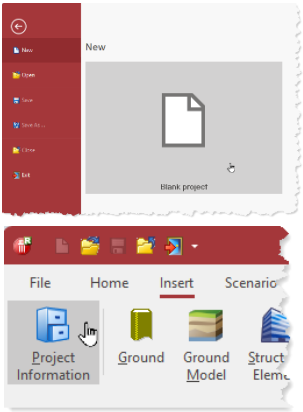
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that appears. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

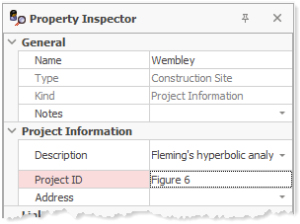
Step 1 – create the project information

In Step 1, you will enter project information about the site and the Engineer.

1. Click **New** on the program's **File** menu to create a blank project. Alternatively, click on the **New** button on the **Quick-access toolbar**, located on the right-hand side of Repute's icon in the tile bar.
2. Open the Stockyard's Project Information group by selecting the **Insert** tab on Repute's ribbon and then clicking on the **Project Information** button.



3. In the **Stockyard**, hold the `Ctrl` key down and click on the item labelled `Construction Site`. A newly created site will appear in the **Project Manager** (under Project Information > Project Information).
4. In the **Project Manager**, double-click on the newly created site (`Site 1`) to open its Property Inspector.
5. In the **Property Inspector**, change the **Name** to `Wembley`; enter `Fleming's hyperbolic analysis` in the **Description** box and click **OK** to confirm what you have typed; and, finally, enter `Figure 6` in the **Project ID** box.



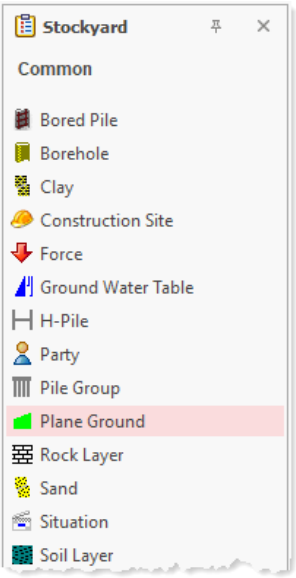
6. Returning to the **Stockyard**, create a Party by holding down the `Ctrl` key and clicking on the item labelled `Party`. A newly created party will appear in the **Project Manager** (under Project Information > Project Information) as `Party 1` and the Property Inspector will display its default properties.
7. In the **Property Inspector**, change the **Name** of the newly created party to `Whitaker and Cooke` and the role to `Engineer` (if not already selected).

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 1.rptz`

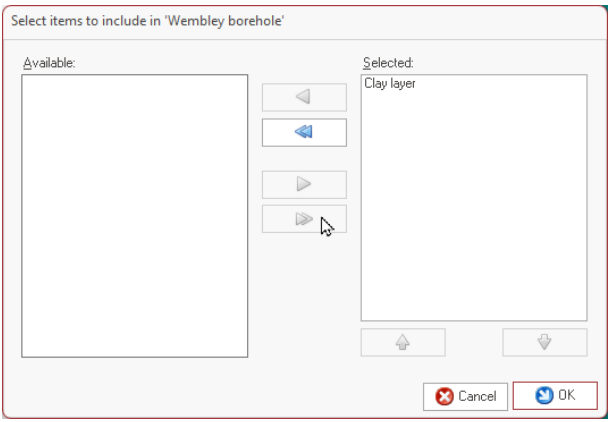
Step 2 – create the ground conditions

In Step 2, you will define the site's ground conditions.



1. Returning to the **Stockyard**, click on the button labelled **Common** to open the Common group, where you will find (amongst other items) Plane Ground.
2. Hold the **Ctrl** key down and click on the item labelled `Plane Ground` to create it.
3. In the **Property Inspector**, change the **Name** `Ground Surface 1` to `Horizontal ground surface` but leave the other (default) properties unchanged.
4. Back in the **Stockyard**, open the **Grounds** group so that you can create a Clay. You may need to click one of the small buttons at the bottom of the Stockyard to display this group. You can move the mouse over each button in turn to display a tooltip indicating which panels they control. Create the Clay by `Ctrl`-clicking on it.
5. In the **Property Inspector**, change the **Name** of the new Clay to `London Clay` and enter the following properties: under the heading **Strength > Drained Strength**, **Peak friction** = 23° and, under **Strength > Undrained strength**, **Undrained strength** = 100 kPa.
6. In the **Stockyard**, open the **Ground Model** group and create both a `Soil Layer` and a `Borehole`.
7. Select `Layer 1` in the **Project Manager** and then (in the **Property Inspector**) change its **Name** to `Clay layer`, its **Thickness** to 25 m, and select `London Clay` in its **Soil** box (under the heading **Material**).
8. Now select `Borehole 1` in the **Project Manager** and (in the **Property Inspector**) rename it `Wembley borehole`. Next, press the **Select ...** button.

9. In the dialog box that appears, click on the **>>** button to move `Clay layer` from the **Available layers** box to the **Selected layers box**. Click **OK** to confirm the changes.



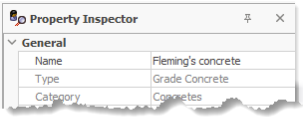
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 2.rptz`

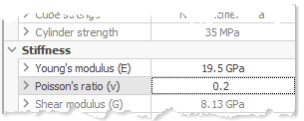
Step 3 – create the pile

In Step 3, you will create a bored pile and specify its cross-section and concrete grade.

1. Right-click on the **Stockyard** and select **Materials > Concretes** from the pop-up menu that appears. Then `Ctrl`-click on the concrete class `C35/45`.
2. In the **Property Inspector**, change the **Name** of the concrete to `Fleming's concrete` and its **Young's modulus (E)** to 19.5 GPa.



3. Open the **Structural Elements** group in the **Stockyard** and create a `Bored Pile`.



4. In the **Property Inspector**, change the **Length** of the pile to 12.2 m and its **Shaft diameter** to 775 mm (its **Base diameter**

will automatically increase to 775 mm). Finally, select `Fleming's concrete` in the **Material name** box.

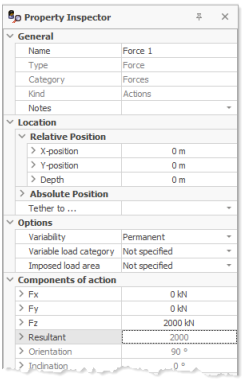
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 3.rptz`

Step 4 – create the force

In Step 4, you will create the force applied to the pile.

1. Open the **Actions** group in the **Stockyard** and create a `Force`.
2. In the **Property Inspector**, change the **Variability** flag (under **Options**) to `Permanent`.
3. Change **Fz** (under **Components of action**) to 2000 kN. The **Resultant** will automatically change to 2000 kN.



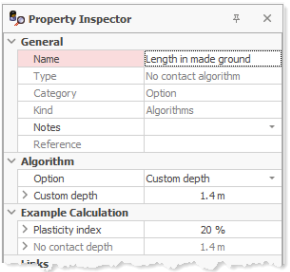
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 4.rptz`

Step 5 – sleeve the pile

In Step 5, you will sleeve the pile through the made ground.

1. Open the **Algorithms** group in the **Stockyard** and create a `No contact algorithm`.
2. In the **Property Inspector**, change the **Name** of the algorithm to `Length in made ground` and its **Custom depth** to 1.4 m. **Algorithm > Option** will automatically change to `Custom depth`. Under the heading **Example Calculation**, the **No contact depth** will also change to



1.4 m (which is depth of ground that the calculation will ignore when calculating shaft friction).

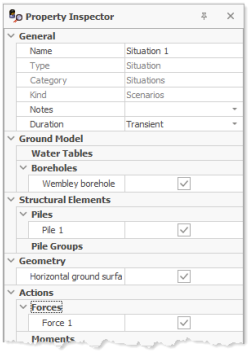
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 5.rptz`

Step 6 – create the scenario and calculation

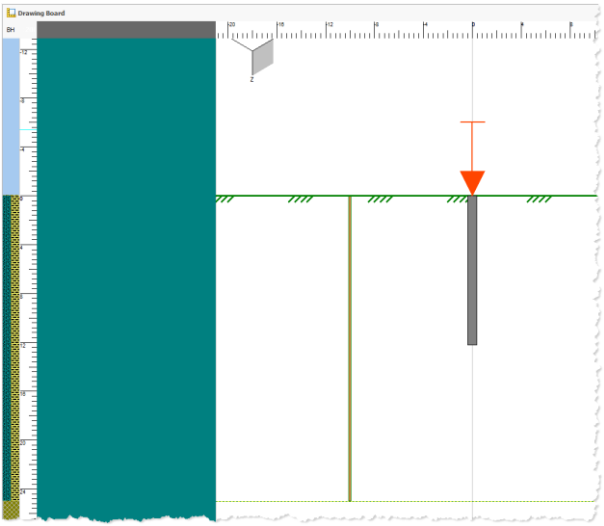
In Step 6, you will create the scenario and calculation and link various items together.

1. Re-open the **Common** group in the **Stockyard** and create a `Situation`.
2. In the **Property Inspector**, change the situation's **Duration** to `Transient` (if not already set).
3. Then, tick the following items so that they appear in this scenario:

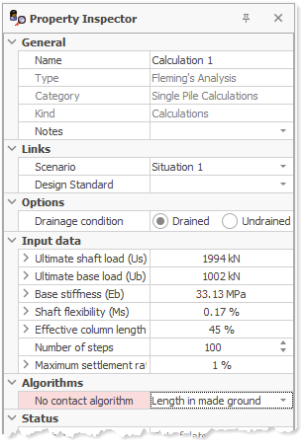


   - `Wembley borehole` (found under **Ground Model > Boreholes**)
   - `Horizontal ground` (under **Geometry**)
   - `Pile 1` (under **Structural Elements > Piles**)
   - `Force 1` (under **Actions > Forces**)

4. As you do so, these items will appear on the **Drawing Board**:



5. Open the **Calculations** group in the **Stockyard** and create a `Fleming's Analysis`.
6. In the **Property Inspector**, set the **Scenario** to `Situation 1` and the **No contact algorithm** to `Length in made ground`.
7. Enter the following values for the calculation's other properties: **Ultimate shaft load (Us)** = 1994 kN; **Ultimate base load (Ub)** = 1002 kN; **Base stiffness (Eb)** = 33.125 MPa; **Shaft flexibility (Ms)** = 0.17%; and **Effective column length (Ke)** = 45%. These values are taken directly from Fleming's paper.



8. Finally, set the **Maximum settlement ratio** to 1%. This controls how much of the load vs displacement curve is generated.

You want to go up to about 8 mm displacement, which is 1% of the pile diameter.

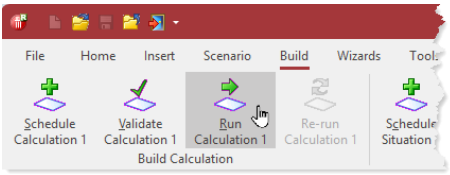
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 2\Step 6.rptz`

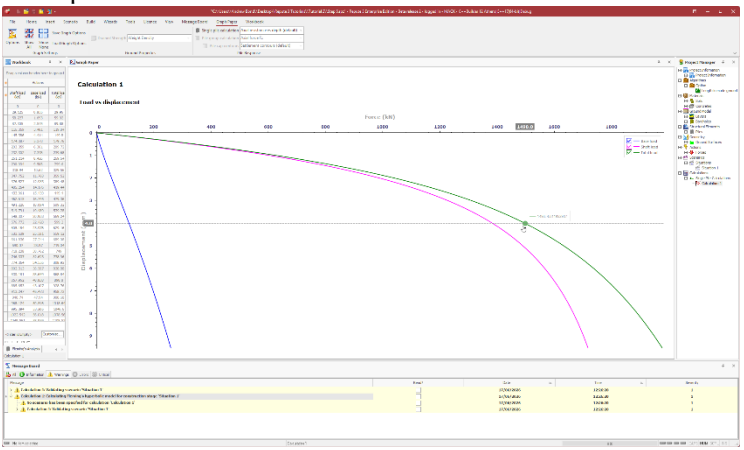
Step 7 – perform and review the calculation

In Step 7, you will perform the calculation and produce a report showing the relationship between load and displacement.

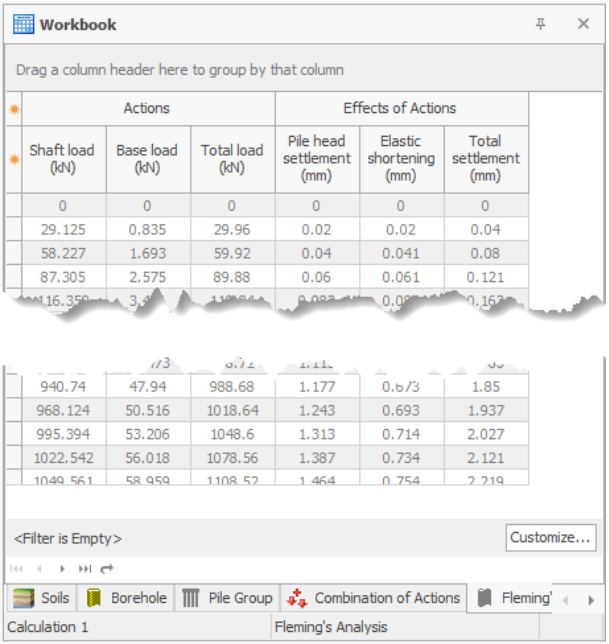
1. Run the calculation by selecting the **Build** tab on Repute's ribbon and then clicking on the button labelled `Run Calculation 1`.



2. Repute performs the calculation and then displays the **Workbook**, **Graph Paper**, **Project Manager**, and **Message Board**. (aka its Check View).
3. The **Graph Paper** displays a graph which is near-identical to Figure 6 in Fleming's paper, showing base load vs displacement (in blue), shaft load (in pink), and total load (in green). If you move your mouse over the lines, you will see spot values as shown below.



4. If you expand the width **Workbook**, you can see the data on which this graph is based.



Step 8 – close the project

In Step 8, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the **File** menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before proceeding. Answer **Yes** or **No** by clicking the appropriate button.
3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

[Tutorials]\Tutorial 2\Tutorial 2.rptz

What next?

Tutorial 3 shows you how to design a single pile in accordance with the requirements of Eurocode 7.

Tutorial 3. Single pile design to Eurocode 7

Introduction

Tutorial 3 shows you how to design a single pile according to the requirements of Eurocode 7. The worked example is taken from Chapter 13 of the book *Decoding Eurocode 7* by Bond and Harris (2008), London: Taylor and Francis.

- Ground conditions comprise 8 m of medium strength sandy CLAY overlying medium dense gravelly SAND. The clay has a representative undrained strength of 45 kPa and a representative weight density of 18.5 kN/m^3. The sand has a representative peak internal friction angle of 36°, zero effective cohesion, and representative weight density of 20 kN/m^3. The sand's constant-volume angle of shearing-resistance is 33°.
- Groundwater at the site is at 1 m depth and skin friction above groundwater will be ignored.
- The pile studied is a 10 m long, 400 mm square pile made of concrete with characteristic weight density of 25 kN/m^3.
- Vertical loads of 650 kN (permanent) and 250 kN (variable) will be applied to the pile.
- You want to determine the minimum pile length required by Eurocode 7 according to the National Annexes published in Ireland and in the United Kingdom.

Full hand calculations for this example are given as Examples 13.1 and 13.2 in Bond and Harris (2008).

This tutorial is written for users of the Standard, Enterprise, and Trial Editions of Repute only. Users of the Professional Edition should look at Tutorials 4-7.

Overview

- In Step 1, you will use the Project Wizard to enter project information, select design standards, and create a scenario to represent short-term conditions.
- In Step 2, you will use the Borehole Wizard to create a borehole containing the clay and sand layers.
- In Step 3, you will add a water table to the scenario.
- In Step 4, you will create a pile and specify its cross-section and concrete grade.
- In Step 5, you will use the Action Wizard to create the forces applied to the pile.
- In Step 6, you will use the Calculation Wizard to create a calculation and specify the design standard to use in that calculation.
- In Step 7, you will specify precise details of how you want the calculation to be performed.
- In Step 8, you will perform the calculations and review the results.
- In Step 9, you will create a second calculation, identical to the first but based on the UK National Annex to Eurocode 7.
- In Step 10, you will close (and optionally save) the project.

