SOLPLAN

FOUNDATION ANALYSIS SOFTWARE

PILE GROUP ANALYSIS · THREE-DIMENSIONAL, NONLINEAR

Solgroup

User Manual

Building, running and reporting three-dimensional pile group analyses with Solgroup.

Release **1.0.10** · Document **SPL-SG-002** · Revision **A**

DOCUMENT TITLE	DOCUMENT NO.	REVISION	ISSUE DATE
Solgroup — User Manual	**SPL-SG-002**	**A**	**2026-11-15**
STATUS	SOFTWARE RELEASE	PAGES	LANGUAGE
Issued for release	**1.0.10**	**154**	**English**

© 2026 Solplan. All rights reserved.

Solplan · Indonesia

Document control

Document	SPL-SG-002 — Solgroup User Manual
Applies to	Solgroup 1.0.10 and later releases until superseded
Revision / date	A / 2026-11-15
Status	Issued for release
Publisher	Solplan

Revision history

Rev.	Date	Description
A	2026-11-15	First issue for Solgroup 1.0.10.

Solgroup documentation set

No.	Title	Content
SPL-SG-001	Solgroup — End-User Licence Agreement	Terms under which Solgroup is licensed to its users.
SPL-SG-002	Solgroup — User Manual	Building, running and reporting three-dimensional pile group analyses with Solgroup.
SPL-SG-003	Solgroup — Technical Reference	Pile-group mechanics, soil-response curves, cap models, numerical solution and verification.
SPL-SG-004	Solgroup — Release Notes	Scope, requirements and known limitations of release 1.0.10.
SPL-SG-005	Solgroup — Third-Party Software Notices	Open-source components distributed with Solgroup and their licences.

This document is uncontrolled when printed or copied. The current revision is the one distributed with the installed release of Solgroup. The information in this document is subject to change without notice and does not represent a commitment on the part of the publisher.

© 2026 Solplan. All rights reserved. Solplan and Solgroup are names used by the publisher for its company and products. Other product and company names mentioned in this document may be trademarks of their respective owners and are used for identification only.

Contents

1 Introduction 6
1.1 Purpose and scope of this manual 6
1.2 What Solgroup computes 6
1.3 Conventions used in this manual 6
1.4 Documentation set 6

2 Installation, licence and start-up 7
2.1 Requirements 7
2.2 Installing 7
2.3 Licence and sign-in 7
2.4 Starting the program 8
2.5 Files and folders 8

3 The Solgroup window 9
3.1 The workflow of an analysis 9
3.2 The menus 10
3.3 The File menu 12

4 Starting a model from a template 14

5 Units 16

6 Getting started 17
6.1 Step 1: lay out the piles 17
6.2 Step 2: define the pile and cap sections 18
6.3 Step 3: define the soil 19
6.4 Step 4: apply the loads 20
6.5 Step 5: define the load cases and the combinations 21
6.6 Step 6: check and run 21
6.7 Step 7: review the results 22
6.8 Step 8: check equilibrium and report 24

7 Tutorial: a jetty trestle on battered piles 25
7.1 Step 1: lay out the bents 25
7.2 Step 2: define the steel pipe 26
7.3 Step 3: describe the seabed 27
7.4 Step 4: berthing, mooring and current 28
7.5 Step 5: run and review 29

8 Drawing, selecting and editing 32
8.1 Coordinates, grids and coordinate systems 32
8.2 Drawing tools 33
8.3 Selecting 34
8.4 Editing 35
8.5 Views and display 37

9 Defining the model 40
9.1 Definition lists 40
9.2 Materials 41
9.3 Pile sections 41
9.4 Cap sections 55
9.5 Soil layers 56
9.6 Soil profile 70
9.7 User-defined curves 72
9.8 Lateral group effect 75
9.9 Lateral reduction sets 77
9.10 Printing depths for p-y curves 77
9.11 Groups 78

© 2026 Solplan. All rights reserved.

9.12 Model tables 78

10 Assigning properties 79
10.1 Pile properties 79
10.2 Cap properties 81

11 Loads, load cases and combinations 83
11.1 Load patterns 83
11.2 Assigning loads 83
11.3 Load cases 84
11.4 Load combinations 88

12 Analysis 91
12.1 Analysis options 91
12.2 Load cases to run 92
12.3 Check Model 93
12.4 Running the analysis 93
12.5 Multi-file analysis 93
12.6 Command Line and the MCP server 94

13 Results 96
13.1 Results on the model 96
13.2 Result graphs and cap response 96
13.3 Tables and curves 108
13.4 Text reports 109
13.5 Checking the results 111

14 Reports and exports 113
14.1 Report Builder 113
14.2 Other exports 114
14.3 Contents of a report 114

15 Options, tools and help 116
15.1 Preferences 116
15.2 Other options 116
15.3 Tools, Help and Window 117
15.4 Keyboard shortcuts 117

16 Worked examples 119
16.1 Example 1: rectangular group under combined load 119
16.2 Example 2: jetty bents with battered piles 121
16.3 Example 3: rigid and flexible caps under two columns 123
16.4 Example 4: group effect of closely spaced piles 125
16.5 Example 5: connection of the pile heads 126
16.6 Example 6: stiffness of the foundation for the structural model 127
16.7 Example 7: pushover of the group 128
16.8 Example 8: piles in moving soil 129
16.9 Example 9: embedded pile cap 130
16.10 Example 10: torsion from the position of a load 130
16.11 Example 11: effect of scour 131
16.12 Example 12: cracking of a concrete pile 132
16.13 Example 13: moment–curvature of a bored pile 133
16.14 Example 14: prestressed concrete pile 134
16.15 Example 15: steel pipe pile and a check by hand 134
16.16 Example 16: unsymmetrical group of bored piles 135
16.17 Example 17: strength increasing with depth 136
16.18 Example 18: ring of piles loaded in two directions 138
16.19 Example 19: breasting dolphin with battered piles 138
16.20 Example 20: jetty on a sloping seabed 139
16.21 Example 21: rectangular section bending about both axes 141
16.22 Example 22: bored pile with a permanent casing 142

© 2026 Solplan. All rights reserved.

16.23 Example 23: unsymmetrical reinforcement 142
16.24 Example 24: editing the model in a spreadsheet 143
16.25 What to try next 144

17 Troubleshooting 145
17.1 Errors found by Check Model 145
17.2 Bearing failure under vertical load 145
17.3 Failure under a very large horizontal load 145
17.4 An analysis that does not converge 146

Appendix A Check Model messages 147

© 2026 Solplan. All rights reserved.

1 Introduction

1.1 Purpose and scope of this manual

This manual explains how to use Solgroup 1.0.10: how to lay out a group of piles and its cap, how to describe the piles and the ground, how to apply loads, how to run the analysis, and how to read, check and report the results. It is written for geotechnical and structural engineers who design pile foundations. It assumes familiarity with the analysis of laterally loaded piles, but not with Solgroup.

The engineering model, the soil-response curves, the numerical solution and the verification of the program are described in the Technical Reference (SPL-SG-003). This manual refers to it where a choice in the program depends on the theory.

1.2 What Solgroup computes

Solgroup analyses a group of piles connected by a pile cap and loaded in three dimensions. It answers the questions that govern the design of such a foundation:

- **Movement of the cap.** For each load case the program finds the three translations and three rotations of the cap that bring the cap into equilibrium with the forces exerted on it by the piles.
- **Distribution of the load.** It reports the axial force, the two shear forces, the two bending moments and the torque at the head of every pile, so that the most heavily loaded piles can be identified.
- **Response along each pile.** It reports the deflection, slope, bending moment, shear force, axial force and soil reaction along every pile, with the depth and value of the maximum moment.

Each pile is a beam-column supported by nonlinear soil springs: lateral p-y curves along the pile, axial load–settlement curves along the shaft and at the tip, and torsional curves. The cap is either rigid, with six degrees of freedom, or flexible, represented by a mesh of plate elements. The soil can also move horizontally and load the piles (free-field soil movement). Two special analyses complete the program: a pushover of the group under increasing load, and the stiffness of the foundation for use in a structural model.

1.3 Conventions used in this manual

Table 1.1 — Typographic conventions

Convention	Meaning
Define ▸ Load Cases…	a menu command, a button or a field in the program
C:\Program Files\Solplan	a file, a folder or an address
F5	a key or a key combination
p, *y*, *t*, *z*	a symbol of the Technical Reference

The figures show Solgroup with SI units. Notes marked as such contain information that prevents an error or explains a result that may otherwise look surprising.

1.4 Documentation set

Table 1.2 — Solgroup documents

No.	Title	Use
SPL-SG-001	End-User Licence Agreement	Terms of use, licence and session rules
SPL-SG-002	User Manual	This document
SPL-SG-003	Technical Reference	Pile and cap models, soil curves, numerical method, verification
SPL-SG-004	Release Notes	Content of the release, requirements, known limitations
SPL-SG-005	Third-Party Software Notices	Open-source components and their licences

© 2026 Solplan. All rights reserved.

2 Installation, licence and start-up

2.1 Requirements

Solgroup runs on 64-bit Windows 10 or Windows 11. The minimum and recommended configurations are listed in the Release Notes (SPL-SG-004). A display of at least 1280 × 720 pixels is needed so that every data window fits on the screen. The analyses of the piles of a group run in parallel; a processor with four or more cores shortens the run of a large group.

2.2 Installing

Install Solgroup with the Setup program supplied by Solplan (Solgroup-1.0.10-Setup.exe). Setup needs administrator rights and installs the program into C:\Program Files\Solplan\Solgroup. No separate Python installation is needed. Setup offers two options: to open .ggp project files with Solgroup (on by default) and to create a desktop shortcut (off by default). The documentation is installed in the docs folder of the installation.

2.3 Licence and sign-in

Use of Solgroup is governed by the End-User Licence Agreement (SPL-SG-001). A Solgroup licence is separate from a licence of another Solplan program. The rules that affect daily work are these:

1. Each user signs in inside Solgroup with a personal solplan.id account: the e-mail address and password registered at solplan.id, or **Continue with Google**, which opens the browser to sign in there and confirm a short code. When the account holds more than one Solgroup licence, Solgroup asks which one to use.
2. An account can be signed in on one computer at a time. Signing in on a second computer ends the session on the first one; the first computer offers to save open work before it closes.
3. A multi-seat (Team) licence lets as many members work at the same time as it has seats; each seat admits up to five members.
4. Solgroup checks the licence every few minutes while it runs. If the licence server cannot be reached, it warns and tries again after three minutes; after a second failure it offers to save open work and closes.
5. To work without a connection, borrow the licence beforehand in **Help ▸ Licence…** for 1 to 30 days (never beyond the end of the licence). Return it there when you are back on-line.
6. Changing the computer date or time, or using tools that misrepresent it, is not permitted. When Solgroup detects an inconsistency it asks for an on-line validation; it never deletes or changes project files.

Table 2.1 — Licence types

Type	Who uses it	What differs
Personal	one user	full use
Team	members of an organisation, five per seat	full use; as many users at a time as seats
Academic	a university, per campus: any number of users who sign in with an e-mail address of the university	teaching and non-commercial research; reports and exported images are marked “Student version”
Trial	anyone, seven days, once per program for each account and computer	results can be viewed and projects saved, but nothing can be exported, printed or copied

Licences and trials are managed under **Account ▸ Licences** at solplan.id. The type of the licence in use is shown on the right of the status bar.

Help ▸ Licence… shows the **Account** that is signed in and the **Licence** in use with its end date (Figure 2.1). While a session is active it offers borrowing the licence for work without a connection and returning it; **Manage account…** opens the account page at solplan.id, and **Sign out** ends the session on this computer.

© 2026 Solplan. All rights reserved.

If the account that is signed in has no active Solgroup licence, Solgroup says so at start-up and offers **My licences** (the licence page of the account, where a trial is started or a licence is bought), **Check again**, **Sign out** (to sign in with a different account) and **Close**. When the account holds more than one licence, **Use another account** in the licence choice signs out in the same way. Signing out of the website does not sign out the program.

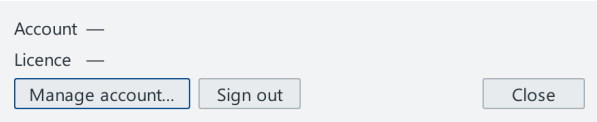


Figure 2.1 — Licence, before signing in

NOTE Project files are always stored on your own computer. The licence service receives account and session data only, never project data (EULA, clause 10).

2.4 Starting the program

Start Solgroup from the Start menu (**Solplan ▸ Solgroup**) or by double-clicking a .ggp file. After the start screen, Solgroup asks you to sign in if no valid sign-in is stored, and then offers the templates of Chapter 4 for a new model. **Cancel** in that window starts a blank model with a grid only.

2.5 Files and folders

Table 2.2 — Files and folders used by Solgroup

File or folder	Content
`*.ggp`	Project. Compressed JSON text with the complete input, but no results: results are recomputed when the project is analysed.
`*.autosave.ggp`	Recovery copy written while a model is edited. After an interruption, Solgroup offers to restore it the next time it starts.
`%APPDATA%\Geostruktur\Solgroup`	Recovery copies of unnamed models and the folder logs with the diagnostic log.

Saving is atomic: Solgroup writes a temporary file and replaces the previous file only when the write has succeeded. An interrupted save therefore never damages the existing project. When you close a project with unsaved changes, Solgroup asks whether to save them.

© 2026 Solplan. All rights reserved.

3 The Solgroup window

Figure 3.1 shows the main window with a rectangular group in the plan view and in a 3D view. Its parts are:

- **Menu bar and tool bars** with every command. The **Main** tool bar holds the file, view and analysis commands; the **Draw & Select** tool bar holds the drawing and selection tools. **Options ▸ Reset Toolbar Layout** restores their positions.
- **View panes.** One, two or four panes (**View ▸ Pane Layout**), each showing the plan (**XY**), an elevation (**XZ**, **YZ**) or a 3D view. Each pane can show the model, the loads, the assigns or the results.
- **Model Explorer**: the objects of the model in a tree (piles, cap, sections, soil, loads, cases). A click selects the object; a double click opens its data.
- **Properties**: the data of the selected object, which can be edited directly.
- **Workflow**: the steps of an analysis in order, with the state of each step and a command to open it.
- **Messages**: the findings of Check Model and the messages of the last run. A click on a message selects the objects it concerns.
- **Command**: the Command Line, in a tab beside *Messages*, where the model is built, changed and analysed with text commands (Section 12.6).
- **Status bar**: the selection, the snap mode, the **Command** button that shows the Command Line, the units of the model (which can be changed there), the progress of a run and the licence type.

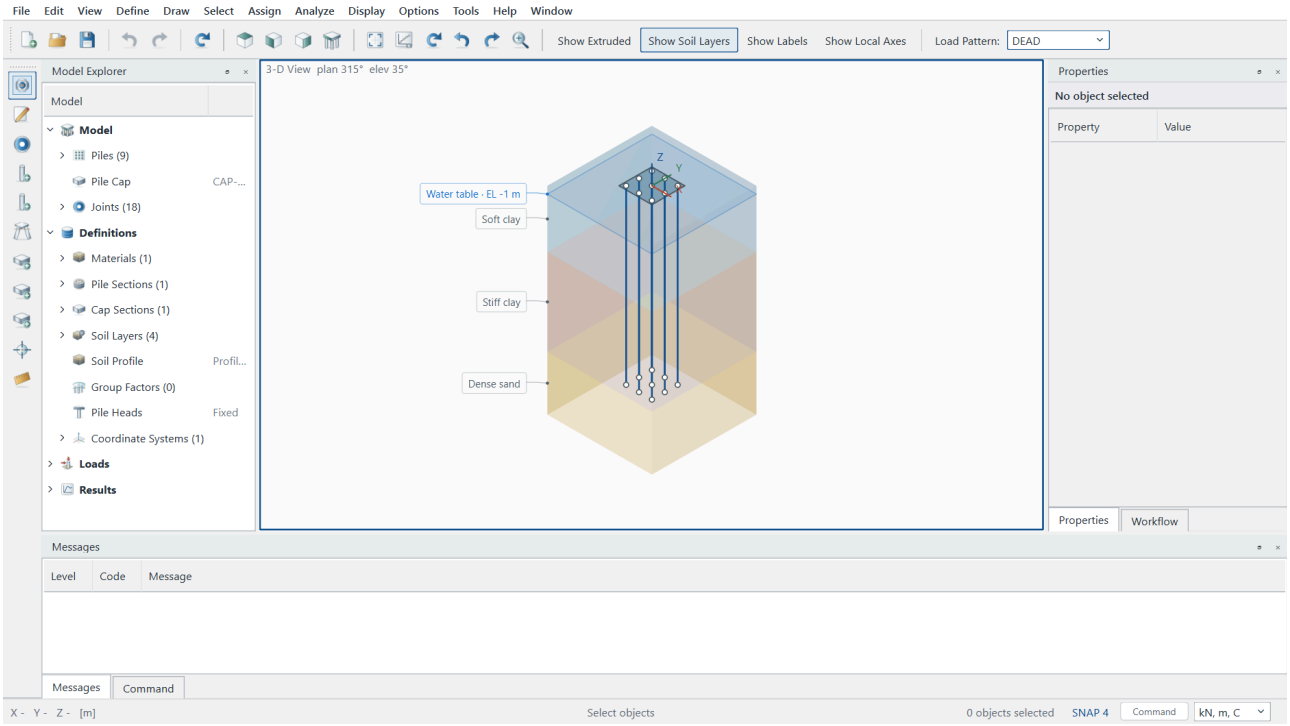


Figure 3.1 — Main window with a rectangular group

The panels are shown or hidden with the **Window** menu. Their arrangement is kept between sessions.

3.1 The workflow of an analysis

An analysis follows the order of the **Workflow** panel and of the menus:

1. **Lay out the piles and the cap**: from a template (Chapter 4) or by drawing (Chapter 8).
2. **Define the properties**: materials, pile and cap sections, soil layers and soil profile (**Define** menu, Chapter 9).

© 2026 Solplan. All rights reserved.

3. **Assign** the properties to the piles and the cap: sections, head connection, batter, length, cap behaviour (**Assign** menu).
4. **Define the loading**: load patterns, load cases and load combinations, and assign the loads (Chapter 11).
5. **Set the analysis options, check the model and run** (**Analyze** menu, Chapter 12).
6. **Review and report the results** (**Display** menu, Chapter 13).

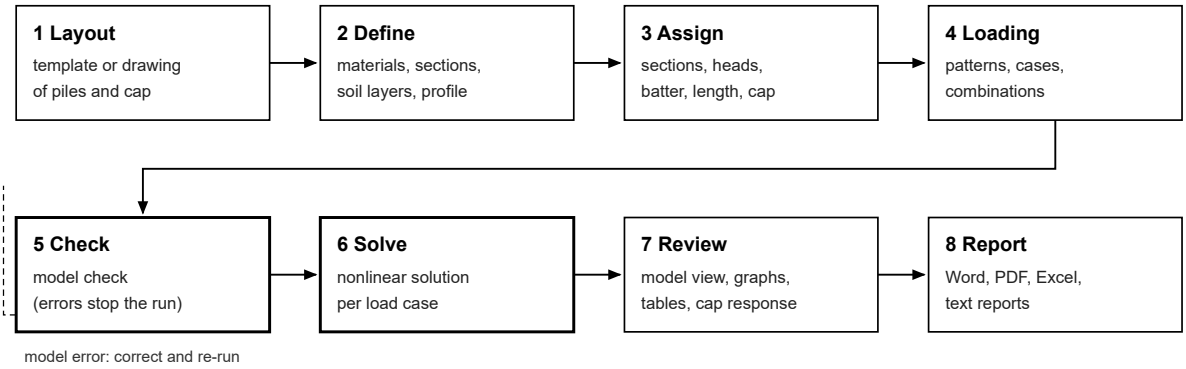


Figure 3.2 — Workflow of an analysis

3.2 The menus

The menu bar holds every command of the program; the most frequent ones are repeated on the tool bars. Table 3.1 gives the content of each menu, and the figures show the menus as they open, with a model that has results, so that the commands for results are available. A command followed by … opens a dialog; a triangle opens a submenu. The commands are described in the chapters that follow.

Table 3.1 — Content of the menus

Menu	Content
File	new, open and save models; import and export of model tables and of a single-pile model; reports; printing
Edit	undo and redo; cut, copy, paste and delete; move, replicate, divide and join piles; labels; duplicate piles; shifting the piles, the soil elevations and the loading together
View	plan, elevations and 3D view; pane layout; zoom and rotation; what the panes show (grid, axes, extruded piles, soil layers, labels, local axes) and its transparency
Define	materials and sections, soil layers, profiles and curves, coordinate systems and grids, groups, load patterns, load cases and load combinations
Draw	drawing of piles, caps, special joints, reference points and lines; snap; labels; the grid; default draw properties
Select	selection by window, line or polygon, by property, by object type, by coordinates and by group
Assign	properties of piles and cap; joint, cap and pile loads; free-field soil movement; copying assigns between piles
Analyze	analysis options, cases to run, Check Model, Run Analysis and the analysis of several model files
Display	loads, assigns, tables, soil and moment-curvature curves, results in the canvas, result graphs, text reports and the Report Builder
Options	preferences, rebuilding the layout from a template, colours, display options and graph settings
Tools	model statistics and the folder of the model
Help	licence and version
Window	showing and hiding the panels and the tool bars

© 2026 Solplan. All rights reserved.

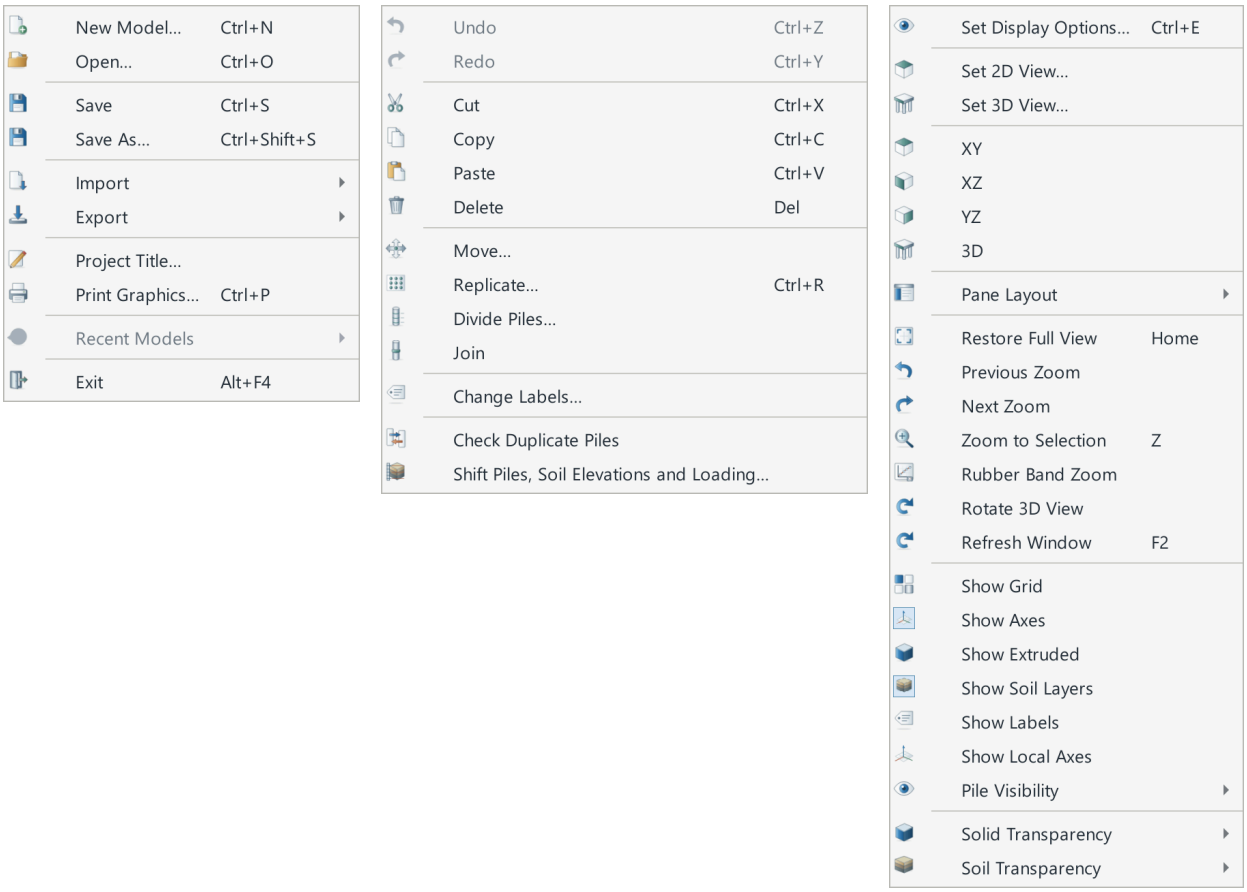


Figure 3.3 — The File, Edit and View menus

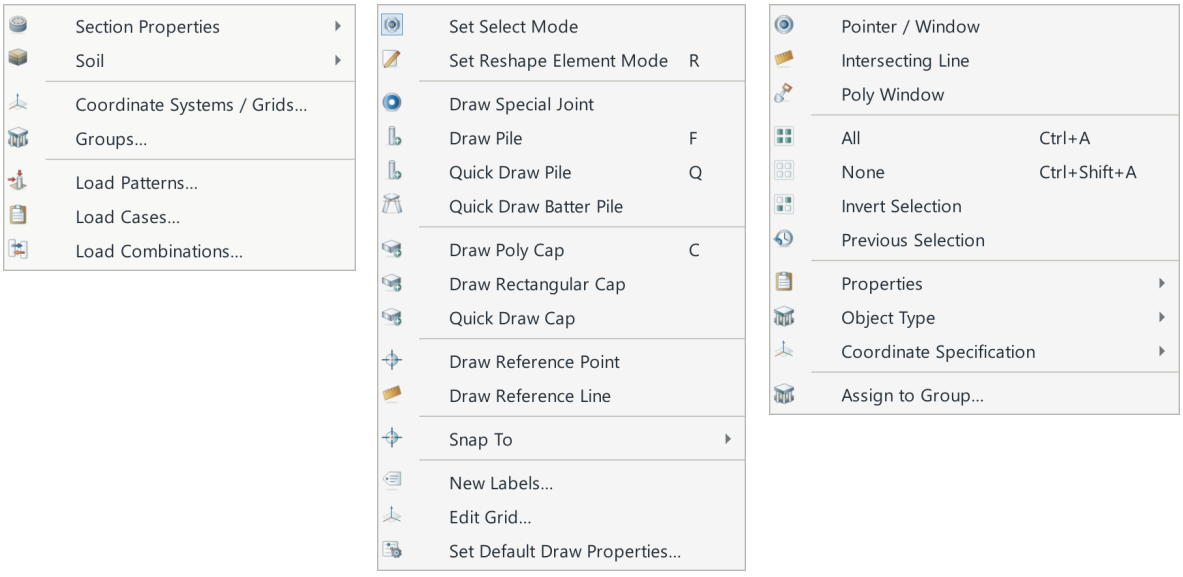


Figure 3.4 — The Define, Draw and Select menus

© 2026 Solplan. All rights reserved.