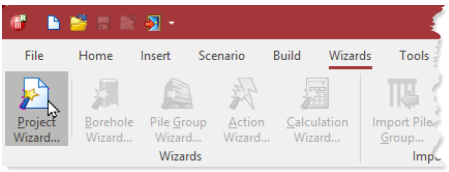
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

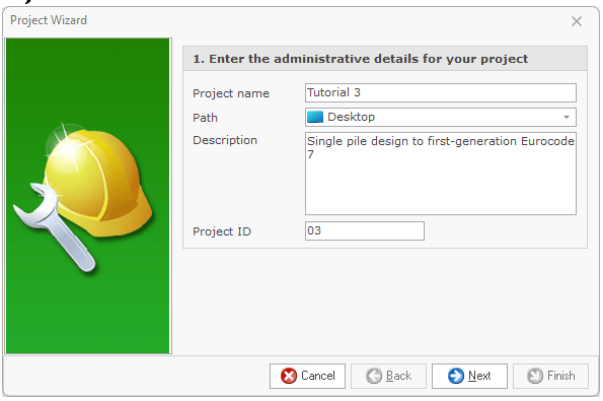
Step 1 – create the project

In Step 1, you will use the Project Wizard to enter project information, select design standards, and create a scenario to represent short-term conditions.

1. Open the **Project Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Project Wizard** button.

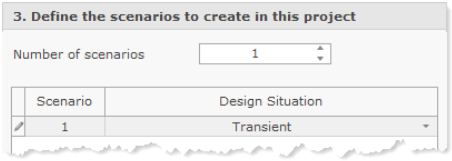


1. When the Wizard appears, type `Tutorial 3` in the **Project name** box. Choose the folder where you want to save this project by using the **Path** control. By default, Repute will save the project on your `Desktop`.
2. Enter `Single pile design to first-generation Eurocode 7: 2004` in the **Description** box and `03` in the **Project ID** box.



3. Click **Next** to display page 2 of the Wizard. The design standards that appear here depend on which edition of Repute you are running (the Enterprise Edition supports more design standards than the Standard and Professional editions).
4. Select `BS EN 1997-1:2004` and `IS EN 1997-1:2005` by clicking on the relevant check boxes (a tick mark appears next to a standard when it is selected).

5. Click **Next** to display page 3. Keep the **Number of scenarios** as `1` but change the **Design Situation** of Scenario 1 to `Transient`.



6. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
7. Click **Finish** to generate the project. The Project Wizard then:
   - creates `Site 1`, `Ground Surface 1`, `Design Standards 1` and 2, and `Situation 1`
   - adds `Ground Surface 1` to `Situation 1`
   - creates a new project named `Tutorial 3.rptz` containing all these items
8. Finally, rename the design standards to make it easier to identify them later, as follows ...
9. In the **Project Manager**, right-click on `Design Standard 1` (under **Design Standards > Limit State Standards**) and select **Properties ...**. The **Property Inspector** will appear. Change the **Name** of this standard to `EC7 with UK NA`.
10. Repeat the previous step for `Design Standard 2`, renaming it `EC7 with Irish NA`.

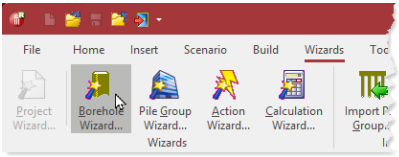
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 1.rptz`

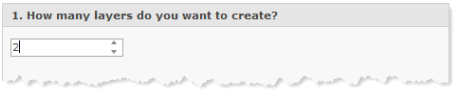
Step 2 – create the borehole

In Step 2, you will use the Borehole Wizard to create a borehole containing the clay and sand layers.

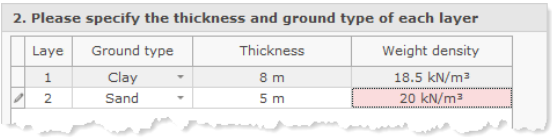
1. Open the **Borehole Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Borehole Wizard** button.



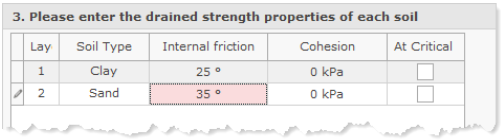
2. When the Wizard appears, set the number of layers to 2.



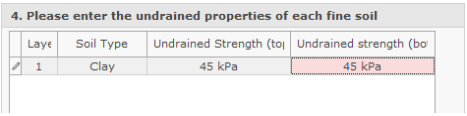
3. Click **Next** to display page 2 of the Wizard. Change Layer 1's **Ground type** to `Clay`, its **Thickness** to 8 m, and its **Weight density** to 18.5 kN/m^3. Leave Layer 2's **Ground type** as `Sand` and its **Weight density** as 20 kN/m^3 but change its **Thickness** to 5 m.



4. Click **Next** to display page 3. Leave the clay's properties unchanged but change the sand's **Internal friction** to 35°. Leave its **Cohesion** at 0 kPa and leave the **At Critical State?** box unticked.



5. Click **Next** to display page 4. The sand does not appear on this page since – as a coarse soil – it does not have undrained strength. Change the **Undrained Strength (top)** and **Undrained Strength (bottom)** of the clay to 45 kPa.



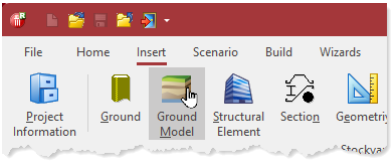
6. Click **Next** to display page 5. Since the ground profile does not include rock, there is nothing to set on this page.
7. Click **Next** to display page 6. Leave the stiffness properties (i.e. **Shear Modulus**) of both layers unchanged.
8. Click **Next** to display page 7. Tick `Situation 1` to add the new borehole to that scenario.
9. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
10. Click **Finish** to generate the borehole. **The Borehole Wizard** then:
    - creates `Soils 1` and 2, `Layers 1` and 2, and `Borehole 1`
    - links `Layer 1` to `Soil 1` and `Layer 2` to `Soil 2`
    - adds `Layers 1` and 2 to `Borehole 1`

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 2.rptz`

Step 3 – add a water table

In Step 3, you will add a water table to the scenario.



1. Open the **Stockyard**'s Ground Model panel by selecting the **Insert** tab on Repute's ribbon and then clicking on the **Ground Model** button.
2. In the **Stockyard**, hold the `Ctrl` key down and click on the item labelled `Ground Water Table`. The water table will appear in the **Project Manager** (under Ground Model > Water Tables).

3. In the **Property Inspector** (under Location > Relative Position), change the **Depth** of `Water Table 1` to 1 m.
4. In the **Project Manager**, select `Situation 1` (under Scenarios > Situations).
5. In the **Property Inspector**, place a tick next to `Water Table 1` (under Ground Model) to add the water table to this scenario. The **Drawing Board** will refresh.
6. In this step, you have:
   - created `Water Table 1`
   - added `Water Table 1` to `Situation 1`

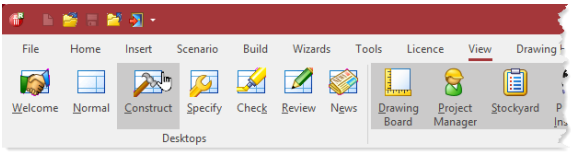
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 3.rptz`

Step 4 – create the pile

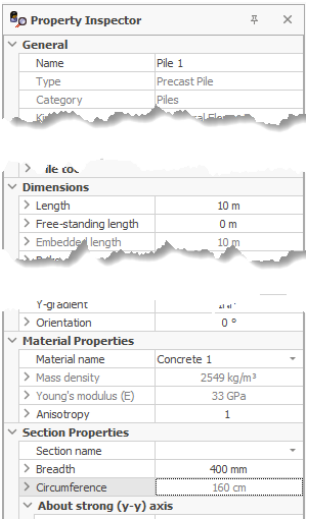
In Step 4, you will create a pile and specify its cross-section and concrete grade.

1. Display Repute's **Construct desktop** by selecting the **View** tab and clicking on the **Construct** button. The program will display the **Drawing Board**, **Project Manager**, and **Stockyard**.



2. Right-click anywhere inside the **Stockyard** to display its context menu and select the **Materials > Concretes** command. The **Concretes** group will open.
3. Create the concrete by holding the `Ctrl` key down and clicking on `C30/37`. (When the `Ctrl` key is pressed, Repute automatically creates an instance of the item that is selected in the **Stockyard**.) The concrete will appear in the **Project Manager** (under Materials > Concretes).

4. Right-click inside the **Stockyard** and select the **Sections > Simple Sections** command. The **Simple Sections** group will open.
5. Create a square section by holding the `Ctrl` key down and clicking on `Square Section`. The section will appear in the **Project Manager** (under Sections > Simple Sections).
6. In the **Project Manager**, right-click on `Section 1` and select **Properties ...**. The **Property Inspector** will appear.
7. In the **Property Inspector**, change the **Breadth** (under Dimensions) of the section to 400 mm.
8. Next, click on the **Stockyard**'s Structural Elements caption (near the bottom of the Stockyard). The **Structural Elements** group will open.
9. Create the pile by holding the `Ctrl` key down and clicking on `Precast Pile`. The pile will appear in the **Project Manager** (under Structural Elements > Piles).
10. In the **Property Inspector**, change the **Length** (under Dimensions) of the pile 10 10 m; the **Material name** (under Material Properties) to `Concrete 1`; and the **Section name** (under Section Properties) to `Section 1`. Leave all other properties of the pile unchanged.



11. Returning to the **Project Manager**, select `Situation 1` (under Scenarios > Situation).
12. In the **Property Inspector**, place a tick next to `Pile 1` (under Structural Elements > Piles) to add the pile to this scenario. The **Drawing Board** will refresh.
13. In this step, you have:
    - created `Concrete 1`, `Section 1`, and `Pile 1`
    - linked `Pile 1` to `Concrete 1` and `Section 1`

- added `Pile 1` to `Situation 1`

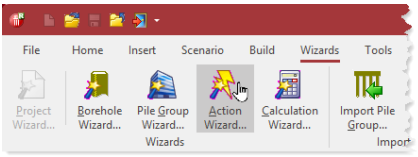
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 4.rptz`

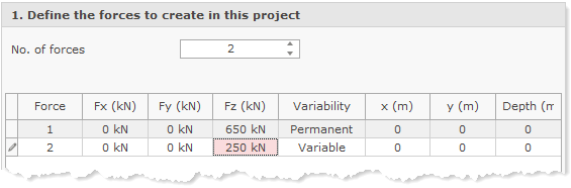
Step 5 – create the forces

In Step 5, you will use the Action Wizard to create the forces applied to the pile.

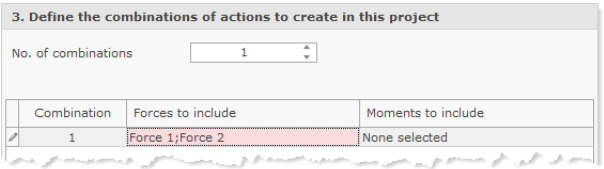
1. Open the **Action Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Action Wizard** button.



2. When the Wizard appears, increase the **No. of forces** to 2; `Force 1` and `Force 2` will be created.
3. Set the **Fz** value for `Force 1` to 650 kN and change its **Variability** to `Permanent`. Then set the **Fz** value for `Force 2` to 250 kN but keep its **Variability** as `Variable`. Leave all other properties unchanged (at zero).



4. Click **Next** to display page 2 of the Wizard. Since no moments are applied to the pile, leave the **No. of moments** as 0.
5. Click **Next** to display page 3. Increase the **No. of combinations** to 1; `Combination 1` will be created. Click on the label `None selected` under the column **Forces to include** and then tick `Force 1` and `Force 2`.



6. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
7. Click **Finish** to generate the actions and their combination. The **Actions Wizard** will then:
   - create `Forces 1` and `2` and `Combination 1`
   - add `Forces 1` and `2` to `Combination 1`
8. Returning to the **Project Manager**, select `Situation 1` (under Scenarios > Situations).
9. Then, in the **Property Inspector**, place a tick next to `Combination 1` (under Actions) to add the combination to this scenario. The **Drawing Board** will refresh.

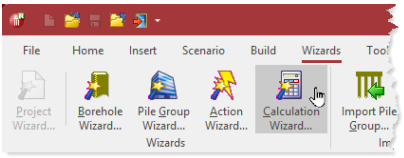
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 5.rptz`

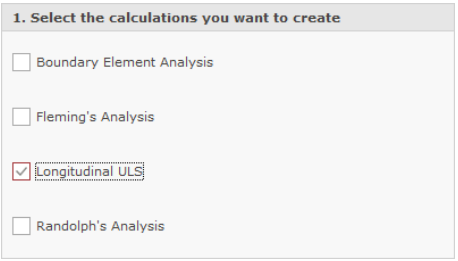
Step 6 – create the calculation

In Step 6, you will use the Calculation Wizard to create a calculation and specify the design standard to use in that calculation.

1. Open the **Calculation Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Calculation Wizard** button.



2. When the Wizard appears, tick `Longitudinal ULS`. (The calculations that appear here depend on which edition of Repute you are running. The Enterprise Edition provides more calculations than the Standard and Professional editions.)



3. Click **Next** to display page 2 of the Wizard. Select `EC7 with Irish NA`.

4. Click **Next** to display page 3. Tick `Situation 1`.
5. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
6. When you are ready, click **Finish** to generate the first calculation. The **Calculation Wizard** then:
   - creates `Calculation 1`
   - links `Situation 1` and `EC7 with Irish NA` to `Calculation 1`

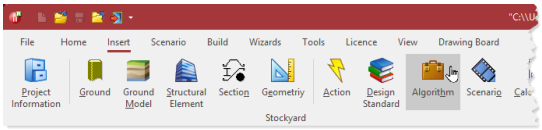
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 6.rptz`

Step 7 – customize the calculation

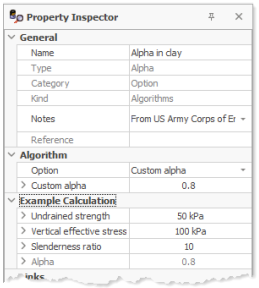
In Step 7, you will specify precise details of how the calculation is to be performed, following the decisions made by Bond and Harris, 2008.

1. Open the **Stockyard**'s Algorithms group by selecting the **Insert** tab on Repute's ribbon and then clicking on the **Algorithm** button.



2. Hold the `CTRL` key down and click on `No contact algorithm` in the **Stockyard**. Then, in the **Property Inspector**, change the **Name** of the new algorithm to `No skin friction` and enter `Ignore skin friction above water table` in its **Notes** field. Set **Custom depth** to 1.0 m. **Algorithm > Option** will automatically change to `Custom depth`.

3. Hold the `CTRL` key down and click on `Alpha` in the **Stockyard**. Change the **Name** of the new algorithm to `Alpha in clay` and enter `From US Army Corps of Engineers` in its **Notes** field. Enter a value of 0.8 into the **Custom alpha** box. **Algorithm > Option** will automatically change to `Custom alpha`.

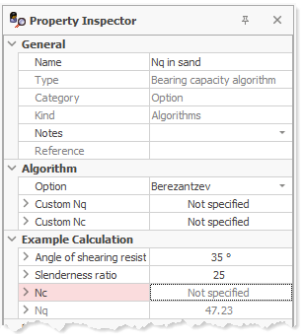


4. Hold the `CTRL` key down and click on `Earth pressure coefficient` in the **Stockyard**. Change the **Name** of the new algorithm to `Ks in sand` and enter `Ks = Nq/50` in its **Notes** field. Enter a value of 1.59 into the **Custom compression coefficient** box. **Algorithm > Option** will automatically change to `Custom earth pressure coefficient`.

5. Hold the `CTRL` key down and click on `Wall friction` in the **Stockyard**. Change the **Name** of the new algorithm to `Delta in sand` and enter `Critical state friction angle` in its Notes field. Enter a value of 33° into the **Custom**

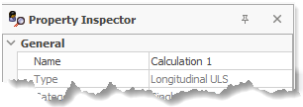
friction value box. **Algorithm > Option** will automatically change to `Custom friction value`.