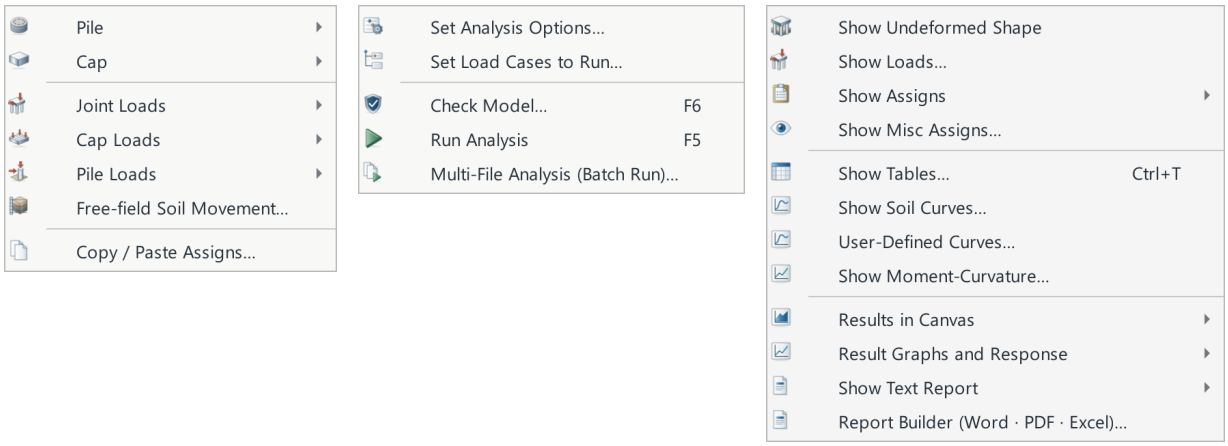


Figure 3.5 — The Assign, Analyze and Display menus

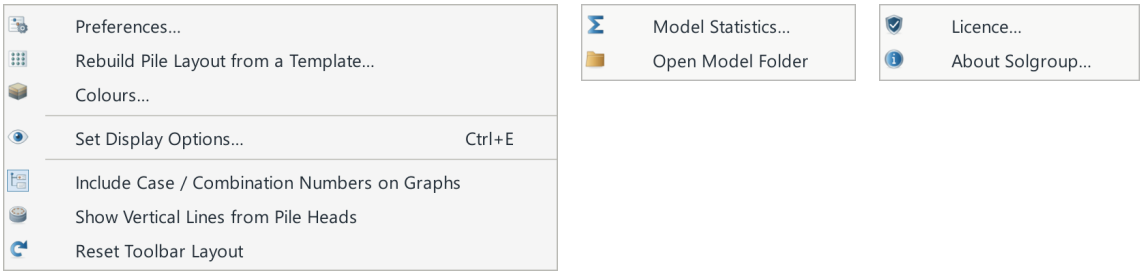


Figure 3.6 — The Options, Tools and Help menus

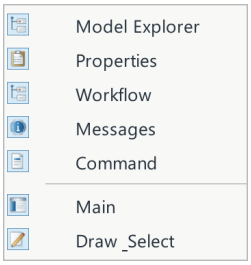


Figure 3.7 — The Window menus

3.3 The File menu

The **File** menu holds the commands for model files, data exchange and output:

Table 3.2 — Commands of the File menu

Command	Use
New Model… (Ctrl+N)	starts a model from a template (Chapter 4)
Open… (Ctrl+O), Save (Ctrl+S), Save As… (Ctrl+Shift+S)	open and save model files (Section 2.5). Unsaved changes are offered for saving before another model is opened or the program is closed.
Import ▸ Model Tables (CSV / Excel)…	reads model tables edited in a spreadsheet back into the model; the values of matching rows are updated, and a summary lists the values updated, the rows matched, the rows not found and any problem (Section 9.12)
Import ▸ Solpile Model (.lpx)…	one Solpile pile becomes the section catalogue, the soil profile and one pile at the origin
Export ▸ Report Builder (Word · PDF · Excel)…	choose the report sections, preview them and export a Word, PDF or Excel report (Chapter 14)
Export ▸ Model Tables (CSV)…; Model Tables (Excel)…	every table of Show Tables, as CSV files or in one Excel workbook with one sheet per table, to be edited and read back

© 2026 Solplan. All rights reserved.

Command	Use
Export ▸ Text Reports (.txt)…	the input echo, the output results and the summary results reports as three text files
Export ▸ View Bitmap…	saves the active view as a PNG or BMP image, by default named after the model in its folder
Project Title…	the title of the project, shown in the window title and the reports
Print Graphics… (Ctrl+P)	prints the active view
Exit (Alt+F4)	closes the program

The export and print commands need a licence that allows output; with a trial licence they report that output is not available (Section 2.3).

© 2026 Solplan. All rights reserved.

4 Starting a model from a template

File ▸ New Model… opens the template window (Figure 4.1). Choose a layout, enter its dimensions and confirm with **OK**. The preview on the right is updated as the values change. The templates are:

Table 4.1 — Templates for a new model

Template	Layout and parameters (defaults)
Blank	a grid only, without objects
Grid Only	a grid with the number of bays and the bay spacing in X and Y (4 bays of 3 m each way)
Rectangular Pile Group	$n_x \times n_y$ piles at spacings s_x, s_y (3 × 3 at 1.8 m), the most common case
Staggered / Zig-zag Group	rows of piles with alternate rows offset: piles per row, number of rows, spacing, row spacing and offset (3 × 3 at 1.8 m, offset 0.9 m)
Circular (Single Ring)	a ring of piles for a dolphin, a stack or a circular pier: radius, number of piles, start angle (3 m, 8 piles)
Circular (Multi Ring)	several rings: number of rings, radius of the first ring and increment of radius, piles in the first ring and increment of piles per ring (2 rings, 2 m + 2 m, 6 + 6 piles)
Radial Batter Group	a ring of piles battered outward or inward (3 m, 8 piles, outward)
Triangular Pile Group	three piles under a triangular cap with clipped corners, or more rows for larger groups (2 rows, spacing 1.8 m)
Line / Single Row Group	one row of piles: number, spacing and direction of the row (5 piles at 1.8 m)
Trestle / Bent (jetty)	a row of jetty bents whose outer pile pair is battered: number of bents, bent spacing, piles per bent, pile spacing within a bent, batter of the outer pair (4 bents at 6 m, 4 piles at 3 m, 9.46°); its pile heads and deck are 4 m above the seabed (ground at 0) and a new model puts the water 1 m below the deck
Custom from Table	X, Y, batter and length of each pile pasted from the clipboard or read from a CSV file

Every template that creates piles also takes the **Pile length** (24 m), the **Pile head elevation** (0; 4 m for the jetty), the **Batter angle from vertical** (0) and the **Cap edge offset** (0.5 m), the distance from the outer piles to the edge of the cap; **Create cap automatically from pile extent** creates a cap that encloses the piles. For the rectangular group, the layout is given by the **Number of piles in X**, the **Number of piles in Y**, the **Spacing in X** and the **Spacing in Y**. The preview can show the layout in 3D or in Plan (X-Y).

© 2026 Solplan. All rights reserved.

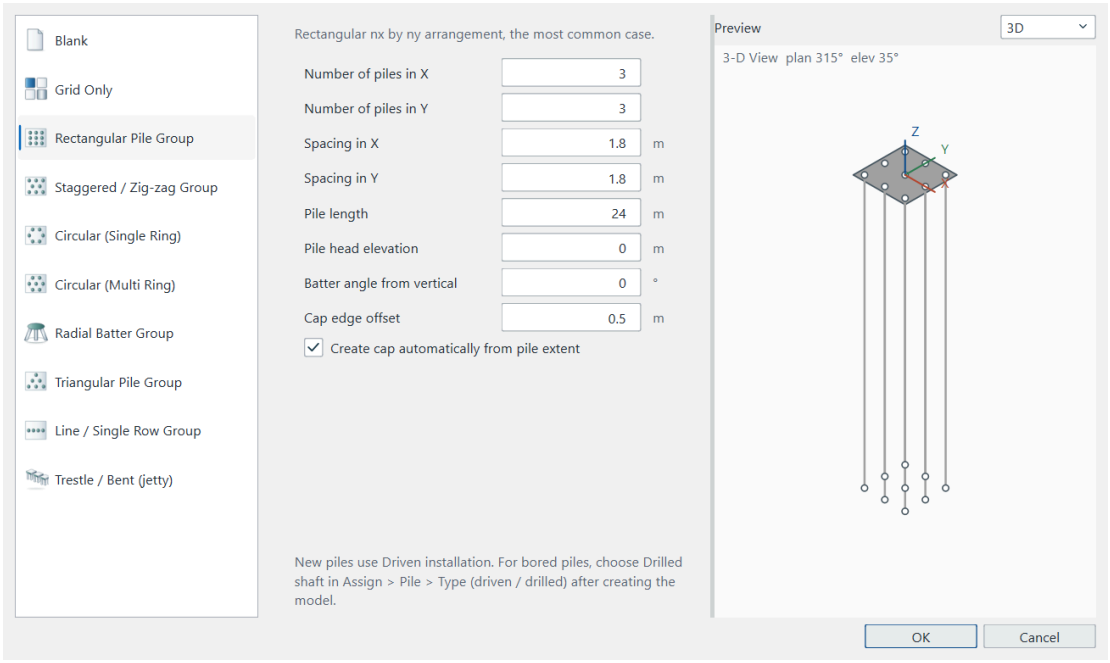


Figure 4.1 — Template window for a new model

A new model contains one concrete material, one soil layer over the whole soil profile, and no pile or cap section; the piles and the cap are drawn grey until a section is assigned. The steps of Chapter 9 and Chapter 10 complete it.

Options ▸ Rebuild Pile Layout from a Template… replaces the piles of an existing model by a new layout and keeps the soil, the sections and the load cases. Its window is the template window, with the pile section of the new piles chosen from the sections of the model.

NOTE A group can contain at most 2 500 piles. The template window rejects a layout above that limit before it is built.

© 2026 Solplan. All rights reserved.

5 Units

Every value is entered and displayed in the units of the model. Six unit systems are available: kN, m; kgf, m; tonf, m; N, mm; kip, in; and kip, ft. The values are stored in kN and m, so that a change of units converts the display without loss. The units are chosen in **Options ▸ Preferences…** (**Display units**) and on the status bar, and they are saved with the model. Results are always shown in the units of the model.

The preferences also set the joint merge tolerance, the sign convention of moments in graphs, the autosave interval, the number of undo steps kept, and the snap tolerance and fine grid of the drawing tools (Section 15.1).

NOTE Switching between an SI-based unit system and a US customary one also switches the unit system in which the dimensional constants of the published soil relations are evaluated. The results can therefore differ slightly between the two, and they are cleared when the switch is made.

The same window sets the joint merge tolerance, the sign convention of moments in the reports, the autosave interval, the number of undo steps kept, and the snap tolerance and fine grid of the drawing tools.

© 2026 Solplan. All rights reserved.

6 Getting started

This chapter builds and analyses a small pile group from start to finish, so that the main steps of every analysis are seen once in order. The model is a group of six bored piles of 0.8 m diameter, two in X and three in Y at 2.4 m spacing, 20 m long, under a rigid cap, in a layered soil of soft clay, stiff clay and dense sand. It carries a dead load and a live load at the cap, and the factored combination 1.2 D + 1.6 L is analysed. Each step names the command, the values to type and what the window shows afterwards. The dialogs themselves are described in detail in the following chapters; the references point to them. All numbers below were computed by Solgroup 1.0.10 with this model.

6.1 Step 1: lay out the piles

1. Choose **File** ▸ **New Model…** (Ctrl+N) and select *Rectangular Pile Group*.
2. Type Number of piles in X = 2, Number of piles in Y = 3, Spacing in X = 2.4 m, Spacing in Y = 2.4 m, Pile length = 20 m and Cap edge offset = 0.6 m. Leave the pile head elevation at 0, the batter at 0 and *Create cap automatically from pile extent* ticked, and click **OK**.

The window shows the six piles and the cap in a 3D view, inside the soil of the new model: one layer of soft clay, drawn transparently, which Step 3 divides into three (Figure 6.1). The piles and the cap are grey because a new model has no section yet. The **Model Explorer** on the left lists the objects and the definitions; the **Properties** panel on the right shows the data of the selected object.

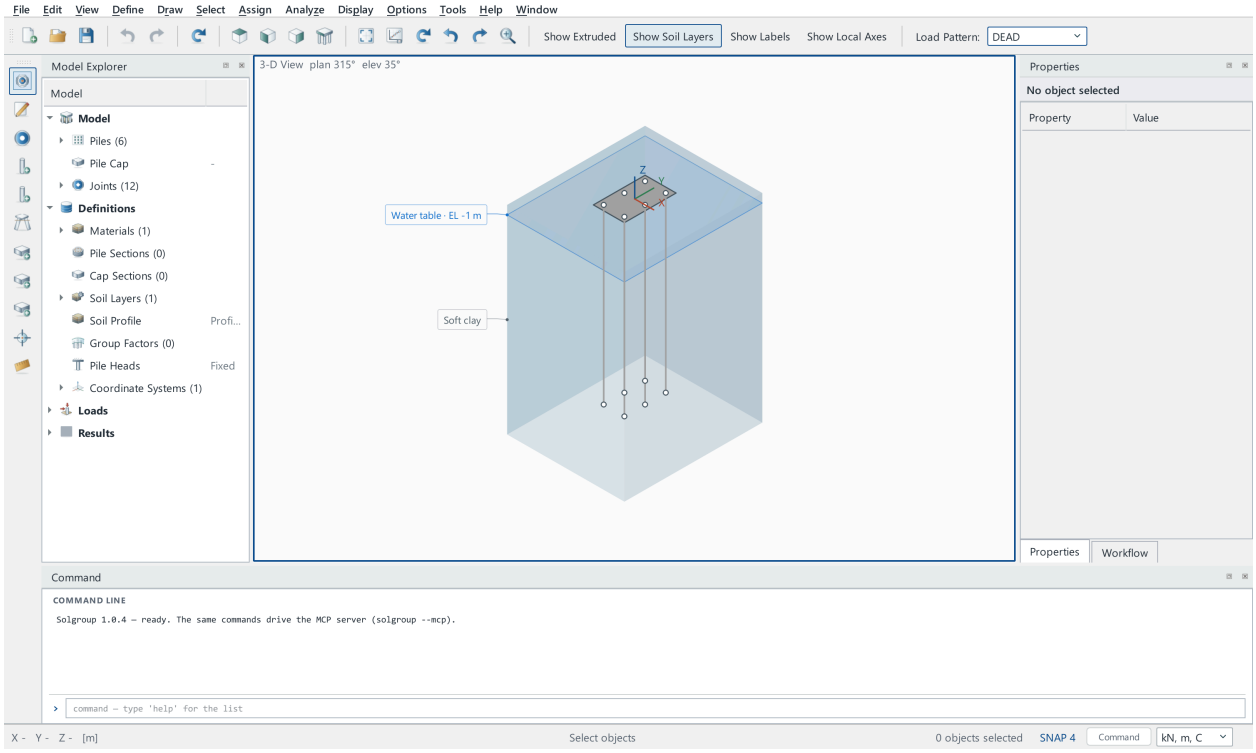


Figure 6.1 — The new model in the 3D view

Click **XY** on the tool bar (or **View** ▸ **XY**) to see the plan (Figure 6.2). The piles are labelled P1 to P6 from the corner with the smallest X and Y; the cap is centred on the global origin. The reference point of the cap loads is the centroid of the cap at its elevation, here the origin.

© 2026 Solplan. All rights reserved.

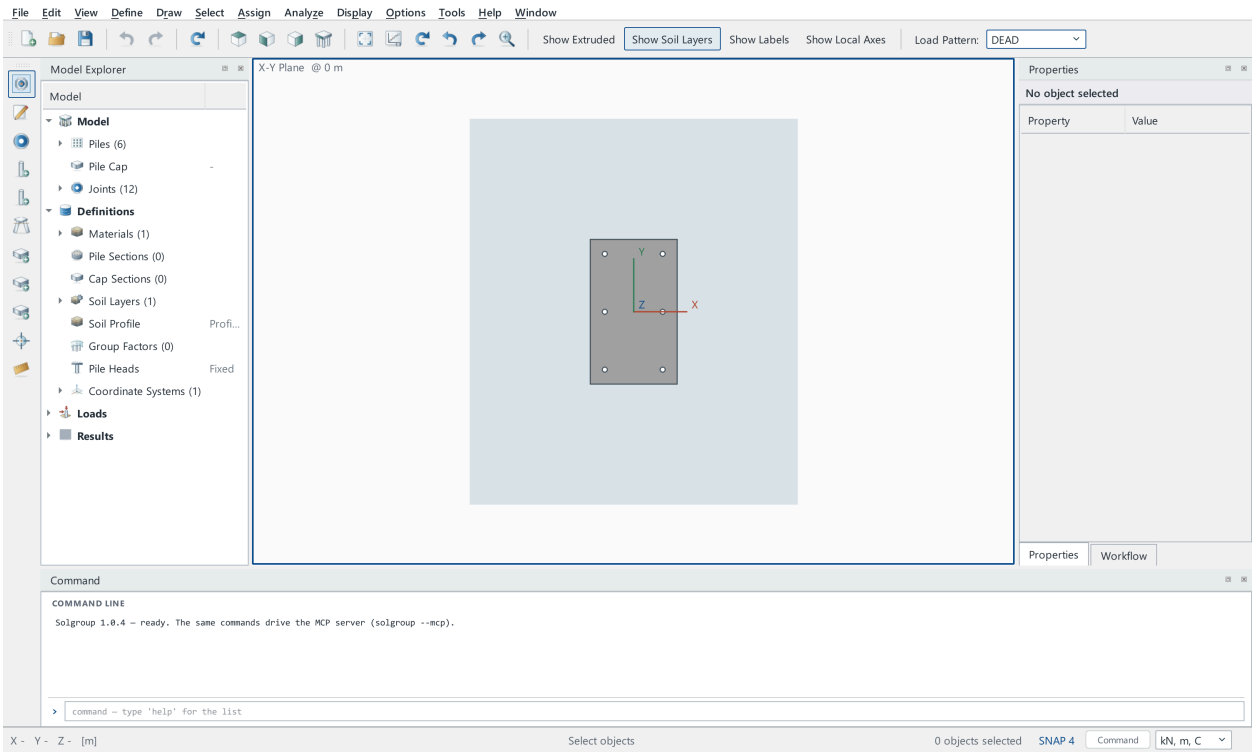


Figure 6.2 — Plan view with the pile labels

6.2 Step 2: define the pile and cap sections

A new model has one concrete material and no section: the piles and the cap stay grey, and **Check Model** reports them as errors, until a section is assigned to them. The bored piles are modelled with a nonlinear reinforced-concrete section, which needs a reinforcing steel:

1. **Define** ▸ **Section Properties** ▸ **Materials…**, **Add New…**: name *Rebar 420*, type Rebar, Yield strength Fy = 420 MPa (420 000 kN/m^2), Ultimate strength Fu = 525 MPa; **OK** (Section 9.2).
2. **Define** ▸ **Section Properties** ▸ **Pile Sections…**, **Add New…**: family Concrete, shape Round, **OK**. Name *Bored pile D800*; Dimensions: Outside diameter, D = 0.8 m; Material: the concrete and *Rebar 420*; Rebars: pattern Circle, 16 bars, cover 0.075 m; Nonlinear: *Enter values manually* with the thrusts 0, 1 500 and 3 000 kN, which span the axial forces expected in the piles (Section 9.3). Check the curves with **Show moment-curvature…** and click **OK** twice.
3. Select every pile (**Select** ▸ **All**, Ctrl+A). Choose **Assign** ▸ **Pile** ▸ **Section…**, select *Bored pile D800* and click **Apply**. Then **Assign** ▸ **Pile** ▸ **Type (driven / drilled)…**, select *Drilled shaft* and click **Apply**: the installation type selects the axial load-transfer curves of bored piles.
4. **Define** ▸ **Section Properties** ▸ **Cap Sections…**, **Add New…**: name *CAP-1000*, thickness 1.0 m, material the concrete; **OK** twice. Select the cap (**Select** ▸ **All** selects it with the piles, or click *Pile Cap* in the Model Explorer), then **Assign** ▸ **Cap** ▸ **Cap Section…**, select *CAP-1000* and click **OK**. For a rigid cap the thickness only places the cap and draws it; it matters for a flexible cap (Section 16.3).

Display ▸ **Show Assigns** ▸ **Pile Section** colours the piles by section, with a legend, so that the assignment can be checked at a glance (Figure 6.3); the piles are drawn as extruded solids with **Show Extruded**.

© 2026 Solplan. All rights reserved.

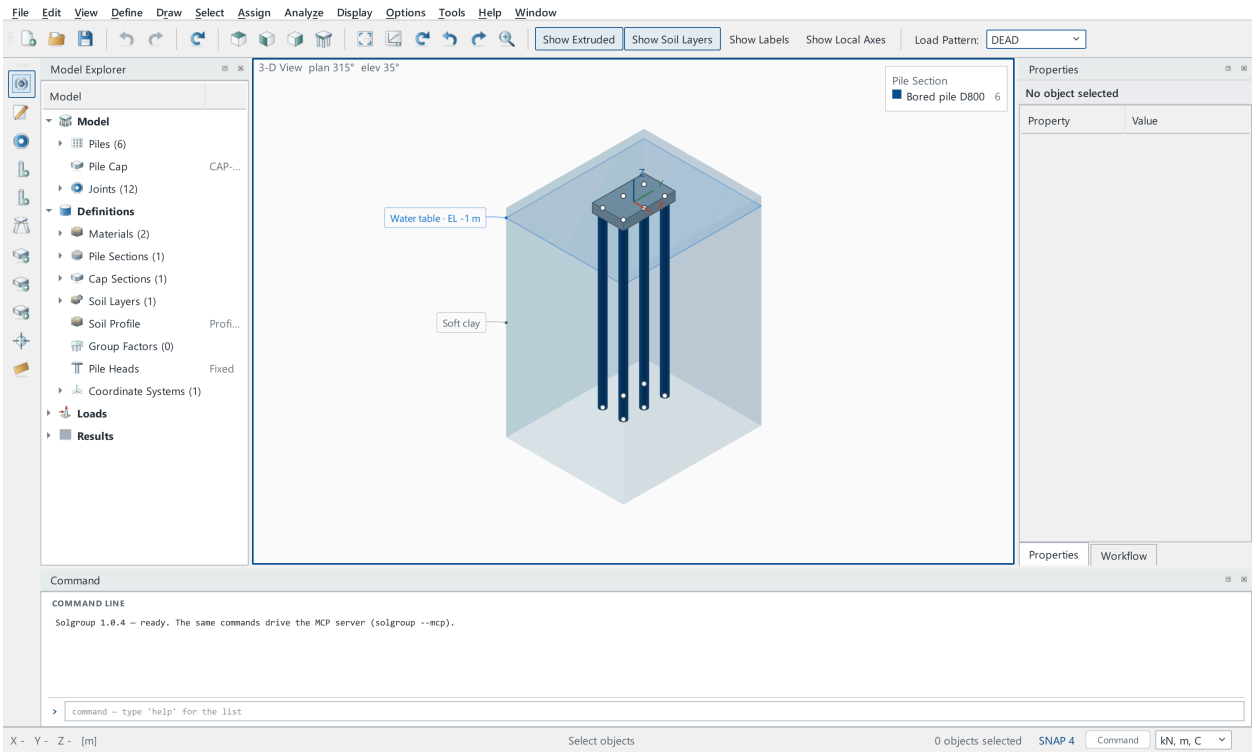


Figure 6.3 — Piles coloured by their section

6.3 Step 3: define the soil

A new model has one soil layer, *Soft clay* (Soft Clay (Matlock), unit weight 15 kN/m³ above and 16 kN/m³ below the water table, c_u = 25 kPa, ε_{50} = 0.02), over the whole profile from the ground surface at elevation 0 to −40 m, with the water table at −1 m. Two layers are added below it:

1. **Define** ▸ **Soil** ▸ **Soil Layers…**, **Add New…**: name *Stiff clay*, model Stiff Clay without Free Water, unit weight 17.5 / 19 kN/m³, c_u = 120 kPa, ε_{50} = 0.005; **OK**. **Add New…** again: name *Dense sand*, model API Sand (O'Neill), unit weight 18 / 20.5 kN/m³, φ = 38°, k = 61 000 kN/m³; **OK** twice. The preview of each layer draws its p-y curve (Section 9.5).
2. **Define** ▸ **Soil** ▸ **Soil Profile…**: three rows, soft clay from 0 to −8 m, stiff clay from −8 to −20 m and dense sand from −20 to −40 m; the water table stays at −1 m (Section 9.6). **OK**.

In the 3D view the layers are drawn around the piles with their names, and the water table is marked (Figure 6.4).

© 2026 Solplan. All rights reserved.

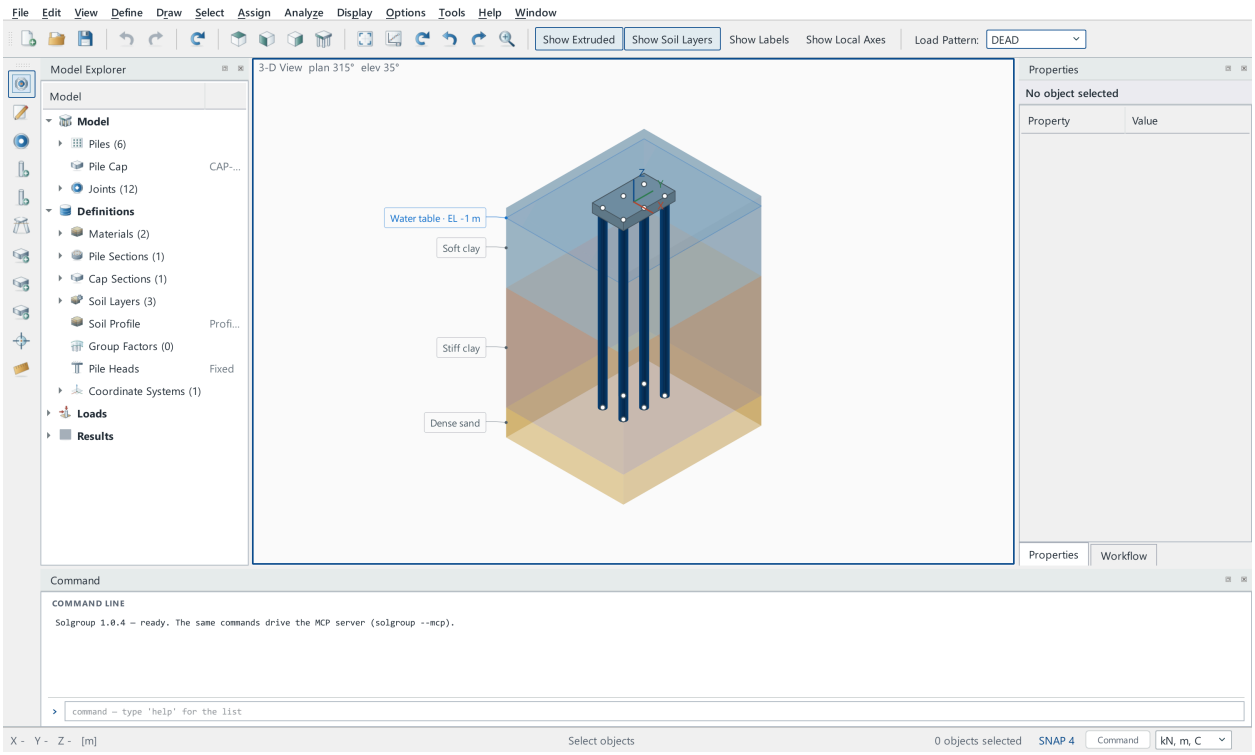


Figure 6.4 — The soil layers around the group

6.4 Step 4: apply the loads

A new model has the load patterns DEAD and LIVE. DEAD carries the self weight of the cap (self weight multiplier 1): the unit weight of its concrete times its volume, computed by Solgroup. Nothing else is typed into DEAD. The column, which stands 0.5 m off the middle of the cap in X, brings the weight of the floors, finishes and services above it: a superimposed dead load, in a pattern of its own.

1. **Define ▸ Load Patterns…**, **Add New…**: name *SDL*, type SUPERDEAD, self weight multiplier 0 (Section 11.1). SUPERDEAD takes the dead-load factors in the ACI 318 and ASCE 7 combinations and the DW factors under AASHTO LRFD.
2. The column is marked by a reference point: **View ▸ XY**, **Draw ▸ Draw Reference Point**, a click near the column, then its exact position in the Properties panel: X = 0.5 m, Y = 0.
3. Pick the reference point in the Model Explorer and choose **Assign ▸ Cap Loads ▸ Point Forces / Moments…** with *Apply at* = **Selected points / pile-head joints**. Pattern SDL: FZ = −4,500 kN (downward is negative); **Apply**. Apply clears the selection, so the point is picked again for the pattern LIVE: FX = 300 kN and FZ = −1,200 kN; **Apply** (Section 11.2).