6. Hold the CTRL key down and click on `Bearing capacity algorithm` in the **Stockyard**. Change the **Name** of the new algorithm to `Nq in sand`. Change the **Option** to `Berezantzev`. In the **Example Calculation** group, change the **Angle of shearing resistance** to 35° and the **Slenderness ratio** to 25. The value of **Nq** will update (to 47.23), while **Nc** remains `Not specified`. Berezantzev's algorithm is only applicable to coarse soils, hence N_c is not available.

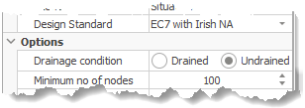


7. In the **Project Manager** (under Calculations > Single Pile Calculations), select `Calculation 1`.

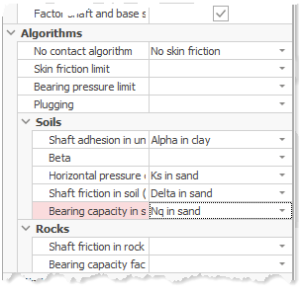
8. In the **Property Inspector**, link all the algorithms you have created to the calculation.



9. Under **Algorithms > No contact algorithm**, select `No skin friction`



10. Under **Soils >**:
    - **Shaft adhesion in undrained soil** – select `Alpha in clay`
    - **Horizontal pressure coefficient in sand (Ks)** – select `Ks in sand`
    - **Shaft friction in soil (delta/phi)** – select `Delta in sand`
    - **Bearing capacity in soil** – select `Nq in sand`



11. Finally, set the **Drainage condition** to `Undrained` to tell the program to base the calculation in clay on total stresses.

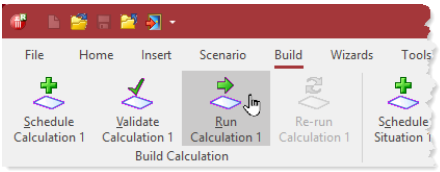
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 7.rptz`

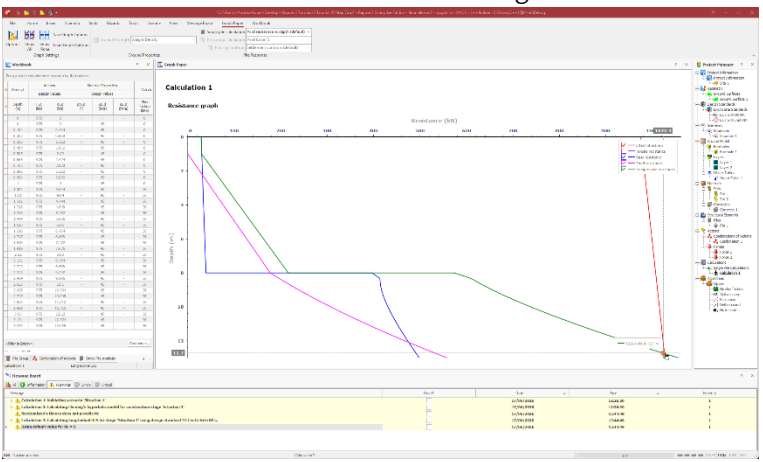
Step 8 – perform and review the calculations

In Step 8, you will perform the calculation and review the results.

1. Run the calculation by selecting the **Build** tab on Repute's ribbon and then clicking on the `Run Calculation 1` button.



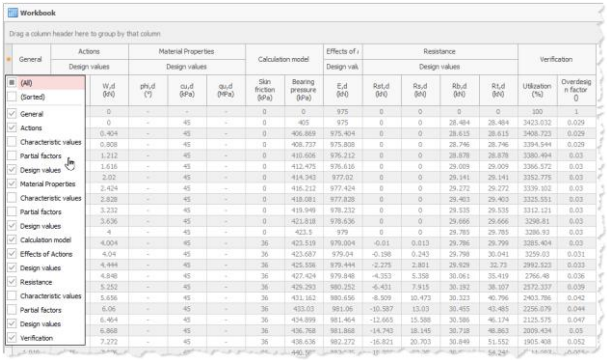
2. Repute will perform the specified calculation and then displays the **Workbook**, **Graph Paper**, **Project Manager**, and **Message Board**. Your screen will now look something like this:



3. The **Graph Paper** shows the:
   - effect of actions (red line) – i.e. the sum of the applied forces and self-weight of the pile – increasing with depth
   - base resistance R_b (blue line) and shaft resistance R_s (pink line) also increasing with depth

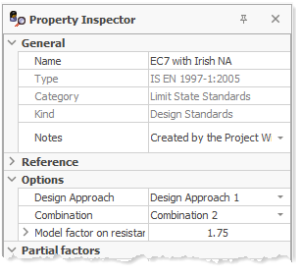
- total compressive resistance $R_c = R_s + R_b$ (green line) exceeding the effect of actions ($R_c > E$) at a depth of 12.7 m

4. The **Workbook** (left panel) shows the same information in tabular format. The Workbook contains a lot more information than is initially shown. To display this additional information, click on the button in the top-left-hand corner (labelled *) and select the data you want to see. The picture here shows results for the depth 10 m.



5. The results calculated by Repute differ from those given in the book by Bond and Harris (2008), where a model factor on resistance of 1.5 was used (following the draft Irish National Annex). Instead, Repute uses the value 1.75 that appears in the published version of that National Annex.

6. You can check this by selecting the **View** tab on Repute's ribbon and then clicking on the **Specify** button. Then, select `EC7 with Irish NA` in the **Project Manager** to display its properties in the **Property Inspector** – the value given for **Options > Model factor on resistance** is 1.75.



You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 3\Step 8.rptz`

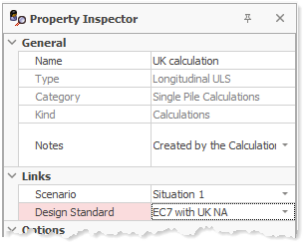
A Repute project that reproduces Bond and Harris's results can be found here:

`[Examples]\Bond and Harris (2008) Example 13.1.rptz`

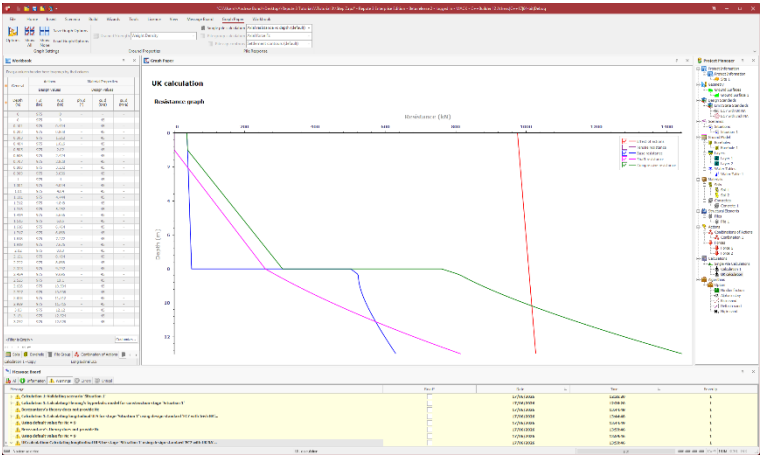
Step 9 – change the design standard

In Step 9, you will create a second calculation, identical to the first but based on the UK National Annex to Eurocode 7.

1. Returning to the **Project Manager**, create a copy of `Calculation 1` by right-clicking on it (look under Calculations > Single Pile Calculations) and selecting the **Edit > Duplicate** command.
2. In the **Property Inspector**, change the **Name** of the newly created calculation to `UK calculation` and its **Design Standard** to `EC7 with UK NA`.



3. In the **Project Manager**, select the Design Standard `EC7 with UK NA` (under Design Standards > Limit State Standards).
4. Then in the **Property Inspector**, change **Options > Pile testing** to `Investigation tests`. The values of the model factor and the resistance factors will change. This setting is appropriate when the calculated resistance will be checked by static pile load tests taken to ultimate load.
5. In the **Project Manager**, select the `UK calculation` (under Calculations > Single Pile Calculations) and then run the calculation by selecting the **Build** tab on Repute's ribbon and clicking on the `Run UK calculation` button. Your screen will now look something like this:



You will find everything you've done so far in the file:

[Tutorials]\Tutorial 3\Step 9.rptz.

Step 10 – close the project

In Step 10, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the File menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before proceeding. Answer **Yes** or **No** by clicking the appropriate button.
3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

[Tutorials]\Tutorial 3\Tutorial 3.rptz

What next?

Tutorial 4 shows how the boundary element method can be used to analyse a pile group.

Tutorial 4. Pile group in clay and sand

Introduction

Tutorial 4 demonstrates how the boundary element method can be used to analyse a pile group. The worked example involves a group of four bored piles in stiff clay overlying dense sand:

- Ground conditions comprise 8 m of stiff clay overlying dense sand. Both soils will be modelled as linear elastic materials.
- The stiff clay has a vertical Young's modulus of 40 MPa and a horizontal modulus of 20 MPa, with Poisson's ratio equal to 0.5.
- The dense sand has the same Young's modulus vertically and horizontally, equal to 50 + 10 z MPa (where z is the depth below the top of the sand in metres), and Poisson's ratio equal to 0.3.
- The bored piles are installed on a 2 x 2 grid, at 3 m spacing (centre-to-centre). Each pile is 20 m long, 1050 mm in diameter, with a Young's modulus (vertically and horizontally) of 30 GPa.
- A characteristic vertical force of 12 MN (permanent), horizontal force of 1 MN (variable), and moment of 2 MNm (variable) are applied at the centre of the pile group.
- You want to determine the displacement and rotation of the pile cap under serviceability conditions.

This tutorial is written for users of the Professional, Enterprise, and Trial Editions of Repute only. Users of the Standard Edition should look at Tutorials 1-3.

Overview

- In Step 1, you will use the Project Wizard to enter project information and create a scenario to represent a persistent design situation.
- In Step 2, you will use the Pile Group Wizard to create a 2 x 2 pile group and specify the piles' properties.
- In Step 3, you will use the Borehole Wizard to create a borehole containing stiff clay and dense sand layers.
- In Step 4, you will enter the stiffness parameters for Soils 1 and 2.

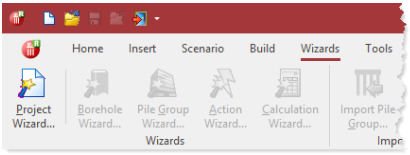
- In Step 5, you will use the Action Wizard to create the actions applied to the pile group.
- In Step 6, you will use the Calculation Wizard to create the calculation you want Repute to perform.
- In Step 7, you will perform the calculation and review the results.
- In Step 8, you will produce a report summarising the calculation.
- In Step 9, you will close and (optionally) save the project.

If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

Step 1 – create the project

In Step 1, you will use the Project Wizard to enter project information and create a scenario to represent the design situation to be analysed.



1. Open the **Project Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Project Wizard** button.
2. When the Wizard appears, type `Tutorial 4` in the **Project name** box. Choose the folder where you want to save this project by using the **Path** control. If you do not change the setting here, the file will be saved on your `Desktop`.
3. Enter `Pile group in clay and sand` in the **Description** box and `04` in the **Project ID** box.
4. Click **Next** to display page 2 of the Wizard. Since we are not going to use a design standard, there is nothing to set on this page.

5. Click **Next** to display page 3. In the table, change the **Design Situation** of Situation 1 to `Persistent` (if it is not already set).
6. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
7. When you are ready, click **Finish** to generate the project. The **Project Wizard** then:
   - creates `Site 1`, `Ground Surface 1`, and `Situation 1`
   - adds `Ground Surface 1` to `Situation 1`
   - creates a new project `Tutorial 4.rptz` containing all these items

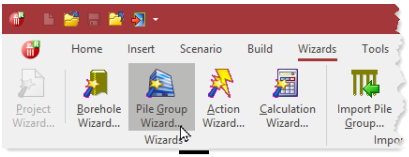
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 4\Step 1.rptz

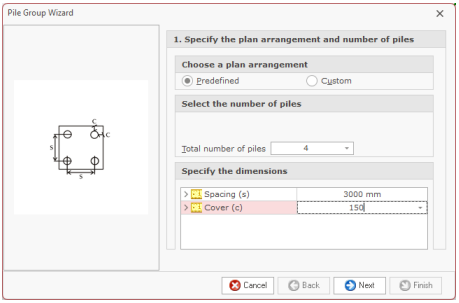
Step 2 – create the pile group

In Step 2, you will use the Pile Group Wizard to create a 2 x 2 pile group and specify the piles' properties.

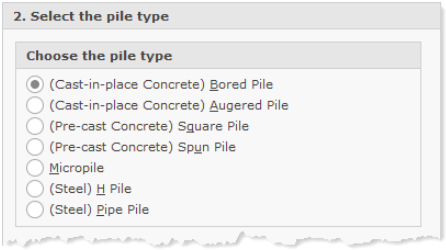
1. Open the **Pile Group Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Pile Group Wizard** button.



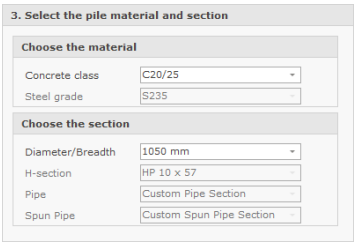
2. When the Wizard appears, choose the `Predefined` plan arrangement and change the **Total number of piles** to 4. The picture on the left-hand side of the Wizard will change to show you the default 2 x 2 pile arrangement. Change the **Spacing** between the piles to `3000` mm but leave the **Cover** at its default value (150 mm).



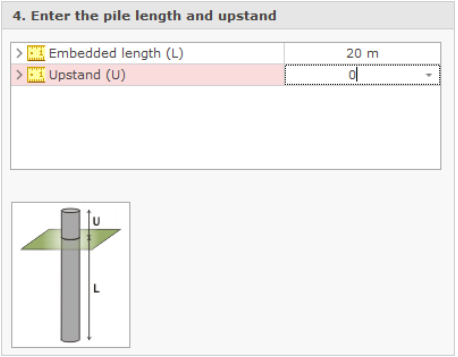
3. Click **Next** to display page 2 of the Wizard. If not already set, set the pile type to `(Cast-in-place Concrete) Bored Pile`.



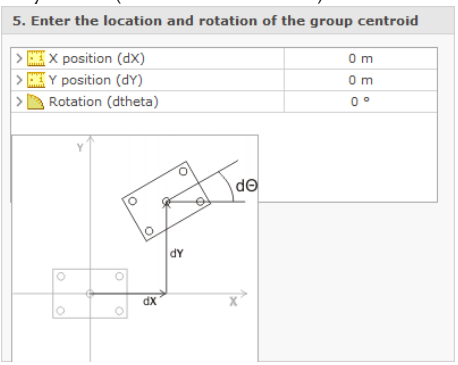
4. Click **Next** to display page 3. Change the **Concrete class** to `C20/25` and the **Diameter/Breadth** to `1050` mm. By default, this concrete's Young's modulus will be set to 30 GPa (not shown).



5. Click **Next** to display page 4. Change the **Embedded length (L)** to `20` m but leave the **Upstand (U)** as 0 m.

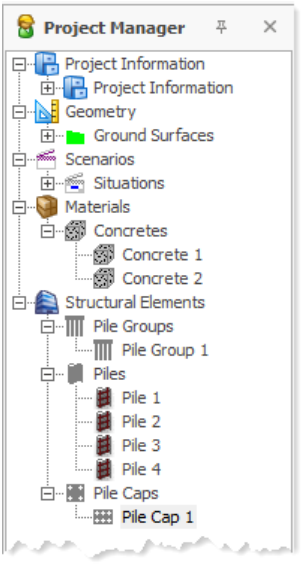


6. Click **Next** to display page 5. Leave all the values on this page as zero – this will position the pile group centrally in the co-ordinate system (and not rotated).



7. Click **Next** to display page 6. Tick `Situation 1` to add the pile group to the scenario.
8. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.

9. When you are ready, click **Finish** to generate the pile group. The **Pile Group Wizard** then:
   - creates `Piles 1` to `4`, `Pile Group 1`, `Pile Cap 1`, and `Concretes 1` and `2`
   - links `Concrete 1` to `Piles 1` to `4`
   - links `Concrete 2` to `Pile Cap 1`
   - adds `Piles 1` to `4` and `Pile Cap 1` to `Pile Group 1`
   - adds `Pile Group 1` to `Situation 1`



Select Pile Cap 1 in the Project Manager and then, in the **Property Inspector**, set its **Thickness** to `1.5` m and press `ENTER`.

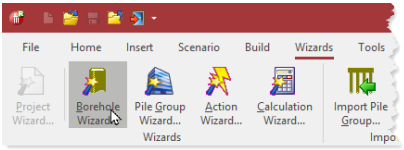
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 4\Step 2.rptz

Step 3 – create the borehole

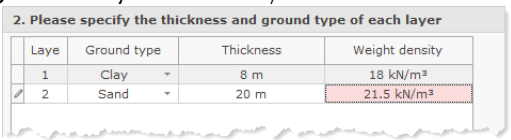
In Step 3, you will use the Borehole Wizard to create a borehole containing stiff clay and dense sand layers.

1. Open the **Borehole Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Borehole Wizard** button.

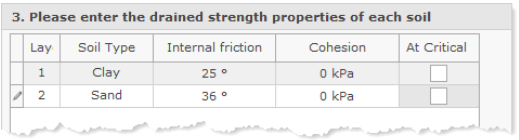


2. When the Wizard appears, increase the number of layers to 2.
3. Click **Next** to display page 2 of the Wizard. Change the **Ground type** of `Layer 1` to `Clay`, its **Thickness** to 8 m, and its **Weight density** to 18 kN/m^3. Leave the **Ground type** of

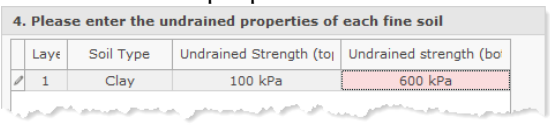
`Layer 2` as `Sand` but change its **Thickness** to 20 m and its **Weight density** to 21.5 kN/m³.



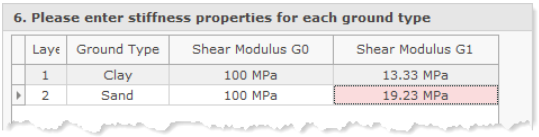
4. Click **Next** to display page 3. Change the **Internal friction** angle of `Layer 1` to 25° and that of `Layer 2` to 36°. Leave the **Cohesion** of both soils as 0 kPa and the **At Critical State?** boxes unchecked.



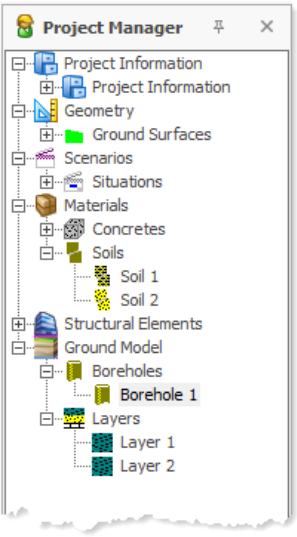
5. Click Next to display page 4. Change the **Undrained strength (top)** of `Layer 1` to 100 kPa and its **Undrained strength (bottom)** to 600 kPa. This strength increase will occur over the full thickness of `Layer 1`, i.e. 8 m (as specified above). `Layer 2` does not appear on this page because – as a sand – it has no undrained properties.



6. Click **Next** to display page 5. Since the ground profile does not include rock, there is nothing to set on this page.
7. Click **Next** to display page 6. For `Layer 1`, to obtain a large-strain undrained Young's modulus E_u = 40 MPa, enter its **Shear modulus G1** as `13.33` MPa, which is equal to $E_u/2(1 + \nu_u) = E_u/3$, with a Poisson's ratio ν_u = 0.5. Leave its **Shear Modulus G0** value as 100 MPa.
8. For `Layer 2`, to obtain a large-strain drained Young's modulus E' = 50 MPa, enter its **Shear modulus G1** as `19.23` MPa – which is equal to $E'/2(1 + \nu) = E'/2.6$, with a Poisson's ratio ν = 0.3. Leave its **Shear Modulus G0** value as 100 MPa.



9. Click **Next** to display page 7. Tick `Situation 1` to add the borehole to the scenario.
10. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
11. When you are ready, click **Finish** to generate the borehole. The **Borehole Wizard** then:
    - creates `Soils 1-2`, `Layers 1-2`, and `Borehole 1`
    - links `Layer 1` to `Soil 1` and `Layer 2` to `Soil 2`
    - adds `Layers 1-2` to `Borehole 1` and `Borehole 1` to `Situation 1`



You will find everything you've done so far in the file:

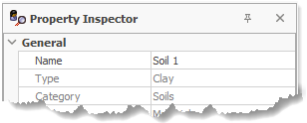
[Tutorials]\Tutorial 4\Step 3.rptz

Step 4 – modify the soils' stiffness properties

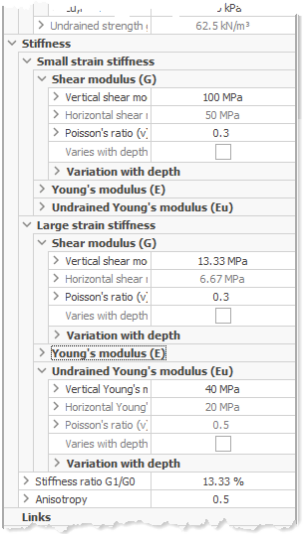
In Step 4, you will modify the stiffness properties of Soils 1 and 2.

1. Select `Soil 1` in the **Project Manager** (under Materials > Soils).
2. In the **Property Inspector**, under the heading **Stiffness > Large Strain Stiffness > Undrained Young's modulus (Eu)**, change the **Vertical Young's modulus** from 39.99 MPa to `40` MPa.

3. Type `0.5` in the **Stiffness > Anisotropy** box to change the **Horizontal Young's modulus (Eh)** value to 20 MPa (i.e. 40 MPa x 0.5).

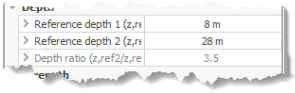


4. The **Stiffness ratio G1/G0** is shown as 13.33%, which is obtained by dividing the large strain stiffness G_1 = 13.33 MPa by the small strain stiffness G_0 = 13.33 MPa. You will find the value of G_0 under the heading **Stiffness > Small Strain Stiffness > Shear modulus (G)**.



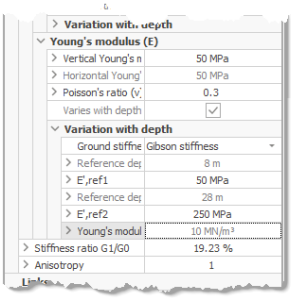
5. Back in the **Project Manager**, select `Soil 2`. The **Property Inspector** will change to display its properties.

6. Under the heading **Depth**, check that **Reference depth 1 (z,ref1)** is equal to 8 m (this is the top of the soil layer) and **Reference depth 2 (z,ref2)** is equal to 28 m (this is the bottom of the layer).



7. Under the heading **Stiffness > Large strain stiffness > Young's modulus**, check that the **Poisson's ratio (v)** of this soil is equal to 0.3.

8. Under the same heading, expand the heading **Variation with depth** and enter `250` MPa into the **E',ref2** box. The value of **Young's modulus gradient (dE/dz)** will change to 10 MN/m^3 – equal to (**E',ref1** – **E',ref2**) / (**z,ref1** – **z,ref2**) = (250 MPa – 50 MPa)/(28 m – 8 m).



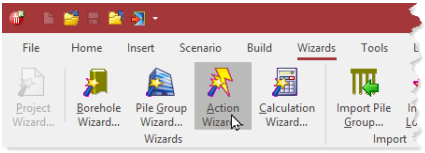
You will find everything you've done so far in the file:

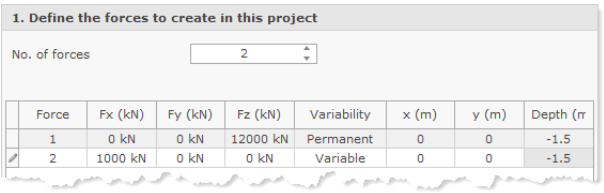
[Tutorials]\Tutorial 4\Step 4.rptz

Step 5 – create the forces and moments

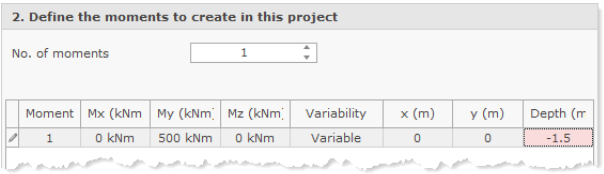
In Step 5, you will use the Action Wizard to create loads applied to the pile group.

1. Open the **Action Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Action Wizard** button.



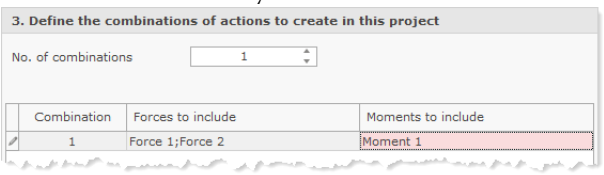


2. When the Wizard appears, increase the **No. of forces** to 2. Set the **Fz** value of `Force 1` to `12000` kN, its **Variability** to `Permanent`, and its **Depth** to –1.5 m. Set the **Fx** value of `Force 2` to `1000` kN, its **Fz** value to `0` kN, and its **Depth** to –1.5 m but leave its **Variability** as `Variable`.
3. Click **Next** to display page 2 of the Wizard. Increase the **No. of moments** to 1. Set the **My** value of `Moment 1` to 500 kNm and its **Depth** to –1.5 m but leave its **Variability** as `Variable`.

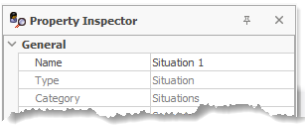


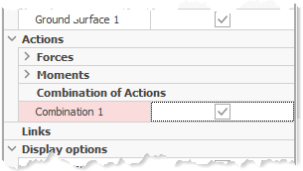
4. Click **Next** to display page 3. Increase the **No. of combinations** to 1. Under the column **Forces to include**, click the text `None selected` and then tick `Force 1` and `Force 2`. Click away. Then, under the column **Moments to**

include, click the text `None selected` and then tick **Moment 1**. Click away.



5. Click **Next** to display the last page. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
6. When you are ready, click Finish to generate the actions and their combination. The **Actions Wizard** then:
   - creates `Forces 1 and 2`, `Moment 1`, and `Combination 1`
   - adds `Forces 1 and 2` and `Moment 1` to `Combination 1`
7. Going back to the **Project Manager**, select `Situation 1` (under Scenarios > Situations).
8. Then, in the **Property Inspector**, tick `Combination 1` (under Actions) to add the combination to this scenario. The **Drawing Board** will automatically update.





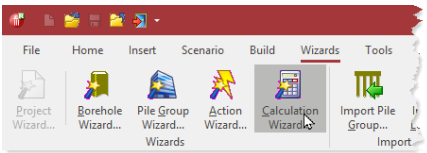
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 4\Step 5.rptz

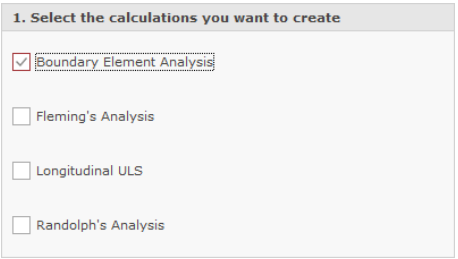
Step 6 – create the calculation

In Step 6, you will use the Calculation Wizard to create the calculation you want Repute to perform.

1. Open the **Calculation Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking the **Calculation Wizard** button.

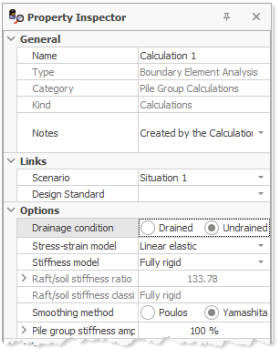


2. When the Wizard appears, select `Boundary Element Analysis`. (The calculations that appear here depend on which edition of Repute you are running. The Enterprise Edition provides more calculations than the Standard and Professional editions.)



3. Click **Next** to display page 2 of the Wizard. Since there are no design standards to select from, there is nothing to do on this page.
4. Click **Next** to display page 3. Select `Situation 1` to link the boundary element analysis to that scenario.
5. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
6. When you are ready, click **Finish** to generate the calculation. The **Calculation Wizard** then:
   - creates `Calculation 1`
   - links `Calculation 1` to `Situation 1`

7. In the **Property Inspector**, for `Calculation 1`, change the **Stress-strain model** (under Options) to `Linear-elastic` and the **Stiffness model** to `Fully rigid` to match the **Raft/soil stiffness classification**.
8. Set the **Drainage condition** to `Undrained` (although the strength of the ground is irrelevant in a linear-elastic analysis, the drainage condition does affect the stiffness used).



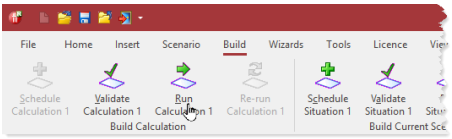
You will find everything you've done so far in the file:

[Tutorials]\Tutorial 4\Step 6.rptz

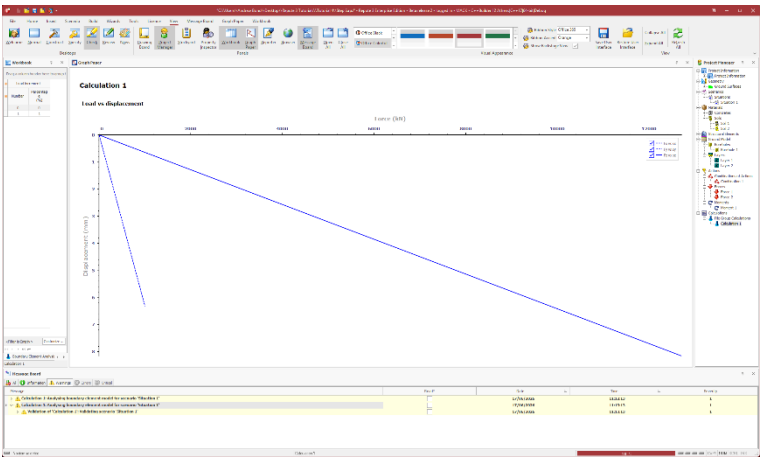
Step 7 – perform and review the calculation

In Step 7, you will perform the calculation and review the results.

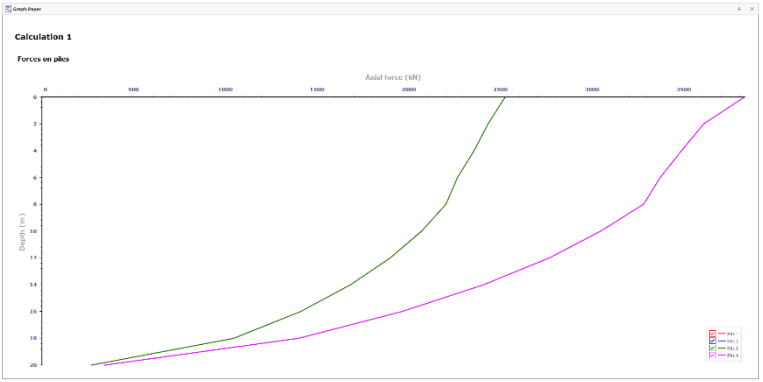
1. Run the calculation by selecting the **Build** tab on Repute's ribbon and clicking on the `Run Calculation 1` button.



2. Repute will run `Calculation 1` and then change the display to show the **Workbook**, **Graph Paper**, **Project Manager**, and **Message Board**. You can switch to this display at any time by clicking on the **Check** button on Repute's **View** tab.
3. Your screen will now look something like this:



4. To display the force distribution along the piles, select the **Graph Paper** tab and, in the drop-down box labelled **Pile group calculation**, select **Axial force Fz**. The graph will change to display the forces in the piles:



Step 8 – close the project

In Step 8, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the **File** menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before

proceeding. Answer **Yes** or **No** by clicking the appropriate button.

3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

[Tutorials]\Tutorial 4\Tutorial 4.rptz

What next?

Tutorial 5 shows you how to set up a non-linear boundary element analysis of a pile group in stiff clay overlying rock.