No moment is typed. Acting 0.5 m off the centre, the 4,500 kN also turn the cap with 2,250 kN·m about Y: Solgroup takes it from the position of the load. **Display ▸ Show Loads…** with *All loads / all patterns* and *Show load values* draws the loads with their values at the reference point (Figure 6.5).

© 2026 Solplan. All rights reserved.

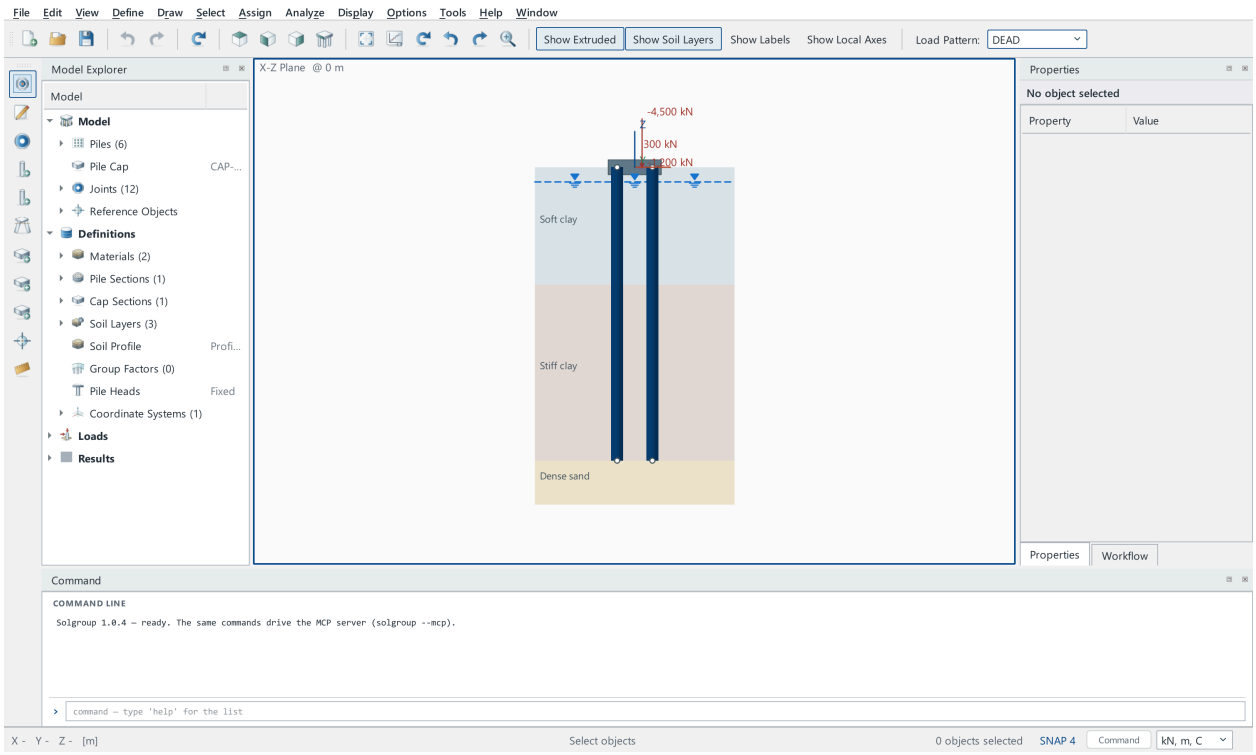


Figure 6.5 — The column loads of the patterns SDL and LIVE at the reference point, 0.5 m off the centre of the cap

6.5 Step 5: define the load cases and the combinations

1. **Define ▸ Load Cases…**, **Add One per Pattern**: one nonlinear static case per pattern, DEAD, SDL and LIVE, each with its pattern at a factor of 1 (Section 11.3). Click **OK**.
2. **Define ▸ Load Combinations…**, **Add Default Design Combos**: the combinations of ACI 318 for the pattern types of the model, *COMB1* = 1.4 DEAD + 1.4 SDL and *COMB2* = 1.2 DEAD + 1.2 SDL + 1.6 LIVE (Section 11.4). **Design Combo Options** switches to ASCE 7 or AASHTO LRFD. A Linear Add combination is analysed as one load set with the factored loads: because the soil is nonlinear, its result is not the sum of the results of its cases.
3. **Analyze ▸ Set Load Cases to Run…**: untick *Run all load cases*, which runs the load cases but not the combinations, keep the three cases ticked and tick both combinations (Section 12.2).

6.6 Step 6: check and run

Analyze ▸ Check Model (F6) lists its findings in the **Command** panel; here there is nothing to fix. **Analyze ▸ Run Analysis** (F5) runs the three cases and the two combinations. At the end a message reports the outcome and the **Command** panel gives a summary of each case (Figure 6.6): all five converged.

© 2026 Solplan. All rights reserved.

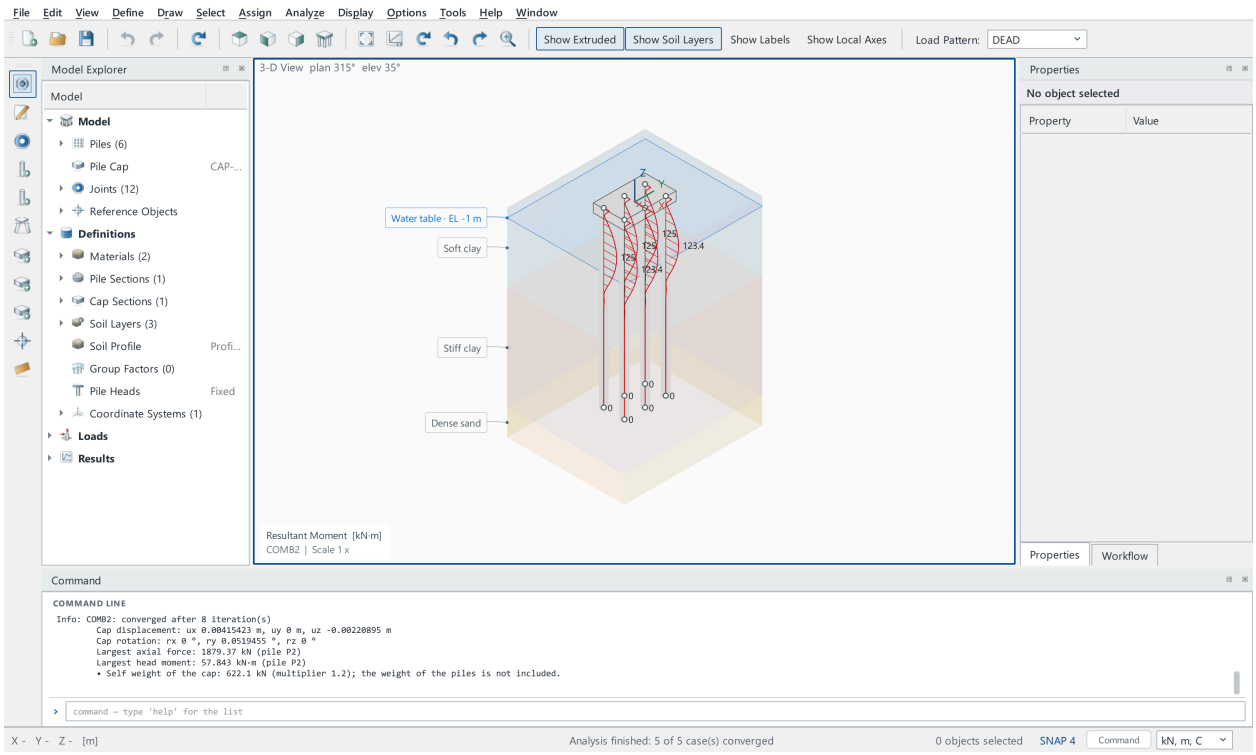


Figure 6.6 — After the run: a summary of each case in the Command panel

6.7 Step 7: review the results

Display ▸ Results in Canvas ▸ Show Deformed Shape… with the combination COMB2 draws the deflected group, magnified for visibility; in the X-Z elevation the cap moves in the direction of the horizontal load and rotates, and the piles bend near their heads (Figure 6.7).

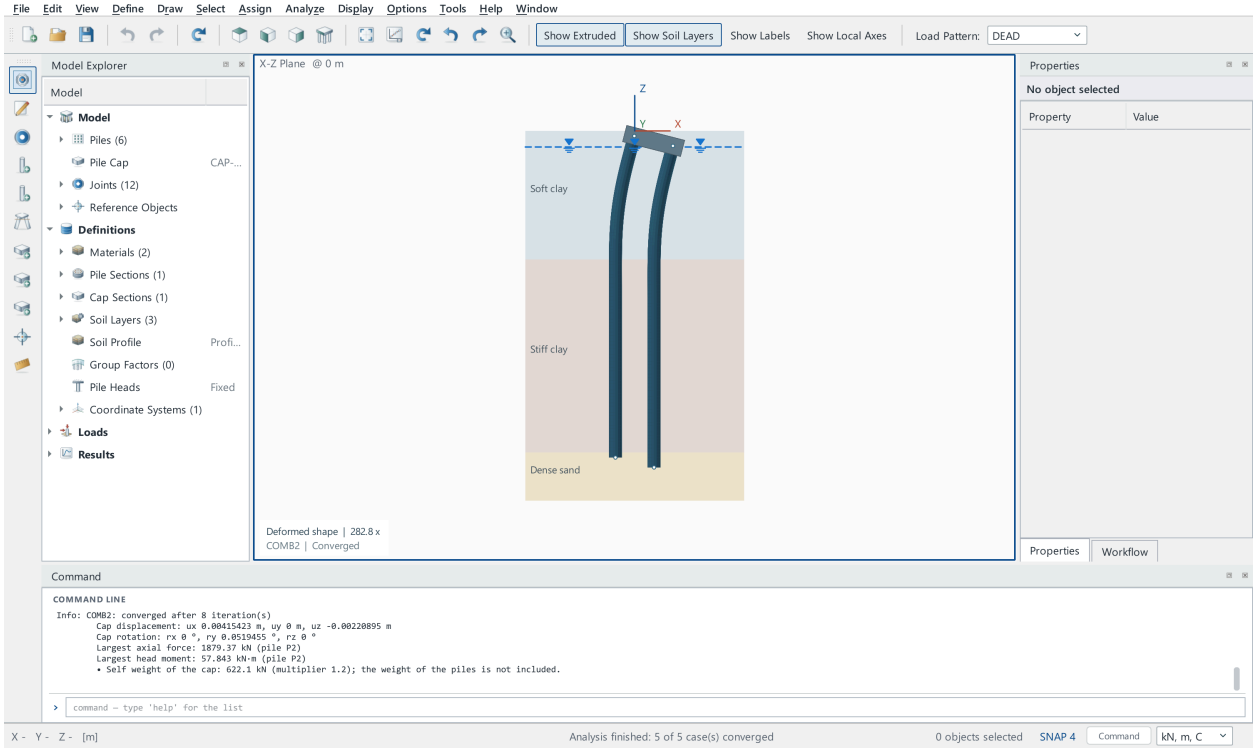


Figure 6.7 — Deformed shape of COMB2 in the X-Z plane

© 2026 Solplan. All rights reserved.

Under the DEAD case alone, the weight of the cap at its centre, every pile takes the same 86.4 kN and no moment. For COMB2 the cap moves 4.15 mm horizontally and −2.21 mm vertically and rotates 0.907×10^{-3} rad, mostly because the column is off centre (**Display ▸ Result Graphs and Response ▸ Show Cap Response…**). The axial forces of the piles range from 768 kN on the far side to 1,879 kN on the side of the column.

Display ▸ Results in Canvas ▸ Force Diagrams in Canvas… draws the bending moment along every pile (Figure 6.8). At the head of pile P1 the turn of the cap and the push of the live load act against each other, so the head moment is small, 55.8 kN·m; the moment changes sign at about 0.8 m below the head and reaches its largest value, 125 kN·m, at 4.2 m, in the soft clay, before it dies out further down. **Show Contour Plot…** colours the piles by the same quantity, with a colour scale (Figure 6.9).

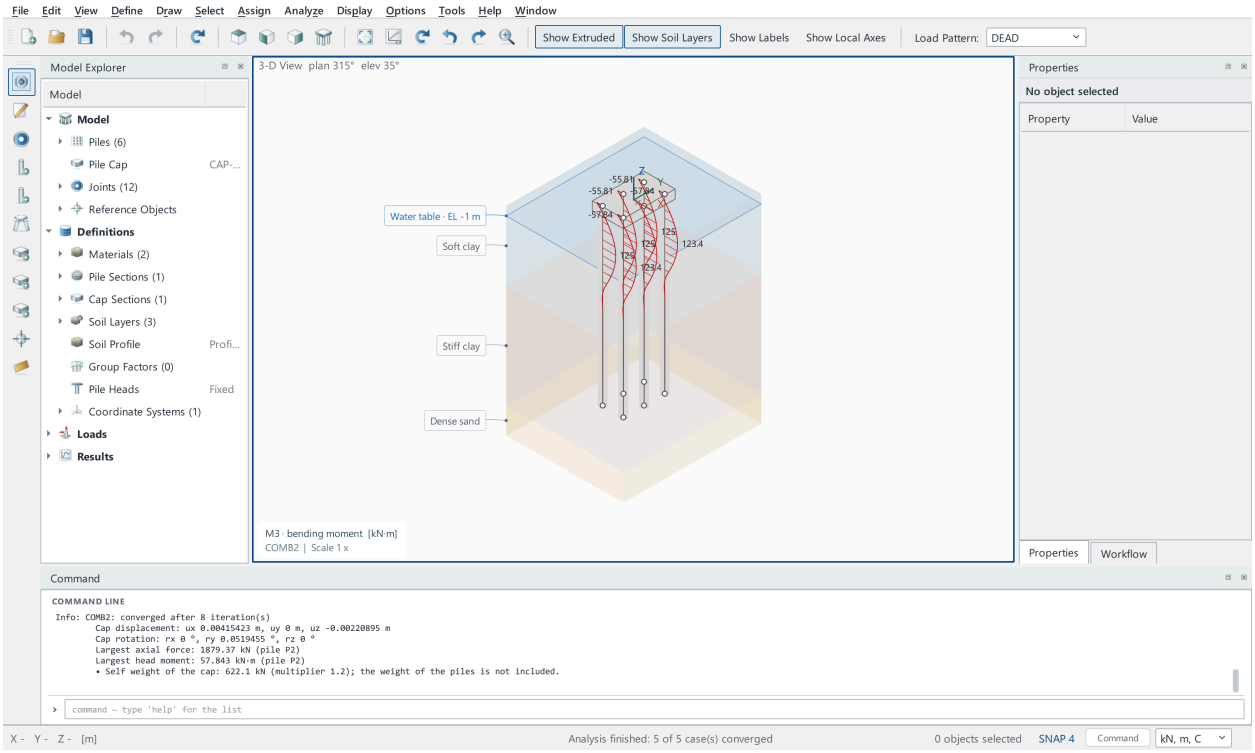


Figure 6.8 — Bending moment along the piles for COMB2

© 2026 Solplan. All rights reserved.

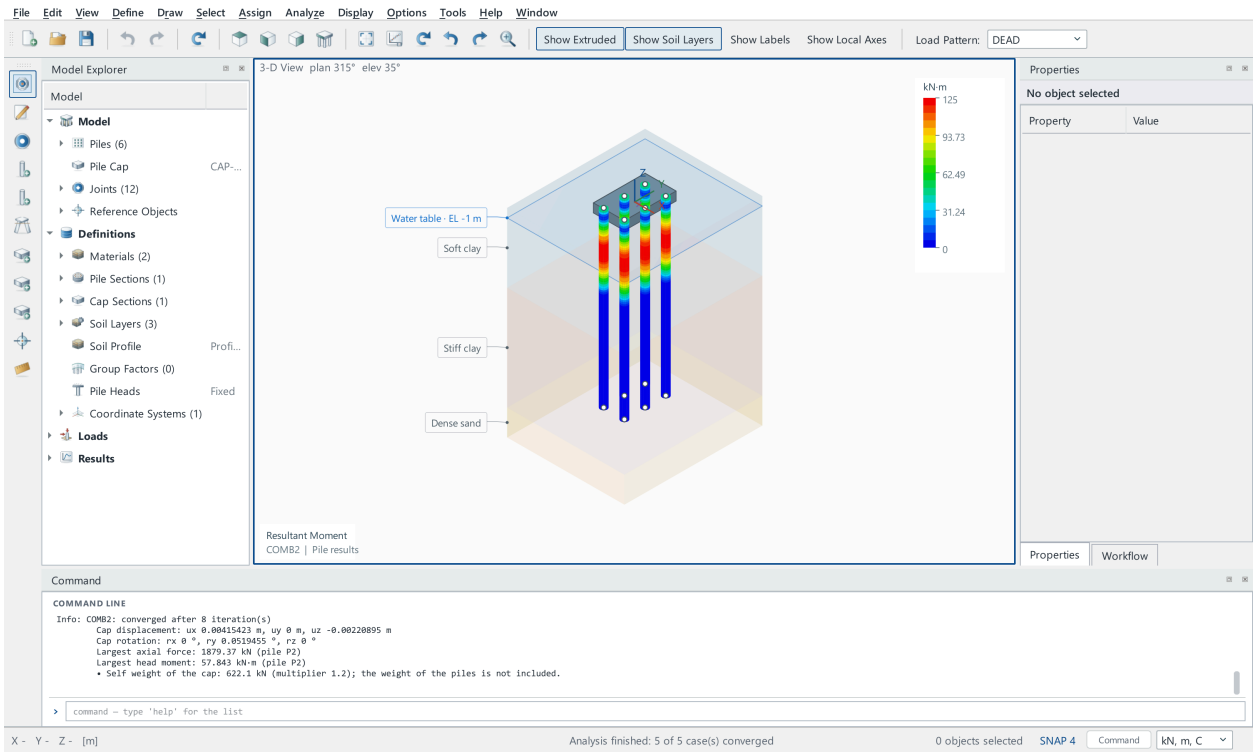


Figure 6.9 — Resultant moment as a contour plot

6.8 Step 8: check equilibrium and report

Before results are used, check that the pile-head forces balance the loads. For COMB2 the vertical load is 1.2 × (4,500 + 518.4) + 1.6 × 1,200 = 7,942.1 kN, the column loads plus the weight of the cap, and the sum of the axial forces of the piles is 7,942.1 kN; the horizontal load is 1.6 × 300 = 480 kN and the sum of the pile-head shears is 480.0 kN. The values are listed in the summary results (**Display ▸ Show Text Report ▸ Summary Results…**) and in the tables of **Display ▸ Show Tables…** (Section 13.5).

Finally, **File ▸ Export ▸ Report Builder (Word · PDF · Excel)…** assembles a report of the model and its results, and **File ▸ Save** (Ctrl+S) keeps the model in a .ggp file; the results are computed again by a run (Chapter 14).

© 2026 Solplan. All rights reserved.

7 Tutorial: a jetty trestle on battered piles

This second tutorial builds a marine structure: three bents of a jetty trestle on steel pipe piles, the outer pair of each bent battered, standing in water above a soft seabed. It adds to the first tutorial the template of a trestle, a steel section, a soil profile with the water above the ground, a load pattern of its own for the berthing load, and the reading of the results of battered piles. The steps that are the same as in the first tutorial are given briefly; all numbers were computed by Solgroup 1.0.10 with this model.

7.1 Step 1: lay out the bents

1. Choose **File** ▸ **New Model…** and select *Trestle / Bent (jetty)*.
2. Type Number of bents = 3, Bent spacing = 6.0 m, Piles per bent = 4, Pile spacing within a bent = 3.0 m and Batter of outer pile pair = 9.46°, which is a batter of 1:6. Type Pile length = 32 m, Pile head elevation = 4.0 m and Cap edge offset = 0.8 m, and click **OK**.

The bents run in Y and the piles of a bent in X (Figure 7.1). The template numbers the piles bent by bent: P1 to P4 in the first bent, P5 to P8 in the second and P9 to P12 in the third. The outer piles of each bent, P1, P4, P5, P8, P9 and P12, are battered outward: the tips of P1, P5 and P9 point to −X, those of P4, P8 and P12 to +X. The X-Z elevation shows the batter (Figure 7.2); at this stage the soil is still that of a new model.

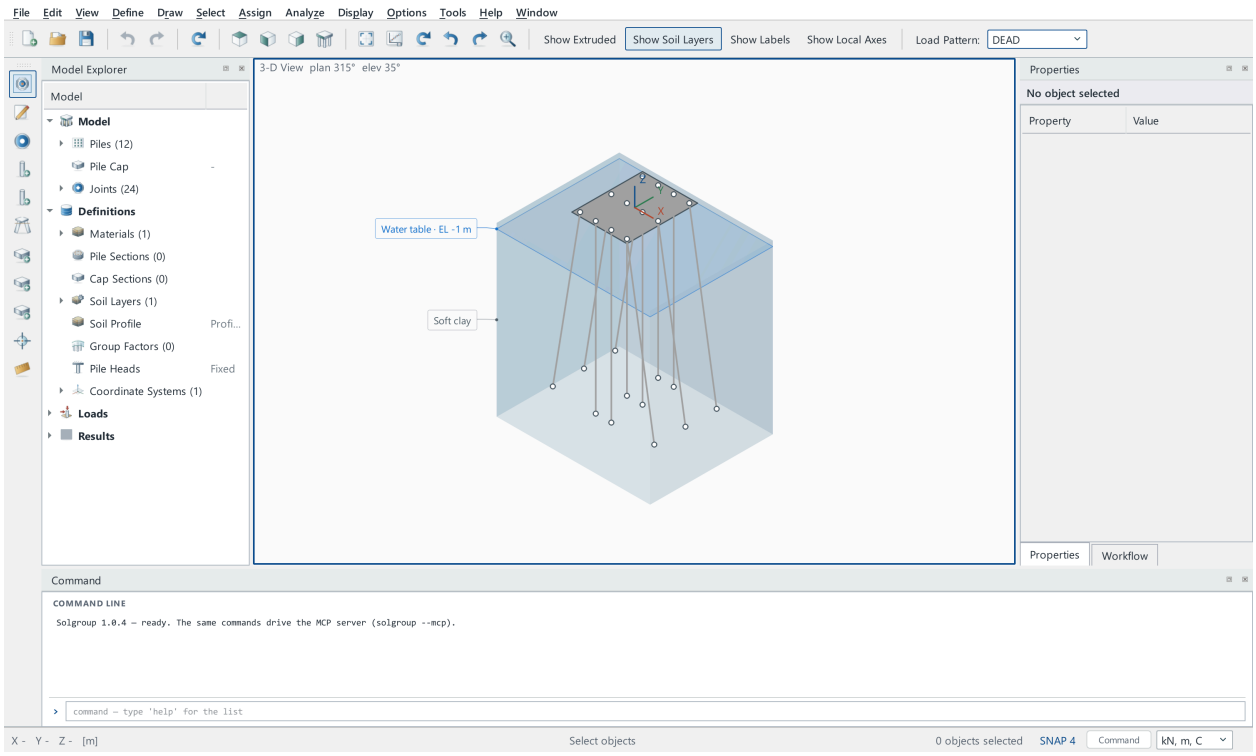


Figure 7.1 — The trestle in the 3D view

© 2026 Solplan. All rights reserved.

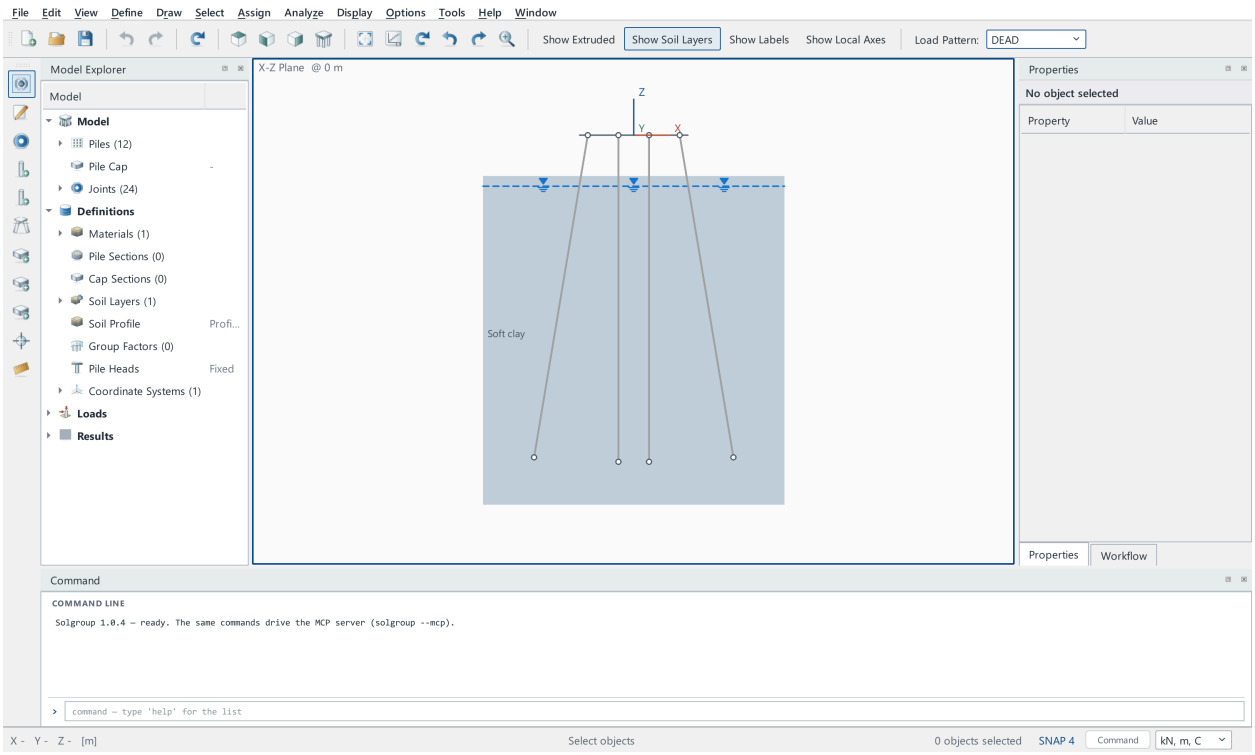


Figure 7.2 — X-Z elevation of a bent with the battered outer pair

7.2 Step 2: define the steel pipe

1. **Define** ▸ **Section Properties** ▸ **Materials…**, **Add New…**: name *Steel 355*, type Steel, modulus of elasticity 200 000 MPa, Poisson's ratio 0.3, Yield strength Fy = 355 MPa, Ultimate strength Fu = 490 MPa.
2. **Define** ▸ **Section Properties** ▸ **Pile Sections…**, **Add New…**: family Steel, shape Pipe. Name *Pipe 812x16*; Outside diameter, D = 0.812 m; Wall thickness, t = 0.016 m; material *Steel 355*; behaviour *Elastic (no yield)*. The area, the moments of inertia and the perimeter are computed from the dimensions.
3. Select all piles and assign the section with **Assign** ▸ **Pile** ▸ **Section…**. The piles are driven, which is the type of a new pile.
4. As in the first tutorial, the cap needs a section: **Define** ▸ **Section Properties** ▸ **Cap Sections…**, **Add New…**: name *CAP-1000*, thickness 1.0 m, the concrete; then select the cap and assign it with **Assign** ▸ **Cap** ▸ **Cap Section…**.

The piles are now drawn in the colour of their section (Figure 7.3).

© 2026 Solplan. All rights reserved.