Tutorial 5. Pile group in stiff clay over a rigid layer

Introduction

This tutorial demonstrates non-linear boundary element analysis of a pile group. It shows you how to:

- Specify a non-linear analysis
- Introduce a rigid layer into the calculations
- Produce a load displacement graph for the pile cap
- Print the results of your calculations

The worked example involves the analysis of a group of 4 piles installed in stiff clay overlying a rigid layer.

- The ground conditions at the site comprise 35 m of stiff clay (Young's modulus 75 + 10 *z* MPa vertically and half that horizontally, where *z* is the depth below the top of the layer; Poisson's ratio 0.5) overlying a rigid layer.
- A non-linear soil model will be used for the clay, with unit weight 19.8 kN/m^3, undrained strength 75 + 10 *z* kPa, and adhesion factor 0.5.
- The water table is at the ground surface.
- The piles will be installed on a 2 x 2 grid, at 3 m spacing (centre-to-centre) along the edge of the grid. Each pile is 20 m long, 1.05 m in diameter, with a free-standing length of 0.5 m and Young's modulus of 30 GPa (both axially and laterally). This is the same pile group as was used in Step 2 of Tutorial 4.
- You are interested in the displacements and rotation of the pile cap under a combined vertical load of 20 MN, horizontal load of 2 MN, and moment of 3 MNm. The loads will be applied at the centre of the pile cap.
- Hyperbolic curve fitting constants of 0.5 (for the shaft), 0.99 (for the base), and 0.9 (for lateral response) should be used.

This tutorial is written for users of the Professional, Enterprise, and Trial Editions of Repute only. Users of the Standard Edition should look at Tutorials 1-3.

Overview

- In Step 1, you will use the Project Wizard to enter project information and create a scenario to represent the design situation to be analysed.
- In Step 2, you will create four piles and connect them together in a pile group.
- In Step 3, you will create a stiff clay and specify its properties.
- In Step 4, you will create a layer, borehole, and water table.
- In Step 5, you will create a force, moment, and combination of actions (to combine the force and moment).
- In Step 6, you will create the calculation that you want Repute to perform.
- In Step 7, you will perform the calculation.
- In Step 8, you will export results to a Microsoft Excel spreadsheet.
- In Step 9, you will close and (optionally) save the project.

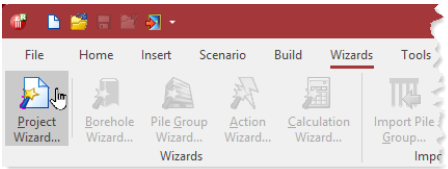
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

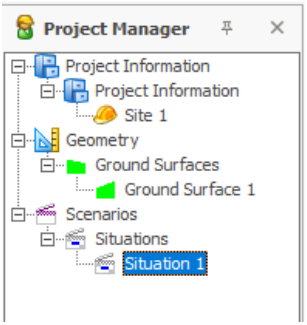
Step 1 – create the project

In Step 1, you will use the Project Wizard to enter project information and create a scenario to represent the design situation to be analysed.

1. Open the **Project Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Project Wizard** button.



2. When the Wizard appears, type `Tutorial 5` in the **Project name** box. Choose the folder where you want to save this project by using the **Path** control. (If you do not change the setting here, it will be saved on your `Desktop`.) Enter `Pile group in stiff clay over a rigid layer` in the **Description** box and `05` in the **Project ID** box.
3. Click **Next** to display page 2 of the Wizard. Since we are not going to use a design standard, there is nothing to set on this page.
4. Click **Next** to display page 3. Since the **Design Situation** of `Situation 1` is already set to `Persistent`, there is nothing to change on this page.
5. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
6. Click **Finish** to generate the project. The **Project Wizard** then:
   - creates `Site 1`, `Ground Surface 1`, and `Situation 1`
   - adds `Site 1` and `Ground Surface 1` to `Situation 1`
   - creates a new project with `Tutorial 5.rptz` with all these items in it



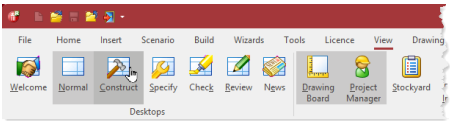
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 5\Step 1.rptz`

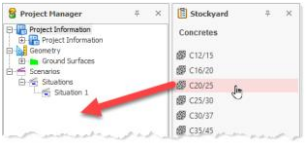
Step 2 – create the pile group

In Step 2, you will create four piles and connect them together in a pile group.

1. Switch to Repute's **Construct Desktop** by selecting the **View** tab on Repute's ribbon and clicking on the **Construct** button. The **Construct Desktop** displays the **Drawing Board**, **Project Manager**, and **Stockyard**.

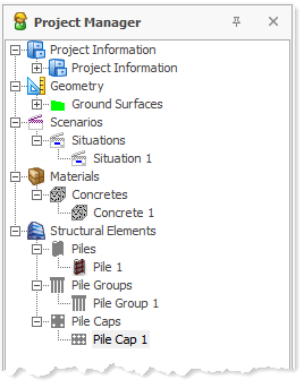


2. Right-click anywhere in the **Stockyard** and select **Materials > Concretes** from the pop-up menu to open the **Concretes** group. Then:



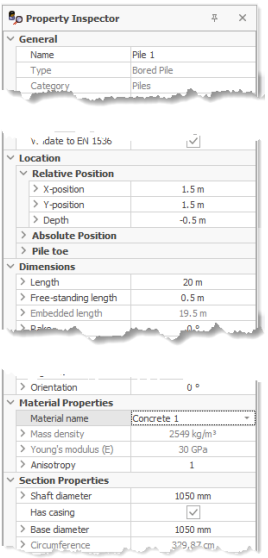
   - click on `C20/25`, keeping the left mouse button pressed
   - drag the cursor away from the **Stockyard** (the cursor will change to signal the dragging operation) and position it over the **Project Manager**
   - release the mouse button to drop the concrete in the **Project Manager** (where it will appear as `Concrete 1`, under the heading Materials > Concretes).

3. Back in the **Stockyard**, click on the heading **Structural Elements** to open the **Structural Elements** group. Then drag a `Bored Pile` to the **Project Manager** as described above. Repeat for a `Pile Group` and a `Rectangular Pile Cap`.

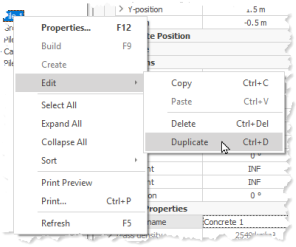


4. Switch to Repute's **Specify Desktop** by selecting the **View** tab and clicking on the **Specify** button. The program will display its **Drawing Board**, **Project Manager**, and **Property Inspector**.
5. In the **Project Manager**, select `Pile 1` (under Structural Elements > Piles).

6. In the **Property Inspector**:
   - under Location > Relative Position, change the pile's **X-position** and **Y-position** to `1.5` m and its **Depth** to –`0.5` m
   - under Section Properties, tick the box **Has casing** and change the pile's **Shaft diameter** to `1050` mm. The pile's **Base diameter** will change to 1050 mm as well
   - under Material Properties, select `Concrete 1` in the **Material name** box
   - under Dimensions, change the pile's **Length** to `20` m (if not already set) and its **Free-standing length** to `0.5` m

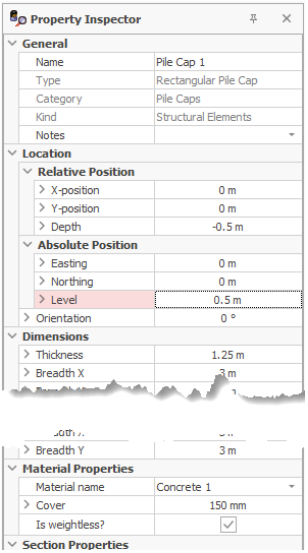


7. Next, duplicate `Pile 1` by right-clicking on it in the **Project Manager** and selecting **Edit > Duplicate** on the pop-up menu. `Pile 1 - Copy` will appear in the **Project Manager** and its properties will be displayed in the **Property Inspector**.

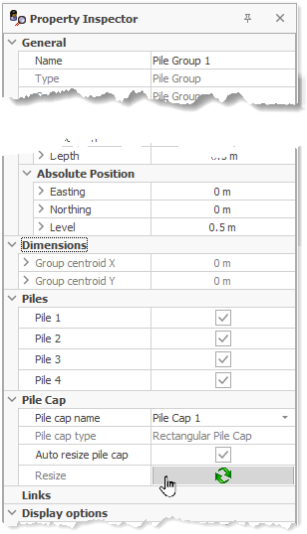


8. In the **Property Inspector**, change the **Name** of the copied pile to `Pile 2` and its `X-position` to –`1.5` m. Leave all its other properties unchanged.
9. Duplicate `Pile 2` via the **Edit > Duplicate** command (as described above) or – even quicker – select `Pile 2` and then press `CTRL+D` to duplicate it.
10. In the **Property Inspector**, change the **Name** of the copied pile to `Pile 3` and its **Y-position** to –`1.5` m, keeping its **X-position** as –1.5 m.

11. Finally, duplicate `Pile 3`, changing the duplicate's **Name** to `Pile 4` and its `X-position` to (plus) 1.5 m, keeping its **Y-position** as −1.5 m.
12. Select `Pile Cap 1` in the **Project Manager** and then, in the **Property Inspector**, change its **Thickness** to 1.25 m and **Level** (under Location > Absolute Position) to 0.5 m. The pile cap's **Depth** changes to –0.5 m when you change its **Level**.
13. Under Material Properties, set the **Material name** to `Concrete 1` and tick the box **Is weightless?**
14. Note that the cap's **Breadth X** and **Breadth Y** values are both 3 m.
15. Next, select `Pile Group 1` in the **Project Manager** and, in the **Property Inspector**:
    - change its **Level** to 0.5 m
    - tick all four piles (under the heading Piles)
    - in the dropdown box **Pile Cap > Pile cap name**, select `Pile Cap 1`



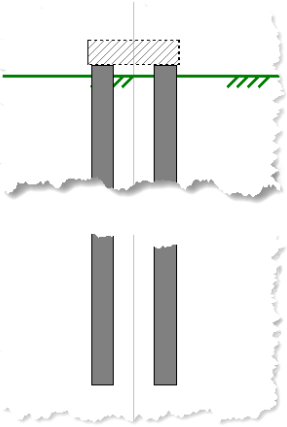
16. Click the **Pile Cap > Resize** button to expand the cap's dimensions to reflect the piles' positions (if you reselect `Pile Cap 1` in the **Project Manager**, you will see that its **Breadth X** and **Breadth Y** values have increased to 4.35 m).



17. Finally, in the **Project Manager**, select `Situation 1` (under Scenarios > Situations) and tick `Pile Group 1` in its **Property Inspector** to add the pile group to the scenario.

18. In this step you have:
    - created `Concrete 1`, `Pile 1`, `Pile Cap 1`, and `Pile Group 1`
    - linked `Concrete 1` to `Pile 1` and `Pile Cap 1`
    - duplicated `Pile 1` to create `Piles 2-4` and changed their locations
    - added `Piles 1-4` and `Pile Cap 1` to `Pile Group 1`
    - added `Pile Group 1` to `Situation 1`

19. The Drawing Board will now look something like this (note that the hatching in the pile cap indicates that it is weightless):



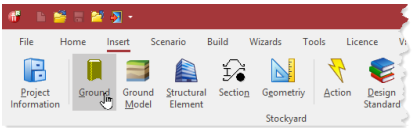
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 5\Step 2.rptz`

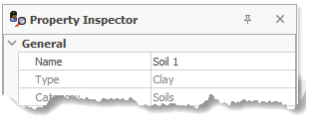
Step 3 – create the soil

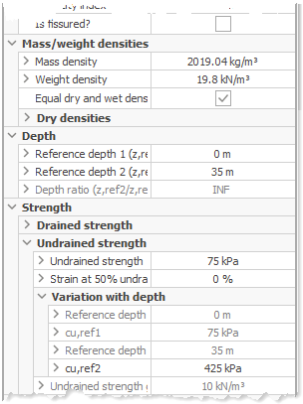
In Step 3, you will create a stiff clay and specify its properties.

1. Open the **Stockyard**'s Grounds group by selecting the **Insert** tab on Repute's ribbon and clicking on the **Ground** button.



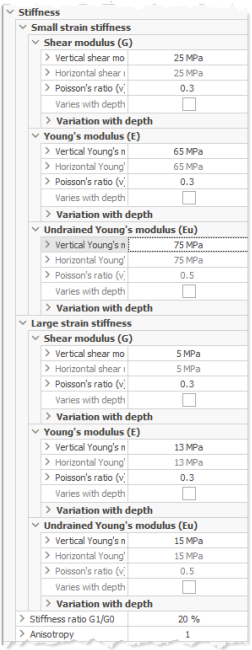
2. In the **Stockyard**:
   - click on `Clay` while keeping the left mouse button pressed
   - drag the cursor away from the **Stockyard** over the **Project Manager** (The cursor will change to signal you are dragging)
   - release the mouse button when the cursor is over the **Project Manager** to drop the clay there (it will appear as `Soil 1` under the heading Materials > Soils)
3. In the **Property Inspector** for `Soil 1`:





   - if necessary, untick **Strict validation** (under Classification)
   - change the clay's **Weight density** (under Mass/weight densities) to 19.8 kN/m^3 – its **Mass density** will change accordingly
   - set **Reference depth 2 (z,ref2)** (under Depth) to `35` m – this is the bottom of the soil layer
   - change the clay's **Undrained strength** (under Strength > Undrained strength) to `75` kPa
   - open the heading **Undrained strength > Variation with depth** and set **cu,ref2** to `425` kPa – the **Undrained strength gradient (dcu/dz)** will change to 10 kN/m^3

4. Under the heading **Stiffness > Small strain stiffness > Undrained Young's modulus (Eu)**:
   - set the clay's **Vertical Young's modulus** to 75 MPa
   - the soil's **Shear modulus (G) > Vertical shear modulus** will change to 25 MPa and its (drained) **Young's modulus (E) > Vertical Young's modulus** will change to 65 MPa.
   - in addition, the **Large strain stiffness > Undrained Young's modulus (Eu)** will change to 15 MPa since the **Stiffness ratio G1/G0** (= E_{u1}/E_{u0}) is set to 20%



5. Open the heading **Stiffness > Small strain stiffness > Undrained Young's modulus (Eu) > Variation with depth** and enter 425 MPa as the value of **Eu,ref2** – the **Young's modulus gradient (dE/dz)** will change to 10 MN/m^3

6. Finally, change the clay's **Anisotropy** to 0.5 – the clay's **Small strain stiffness > Undrained Horizontal Young's modulus (Eh) > Horizontal Young's modulus** changes from 75 MPa to 37.5 MPa

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 5\Step 3.rptz`

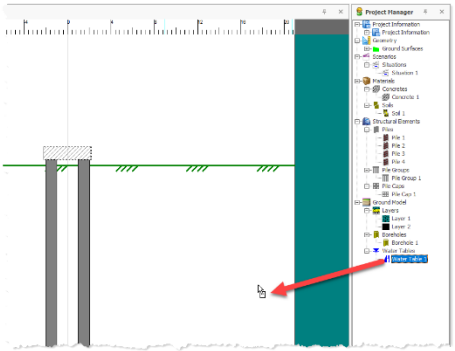
Step 4 – create a layer, borehole, and water table

In Step 4, you will create a layer and borehole to hold the stiff clay and a water table (at ground surface).

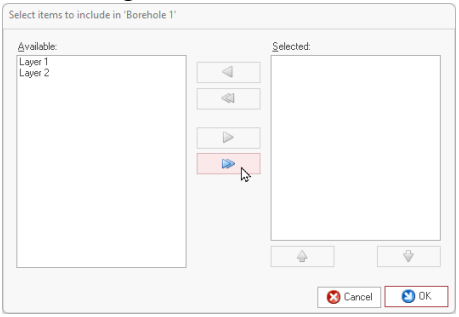
1. In the **Stockyard**, open the **Ground Model** group and then create a Soil Layer, a Rigid Layer, a Borehole, and a Ground

Water Table by dragging-and-dropping these items from the **Stockyard** to the **Project Manager**.

2. Add `Water Table 1` to the scenario by dragging it from the **Project Manager** to the **Drawing Board**.



3. Select `Layer 1` in the **Project Manager** and then, in the **Property Inspector**, change its **Thickness** to `35` m and its **Soil** to `Soil 1`.
4. Select `Borehole 1` in the **Project Manager** and, in the **Property Inspector**, press the **Select ...** button. In the dialog box that appears, click on the **>>** button to move `Layers 1` and `2` from the **Available** to the **Selected** box. Click **OK** to confirm the changes.



5. Add `Borehole 1` to the scenario by dragging it from the **Project Manager** to the **Drawing Board**. The borehole column will appear on the left-hand side of the **Drawing Board**.
6. In this step you have manually:

- created `Layers 1-2`, `Borehole 1`, and `Water Table 1`
- linked `Layer 1` linked `Soil 1`
- added `Layers 1-2` to `Borehole 1`
- added `Water Table 1` and `Borehole 1` to `Situation 1`

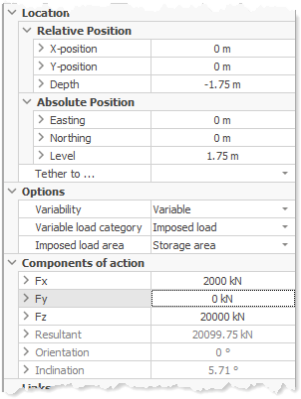
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 5\Step 4.rptz`

Step 5 – create forces and moments

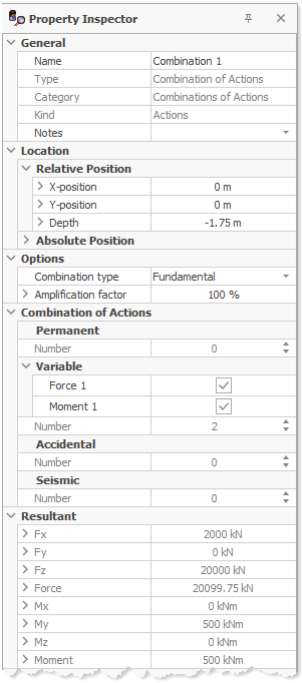
In Step 5, you will create a force, moment, and combination of actions (to combine the force and moment).

1. In the **Stockyard**, open the **Actions** group and then create a Force, a Moment, and a Combination of Actions by dragging-and-dropping these items from the **Stockyard** to the **Project Manager**.
2. Select `Force 1` in the **Project Manager** and, in its **Property Inspector**, change its **Level** (under Location > Absolute Position) to `1.75` m – the **Depth** will change to –1.75 m as you do this (the change in level is necessary to place the force on top of the pile cap).



3. Change the force's **Fx** value to `2000` kN and its **Fz** value to `20 000` kN – the **Resultant** will change to 20 099.75 kN.
4. Select `Moment 1` in the **Project Manager** and, in the **Property Inspector**, change its **Level** to `1.75` m but leave its **My** value as 500 kNm.

5. Select `Combination 1` in the **Project Manager** and, in the **Property Inspector**, tick both `Force 1` and `Moment 1`. Note that the **My** value is shown as 4000 kNm because there is an additional moment due to **Fx** being at a different level to the combination. This moment is equal to 2000 kN x 1.75 m = 3500 kNm. If you change the **Level** of `Combination 1` to `1.75` m, then **My** will revert to 500 kNm as the lever arm of **Fx** becomes zero.



6. Finally, drag-and-drop `Combination 1` from the **Project Manager** onto the **Drawing Board**.
7. In this step you have manually:
   - created `Force 1`, `Moment 1`, and `Combination 1`
   - added `Force 1` and `Moment 1` to `Combination 1`
   - added `Combination 1` to `Situation 1`

You will find everything you've done so far in the file:

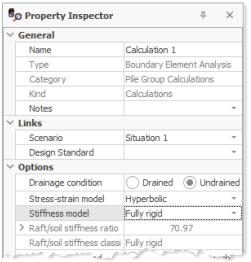
`[Tutorials]\Tutorial 5\Step 5.rptz`

Step 6 – create the calculation

In Step 6, you will create a boundary element analysis of the pile group.

1. In the **Stockyard**, open the **Calculations** group by right-clicking and selecting **Calculations** from the pop-up menu.
2. Drag-and-drop a Boundary Element Analysis from the **Stockyard** to the **Project Manager**.

3. In the **Property Inspector**, change the **Scenario** (under Links) to `Situation 1`. Then, change the **Drainage condition** (under Options) to `Undrained`, the **Stress-strain model** to `Hyperbolic`, and the **Stiffness model** to `Fully rigid`.



4. In this step you have:
   - created `Calculation 1`
   - linked `Calculation 1` to `Situation 1`

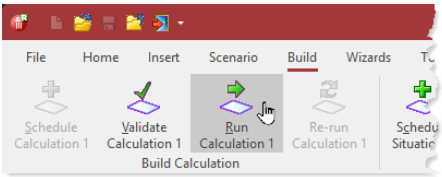
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 5\Step 6.rptz`

Step 7 – perform and review the calculation

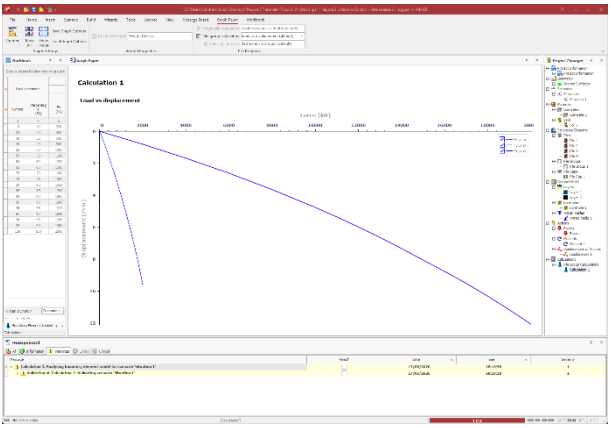
In Step 7, you will perform the calculation and review the results.

1. Run the calculation by selecting the Build tab on Repute's ribbon and then clicking on the **Run Calculation 1** button.



2. Repute will perform the calculation and then change its display to show its **Workbook**, **Graph Paper**, **Project Manager**, and **Message Board**. You can switch to this display at any time by clicking on the **Check** button on Repute's **View** tab.

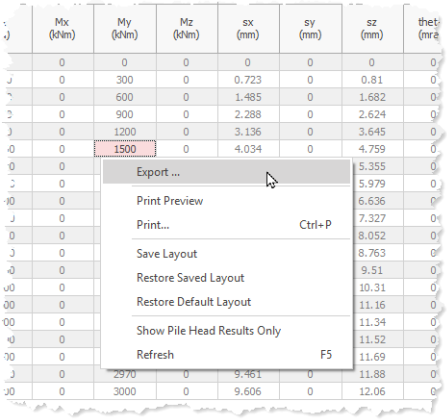
3. Your screen will now look something like this:



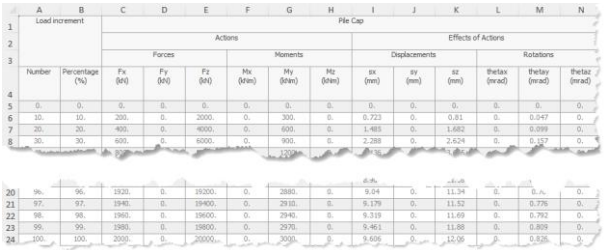
Step 8 – export the results

In Step 8, you will export results to a Microsoft Excel spreadsheet.

1. Right-click in the **Workbook** to reveal its context menu and select the **Export ...** command. Only the data that is currently displayed in the **Workbook** will be exported. To include other results, click on the asterisk * in the top left corner of the worksheet and select the results you want to include.



2. Enter the **File name** for the spreadsheet, navigate to the folder where you want to save it, and then click on the **Save** button. By default, Repute will save the data in a Microsoft Excel workbook (*.xlsx) file.
3. If you open that workbook in Microsoft Excel, it should look something like the image below:



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Load increment		Pile Cap											
2			Actions						Effects of Actions					
3			Forces			Moments			Displacements			Rotations		
4	Number	Percentage (%)	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)	sx (mm)	sy (mm)	sz (mm)	thetax (mrad)	thetay (mrad)	thetaz (mrad)
5	0.	0.	0.	0.	0.	0.	0.	0.	0.	0.	0.	0.	0.	0.
6	10.	10.	200.	0.	2000.	0.	300.	0.	0.723	0.	0.81	0.	0.047	0.
7	20.	20.	400.	0.	4000.	0.	600.	0.	1.483	0.	1.662	0.	0.099	0.
8	30.	30.	600.	0.	6000.	0.	900.	0.	2.288	0.	2.624	0.	0.157	0.
20	96.	96.	1920.	0.	19200.	0.	2880.	0.	9.04	0.	11.34	0.	0.76	0.
21	97.	97.	1940.	0.	19400.	0.	2910.	0.	9.179	0.	11.52	0.	0.776	0.
22	98.	98.	1960.	0.	19600.	0.	2940.	0.	9.319	0.	11.69	0.	0.792	0.
23	99.	99.	1980.	0.	19800.	0.	2970.	0.	9.461	0.	11.88	0.	0.809	0.
24	100.	100.	2000.	0.	20000.	0.	3000.	0.	9.606	0.	12.06	0.	0.826	0.

You will find the exported data in Microsoft Excel format in the file:

```
[Tutorials]\Tutorial 5\Tutorial 5.xlsx
```

Step 9 – close the project

In Step 9, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the **File** menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before proceeding. Answer **Yes** or **No** by clicking the appropriate button.
3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

```
[Tutorials]\Tutorial 5\Tutorial 5.rptz
```

What next?

Tutorial 6 looks at the more complicated case of an asymmetric pile group under 3-dimensional loading.

Tutorial 6. Asymmetric pile group under 3D loading

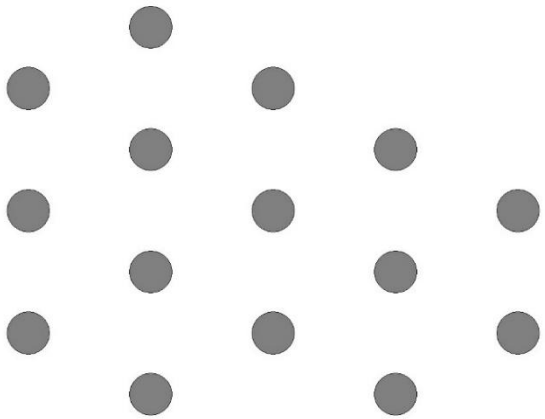
Introduction

Tutorial 6 considers a more complicated project: that of an asymmetric pile group under general 3-dimensional loading. This tutorial shows you how to:

- Modify an existing project
- Change the location of individual piles within a pile group
- Specify 3D loading

The worked example[1] involves the analysis of a group of 15 piles installed in stiff clay overlying dense sand.

- The ground conditions are identical to those in Tutorial 5.
- The piles will be installed on an irregular grid, spaced in rows 3 m apart in the X-direction and 1.5 m in the Y-direction.
- Each pile is 20 m long, 1 m in diameter, and has a Young's modulus of 30 GPa (both axially and laterally).



You are interested in the displacements and rotation of the pile cap under a combined vertical load of 50 MN, horizontal loads of 10 MN (in the X

[1] Suckling, T. P. & Lee, J. (2001). *Piled foundation aspects of the bridge slide at Newark Dyke*, Underground Construction 2001, Int. Symposium & Exhibition, London, pp 361-372.

direction) and 7 MN (in the Y direction), and moments of 5 MNm (in the XZ plane) and 3 MNm (in the YZ plane). The vertical load will be applied on the pile cap at the location of Pile 9.

This tutorial is written for users of the Professional, Enterprise, and Trial Editions of Repute only. Users of the Standard Edition should look at Tutorials 1-3.

Overview

- In Step 1, you will open and modify an existing project.
- In Step 2, you will delete the existing pile group from the project.
- In Step 3, you will create a new pile group.
- In Step 4, you will move the piles to their final positions.
- In Step 5, you will modify the actions on the pile group.
- In Step 6, you will perform the calculation and review the results.
- In Step 7, you will close and (optionally) save the project.

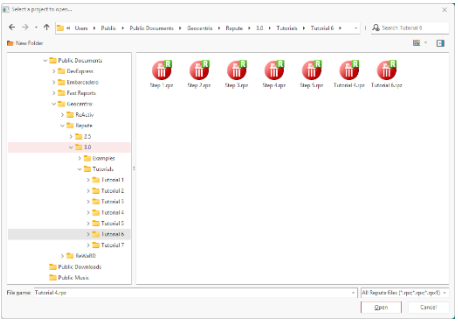
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

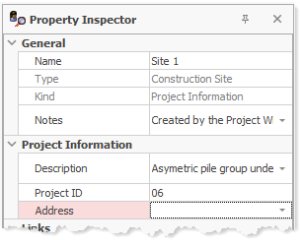
Step 1 – open and modify an existing project

In Step 1, you open an existing project and modify its project information.

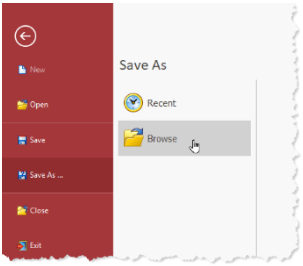
1. Click on the **File > Open** command and, in the dialog box that appears, navigate to the folder [Tutorials]\Tutorial 6 and select the file `Tutorial 4.rptz` (this is a copy of the file that was saved at the end of Tutorial 4).



2. Click on the **Open** button to open this project.
3. In the **Project Manager**, double-click on `Site 1` (under Project Information > Project Information) to display its properties. Then, in the **Property Inspector**, change the **Description** to `Asymmetric pile group under 3D loading` and the **Project ID** to `06`.



4. Save the project under a different name, by clicking **File > Save As > Browse**. In the dialog box that appears, change the **File name** to `Tutorial 6` and click on the **Save** button. Unless you choose another location, the file will automatically be saved in the folder [Tutorials]\Tutorial 6.



You will find everything you've done so far in the file:

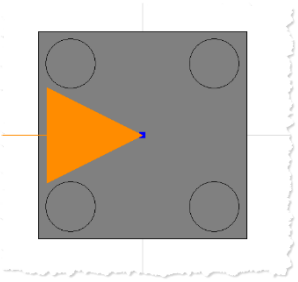
`[Tutorials]\Tutorial 6\Step 1.rptz`

Step 2 – delete the existing pile group

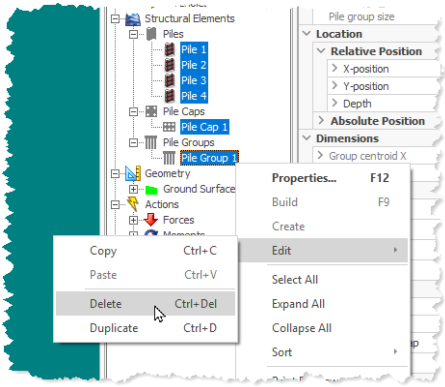
In Step 2, you will delete the existing pile group from the project.

1. `Right-click` anywhere on the **Drawing Board** and select **Plan**.

2. Increase the scale of the drawing by right-clicking on one of the rulers at the edge of the **Drawing Board** and selecting **1:50**.
3. `Click` on the top-left pile (`Pile 1`) in the pile group. A blue selection rectangle will appear with square handles at each corner and `Pile 1` will be highlighted in the **Project Manager**.



4. Hold the `CTRL` key down and then, in the **Project Manager**, select (one at a time) `Piles 1, 2, 3, and 4`; `Pile Cap 1`; and `Pile Group 1`.
5. When all six items are highlighted, right-click on the **Project Manager** and select **Edit > Delete** on the pop-up menu that appears. Click **Yes to All** to confirm that you want to delete all these items from the project and they will then disappear from both the **Project Manager** and the **Drawing Board**.



6. Finally, delete `Concretes 1 and 2` in the same way.

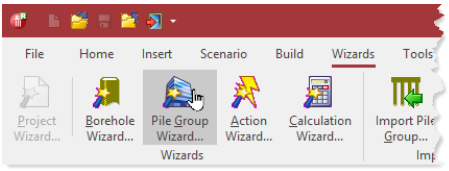
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 6\Step 2.rptz`

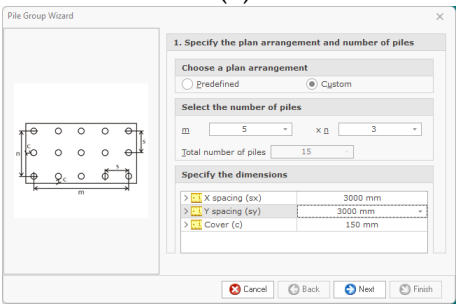
Step 3 – create a new pile group

In Step 3, you will create a new pile group to replace the old one.