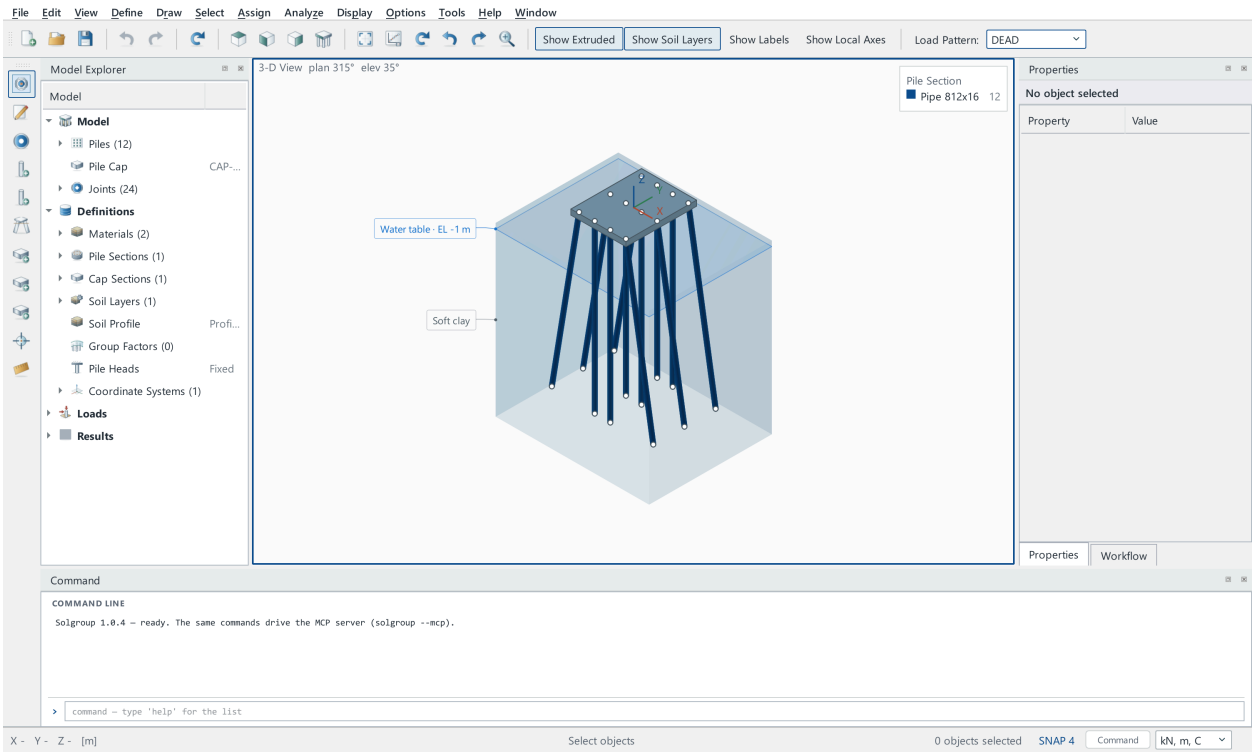


Figure 7.3 — The piles coloured by their section

7.3 Step 3: describe the seabed

The seabed is at elevation 0, with 6 m of soft clay over medium sand, and the water stands 3 m above the seabed. The soft clay is the layer of the new model, with c_u = 25 kPa and ε_{50} = 0.02; the medium sand is added:

1. **Define ▸ Soil ▸ Soil Layers…**, **Add New…**: name *Medium sand*, model Sand (Reese), unit weight 17 / 19.5 kN/m^3, φ = 33° and k = 24,400 kN/m^3 (Section 9.5).
2. In **Define ▸ Soil ▸ Soil Profile…** (Section 9.6) keep the Ground surface at 0 and type the Water table at +3.0 m, above the ground: the whole soil is then below water and its unit weight below water is used.
3. Set the rows to *Soft clay* from 0 to −6 m and *Medium sand* from −6 to −60 m.

The piles stand 4 m free above the seabed, through the water, which gives no lateral resistance (Figure 7.4).

© 2026 Solplan. All rights reserved.

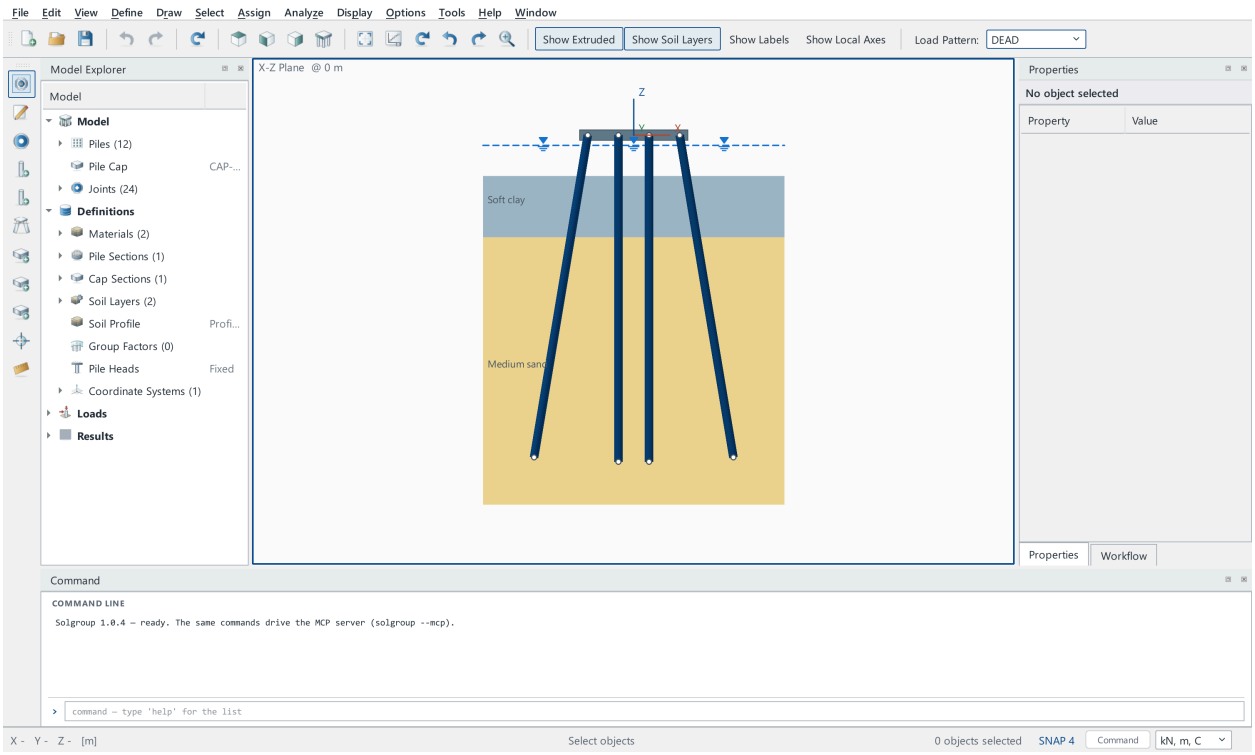


Figure 7.4 — The seabed profile with the water table above the ground

7.4 Step 4: berthing, mooring and current

The dead load needs nothing typed: the deck of this trestle is the cap itself, one metre of concrete, and its weight enters the pattern DEAD through the self weight multiplier of 1. A deck load typed as well would count the same concrete twice. The ship does not push on the middle of the cap: it hits a fender on the berthing face, and the mooring line pulls from a bollard on the same face. The current loads the piles themselves, in the water.

1. **Define ▸ Load Patterns…**, **Add New…**: *BERTH*, type BERTHING; *MOOR*, type MOORING; *CURRENT*, type CURRENT. Select LIVE, which this model does not use, and click **Delete**.
2. Two reference points (**View ▸ XY**, **Draw ▸ Draw Reference Point**, then the exact position, label and elevation in the Properties panel): *Fender* at (−5.3, 3.0, +4) m, on the minus X edge three metres along the jetty, and *Bollard* at (−5.3, −6.0, +4) m, at the first bent. A click can snap to a battered pile below the cap, so check the elevation.
3. Pick *Fender* in the Model Explorer, **Assign ▸ Cap Loads ▸ Point Forces / Moments…** at **Selected points / pile-head joints**: pattern BERTH, FX = 700 kN, across the jetty. Pick *Bollard*: pattern MOOR, FX = −400 kN. The fender lies three metres off the middle line of the cap, so the berthing force also twists the cap about Z with 2,100 kN·m: nothing to type, Solgroup takes it from the position.
4. Select all piles, **Assign ▸ Pile Loads ▸ Distributed Lateral Load…**: pattern CURRENT, Global X, from elevation +4 (the heads) to 0 (the seabed), 3.0 kN/m at the top and 1.5 kN/m at the bottom (Section 11.2).
5. **Define ▸ Load Cases…**, **Add One per Pattern**: DEAD, BERTH, MOOR and CURRENT. **Define ▸ Load Combinations…**, **Add New…**: *D+B+C* = DEAD + BERTH + CURRENT and *D+M+C* = DEAD + MOOR + CURRENT, Linear Add, factors 1.0; tick the cases and both combinations in **Analyze ▸ Set Load Cases to Run…**.

© 2026 Solplan. All rights reserved.

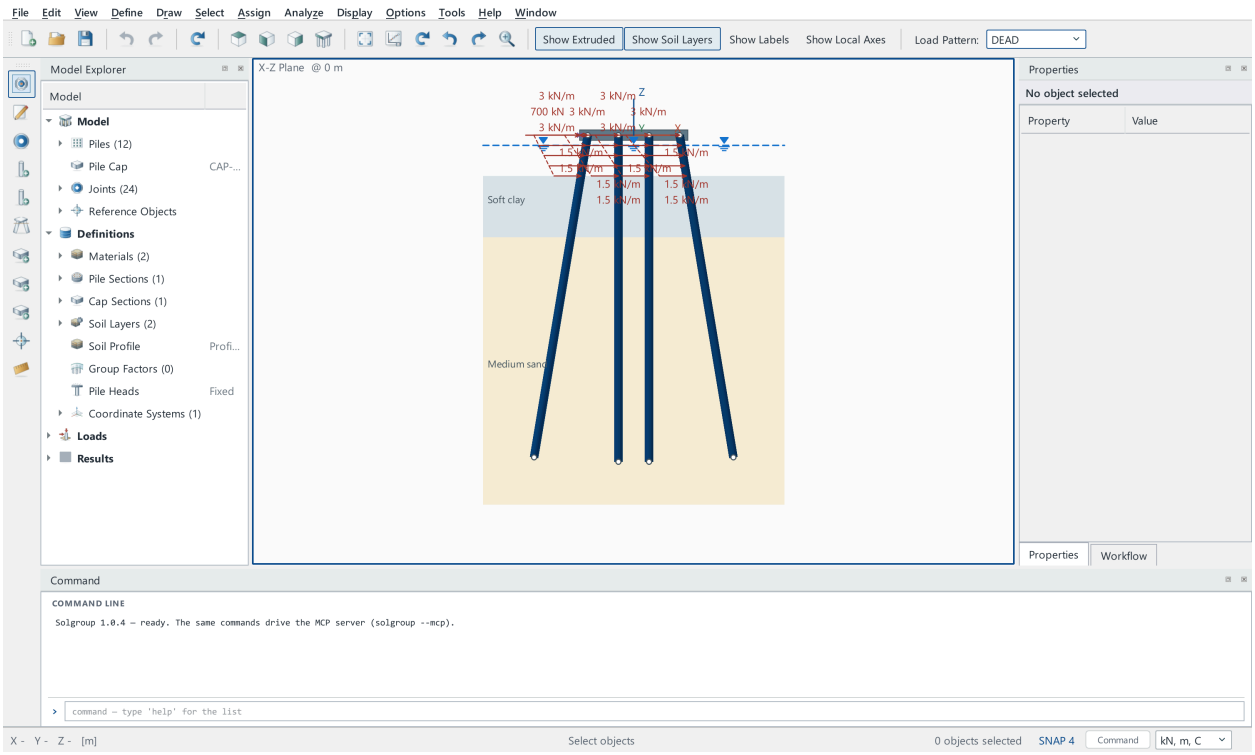


Figure 7.5 — The berthing and mooring forces at the fender and the bollard, and the current on the piles through the water (X-Z elevation)

7.5 Step 5: run and review

Analyze ▸ **Check Model** finds nothing to fix: a water table above the ground is normal for a marine structure. **Analyze** ▸ **Run Analysis** runs the cases and combinations; the Command panel lists them as converged (Figure 7.6).

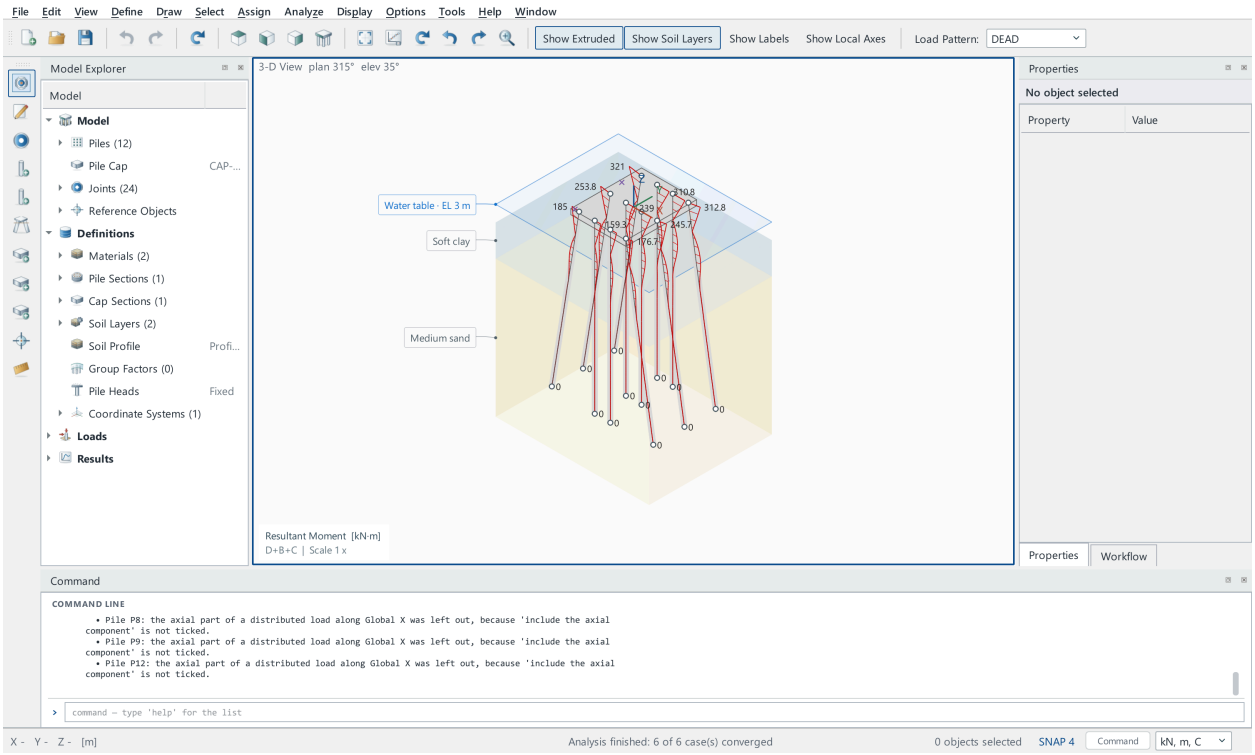


Figure 7.6 — After the run

© 2026 Solplan. All rights reserved.

Under DEAD alone, the weight of the deck, every pile carries about 288 kN. In the X-Z elevation of the deformed shape of D+B+C (Figure 7.7) the cap moves 4.68 mm in X and turns −0.020° about Z; the piles bend over their free length and in the soft clay, and the bending dies out in the sand (Figure 7.8).

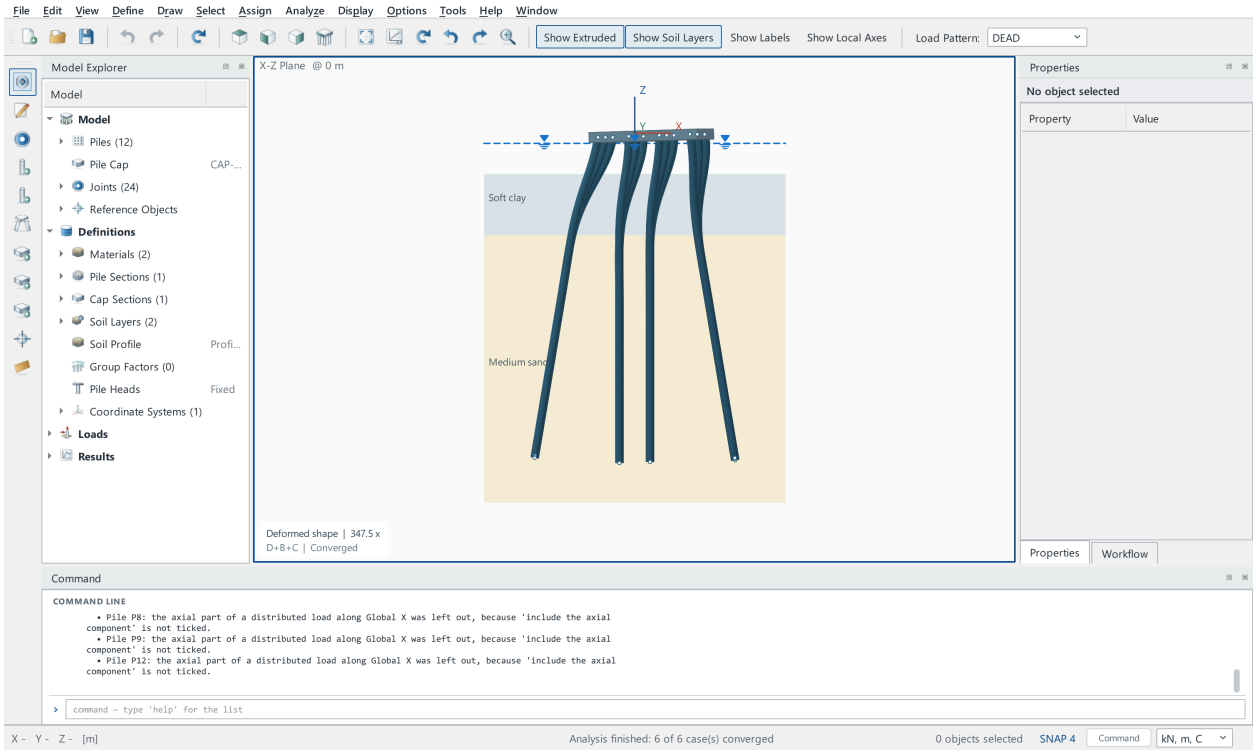


Figure 7.7 — Deformed shape of D+B+C in the X-Z plane

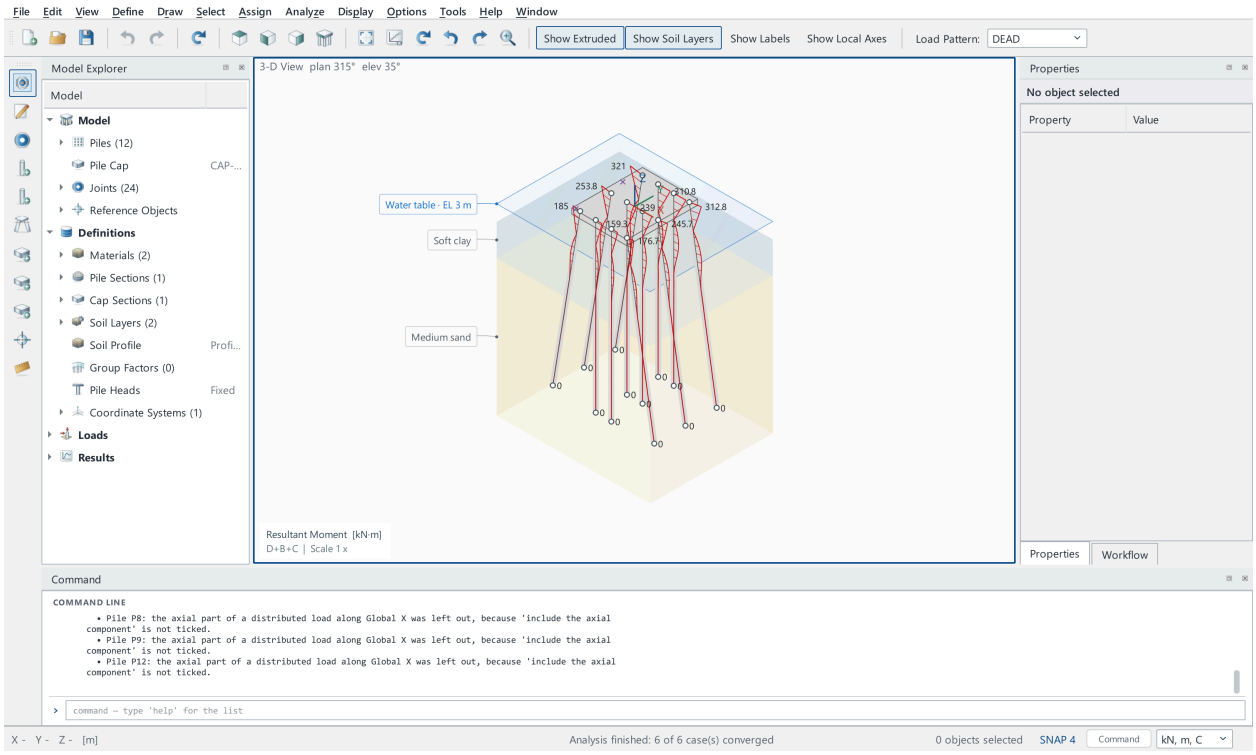


Figure 7.8 — Resultant moment along the piles for D+B+C, with the largest and smallest value of each pile

The pile-head forces are listed by **Display ▸ Show Tables…**, table *Pile Head Results*, one row per case and pile (Figure 7.9). For the first bent, far from the fender, and the third bent, next to it:

© 2026 Solplan. All rights reserved.

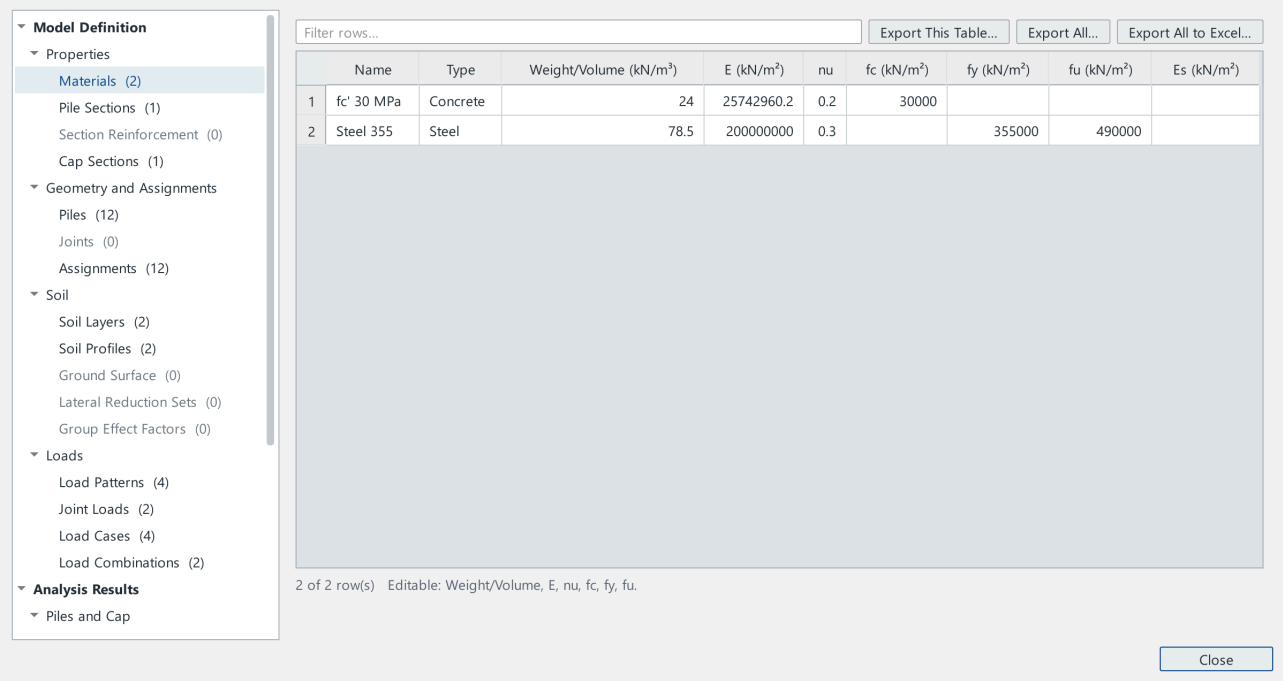


Figure 7.9 — Show Tables, Pile Head Results

Table 7.1 — Pile-head forces of the first and third bents

Pile	Batter	Axial, BERTH (kN)	Axial, D+B+C (kN)	Shear, D+B+C (kN)	Axial, D+M+C (kN)
P1	tip to −X	8.4	287.3	38.7	453.9
P4	tip to +X	−9.5	288.9	37.0	121.4
P9	tip to −X	−243.5	23.1	66.5	227.9
P12	tip to +X	244.3	546.0	64.5	349.0

The case BERTH shows how the battered pairs work, and the twist. Under the force in +X the pile whose tip points to +X is compressed and the pile whose tip points to −X is pulled (a negative axial force is tension): the horizontal components of their axial forces resist the load together with the shears. The third bent, next to the fender, takes most of it, P12 compressed with 244.3 kN and P9 pulled with −243.5 kN, while the first bent, nine metres from the fender, hardly feels it. In D+B+C the weight of the deck and the current are added: P12 carries 546.0 kN against 288.9 kN in P4 at the far end, and the head shears grow towards the fender as well. Mooring turns it around: the bollard is at the first bent, and in D+M+C the pile P1, leaning towards minus X, takes the most, 453.9 kN.

The batter of 1:6 is gentle, so in this trestle the shears still carry most of the berthing force; a steeper batter, or battered piles in both directions of the load as in Example 19 (Section 16.19), would shift more of it to axial forces. To continue, compute the foundation stiffness of the trestle for the deck model (Example 6, Section 16.6), or apply the berthing load in steps with a pushover case (Example 7, Section 16.7).

© 2026 Solplan. All rights reserved.

8 Drawing, selecting and editing

Piles, caps and the other objects of a model are drawn in the model view and edited with the commands of the **Draw**, **Select**, **Edit** and **View** menus. A model started from a template (Chapter 4) already contains the piles and the cap; this chapter is needed to change the layout or to build an irregular one.

8.1 Coordinates, grids and coordinate systems

The coordinates of the model are global: X and Y horizontal, Z vertical and positive upward. Site coordinates, for example of a map grid, can be used directly; the results do not depend on the position of the group. The status bar shows the coordinates of the cursor and of the point it snaps to; a marker at the cursor (Joint, Grid Intersection, End, Midpoint, Intersection) shows the kind of point, in the drawing tools and in the selection mode alike.