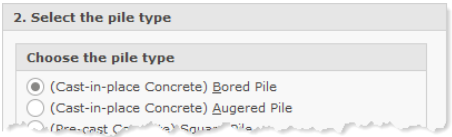
1. Open the **Pile Group Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Pile Group Wizard** button.



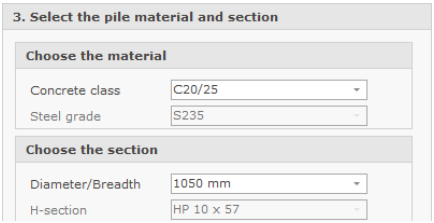
2. When the Wizard appears, choose the **Custom** plan arrangement and change the number of pile rows to **m** = 5 by **n** = 3. (The picture on the left-hand side of the Wizard will change to show you a generic rectangular pile arrangement.) Enter `3000` mm as both the **X spacing (sx)** and the **Y Spacing (sy)** but leave the **Cover (c)** as 150 mm.



3. Click **Next** to display page 2 of the Wizard. Choose `(Cast-in-place Concrete) Bored Pile` as the **pile type**.

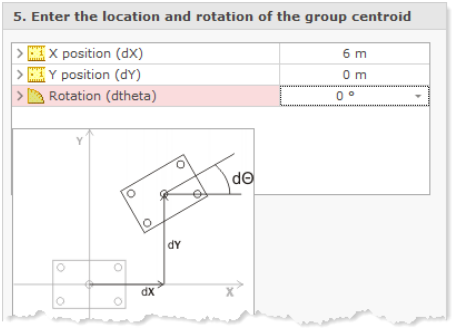


4. Click **Next** to display page 3. Change the **Concrete class** to `C20/25` and the **Diameter/Breadth** to `1050` mm. By default, the concrete will be assigned a Young's modulus of 30 GPa.



5. Click **Next** to display page 4. Change the **Embedded length (L)** to `20` m but leave the **Upstand (U)** as 0 m.

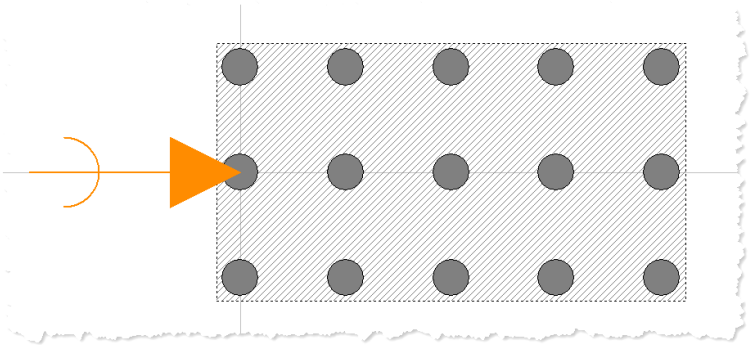
6. Click **Next** to display page 5. Change the pile group's **X position (dX)** to `6` m but leave its **Y position (dY)** as 0 m and its **Rotation (dθ)** as 0°. This will move the piles in the group closer to their final positions.



7. Click **Next** to display page 6. Tick `Situation 1` to add the pile group to the scenario.
8. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
9. When you are ready, click **Finish** to generate the pile group. The **Pile Group Wizard** then:
   - creates `Piles 1-15`, `Pile Cap 1`, `Pile Group 1`, and `Concretes 1-2`

- links `Piles 1-15` to `Concrete 1` and `Pile Cap 1` to `Concrete 2`
- adds `Piles 1-15` and `Pile Cap 1` to `Pile Group 1`
- adds `Pile Group 1` to `Situation 1`

10. Select `Pile Cap 1` in the **Project Manager** and, in the **Property Inspector**, tick the box **Is weightless?** (under Material Properties).
11. Select `Situation 1` in the **Project Manager** and, in the **Property Inspector**, change the **Page size** to A3, **Page orientation** to Landscape, and **Scale** to 1:100. The Drawing Board will now look something like this:



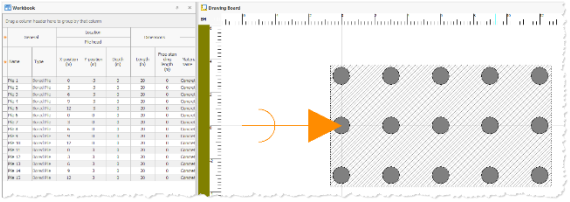
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 6\Step 3.rptz`

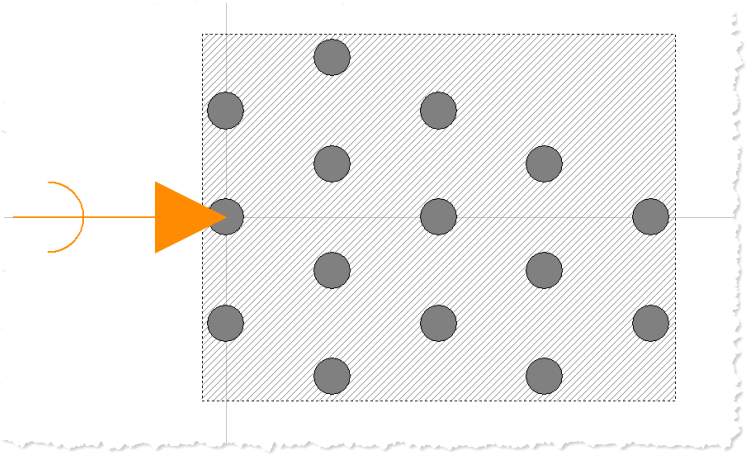
Step 4 – move the piles to their final positions

In Step 4, you will move the piles to their final (asymmetric) positions.

1. Select `Pile Group 1` in the **Project Manager** and then, if it is not already visible, open the **Workbook** by clicking on **View > Workbook** on Repute's ribbon. Widen the Workbook so that you can see the columns headed **General**, **Location**, and **Dimensions** while still displaying the **Drawing Board**, like this:



2. In the **Workbook** row for `Pile 2`, select the cell under the column **Y-position (m)** and enter the value `-4.5` and press ENTER. The pile's position on the Drawing Board will change automatically.
3. Repeat the previous instruction for `Pile 4`.
4. Change the **X-position (m)** of `Pile 15` to `+3.0` m.
5. Change the **Y-position (m)** of:
   - `Piles 7 and 9` to `-1.5` m
   - `Piles 12 and 14` to `+1.5` m.
   - `Pile 15` to `+4.5` m.
6. Back in the **Property Inspector** for `Pile Group 1`, click on the **Resize** button to enlarge the pile cap. The **Drawing Board** will now look something like this:



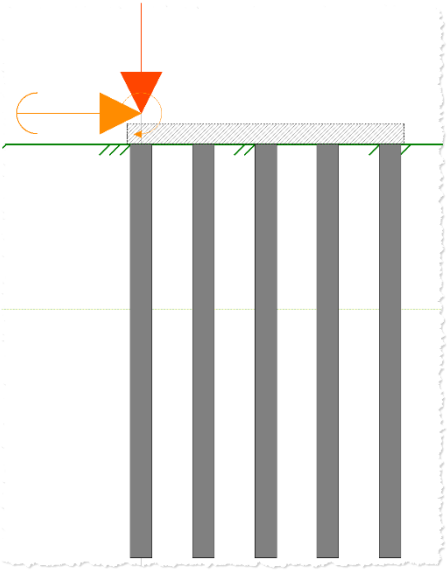
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 6\Step 4.rptz`

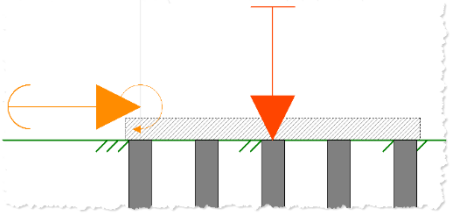
Step 5 – modify the loads

In Step 5, you will modify the actions on the pile group.

1. Firstly, change the viewing plane by right-clicking on the **Drawing Board** and selecting Cross-Section.

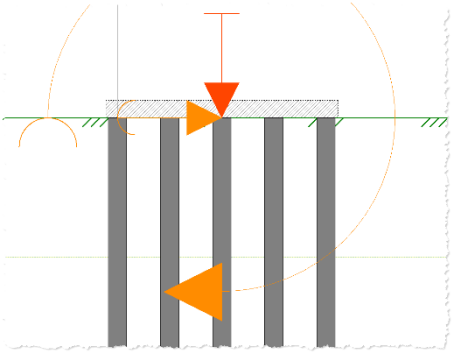


2. Select `Force 1` in the **Project Manager** and then, in its **Property Inspector**, change **Fz** to `50 000` kN and select `Pile 8` in the **Tether to ...** box (this will move `Force 1` so that it sits directly on top of `Pile 8`).



3. Select `Force 2` in the **Project Manager** and, in its **Property Inspector**, change **Fx** to `10 000` kN, **Fy** to `7 000` kN, and select `Pile 8` in the **Tether to ...** box.

4. Select `Moment 1` in the **Project Manager** and, in its **Property Inspector**, change **Mx** to `3 000` kN, **My** to `5 000` kN, and select `Pile 8` in the **Tether to ...** box.
5. The Drawing Board will now look something like this:



You will find everything you've done so far in the file:

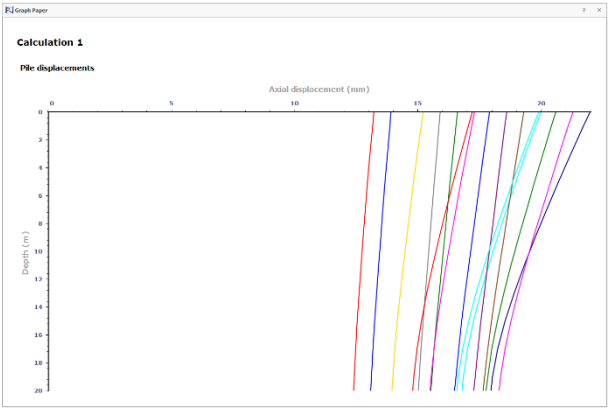
`[Tutorials]\Tutorial 6\Step 5.rptz`

Step 6 – perform and review the calculation

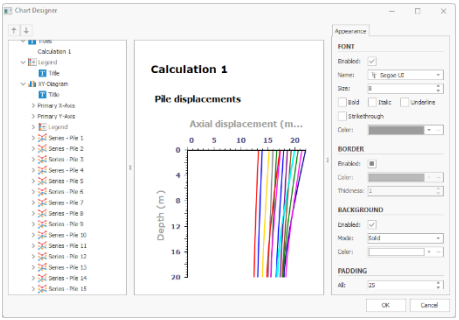
In Step 6, you will perform the calculation and review the results.

1. Right-click on `Calculation 1` in the **Project Manager** and select the **Build** command on the pop-up menu that appears. Repute will perform the calculation and then display its **Workbook**, **Graph Paper**, **Project Manager**, and **Message Board**. You can switch to this display at any time by clicking on the **Check** button on the **View** tab.
2. The load vs displacement graph will now look something like this:

3. You can change the graph that appears on the screen by selecting one of the options in the **Pile Group calculation** drop-down box, which nis located on the **Graph Paper tab**. For example, if you select `Axial displacement sz`, the graph will change to look something like this:



4. You can change the appearance of the graph in the **Chart Designer**, which you can display by right-clicking on the **Graph Paper** and selecting **Options**.



Step 7 – close the project

In Step 7, you will close and – optionally – save the project. (Note: this is not available in the Trial Edition.)

1. Close the project by clicking on the **File** menu's **Close** command.
2. If you have made changes to the project since it was last saved, Repute will ask you if you want to save before proceeding. Answer **Yes** or **No** by clicking the appropriate button.
3. Repute will then (if requested) save and close the project.

You will find a copy of the project in its final form here:

```
[Tutorials]\Tutorial 6\Tutorial 6.rptz
```

What next?

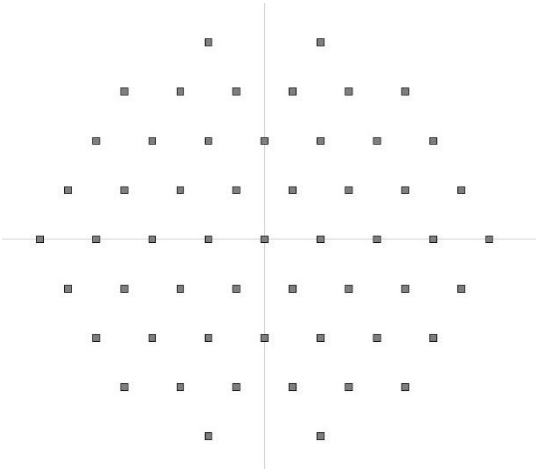
Tutorial 7 looks at a pile group with a fully flexible pile cap, with loads applied directly to the piles.

Tutorial 7. Circular pile group with fully flexible cap

Introduction

Tutorial 7 investigates the settlement of storage tanks founded on soft soils reinforced with driven piles, details of which are given in by Thorburn at al. (1983),[2] as reported by Basile (2021):[3]

- The Ground Model comprises 30 m of clay soil with groundwater located at the ground surface.
- The clay has an undrained strength of 6 kPa at ground surface, increasing to 60 kPa at 30 m. Its small strain undrained Young's modulus increases from 4.5 MPa to 45 MPa over the same depths.
- A total of 55 square precast concrete piles, 29 m long and 250 mm wide are installed on a triangular grid as shown below.



[2] Thorburn S., Laird C.L. and Randolph M.F. (1983). *Storage tanks founded on soft soils reinforced with driven piles*. Proc. Conf. on Recent Advances in Piling, ICE, London, pp 157–164.

[3] Basile, F. (2021). *Influence of cap rigidity on pile group design*. Proc. Piling 2020 Int. Conf., British Geotechnical Association (BGA), ICE Publishing, UK, pp 73-78.

- The precast concrete has a cylinder strength of 25 MPa and Young's modulus of 26 GPa.
- Each pile carries a vertical force of 357 kN.
- The pile cap is circular with diameter 16.55 m and is considered to be fully flexible.
- You want to determine the displacement of the piles under serviceability conditions.

This tutorial shows you how to:

- Import a pile group from data given in an Excel spreadsheet
- Import loads from data given in an Excel spreadsheet
- Specify a pile cap this is fully flexible

This tutorial is written for users of the Professional, Enterprise, and Trial Editions of Repute only. Users of the Standard Edition should look at Tutorials 1-3.

Overview

- In Step 1, you will create the project.
- In Step 2, you will enter the borehole.
- In Step 3, you will complete the ground model.
- In Step 4, you will import the pile group from an Excel spreadsheet.
- In Step 5, you will import the loads from an Excel spreadsheet.
- In Step 6, you will specify a custom alpha value for determining the shaft resistance of the piles.
- In Step 7, you create a boundary element analysis to calculate the settlement of the piles.

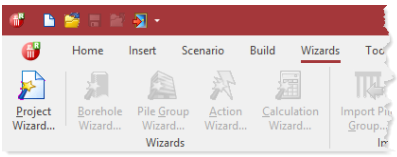
If Repute is not already running, open the program by pressing the Windows key, typing `Repute 3.0` in the search bar, and clicking on the `Repute 3.0` item that should appear. Once the splash screen has disappeared, Repute displays its Welcome screen.

If you have an existing project open, click **Close** on the program's **File** menu. (You will be prompted to save your work if you have not already done so.)

Step 1 – create the project

In Step 1, you will use the Project Wizard to enter project information and create a scenario to represent the design situation to be analysed.