Define ▸ Coordinate Systems / Grids… (also **Draw ▸ Edit Grid…**) lists the coordinate systems of the model with the System name, its Type and the number of Grid lines. **Add New…**, **Add Copy…** and **Modify…** open *Coordinate System / Grid Data* (Figure 8.1):

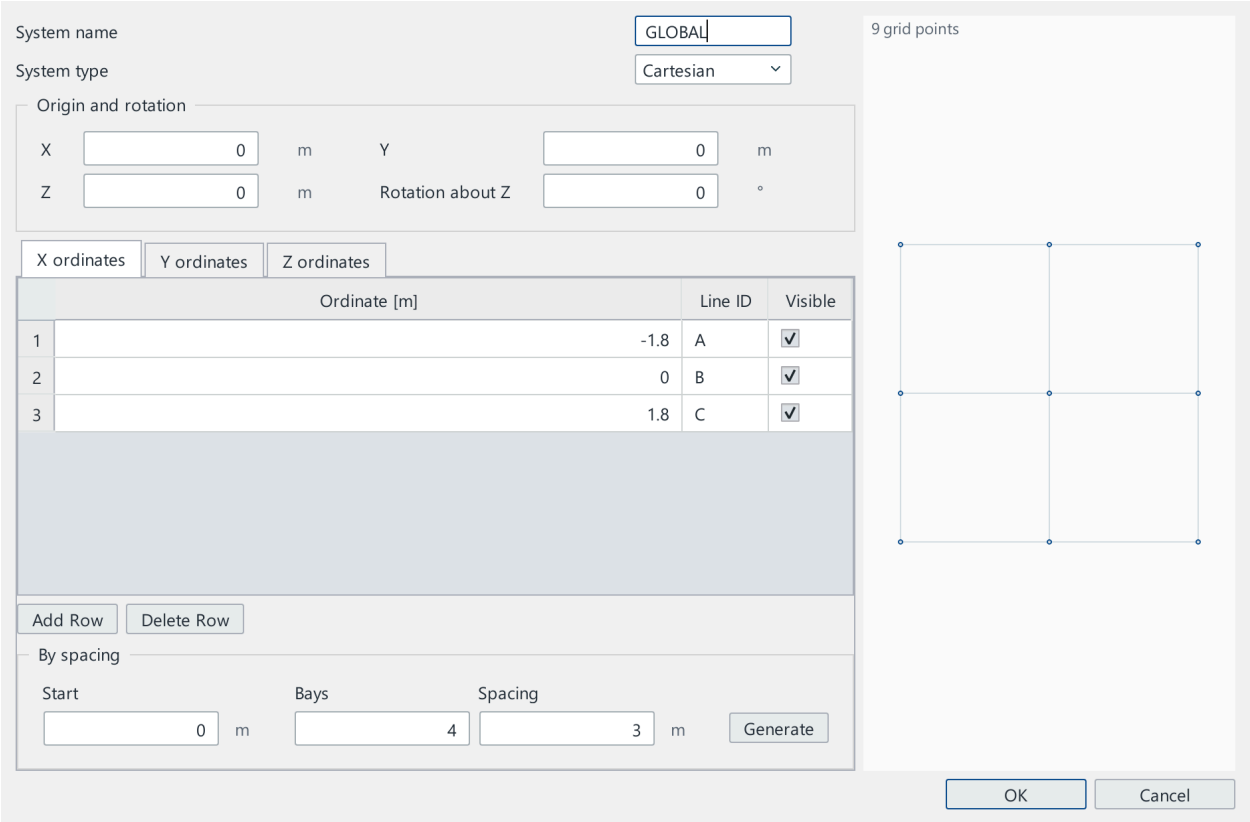


Figure 8.1 — Coordinate System / Grid Data, X ordinates

Table 8.1 — Coordinate System / Grid Data

Field	Meaning
System name	name of the system; the system GLOBAL always exists and cannot be deleted
System type	Cartesian (grid lines at X and Y ordinates) or Cylindrical (radii and angles)
Origin X, Y, Z; Rotation about Z	position and orientation of the system relative to the global axes
X ordinates, Y ordinates, Z ordinates	one tab for each direction. Each row is a grid line: its **Ordinate**, its **Line ID** (the label drawn at the end of the line) and **Visible** (whether it is drawn). **Add Row** and **Delete Row** edit the list.
Start, Bays, Spacing; Generate	replace the ordinates of the tab with an evenly spaced set: the first ordinate, the number of bays and their spacing

© 2026 Solplan. All rights reserved.

The grid lines of the active system are drawn in the model view and are used by the drawing tools to snap objects. Z ordinates define the elevations at which grid points exist; a plan grid without Z ordinates has no grid points.

8.2 Drawing tools

A drawing tool stays active until another tool is chosen or Esc is pressed, which returns to selection. The status bar tells what to click next; Backspace removes the last clicked point.

Table 8.2 — Commands of the Draw menu

Command	Use
Set Select Mode	ends drawing; clicks select objects
Set Reshape Element Mode (R)	select a handle (the head or tip of a pile, a vertex of the cap) and drag it to a new position
Draw Special Joint	click a point to place a joint, for example a point at which loads are applied
Draw Pile (F)	click the pile head, then the pile tip; the pile may be inclined
Quick Draw Pile (Q)	click a grid point to place a vertical pile with the default length and section
Quick Draw Batter Pile	click the pile head, then drag in the direction of the batter; the batter ratio is the default ratio
Draw Poly Cap (C)	click the corners of the cap outline; click the first corner again or right-click to finish
Draw Rectangular Cap	click two opposite corners of the cap
Quick Draw Cap	click to create a cap around the pile heads at the elevation of the click, with the default overhang
Draw Reference Point, Draw Reference Line	place construction aids, which are not part of the analysis model but can serve as load points and snap targets
Snap To	the points the cursor snaps to, each switched on or off: *Joints & Grid Points*, *Midpoints & Ends*, *Intersections*, *Perpendicular*, *Lines & Edges* and *Fine Grid*
New Labels…	renumber the piles (Table 8.6)
Edit Grid…	the coordinate systems and grids (Table 8.1)
Set Default Draw Properties…	the properties given to new piles and caps (Table 8.3)

Piles can be placed anywhere, at any spacing and in any direction; the analysis needs no regular layout, and the group effects are worked out from the actual position of each pile. Every pile head is, however, tied to the cap: there is no pile that stands on its own. A pile drawn outside the cap outline is still tied to a rigid cap, which ignores its outline, and Check Model warns about it; with a flexible cap, a head that is neither at a node nor inside a plate of the mesh stops the run. A pile without a cap is analysed in a separate single-pile model. Two heads at the same point, and a pile that starts on the axis of another, are errors; crossing piles and piles closer than 2.5 diameters are warnings.

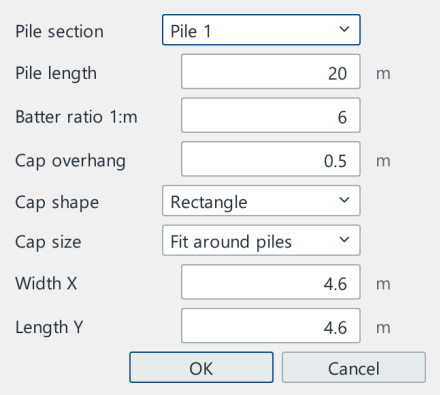


Figure 8.2 — Set Default Draw Properties

© 2026 Solplan. All rights reserved.

Table 8.3 — Set Default Draw Properties

Field	Meaning
Pile section	section of new piles
Pile length	length of piles placed with the quick tools
Batter ratio 1:*m*	batter of piles placed with Quick Draw Batter Pile: one horizontal to *m* vertical
Cap overhang	distance from the outermost pile heads to the edge of a cap drawn with Quick Draw Cap
Cap shape	Rectangle or Circle
Cap size	Fit around piles (from the pile heads and the overhang) or Specify dimensions
Width X, Length Y; Diameter	dimensions of a rectangular or circular cap when they are specified

8.3 Selecting

Most commands of the **Assign** and **Edit** menus act on the selected objects. A click selects an object; a window drawn from left to right selects the objects inside it, and a window drawn from right to left also selects the objects it crosses. The **Select** menu offers further ways:

Table 8.4 — Commands of the Select menu

Command	Use
Pointer / Window	click or window selection, as above
Intersecting Line	draw a line across the objects to select
Poly Window	click the corners of a polygon; right-click to select the objects inside it
All (Ctrl+A), None (Ctrl+Shift+A), Invert Selection, Previous Selection	select all objects, clear the selection, invert it, or restore the previous selection
Properties ▸ by Pile Section, by Material, by Head Connection, by p-y Set	select the piles that have the chosen property
Object Type ▸ All Piles	select every pile
Coordinate Specification ▸ by Specified Plane…	the objects on a plane: XY (constant Z), XZ (constant Y) or YZ (constant X), at the ordinate of the current view, within a tolerance
Coordinate Specification ▸ by Group…, by Label Range…	the piles of a named group, or the piles whose labels fall between two labels
Coordinate Specification ▸ by Row / Ring…	a row along X (constant Y), a row along Y (constant X) or a ring around the plan centre, within a tolerance
Coordinate Specification ▸ Batter Piles Only, Vertical Piles Only	reduce the selection to the battered or to the vertical piles
Assign to Group…	add the selection to a named group, remove it from the group, or replace the group by it; a new name creates a group

The number of selected objects is shown on the status bar. Groups are listed and edited with **Define ▸ Groups…**.

The selection dialogs are small (Figure 8.3). *Select by Specified Plane* asks for the **Plane** (XY plane (constant Z), XZ plane (constant Y) or YZ plane (constant X)) and a **Tolerance**; the ordinate is that of the current view. *Select by Row / Ring* asks how to **Group piles by** (Rows along X (constant Y), Rows along Y (constant X) or Rings around the plan centre) and a **Tolerance** within which piles count as one row, and then lists the rows to choose from. *Select by Group* lists the groups, and *Select by Label Range* asks for the first and the last label. *Assign to Group* asks for the **Group**, an existing one or a new name typed in the box, and the **Action**: Add the selection to the group, Remove the selection from the group, or Replace the group with the selection; the number of selected objects is shown.

© 2026 Solplan. All rights reserved.



Figure 8.3 — Select by Specified Plane

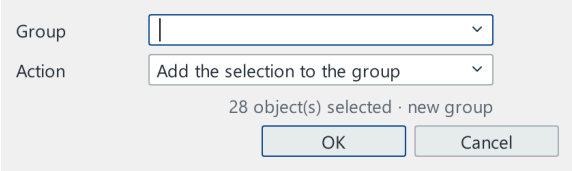


Figure 8.4 — Assign to Group

8.4 Editing

Every change of the model can be undone with **Edit ▸ Undo** (Ctrl+Z) and restored with **Redo** (Ctrl+Y); the number of steps kept is set in the preferences. **Cut**, **Copy** and **Paste** copy the selected objects, and **Delete** removes them.

Table 8.5 — Editing commands

Command	Fields and use
Move…	*dx*, *dy*, *dz*: the translation of the selected objects
Replicate… (Ctrl+R)	copies the selected objects (Figure 8.5). **Linear**: *dx*, *dy*, *dz* and the Number of copies. **Radial**: Rotation axis X and Rotation axis Y of a vertical axis, the Angle per copy and the Number of copies. **Mirror**: the Mirror plane (Y-Z plane (mirror about X), X-Z plane (mirror about Y) or X-Y plane (mirror about Z)) and the Plane ordinate. The local axes rotation of each pile is copied unchanged: for a section that is not symmetric, check the orientation of the copies.
Divide Piles…	divides each selected pile into a number of segments of equal length, joined at new joints
Join	joins selected piles that lie on one line, the tip of each meeting the head of the next, into one pile; piles that are not collinear are not joined
Change Labels…	renumbers the piles (Table 8.6)
Check Duplicate Piles	finds piles identical to an earlier one, with the same geometry and assignments, and offers to delete them
Shift Piles, Soil Elevations and Loading…	moves the model vertically (Table 8.7)

The dialogs of the Edit menu act on the selected objects:

© 2026 Solplan. All rights reserved.

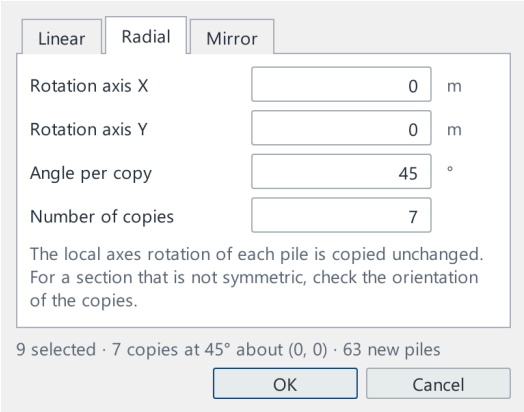


Figure 8.5 — Replicate, Radial tab

Move asks for the translation **dx**, **dy** and **dz**. *Divide Piles* asks for the **Number of segments**. *Change Labels* (Table 8.6) shows the labels that will result before they are applied.

Table 8.6 — Change Labels

Field	Meaning
Ordering	order of numbering: Order by X, then Y; Order by Y, then X; or Order by radius, then angle
Label prefix	text before the number, for example P
Start number	number of the first pile; the dialog shows the labels that will result

Table 8.7 — Shifting Options

Field	Meaning
Absolute Coordinates	the shift is the difference between the Target point elevation and the Reference point elevation
Relative Coordinates	the shift is an Increment (up +), positive upward
Shift All Piles, Shift Soil Layering, Shift Loading	what is moved: the piles with the cap, its mesh, the joints and the elevations assigned to the piles; the boundaries of the soil layers; the elevations of the loads along the piles and the point of application of the cap loads. Only the vertical coordinate changes.

NOTE Moving or replicating the objects of a flexible cap moves its mesh with it. When a later change of the cap outline, the piles or the element size no longer matches the stored mesh, Check Model reports it; generate the mesh again before the run (Section 10.2).

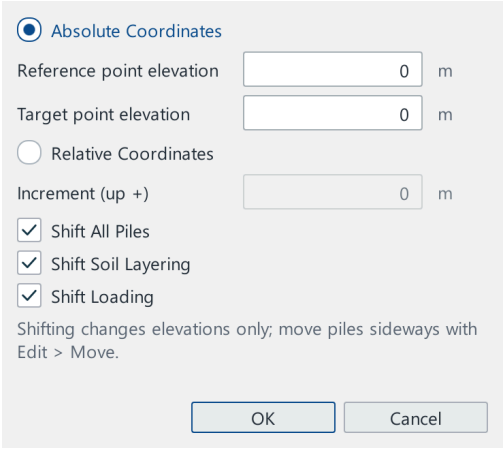


Figure 8.6 — Shifting Options

© 2026 Solplan. All rights reserved.

8.5 Views and display

The model view has up to four panes, each with its own view: **View ▸ Pane Layout** offers Single Pane, Two Panes (stacked), Two Panes (side by side) and Four Panes. The **View** menu applies to the active pane:

Table 8.8 — Commands of the View menu

Command	Use
Set Display Options… (Ctrl+E; also in the Options menu)	every display switch of the active pane in one place (Figure 8.7): the transparency of solids and soil; the assigns shown; the loads shown (point forces, moments, loads along piles, cap area loads, soil movement) with the arrow scale and the number of arrows of a distributed load; grid lines, global axes, bounding box, object labels (optionally hiding overlapping labels), local axes of the piles, extruded solids, soil layers, labels of soil, water and scour, ground and scour markers of each pile, load values beside the arrows; and whether grid lines can be dragged
Set 2D View…	a plan or elevation view: the X-Y, X-Z or Y-Z plane at an ordinate; **Previous ordinate** and **Next ordinate** step through the grid lines
Set 3D View…	the plan and elevation angles of the 3D view and the aperture (0° for an orthographic view; larger values give stronger perspective); buttons for standard views
XY, XZ, YZ, 3D	switch the pane to a standard view
Restore Full View (Home), Previous Zoom, Next Zoom, Zoom to Selection (Z), Rubber Band Zoom, Rotate 3D View	zoom and rotate; the mouse wheel zooms and a drag with the middle button pans
Refresh Window (F2)	redraws the panes
Show Grid, Show Axes, Show Extruded, Show Soil Layers, Show Labels, Show Local Axes	switch these elements on or off; the labels of the piles are hidden in a new model and appear with Show Labels
Pile Visibility	Hide Selected Piles, Hide Other Piles, Invert Pile Visibility or Show All Piles; hidden piles stay in the model and in the analysis
Solid Transparency	opacity of the extruded piles and cap: 100% opaque, 85% opaque, 70% opaque, 55% opaque or 40% opaque; a fully opaque cap hides the pile heads underneath
Soil Transparency	opacity of the soil layers: 0% opaque (hidden), 15% opaque, 30% opaque, 50% opaque or 70% opaque

© 2026 Solplan. All rights reserved.

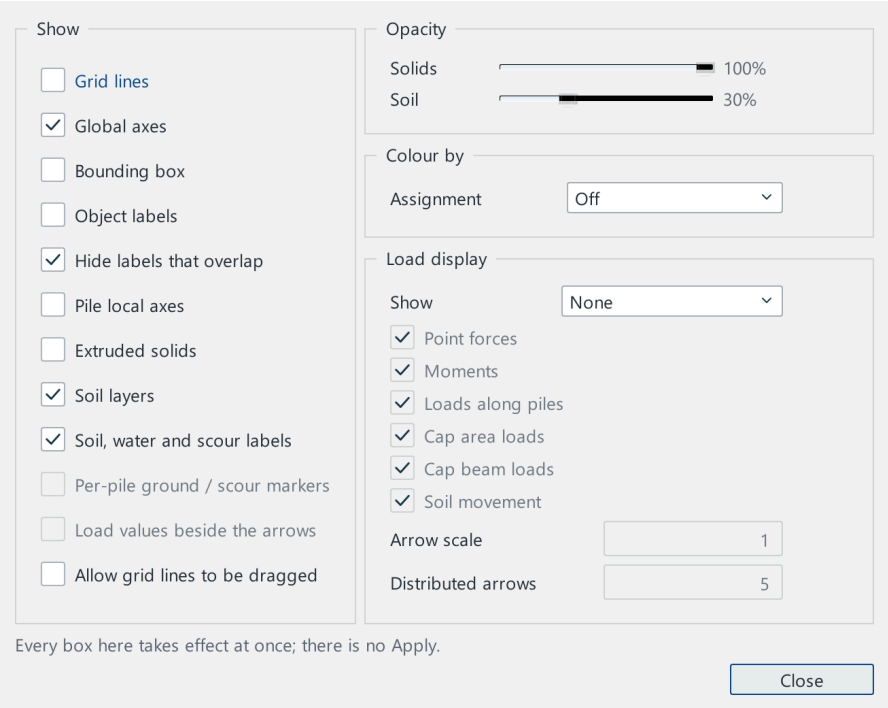


Figure 8.7 — Set Display Options

In *Set Display Options*, **Solids** and **Soil** set the transparency; **Assignment** chooses the assign shown on the piles (Off, Pile Number, Generated L-D Curve, Generated T-R Curve, Pile Section, Material, Pile Type, p-y Set, Head Connection or Lateral Reduction Set); **Show** chooses the loads shown (None, All loads / all patterns, or one load pattern), with the kinds of load (Point forces, Moments, Loads along piles, Cap area loads, Cap beam loads, Soil movement), the **Arrow scale** and the number of **Distributed arrows**. The switches under *Show* are Grid lines (of the active coordinate system), Global axes (the X, Y, Z triad at the origin), Bounding box (the box that encloses every object), Object labels, Hide labels that overlap (labels that would print on top of each other are dropped), Pile local axes (axes 1, 2 and 3 of every pile), Extruded solids (the real section swept along each pile instead of a thick line), Soil layers, Soil, water and scour labels (names and elevations beside the soil model), Per-pile ground / scour markers (the assigned ground and scour elevations around each pile; display only, and unavailable while no pile has a ground elevation of its own; the ground surface itself is always drawn, sloping where the profile has a sloping shape), Load values beside the arrows, and Allow grid lines to be dragged (off by default: a grid line dragged by accident moves everything snapped to it).

Set 2D View asks for the **Plane** (X-Y Plane, X-Z Plane or Y-Z Plane) and the **Plane ordinate**; **Previous ordinate** and **Next ordinate** step through the grid lines. *Set 3D View* asks for the **Plan angle**, the **Elevation angle** and the **Aperture** (0° = orthographic; larger values give stronger perspective); **Home**, **Top**, **Front**, **Side** and **Iso** set standard views.

© 2026 Solplan. All rights reserved.

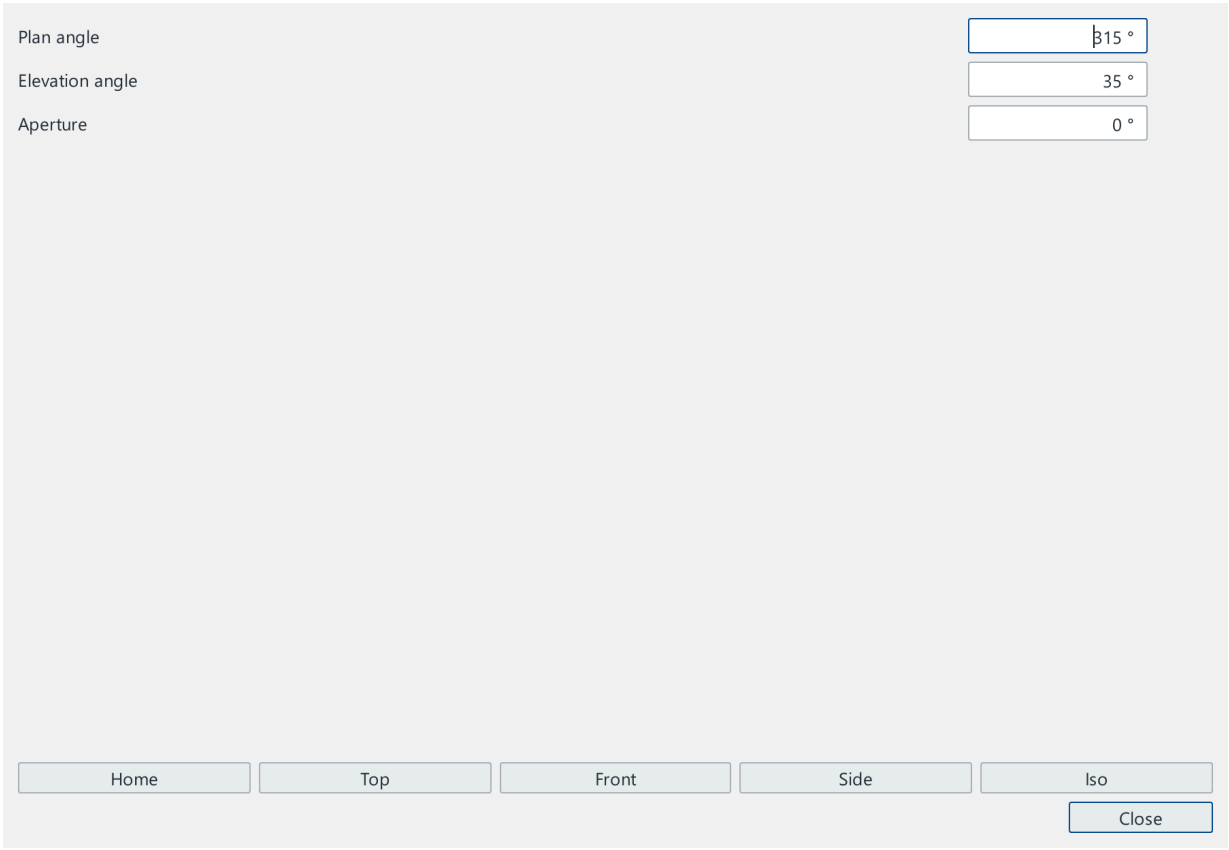


Figure 8.8 — Set 3D View

Display ▸ **Show Assigns** colours the piles by Pile Number, Generated L-D Curve, Generated T-R Curve, section, material, Pile Type, p-y set, head connection, lateral reduction set, axial reduction or group, with a legend. **Show Misc Assigns…** shows the **Object labels** (with **Hide labels that overlap**), the **Local axes of each pile**, and the piles coloured by group membership (**Colour piles by group membership**, available once a group is defined in Define ▸ Groups).

© 2026 Solplan. All rights reserved.

9 Defining the model

The **Define** menu holds the definitions that piles, the cap and the loads refer to by name: materials, sections, soil layers and profiles, curves, group factors, load patterns, cases and combinations, coordinate systems and groups. A definition does nothing until it is used: a pile section acts only on the piles to which it is assigned, a soil layer only where a soil profile places it. This chapter describes the definitions of the structure and the soil, in the order of the menu; load patterns, cases and combinations are described in Chapter 11.

9.1 Definition lists

Every command of the **Define** menu that manages named items opens a definition list (Figure 9.1). The table lists the items with their main properties; the buttons on the right act on the selected item:

Table 9.1 — Buttons of a definition list

Button	Action
Add New…	opens the data form with default values for a new item
Add Copy…	opens the data form with a copy of the selected item, under a new name
Modify…	opens the data form of the selected item (F2, or a double click on the row)
Delete	removes the selected item. An item that is in use cannot be deleted: the button is dimmed and its tooltip names the users, for example the sections that use a material or the piles to which a section is assigned. The GLOBAL coordinate system can never be deleted.

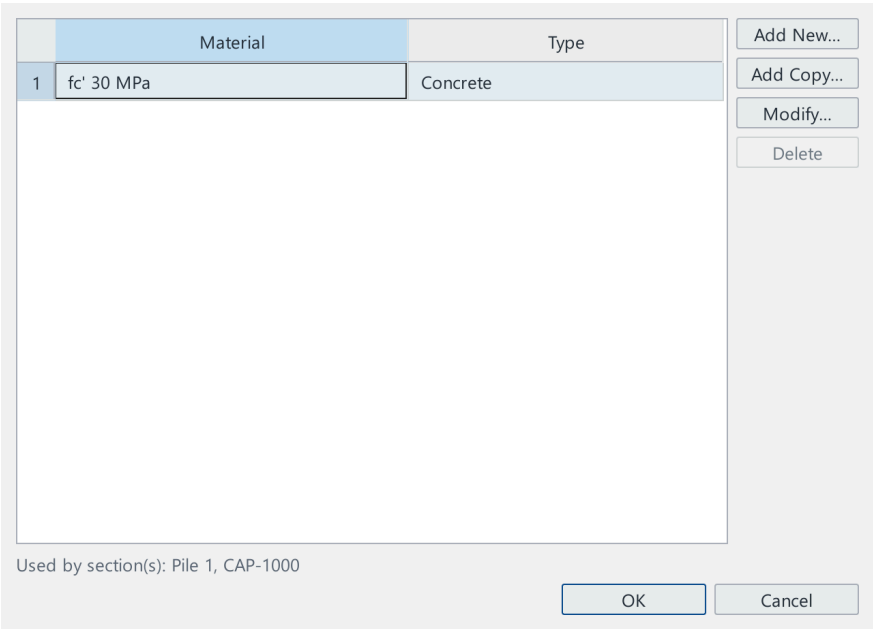


Figure 9.1 — A definition list: Define Materials

Every data form checks its data while it is typed: a missing or duplicate name, or a value outside its range, marks the field with a red border and a message under it, and **OK** stays disabled, with the reason in its tooltip, until the data are valid. Names have at most 64 characters and must be unique within their list. The list works on a copy of the definitions: the changes are written to the model once, when the list is closed with **OK**, so **Cancel** discards the whole editing session, and the session can be undone as one step with **Edit ▸ Undo**.

© 2026 Solplan. All rights reserved.

9.2 Materials

Define ▸ Section Properties ▸ Materials… lists the materials of the model with their type (Figure 9.1). Materials are used by the pile sections (the section material, the reinforcement, the casing and the core) and by the cap sections. The data form, *Material Property Data* (Figure 9.2), contains:

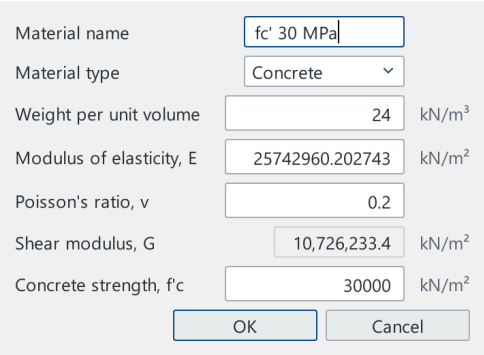


Figure 9.2 — Material Property Data of a concrete

Table 9.2 — Material Property Data

Field	Unit	Description
Material name		name by which the sections refer to the material
Material type		Concrete, Steel, Rebar (reinforcing steel) or Other. The type decides which strengths are asked for. When the type is changed while the unit weight, the modulus and Poisson's ratio still hold the defaults of the previous type, they are replaced by the defaults of the new type; values that have been edited are kept.
Weight per unit volume	force/volume	unit weight of the material, greater than zero. It is stored with the material; the analysis applies no self weight of the piles or the cap (Chapter 11).
Modulus of elasticity, E	stress	Young's modulus, greater than zero. For a rebar material the label reads *Modulus of elasticity, Es*. It gives the elastic bending stiffness *EI* and the axial stiffness *EA* of the sections that use the material.
Poisson's ratio, v		between 0 and 0.5
Shear modulus, G	stress	computed, $G = E/2(1 + v)$; it gives the torsional stiffness *GJ* of the sections
Concrete strength, f'c	stress	compressive strength of concrete; shown for type Concrete only and required there. It is used by the moment–curvature analysis of concrete and prestressed sections.
Yield strength, Fy	stress	shown for types Steel and Rebar only and required there; used by nonlinear steel sections, by the reinforcement and by the plastic moment of steel sections
Ultimate strength, Fu	stress	shown for types Steel and Rebar; zero or not less than F_y

To define a material:

1. Choose **Define ▸ Section Properties ▸ Materials…** and click **Add New…**.
2. Type the name and choose the type; the defaults of the type are filled in.
3. Enter the unit weight, the modulus and the strengths of the material, and click **OK** twice.

9.3 Pile sections

Define ▸ Section Properties ▸ Pile Sections… lists the cross-sections of the piles with their shape and material (Figure 9.3). A section describes the structural shape, the materials and the bending behaviour of a pile; it is assigned to piles with **Assign ▸ Pile ▸ Section…**, or along the length of a pile with **Assign ▸ Pile ▸ Section along Length…**

© 2026 Solplan. All rights reserved.

(Chapter 10).

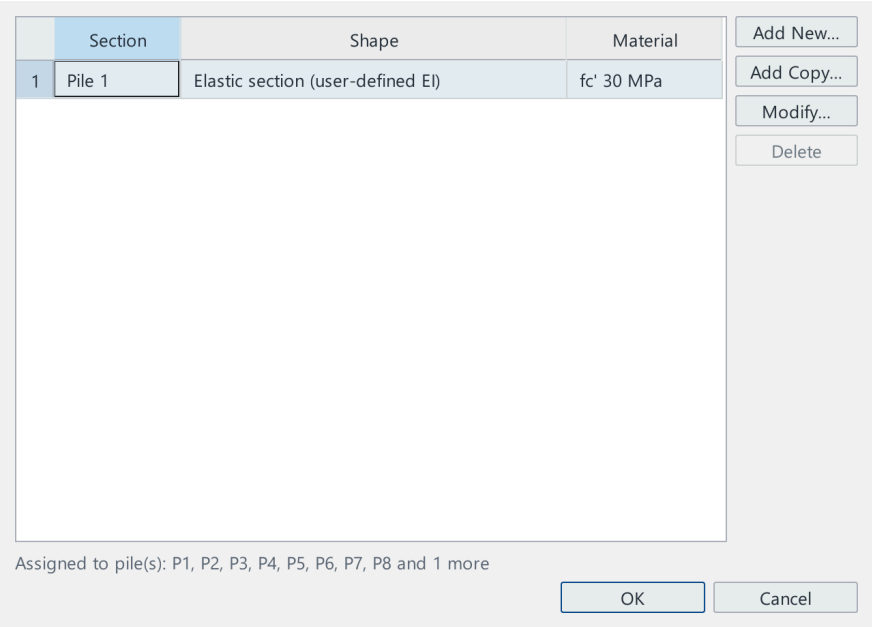


Figure 9.3 — Define Pile Sections

9.3.1 Choosing the shape

Add New… first opens *Add New Pile Section* (Figure 9.4): choose the **Section family** (Concrete, Steel, Prestressed or Other) and click the drawing of the shape; the large drawing on the right and the note under it describe the selected shape. The same window opens from **Change…** in the section form, to change the shape of an existing section: values with the same meaning, such as the diameter or the reinforcement, are kept, and the others take the defaults of the new shape.

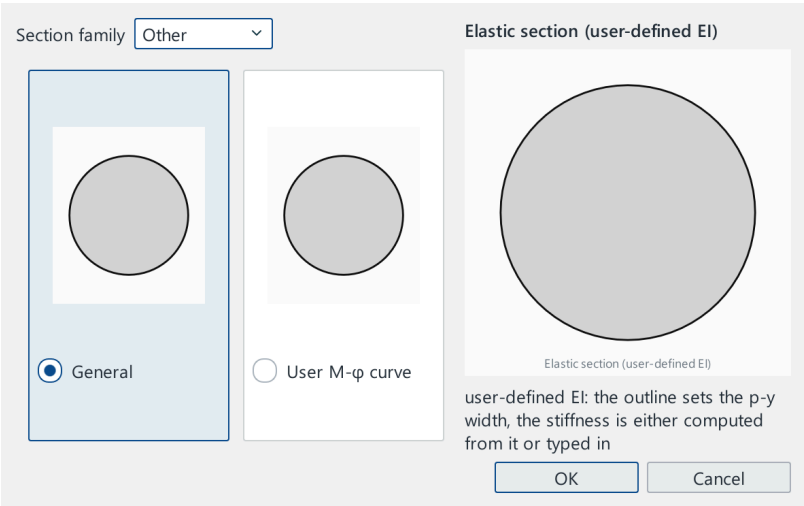


Figure 9.4 — Add New Pile Section

Table 9.3 — Section shapes

Family	Shape	Description	Behaviours
Concrete	Round	bored pile or drilled shaft, solid circular reinforced concrete	nonlinear
	Round with casing	circular concrete inside a steel casing left in place	nonlinear
	Casing and core	steel casing with an embedded steel pipe, H section or catalogue shape	nonlinear

© 2026 Solplan. All rights reserved.

Family	Shape	Description	Behaviours
	Rectangular	rectangular or square reinforced concrete	nonlinear
	Box (any material)	rectangular hollow section of concrete or steel; the void is rectangular and centred	elastic; elastic with moment capacity
Steel	Pipe	driven steel pipe or tubular pile	elastic; elastic with moment capacity; nonlinear
	H or I section	dimensions typed; unrotated, the section bends about its weak axis	elastic; elastic with moment capacity; nonlinear
	Catalog shape	AISC shape (W, M, S, HP, HSS, Pipe) from the steel shape database; dimensions read-only	elastic; elastic with moment capacity; nonlinear
Prestressed	Round; Round with void	solid or hollow (spun pile) circular prestressed concrete	nonlinear
	Square; Square with void	square prestressed concrete with chamfered corners, solid or with a circular void	nonlinear
	Octagonal; Octagonal with void	regular octagon, solid or with a circular void	nonlinear
Other	General	elastic section with user-defined *EI*: the outline sets the p-y width, the stiffness is computed from it or typed	elastic; elastic with moment capacity
	User M-φ curve	user nonlinear bending: moment–curvature or *EI*–moment data typed for each axial thrust	nonlinear

9.3.2 The section form

The form *Pile Section Data* (Figure 9.5) has a header, a column of tabs on the left, the data of the selected tab in the middle, and on the right a scaled drawing of the section with its computed properties. The tabs that are shown depend on the shape and the behaviour; a tab that does not apply is not shown.

© 2026 Solplan. All rights reserved.

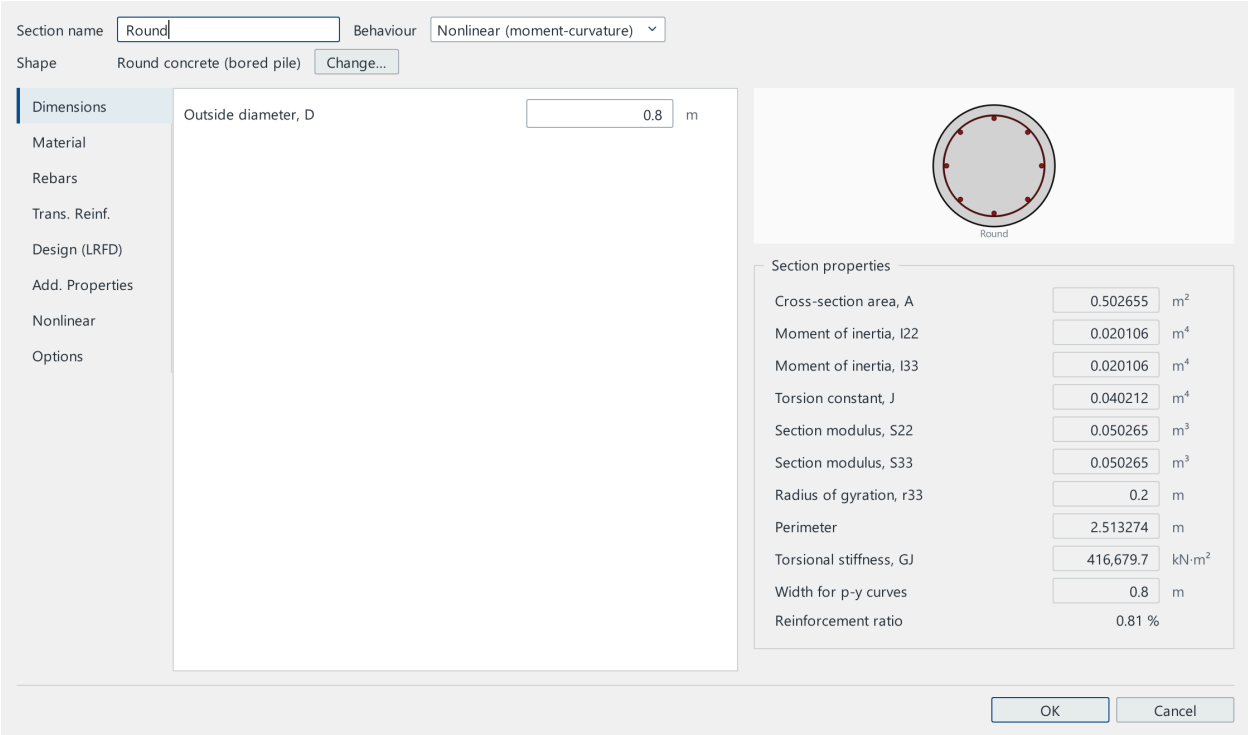


Figure 9.5 — Pile Section Data of a round reinforced concrete section, Dimensions tab

Table 9.4 — Header and section properties

Field	Description
Section name	name used by the assignments
Behaviour	how the bending stiffness is obtained: *Elastic (no yield)* uses the gross properties of the section at every moment; *Elastic with specified moment capacity* adds a plastic hinge where the moment reaches the capacity entered in the Design (LRFD) tab; *Nonlinear (moment-curvature)* computes the moment–curvature relation of the section for a list of axial thrusts, and the bending stiffness of each node follows its bending moment (Technical Reference, chapter “Section mechanics”). Only the behaviours of the shape are offered (Table 9.3).
Shape; Change…	the shape of the section; **Change…** chooses another shape (Section 9.3.1)
Cross-section area, A	gross area, used for the axial stiffness and the tip resistance
Moment of inertia, I22; Moment of inertia, I33	about local axes 2 and 3; I_{33} governs bending in the plane of axis 2, the usual strong direction
Torsion constant, J; Torsional stiffness, GJ	used for the torsional response of the pile
Section modulus, S22; Section modulus, S33; Radius of gyration, r33	elastic section moduli, for the combined stress, and the radius of gyration
Perimeter	used for the side friction of the axial curves
Width for p-y curves	the width *b* used for the soil curves: the diameter of a round section; for a non-circular section, the value of *Width for p-y curves (0 = face width)* or, when it is zero, the face width across the bending plane (for example the flange width when an H section bends about its strong axis)
Reinforcement ratio	area of longitudinal steel divided by the gross area, for concrete sections

The axes of a section follow the local axes of the pile: a width *b* lies along local axis 2 and a depth *h* along local axis 3. The drawing shows the axes, the reinforcement and the strands, so that the orientation can be checked before the section is assigned.

© 2026 Solplan. All rights reserved.

9.3.3 Dimensions

The Dimensions tab holds the dimensions of the shape; the area and the inertias follow from them, except for the General shape when the computation is switched off.

Table 9.5 — Dimensions of each shape

Shape	Fields
Round; Round (prestressed)	Outside diameter, D
Round with casing	Casing outside diameter, D; Casing wall thickness
Casing and core	Casing outside diameter, D; Casing wall thickness; Core / insert (Steel pipe; H section, strong axis; H section, weak axis; AISC shape, strong axis; AISC shape, weak axis); for a pipe core: Core outside diameter and Core wall thickness; for an H core: depth, flange width and thickness, web thickness; for an AISC core: the family and designation; **Fill Core with Concrete** (only for a pipe or a box core)
Rectangular	Width (axis 2), b; Depth (axis 3), h; Rotate 90°
Box (any material)	Outside width (axis 2), b; Outside depth (axis 3), h; Internal width, bi; Internal depth, hi; Width for p-y curves (0 = face width); Tapered; Rotate 90°
Pipe (steel)	Outside diameter, D; Wall thickness, t; Tapered (elastic behaviours)
H or I section	Section depth, d; Flange width, bf; Flange thickness, tf; Web thickness, tw; Width for p-y curves (0 = face width); Rotate 90°
Catalog shape	Shape family (W, M, S, HP, HSS, Pipe); Shape designation; the Depth, d, Flange width, bf, Flange thickness, tf and Web thickness, tw of the shape are shown; Width for p-y curves (0 = face width); Rotate 90°
Round with void (prestressed)	Outside diameter, D; Void diameter, Di
Square; Square with void	Side, b; Corner chamfer; Void diameter (with void); Rotate 90°
Octagonal; Octagonal with void	Width across flats; Void diameter (with void). The corner chamfer is computed so that the shape is a regular octagon.
General	Structural shape (Circular; Pipe (circular hollow); Rectangular; H or I section; General (no outline; type every property)) and the dimensions of that outline, for example Diameter, D; **Compute area and inertia from the dimensions** (ticked: computed and read-only; unticked: typed); Area, A; Moment of inertia, I22 and I33; Torsion constant, J; Perimeter; the Torsional stiffness, GJ = G × J is shown; Tapered
User M-φ curve	Width for p-y curves; Area, A (0 = solid circle of the width); Perimeter (0 = circle of the width); Torsion constant, J (0 = solid circle)

Rotate 90° turns the section 90° about the pile axis: the sides along axes 2 and 3 swap, so the strong and the weak axis swap; the drawing shows the result. An H section that is not rotated bends about its weak axis under a load along axis 2. **Tapered** (elastic behaviours) makes the section change linearly from the pile head to the pile tip: a second set of fields marked — *at tip* holds the dimensions at the tip, for example *Outside diameter, D — at tip* or *Area, A — at tip*, and the properties at each node are interpolated. A nonlinear section cannot be tapered.

© 2026 Solplan. All rights reserved.

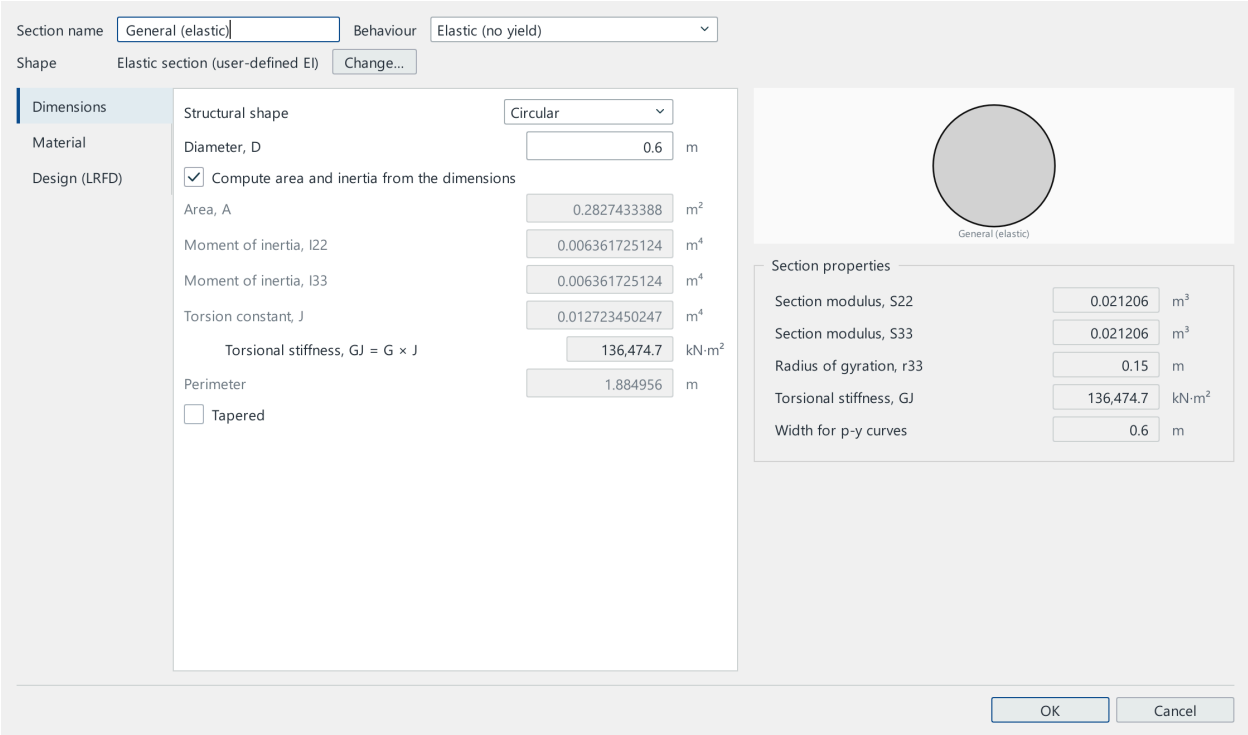


Figure 9.6 — Dimensions of a General (user-defined *EI*) section

9.3.4 Material

The Material tab selects the materials from the material list (Section 9.2) and shows the strengths that the analysis takes from them: the **Section material** (concrete or steel); for reinforced concrete, the **Rebar material**; for cased shafts, the **Casing material** and the **Core material**. The values shown, for example *Concrete strength, f′c*, *Concrete modulus of elasticity, E*, *Reinforcement yield, fy*, *Reinforcement modulus, Es*, *Yield strength, Fy* and *Modulus of elasticity, E*, are read-only: they are changed in the material. **View stress-strain curve…** opens *Stress-Strain Curve* (Figure 9.8), which draws the stress–strain relation of each material of the section used by the moment–curvature analysis; the material is chosen in the list at the top, and the note under the graph names the model and its constants. Negative strain is tension: the concrete carries tension up to cracking and none after it.

© 2026 Solplan. All rights reserved.

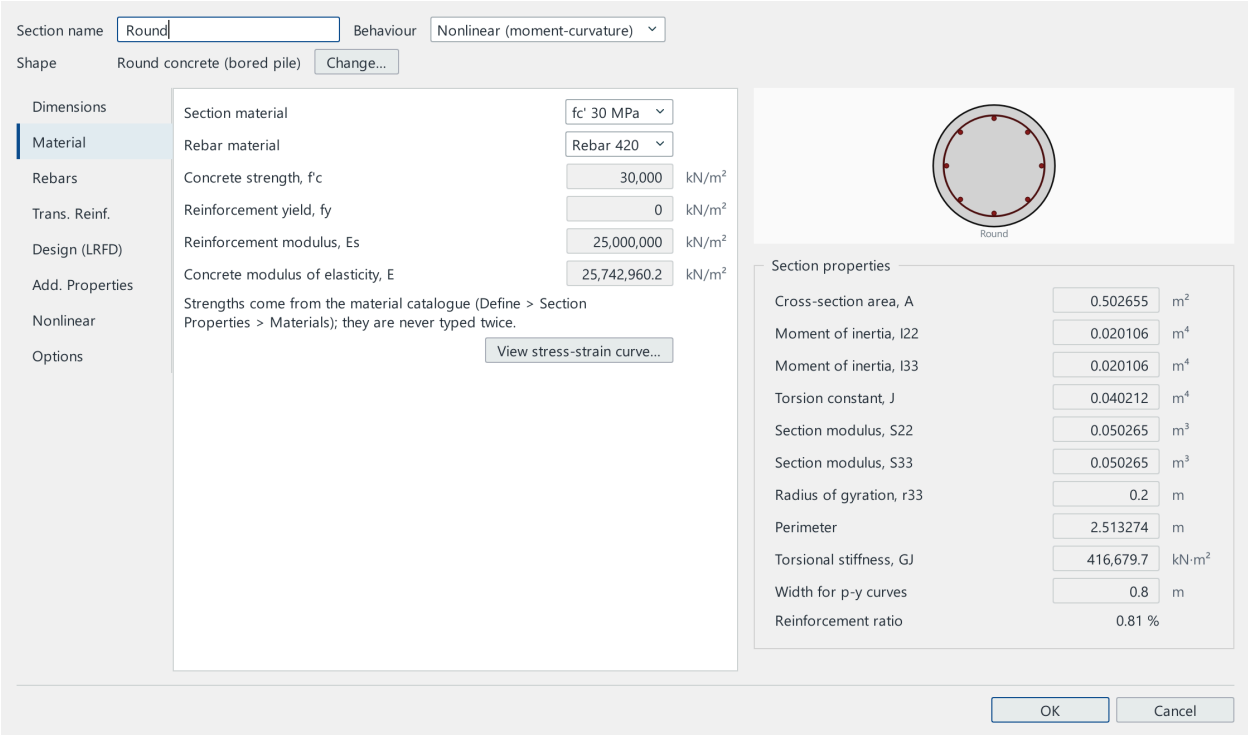


Figure 9.7 — Material tab

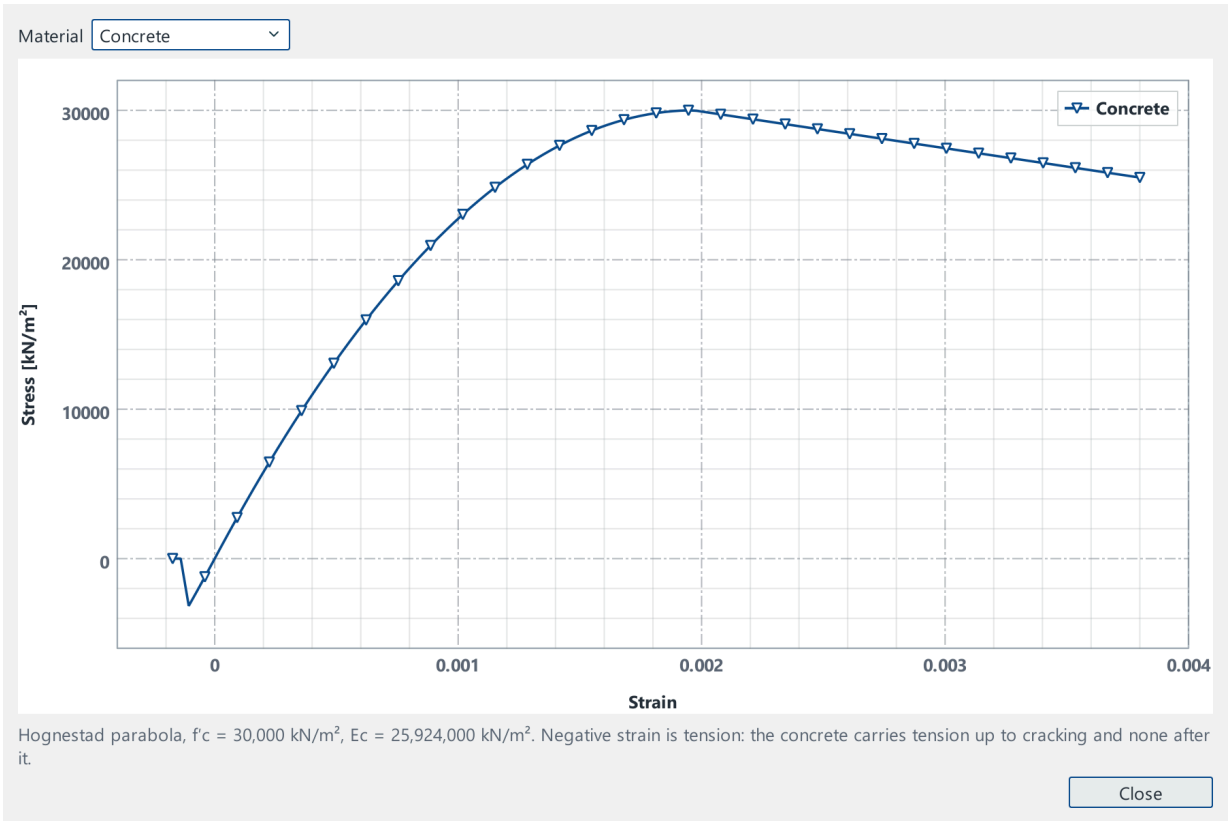


Figure 9.8 — Stress-Strain Curve of the concrete of a bored pile

9.3.5 Rebars

The Rebars tab (concrete shapes) describes the longitudinal reinforcement (Figure 9.9):

Table 9.6 — Rebars tab

Field	Description

© 2026 Solplan. All rights reserved.

Field	Description
Plain Concrete – No Rebars	an unreinforced section; the other fields are dimmed
Bar Size	US (#3 to #18), Canadian (10M to 55M), metric and other bar sizes, each listed with its diameter; for a user-defined size the **Bar Area** is typed and the **Bar Diameter** follows from it
Rebar Pattern	*Circle*: bars equally spaced around a circle; *Rectangle*: bars along the faces of a rectangle, with the numbers of bars on the top and bottom faces and on the left and right faces; *Manual*: bars at coordinates typed in the table
Number of bars	for the circle pattern, the number of bars around the circle
Concrete Cover to Edge of Bar	clear cover from the concrete surface to the outside of the bars
Manual bars table	Bar size, X and Y of each bar from the centre of the section. **Add bar** adds a bar of the section bar size (Insert), **Remove** deletes the selected bars (Delete), and **Fill from circle** replaces the table with the bars of the circle pattern, which can then be edited.

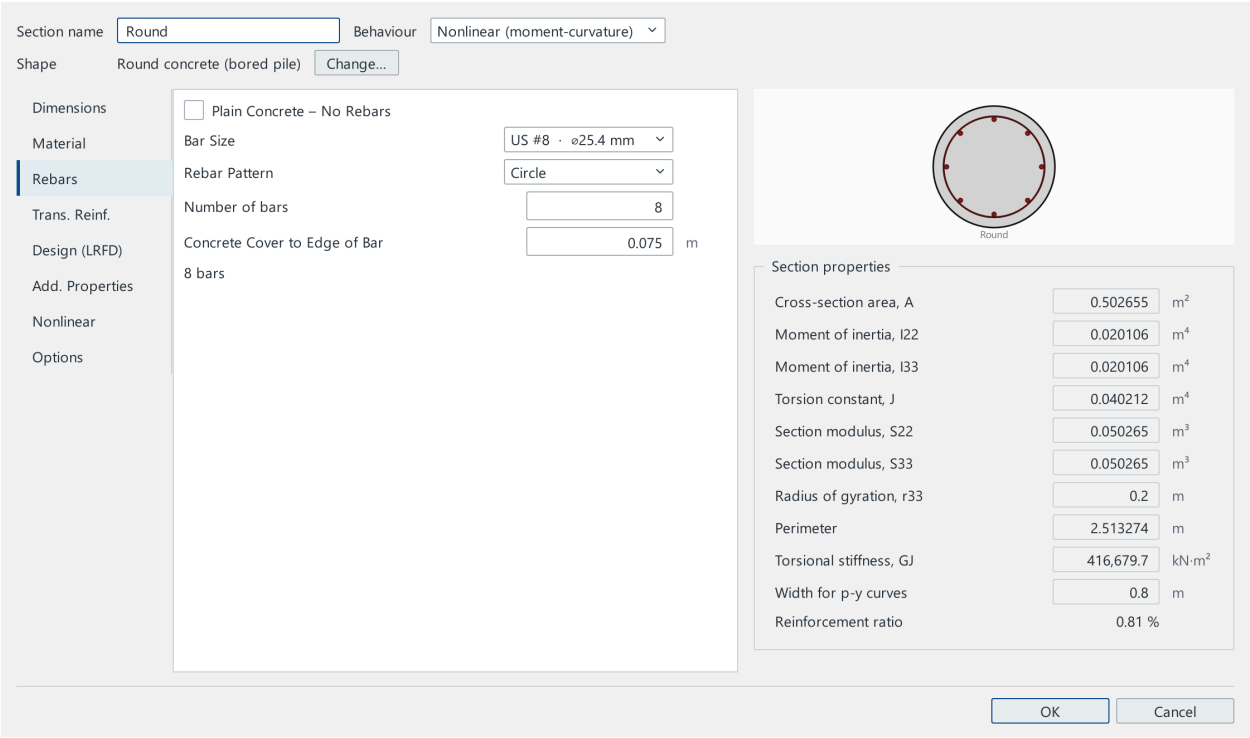


Figure 9.9 — Rebars tab

9.3.6 Trans. Reinf.

The Trans. Reinf. tab (round and rectangular concrete) describes the transverse reinforcement and the options of the confined-concrete and strain-hardening models (Figure 9.10):

Table 9.7 — Trans. Reinf. tab

Field	Description
Consider Confined Concrete (Mander et al. 1988)	uses the confined-concrete model of Mander et al. for the core inside the transverse reinforcement; off, the concrete is unconfined
Rebar Type	Spiral or Hoop. Only confined round sections read it: hoops lower the confinement effectiveness. Rectangular sections always use ties.
Bar Size	size of the transverse bars
Number of Transverse Reinf.; Number of Legs X; Number of Legs Y	for a round section, the number of transverse bars; for a rectangular section, the number of legs of the ties in X and in Y
Spacing of Transv. Reinf.	centre-to-centre spacing along the pile

© 2026 Solplan. All rights reserved.