2. Open the **Project Wizard** by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Project Wizard** button.



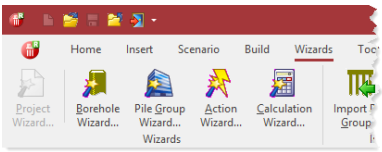
3. When the Wizard appears, type `Tutorial 7` in the **Project name** box. Choose the folder where you want to save this project by using the **Path** control. If you do not change the setting here, the file will be saved on your `Desktop`.
4. Enter `Molasses storage tank` in the **Description** box and `07` in the **Project ID** box.
5. Click **Next** to display page 2 of the Wizard. Since we are not going to use a design standard, there is nothing to set on this page.
6. Click **Next** to display page 3. In the table, change the **Design Situation** of Situation 1 to `Persistent` (if it is not already set).
7. Click **Next** to display the final page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.
8. When you are ready, click **Finish** to generate the project. The **Project Wizard** then:
   - creates `Site 1`, `Ground Surface 1`, and `Situation 1`
   - adds `Ground Surface 1` to `Situation 1`
   - creates a new project `Tutorial 7.rptz` containing all these items

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 1.rptz`

Step 2 – enter the borehole

In Step 2, you will use the Borehole Wizard to define the Ground Model.



1. Next, open the **Borehole Wizard** by selecting the **Wizards** tab on Repute's ribbon and clicking on the **Borehole Wizard** button.
2. When the Wizard appears, increase the number of layers to 1.
3. Click **Next** to display page 2 of the Wizard. Change the **Ground type** of `Layer 1` to `Clay`, its **Thickness** to 30 m, and its **Weight density** to 20.1 kN/m^3.
4. Click **Next** to display page 3. Change the **Internal friction** angle of `Layer 1` to 30° and leave its **Cohesion** as 0 kPa and the **At Critical State?** box unchecked.
5. Click Next to display page 4. Change the **Undrained strength (top)** of `Layer 1` to 6 kPa and its **Undrained strength (bottom)** to 60 kPa. This strength increase will occur over the full thickness of `Layer 1`, i.e. 30 m.
6. Click **Next** to display page 5. Since the ground profile does not include rock, there is nothing to set on this page.
7. Click **Next** to display page 6. Enter the **Shear modulus G0** as `1.5` MPa, which is equivalent to a small-strain undrained Young's modulus (E_{u0}) of 4.5 MPa, since (with a Poisson's ratio $\nu_u = 0.5$) $E_u = 3G$.
8. Enter the **Shear modulus G1** as 0.75 MPa.
9. Click **Next** to display page 7. Tick `Situation 1` to add the borehole to the scenario.
10. Click **Next** to display the last page of the Wizard. If you wish to review any of the settings you have made, click **Back** to return to the relevant page.

11. When you are ready, click **Finish** to generate the borehole. The **Borehole Wizard** then:
    - creates `Soil 1`, `Layer 1`, and `Borehole 1`
    - links `Soil 1` to `Layer 1`
    - adds `Layer 1` to `Borehole 1`
    - adds `Borehole 1` to `Situation 1`

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 2.rptz`

Step 3 – complete the ground model

In Step 3, you will modify the clay's stiffness of the clay so that it increases with depth and add a water table to the Ground Model.

1. Select `Soil 1` in the **Project Manager** (under Materials > Soils).
2. In the **Property Inspector**, close the node **Stiffness > Large strain stiffness** and open the node **Stiffness > Small strain stiffness**.
3. Expand the node **Stiffness > Small strain stiffness > Undrained Young's modulus (Eu) > Variation with depth** and change the value of **Eu,ref2** from `4.5` MPa to `45` MPa and press ENTER
4. Next, open the **Stockyard** by selecting the **Insert** tab on Repute's ribbon and clicking on the **Ground Model** button.
5. In the **Stockyard**, create a Ground Water Table by dragging-and-dropping it from the **Stockyard** to the **Project Manager**.
6. Add `Water Table 1` to the scenario by dragging it from the **Project Manager** to the **Drawing Board**.

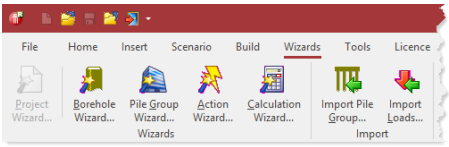
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 3.rptz`

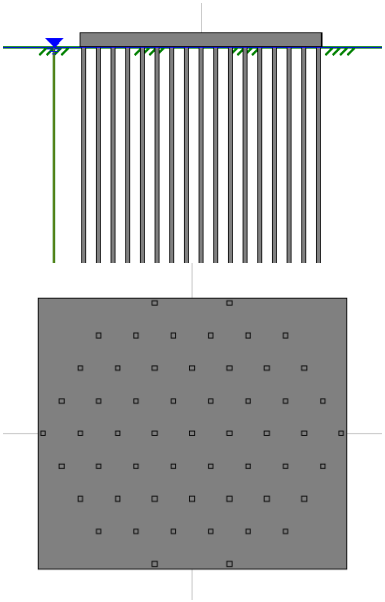
Step 4 – import the pile group from an Excel spreadsheet

In Step 4, you will use the Import Pile Group Wizard to enter details of the 55 piles specified for this project.

Before continuing, we recommend you open the spreadsheet `Pile group from Thorburn et al.xlsx` in Microsoft Excel to see how the pile group data has been formatted so that Repute can import it. You can find this file in the folder `[Tutorials]\Tutorial 7`.

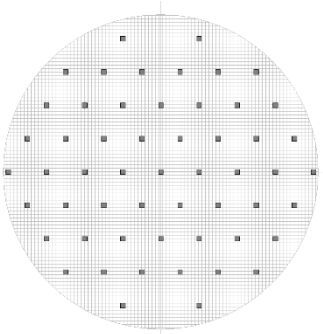


1. Open the Import Pile Group Wizard by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Import Pile Group** button.
2. Navigate to `[Tutorials]\Tutorial 7` and select the file `Pile group from Thorburn et al.xlsx`.
3. When you are ready, click **Open** to import the data from that file. The Wizard then:
   - creates `Piles 01-55`
   - creates the pile group and its pile cap
   - creates concretes for the pile cap and piles
   - creates a square section for the piles
   - adds the piles and the pile cap to the pile group
   - links one of the concretes and the section to each pile
   - links the other concrete to the pile cap
4. Next, expand the **Structural Elements > Pile Groups** node in the **Project Manager** and add the `55-pile group` into the scenario by dragging it to the **Drawing Board**.
5. Double-click on `Situation 1` in the **Project Manager**, under the heading **Scenarios > Situations**.



6. In the **Property Inspector**, change the **Page Size** to A4, **Page Orientation** to Portrait, **Viewing Plane** to Cross-Section, and **Scale** to 1:200. Some or all of these may already be set.
7. To see the pile layout on plan, right-click on the drawing and select **Plan**. Right-click on the ruler and select `1:100` to increase the scale of this view (only).
8. In the **Project Manager**, select `Material for 'Pile 01'` (under **Materials > Concretes**) and then, in the **Property Inspector**, change its **Name** to `Concrete in piles` and its **Young's modulus (E)** to 26 GPa.
9. Select `Material for 'Pile Cap for '55-pile group'',` change its **Name** to `Concrete in pile cap` and its **Young's modulus (E)** to 26 GPa.
10. Select `Section for 'Pile 01'` (under **Sections > Simple Sections**) and then, in the **Property Inspector**, change its **Name** to `Pile section`.
11. In the **Stockyard**, select the heading **Structural Elements** and then create a Circular Pile Cap by dragging-and-dropping it from the **Stockyard** to the **Project Manager**.
12. In the **Property Inspector**, change the **Name** of the new pile cap to `Circular pile cap`, the **Thickness** to 0.3 m, and the **Material name** to `Concrete in pile cap`. Since the weight of the pile cap will be included in the forces subsequently applied to the piles, tick the box **Is weightless?** so that the pile cap weight is not included twice in the calculation.

13. In the **Project Manager**, select `55-pile group` and then, in the **Property Inspector**, set the **Pile cap name** to `Circular pile cap`. Click the Resize button to re-size the cap (the diameter will change to 16.55 m). The cap is shown hatched to remind you it is weightless.



14. Finally, right-click on `Pile Cap for '55-pile group'` in the **Project Manager** and select **Edit > Delete**. Answer **Yes** when prompted to confirm the deletion.

You will find everything you've done so far in the file:

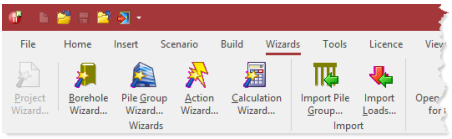
`[Tutorials]\Tutorial 7\Step 4.rptz`

Step 5 – import the loads from an Excel spreadsheet

In Step 5, you will use the Import Loads Wizard to enter details of the loads applied to the 55 piles.

Before continuing, we recommend you open the spreadsheet `Pile loads from Thorburn et al.xlsx` in Microsoft Excel to see how the pile load data has been formatted so that Repute can import it. You can find this file in the folder `[Tutorials]\Tutorial 7`.

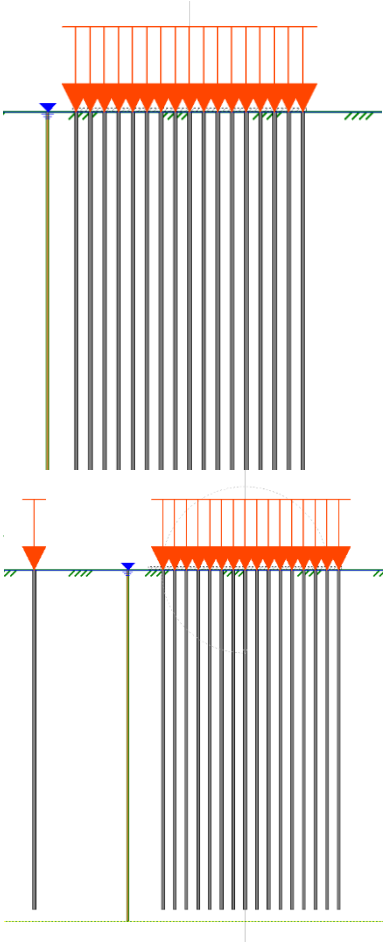
1. Open the Import Loads Wizard by selecting the **Wizards** tab on Repute's ribbon and then clicking on the **Import Loads** button.



2. Navigate to `[Tutorials]\Tutorial 7` and select the file `Pile loads from Thorburn et al.xlsx`.
3. When you are ready, click **Open** to import the data in that file. The Wizard then:

- creates `Forces 01-55`
- creates a combination of actions
- adds the forces to the combination

4. Right-click on the **Drawing Board** and select Cross-Section.
5. Expand the Combination of Actions node in the **Project Manager** and drag `Combined loading` onto the **Drawing Board**.
6. Select `Force 01` in the **Project Manager** and look at its **Tether to ...** value in the **Property Inspector**. This is set to `Pile 01`, which means that the X-position, Y-position, and Depth values of `Force 01` are always kept in sync with those of `Pile 01`.
7. To demonstrate this, select `Pile 01` in the **Project Manager** (under **Structural Elements > Piles**) and change its X-Position to –18 m in the **Property Inspector**. Both the pile and the force that is tethered to it are moved to x = –18 m. Change the X-Position of `Pile 01` back to –8 m to restore its original position.



You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 5.rptz`

Step 6 – specify a custom alpha value

In Step 6, you will specify a custom alpha value to determine the shaft resistance of the piles.

1. Select the **Insert** tab on Repute's ribbon and click on the **Algorithm** button.
2. In the **Stockyard**, create an `Alpha` algorithm by dragging-and-dropping it to the **Project Manager**.
3. In the **Property Inspector**, set the **Custom alpha** value to 0.82. **Algorithm > Option** will change to `Custom`.

You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 6.rptz`

Step 7 – create a boundary element analysis

In Step 7, you will create a boundary element analysis to calculate the settlement of the piles and set some advanced options to ensure a fast calculation.

1. Select the **Insert** tab on Repute's ribbon and click on the **Calculation** button.
2. In the **Stockyard**, create a `Boundary Element Analysis` by dragging-and-dropping it to the **Project Manager**.
3. In the **Property Inspector**, under **Links**, set the **Scenario** to `Situation 1`.
4. Under **Options**, set the **Drainage condition** to `Undrained`, the **Stress-strain model** to `Hyperbolic`, and the **Stiffness model** to `Fully flexible`.
5. Note that the **Raft/soil stiffness ratio** equals 0.12, which indicates that this pile group is `Intermediate` between fully flexible and fully rigid. We are choosing to model the pile group as fully flexible.
6. Expand the **Advanced options > Precision** node and change the **Mesh granularity** to `Medium granularity` (to reduce the **Degrees of freedom**) and the **Number of load increments**

to 25. Together these change will speed the calculation up at the cost of some precision.

7. **Advanced options > Curvature (Hyperbolic model)** node and change the **Shaft factor (Rfs)** to 0.6.
8. Under the heading **Algorithms > Soils**, set the **Shaft adhesion in undrained soil (alpha = fs/cu)** to `Algorithm 1`.

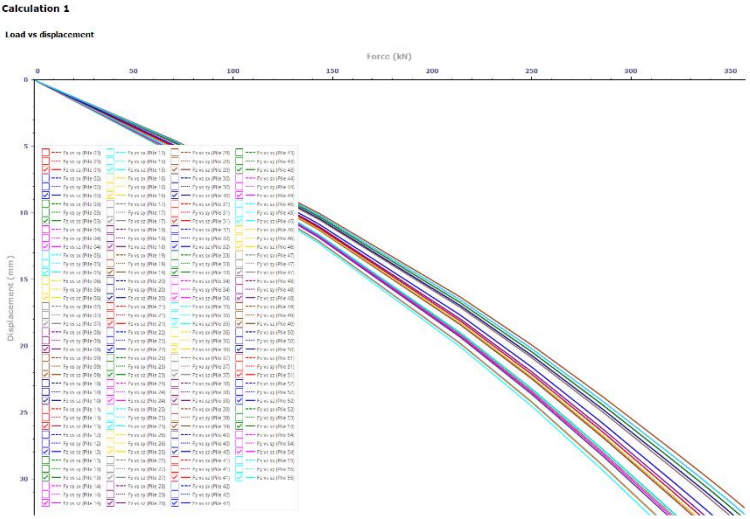
You will find everything you've done so far in the file:

`[Tutorials]\Tutorial 7\Step 7.rptz`

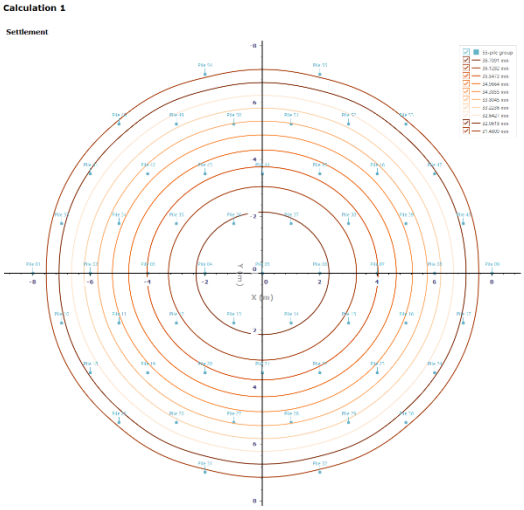
Step 8 – run the analysis

In Step 8, you will run the boundary element analysis and review the result obtained.

1. Select the **Build** tab on Repute's ribbon and click on the **Run All** button.
2. A message box will appear with the message "Please wait – Running calculation 'Calculation 1'". The progress bar at the bottom right-hand corner of Repute's main window tells you how much of the calculation Repute has completed.
3. When the calculation finishes, the Graph Paper will appear showing you load vs displacement curves for all the piles. Right-click on the Graph Paper and select **Options** to display the Chart Designer.
4. Select **Legend** in the left-hand panel and click **Visible** on the right-hand tab sheet to show the graph's legend. Set the **Horizontal** and **Vertical** layouts to `Near`.
5. Click **OK** to close the Chart Designer. You can now tick or untick the curves you want to see on the graph.



6. On the **Graph Paper** tab, click on the text "Settlement contours (default)" next to the label **Pile cap contours** to display contours of pile-head settlement.



You will find a copy of the project in its final form here:

[Tutorials]\Tutorial 7\Tutorial 7.rptz

What next?

Further information about the program's capabilities can be found in the *Repute 3.0 User Manual* and the program's built-in help system. Details of the theory that underpins the program's calculations can be found in the *Repute 3.0 Reference Manual*.

Further resources are available from the Geocentrix website:

www.geocentrix.co.uk/repute