Field	Description
fys of Transv. Reinf. (0 = fy)	yield strength of the transverse steel; zero uses f_y of the longitudinal bars
Use Strain Hardening (longitudinal)	available with confined concrete only (dimmed otherwise): adds strain hardening to the stress–strain curve of the longitudinal bars, with **f_u / f_y**, the strain at the onset of hardening **ε_sh (strain at hardening)** and the **ε_su (ultimate strain)**

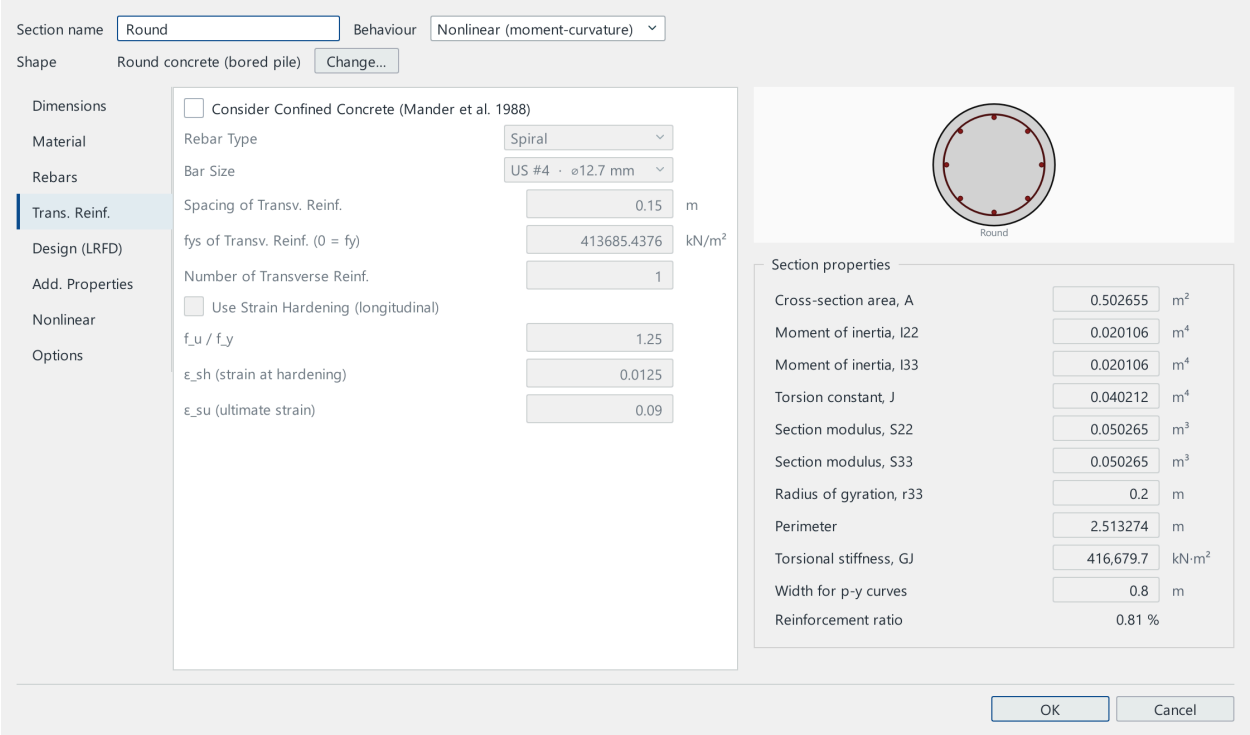


Figure 9.10 — Trans. Reinf. tab

9.3.7 Prestressing

The Prestressing tab (prestressed shapes) describes the strands (Figure 9.11):

Table 9.8 — Prestressing tab

Field	Description
Prestressing Strand Type	User-defined strand; Grade 250, 270 or 300 ksi low-relaxation strand; smooth bars 145 or 160 ksi; deformed bars 150, 157 or 160 ksi; CFCC. A user-defined strand takes the ultimate strength, the modulus and the area typed below the list.
Strand/Bar Size	nominal size with its area
Number of Strands/PS Bars	number of strands in the section
Strand Pattern	Circle or Square. The square layout puts the same number of strands on every side, corners included, so the number of strands is a multiple of 4.
Cover Over Strands	clear cover to the strands
Automatically position strand	places the strands on the pattern from the cover; off, the coordinates are typed in the strand table (X, Y and Area of each strand). **Add strand** and **Remove** edit the table, and **Fill from pattern** replaces it with the strands of the automatic layout.
Prestress Force Before Losses	total jacking force of the strands
Fraction of Loss of Prestress	long-term loss as a fraction of the force before losses, typically 0.10 to 0.20; the value is given by the pile supplier

© 2026 Solplan. All rights reserved.

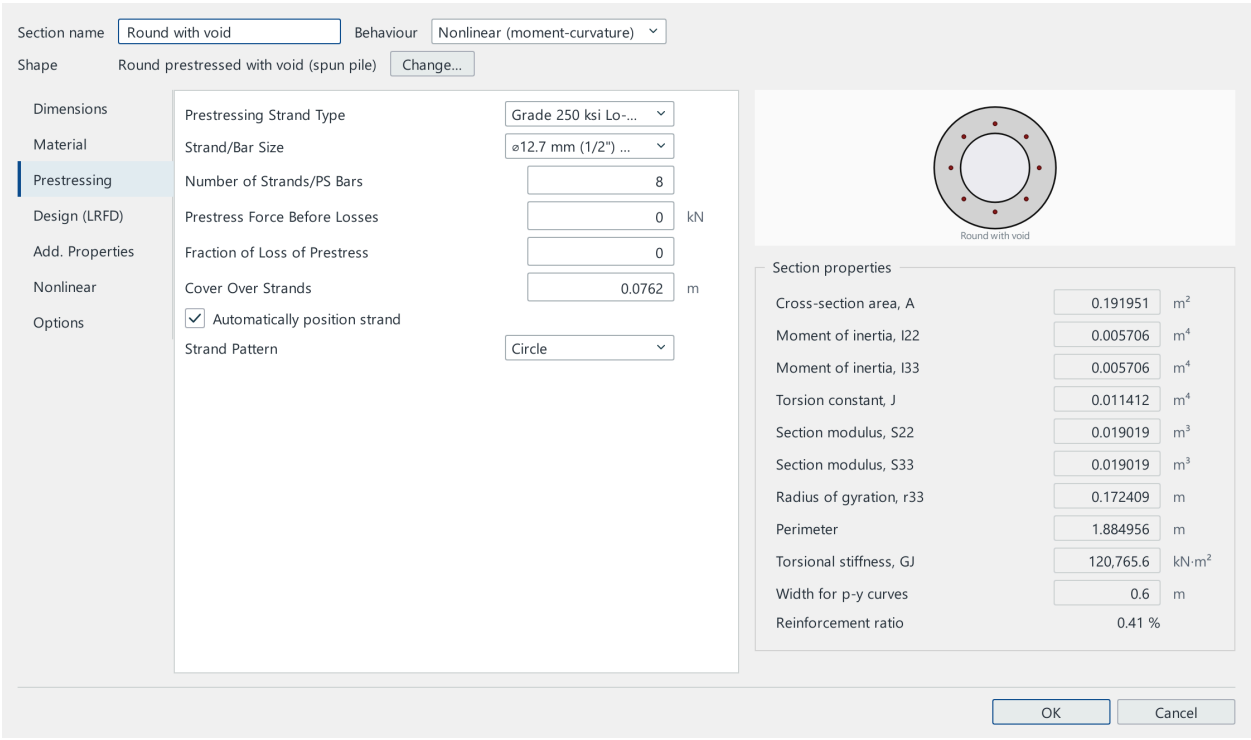


Figure 9.11 — Prestressing tab of a spun pile

9.3.8 Design (LRFD)

The Design (LRFD) tab holds the **Moment capacity about axis 3**, the **Moment capacity about axis 2**, the **Shear capacity, axis 2** and the **Shear capacity, axis 3** of the section. For the behaviour *Elastic with specified moment capacity*, the moment capacities are the plastic hinges of the pile and both must be positive. For every behaviour, the capacities are used for the demand/capacity ratios of load combinations (Chapter 11). A nonlinear section takes its moment capacity from its moment–curvature curve: **Compute moment capacity from M-φ** fills the moment fields from the curves, and the fields are dimmed with that reason in their tooltip.

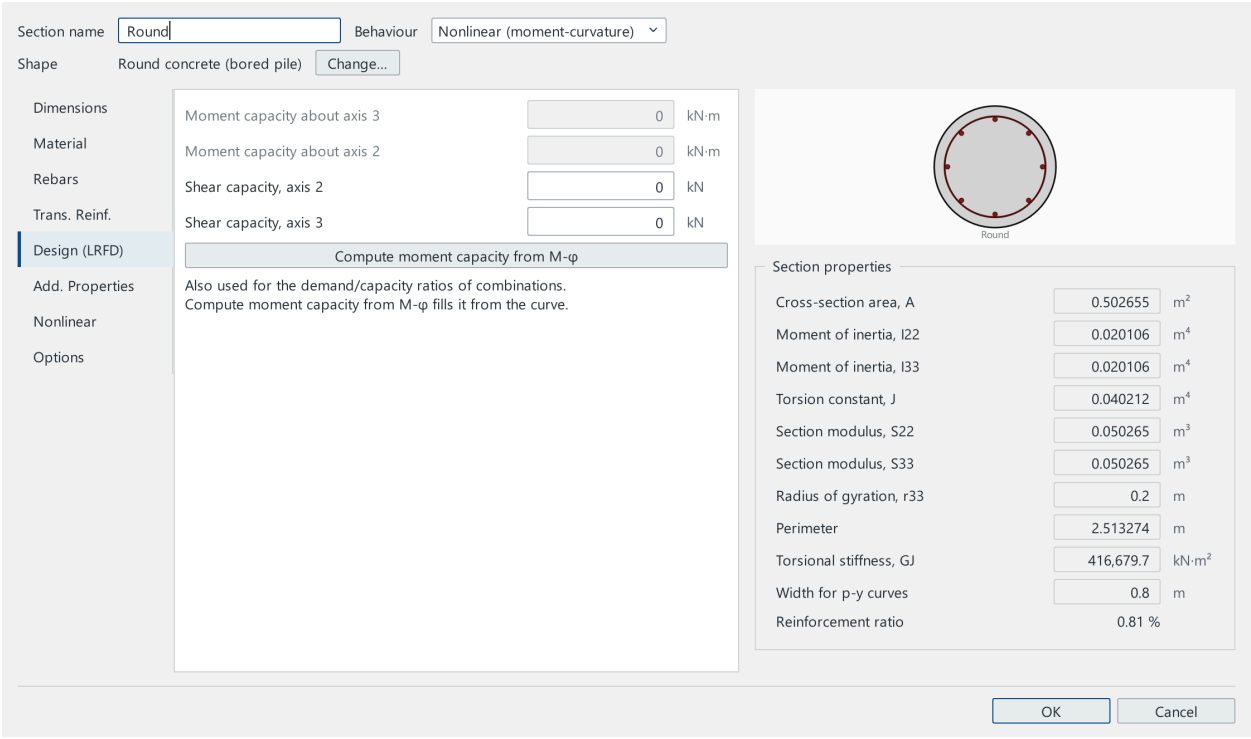


Figure 9.12 — Design (LRFD) tab

© 2026 Solplan. All rights reserved.

9.3.9 Add. Properties

For a nonlinear section, the analysis uses the bending stiffness of the moment–curvature curves and, for the other quantities, a set of equivalent properties shown in the Add. Properties tab. With **Enter the values** off, every value is computed from the section and its material and shown dimmed; ticked, the values can be typed:

Table 9.9 — Add. Properties tab

Field	Use
Eq. diameter	width used by the p-y curves, greater than zero; not used by round shapes, which use their own diameter
Perimeter	side friction of the axial curves; zero means no side friction
Area	tip resistance and axial stiffness
GJ (torsional stiffness)	torsional response
E eq.	axial stiffness = E eq. × Area, and the elastic pile of the equivalent-springs foundation stiffness; bending under load comes from the M-φ curves
Izz (0 = from the eq. diameter); Iyy (0 = from the eq. diameter)	elastic pile of the equivalent-springs foundation stiffness, bending about axes 3 and 2

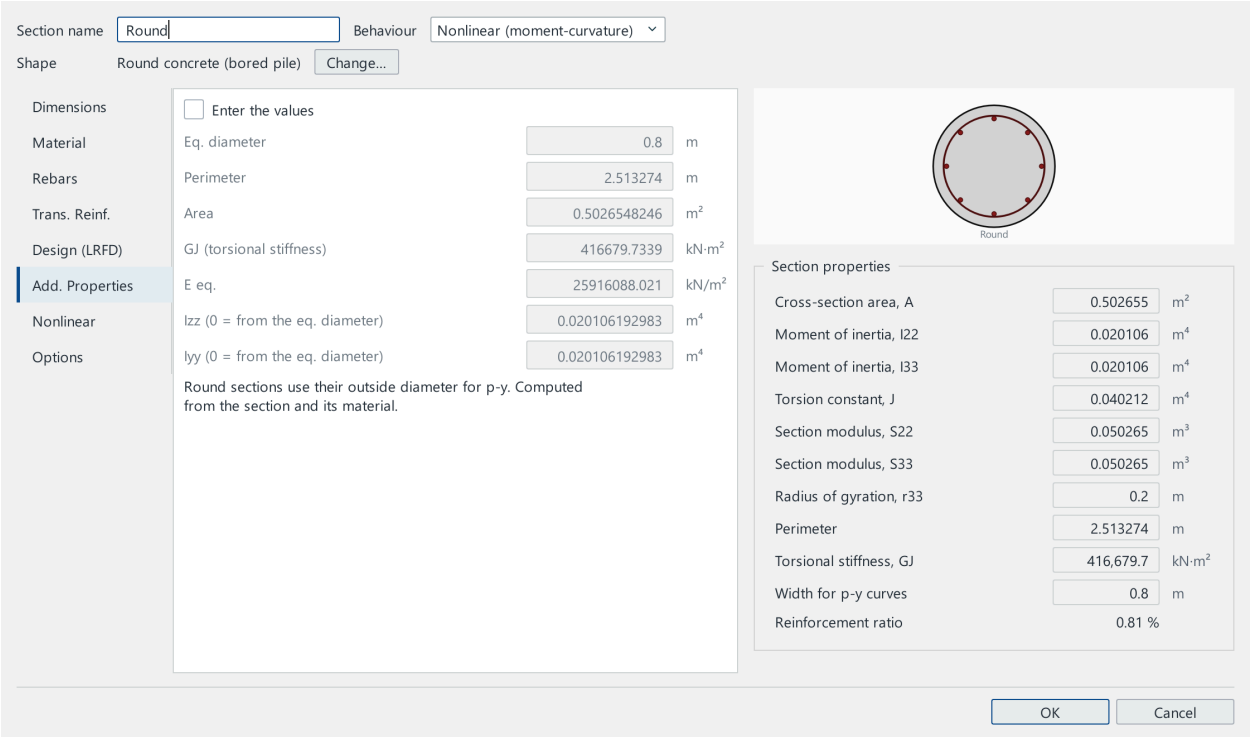


Figure 9.13 — Add. Properties tab

9.3.10 Nonlinear

The Nonlinear tab sets the axial thrusts for which moment–curvature curves are computed. In the analysis, the curve of each node is interpolated linearly between the two thrusts that bracket the axial force of the node, and beyond the first or the last thrust the nearest curve is used, so the list should span the axial forces expected in the piles, tension included where it can occur. The thrusts are given in one of three ways:

- **Set values from the axial capacity**: the list runs between the tension and the compression capacities of the section, moved inward by a small fraction of the range, with **Number of forces** values;
- **Set values from a maximum and a minimum**: **Number of forces** values between **Maximum** and **Minimum**;
- **Enter values manually**: the thrusts are typed in the table (**Add Row**, **Delete Row**); **Number of forces** follows the number of rows.

© 2026 Solplan. All rights reserved.

The two generated lists have at least two values; **Include the zero value** adds a thrust of zero. The **Compression capacity** and the **Tension capacity** of the section are shown. **Show moment-curvature…** computes and draws the curves (Figure 9.15).

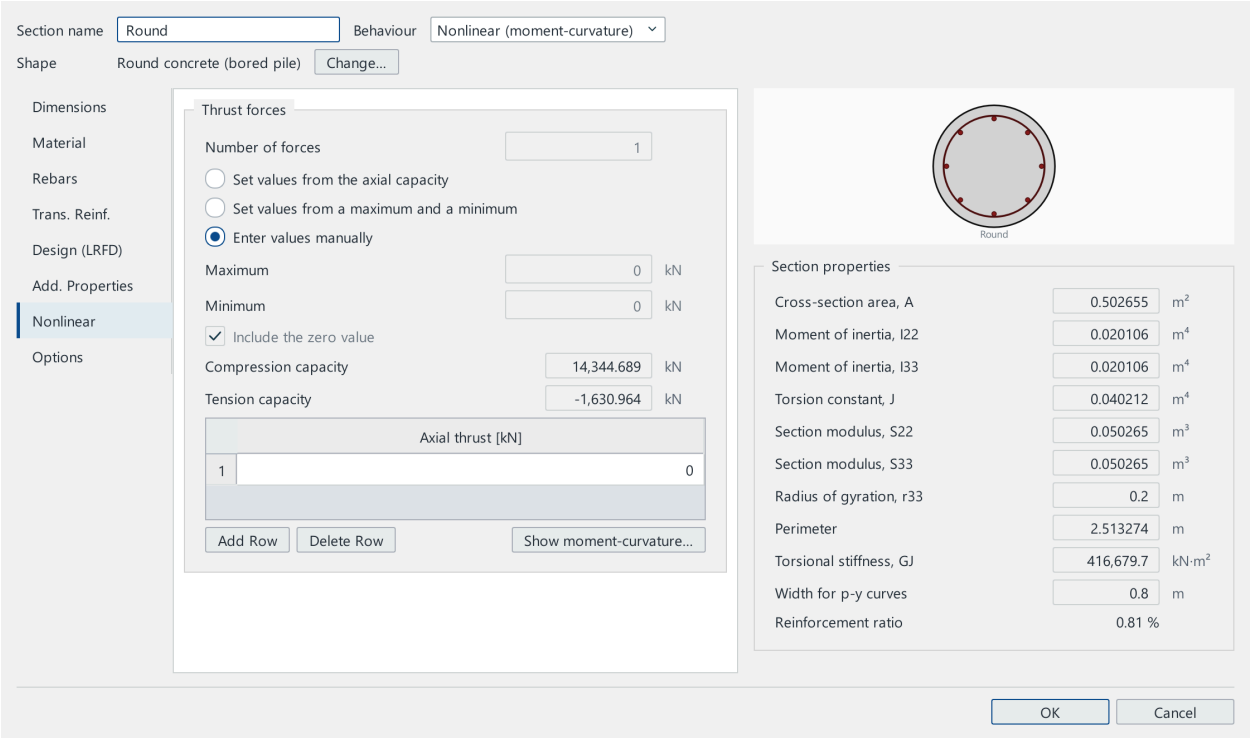


Figure 9.14 — Nonlinear tab

The window *Moment-Curvature* draws one curve for each axial thrust, for bending about axis 3 or about axis 2 as chosen in the list at the top, and states the peak moment. A curve that rises more steeply belongs to a larger compression; the flat part of a curve after cracking is the softening adjustment of the Options tab.

© 2026 Solplan. All rights reserved.

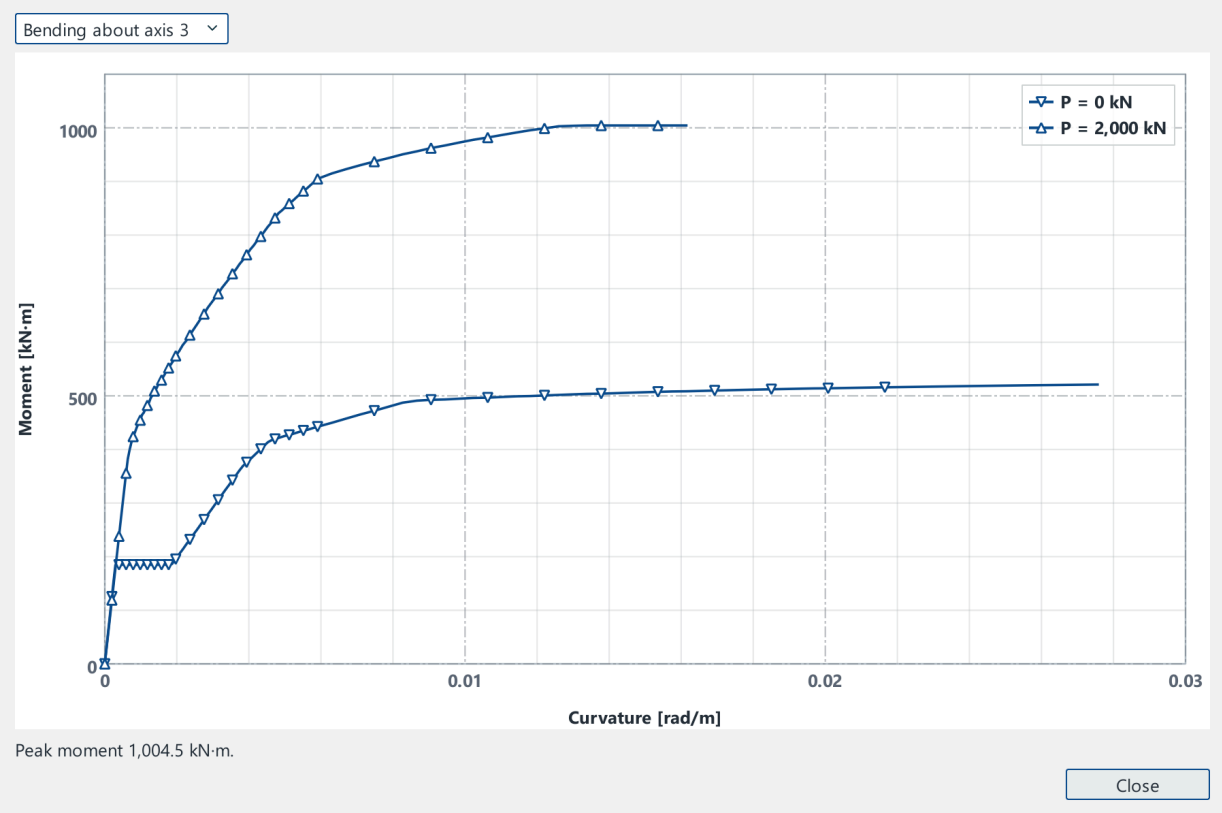


Figure 9.15 — Moment-Curvature of a bored pile at thrusts of 0 and 2 000 kN

For a **User M-φ curve** section, the tab also holds the bending data of the section (Figure 9.16): for axis 3, *Moment-curvature data* or *Elastic, I33*; for axis 2, *Same data as axis 3*, *Its own moment-curvature data* or *Elastic, I22*. The data are typed either as *Moment-curvature sets, one per axial thrust*, or as *User-defined EI against moment*. The set to edit is chosen in the list, and its points are typed in the table, **Curvature, φ** and **Moment, M** (or *EI* and *M*), with **Add Point** and **Delete Point**.

© 2026 Solplan. All rights reserved.

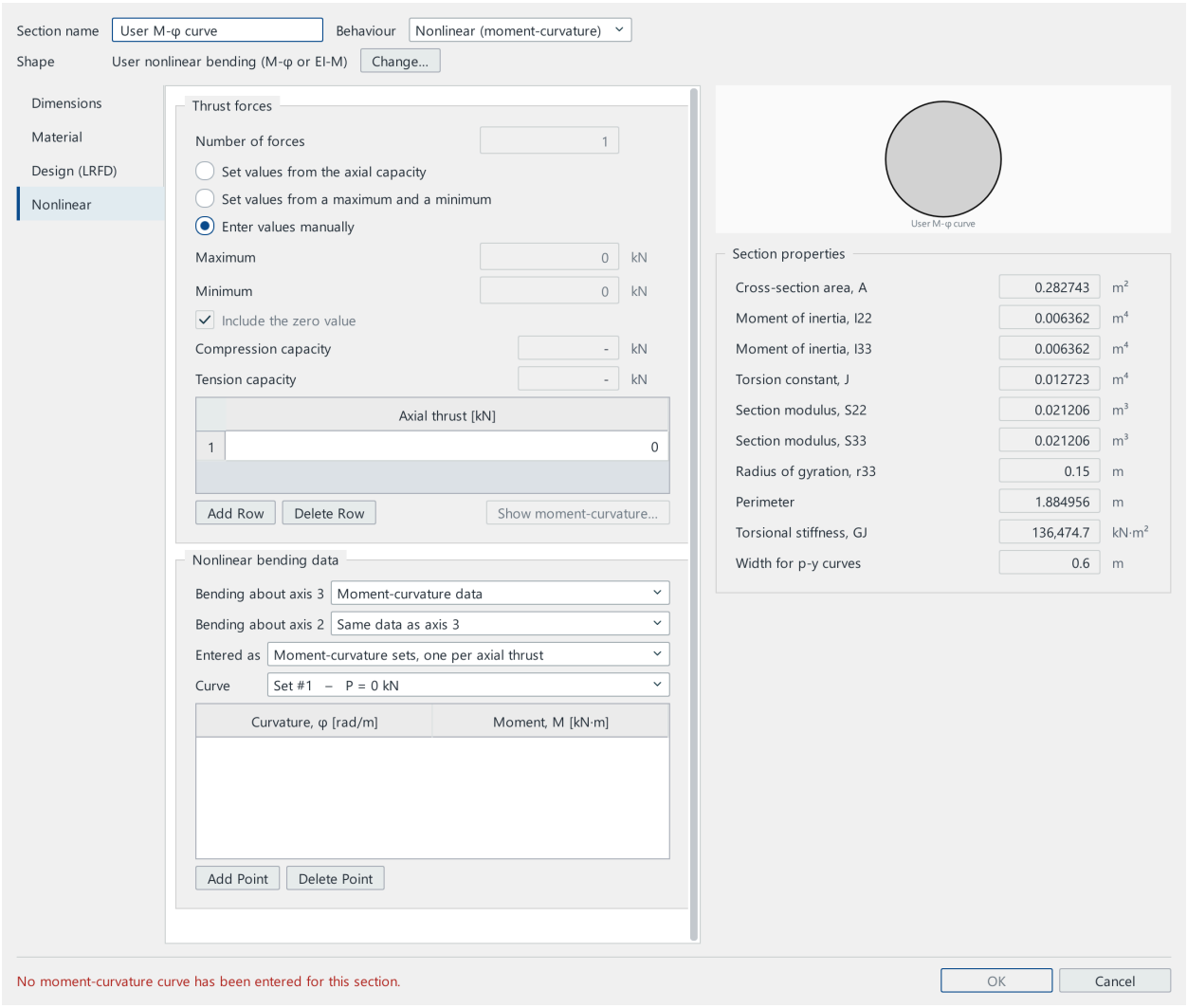


Figure 9.16 — Nonlinear tab of a User M-φ curve section

9.3.11 Options

The Options tab holds the options of the moment–curvature analysis, for the shapes that use them:

- **Adjust softening of the moment-curvature curve** (concrete and prestressed shapes, on by default): the moment of the curve is not allowed to fall once it has reached its peak; points at larger curvature keep the peak moment, so the curve has no descending branch;
- **Compute the equivalent elastoplastic curve (CALTRANS)** (concrete, prestressed and steel shapes): replaces the curve with its equivalent elastoplastic idealisation, with the yield point and the plastic moment of the Caltrans procedure. The idealisation is computed from the raw curve, so the softening adjustment is switched off and dimmed while it is ticked.

© 2026 Solplan. All rights reserved.

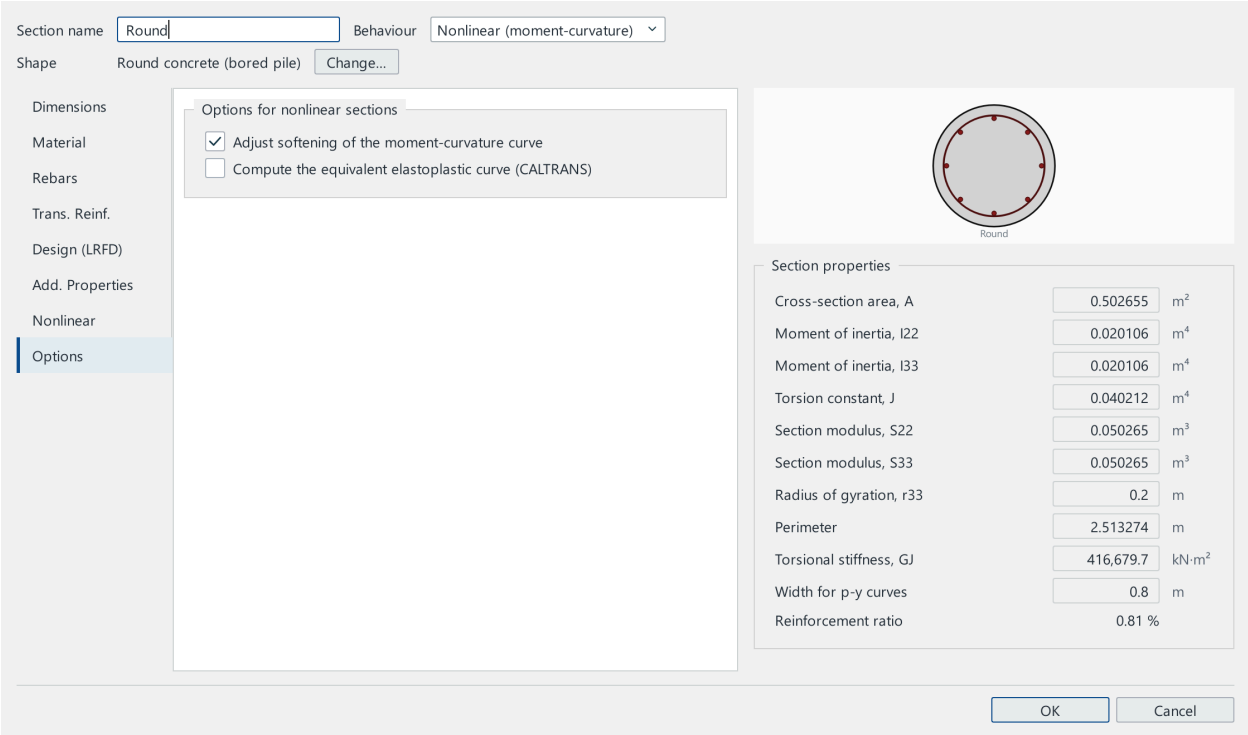


Figure 9.17 — Options tab

To define a bored pile section:

1. In Define Pile Sections click **Add New…**, choose the family *Concrete* and the shape *Round*, and click **OK**.
2. Type the name and the outside diameter in the Dimensions tab.
3. Select the concrete and the rebar materials in the Material tab.
4. Choose the bar size, the pattern, the number of bars and the cover in the Rebars tab, and the transverse reinforcement in the Trans. Reinf. tab.
5. Choose how the axial thrusts are set in the Nonlinear tab, and check the curves with **Show moment-curvature…**.
6. Click **Compute moment capacity from M-φ** in the Design (LRFD) tab if the demand/capacity ratios are wanted, and click **OK**.

9.4 Cap sections

Define ▸ Section Properties ▸ Cap Sections… lists the cap sections with their Cap Section, Thickness and Material columns. The cap section of the model is selected with **Assign ▸ Cap ▸ Cap Section…**; whether the cap is rigid or flexible is set for the whole cap with **Assign ▸ Cap ▸ Cap Behaviour…** (Chapter 10).

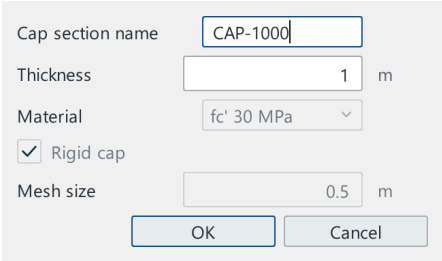


Figure 9.18 — Cap Section Data

Table 9.10 — Cap Section Data

Field	Unit	Description
Cap section name		name of the section

© 2026 Solplan. All rights reserved.

Field	Unit	Description
Thickness	length	thickness of the cap, greater than zero. It is the thickness of the plate elements of a flexible cap, and the default height of the embedded part of a cap whose top elevation is not entered (Chapter 10).
Material		material of the plate elements of a flexible cap; required for a flexible cap and dimmed for a rigid cap
Rigid cap		shows the behaviour of the cap. When the form is opened from the model it is dimmed: the behaviour is set for the whole cap with Assign ▸ Cap ▸ Cap Behaviour.
Mesh size	length	target size of the plate elements when a flexible cap mesh is generated; greater than zero for a flexible cap and dimmed for a rigid cap

A flexible cap whose mesh has been generated keeps the mesh size, the thickness and the material it was generated with: after changing them, regenerate the mesh with **Assign ▸ Cap ▸ Generate Flexible Cap Mesh**. A note in the form says so. A mesh read from another file carries its own plate thickness and material, and the cap section is used only if the mesh is regenerated.

9.5 Soil layers

Define ▸ Soil ▸ Soil Layers… lists the soil layers of the model, with the Layer name and the p-y curve of each. A soil layer is a material without a position: it describes the soil, and the soil profile places it between two elevations (Section 9.6). The same layer may be used more than once in the profile. The data form, *Soil Layer Data* (Figure 9.19), contains the common fields below and the parameters of the chosen p-y curve, described model by model in the following sections.

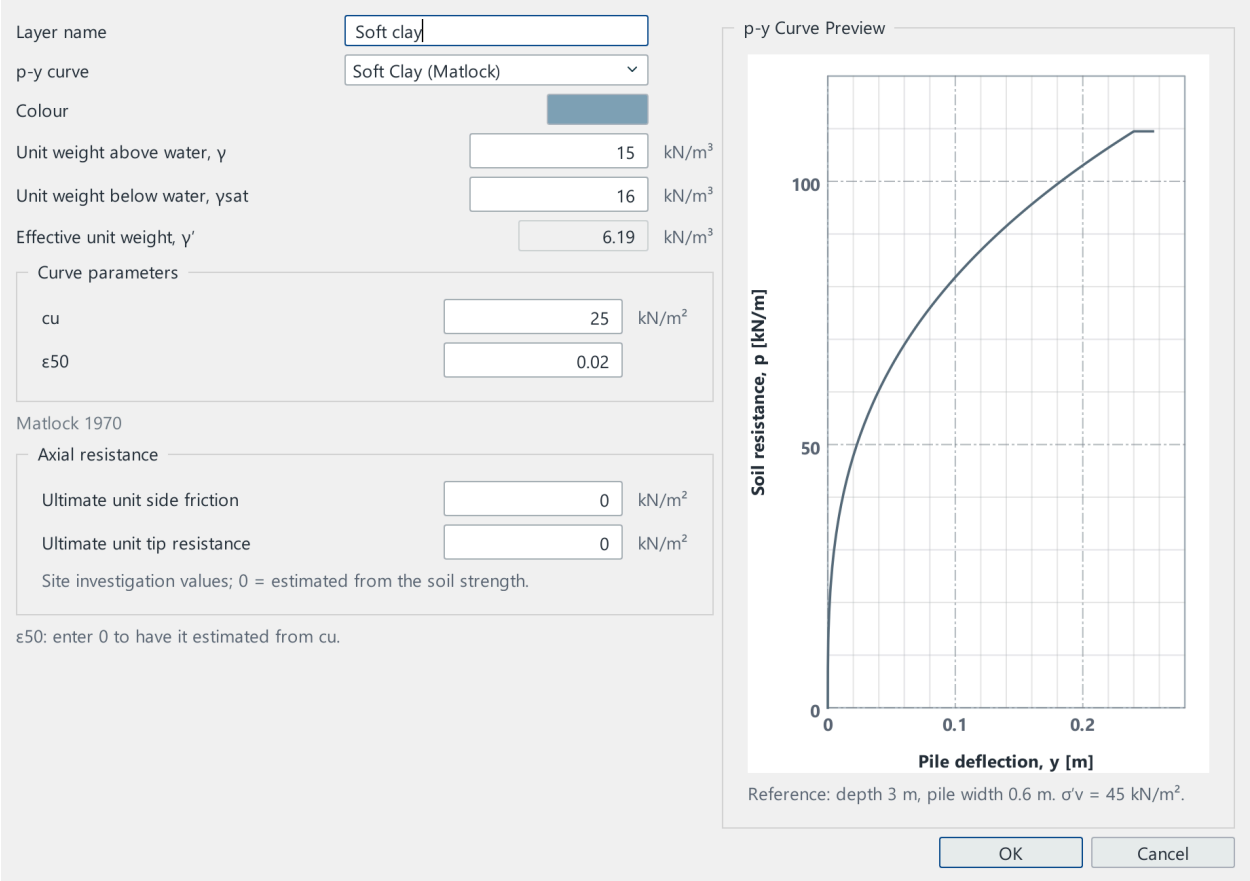


Figure 9.19 — Soil Layer Data with the preview of the p-y curve

© 2026 Solplan. All rights reserved.

Table 9.11 — Soil Layer Data, common fields

Field	Unit	Description
Layer name		name used by the soil profile
p-y curve		the lateral soil model of the layer (Table 9.12). Changing the model keeps the parameters typed for each model during the session, so switching back restores them.
Colour		display colour of the layer in the profile, the views and the reports
Unit weight above water, γ	force/volume	total unit weight above the water table. In a profile without a water table it is used at every depth as the effective unit weight.
Unit weight below water, γsat	force/volume	total unit weight below the water table; used only where the profile has a water table
Effective unit weight, γ′	force/volume	computed, γsat − γw; never entered
Curve parameters		the parameters of the chosen model, with the publication on which the model is based written under them; a note states which values may be entered as zero to have them estimated
Ultimate unit side friction; Ultimate unit tip resistance	stress	site investigation values for the axial load-transfer curves of the piles in this layer; 0 = estimated from the soil strength (Technical Reference, chapter “Soil response for axial loading”)

The **p-y Curve Preview** draws the curve that the analysis builds for the layer, at the reference depth, pile width and vertical effective stress stated under the graph, so that the effect of a parameter can be seen while it is typed. The preview is a check of the input; the curves of the analysis are built at every node with the actual depth, width and stress.

Table 9.12 — Lateral soil models (p-y curves)

Model	Parameters	Reference
Soft Clay (Matlock)	c_u, ε_{50}	Matlock (1970)
Soft Clay (API), User J	c_u, ε_{50}, J	Matlock (1970); API RP 2A
Mod. Soft Clay with Initial k	c_u, ε_{50}, k, J	Matlock (1970)
Stiff Clay with Free Water	c_u, ε_{50}, k	Reese, Cox & Koop (1975)
Stiff Clay without Free Water	c_u, ε_{50}	Welch & Reese (1972)
Stiff Clay with User k	c_u, ε_{50}, k	Welch & Reese (1972)
Sand (Reese)	φ', k	Reese, Cox & Koop (1974)
API Sand (O’Neill)	φ', k	O’Neill & Murchison (1983)
Liquefied Sand (Rollins)	unit weight only	Rollins et al. (2005)
Liquefied Sand Hybrid	$(N_1)_{60cs}$, S_{ur}, ε_{50}	Franke & Rollins (2013)
Silt / c-phi Soil (Cemented Sand)	c, φ', ε_{50}, k	Reese & Van Impe (2011)
Weak Rock	q_u, E_{ir}, k_{rm}, RQD	Reese (1997)
Strong Rock (Vuggy Limestone)	q_u	Reese & Nyman (1978)
Piedmont Residual	field test and its value: N_{60}, q_c, E_D or E_M	Simpson & Brown (2006)
Massive Rock	q_u, m_i, v, GSI, E_i or E_m, ε_{50}	Liang, Yang & Nusairat (2009)
Loess	q_c	Johnson et al. (2006)
Elastic Subgrade	E_s, σ_{max}	linear spring
User-defined p-y	none (curves of the p-y set of the pile)	Section 9.7

NOTE A value of 0 for ε_{50} or k lets the program estimate it from the strength of the layer, as the note under the parameters states. Fields that the chosen model does not use are not shown.

© 2026 Solplan. All rights reserved.

To define a soil layer:

1. Choose **Define** ▸ **Soil** ▸ **Soil Layers…** and click **Add New…**.
2. Type the name, choose the p-y curve and enter the unit weights.
3. Enter the parameters of the model; check the curve in the preview.
4. Enter the ultimate unit side friction and tip resistance if they are known from the site investigation, or leave them at zero, and click **OK**.

The following sections give, for each p-y curve, the parameters asked for in *Soil Layer Data*, with their units in the SI system, their default values and their meaning, and the notes that the form shows. The equations of the models are given in the Technical Reference, chapter “Soil response: p-y curves”.

9.5.1 Soft Clay (Matlock)

Reference: Matlock 1970.

Table 9.13 — Parameters of Soft Clay (Matlock)

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu

ε50: enter 0 to have it estimated from cu.

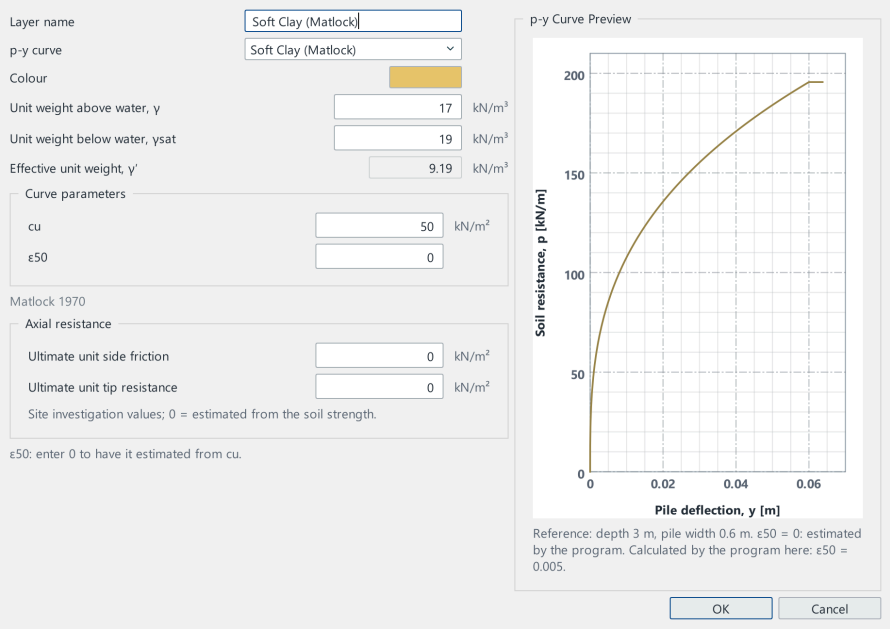


Figure 9.20 — Soil Layer Data, Soft Clay (Matlock)

9.5.2 Soft Clay (API), User J

Reference: Matlock 1970 / API RP-2A, J entered by the user.

Table 9.14 — Parameters of Soft Clay (API), User J

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu
J		0.25	empirical constant in Np = 3 + sigma'v/cu + J z/b; Matlock publishes 0.5 for soft clay

ε50: enter 0 to have it estimated from cu. J: values from 0.25 to 0.50 are usual.

© 2026 Solplan. All rights reserved.

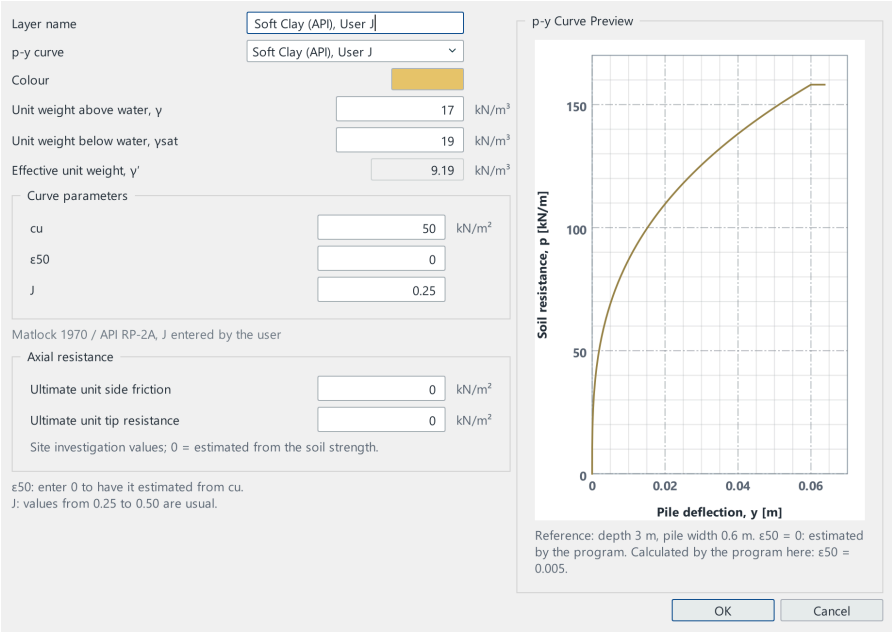


Figure 9.21 — Soil Layer Data, Soft Clay (API), User J

9.5.3 Mod. Soft Clay with Initial k

Reference: Matlock 1970 with an initial straight line of slope k times depth.

Table 9.15 — Parameters of Mod. Soft Clay with Initial k

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'
J		0.5	empirical constant in Np = 3 + sigma'v/cu + J z/b; Matlock publishes 0.5 for soft clay

k and ε50: enter 0 to have them estimated from cu. J: values from 0.25 to 0.50 are usual. J = 0.50 suits soft clay.

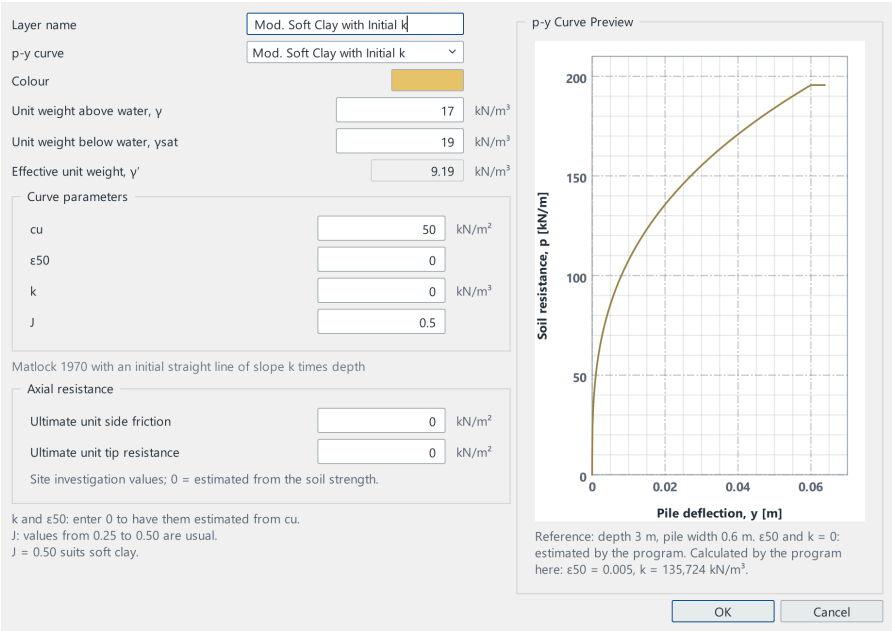


Figure 9.22 — Soil Layer Data, Mod. Soft Clay with Initial k

© 2026 Solplan. All rights reserved.

9.5.4 Stiff Clay with Free Water

Reference: Reese, Cox & Koop 1975.

Table 9.16 — Parameters of Stiff Clay with Free Water

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'

k and ε50: enter 0 to have them estimated from cu.

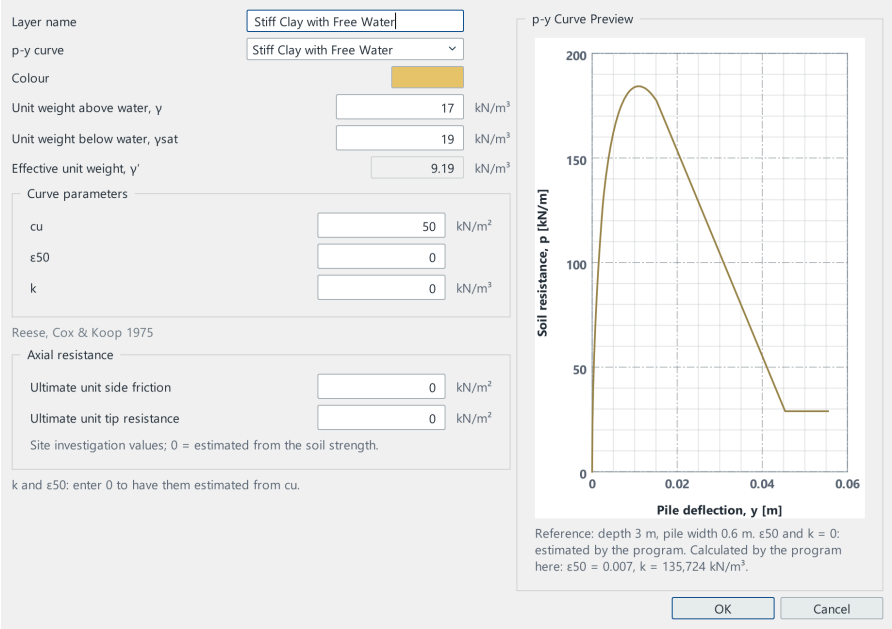


Figure 9.23 — Soil Layer Data, Stiff Clay with Free Water

9.5.5 Stiff Clay without Free Water

Reference: Welch & Reese 1972.

Table 9.17 — Parameters of Stiff Clay without Free Water

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu

ε50: enter 0 to have it estimated from cu.

© 2026 Solplan. All rights reserved.

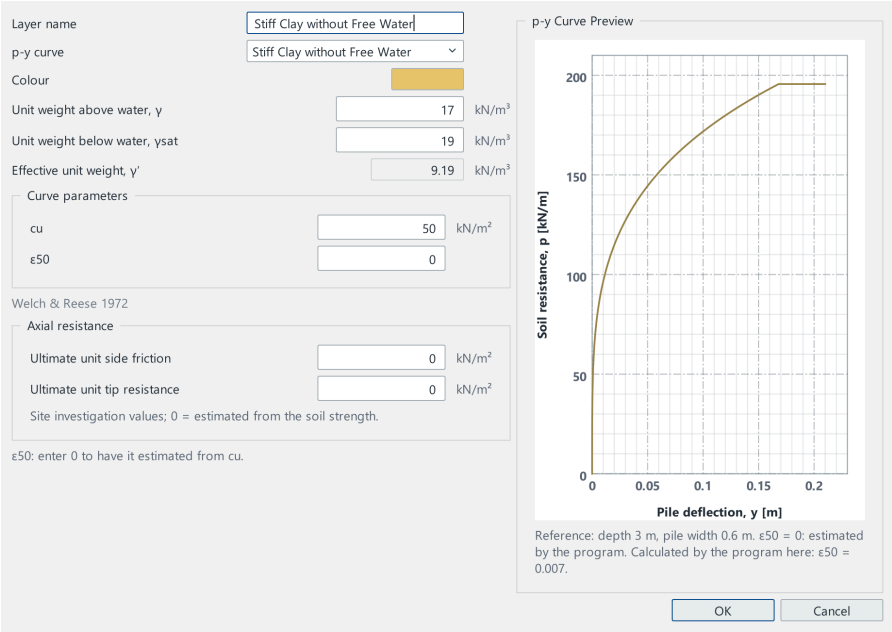


Figure 9.24 — Soil Layer Data, Stiff Clay without Free Water

9.5.6 Stiff Clay with User k

Reference: variant with user-specified k.

Table 9.18 — Parameters of Stiff Clay with User k

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'

Like the stiff clay model without free water, but k sets the initial slope of the p-y curve. k should come from a lateral load test in the field. k and ε50: enter 0 to have them estimated from cu.

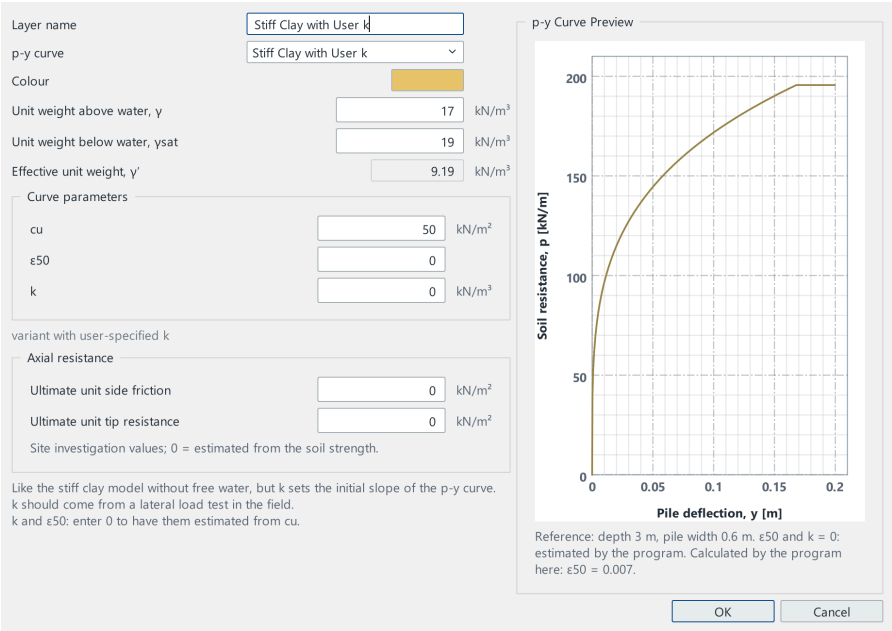


Figure 9.25 — Soil Layer Data, Stiff Clay with User k

© 2026 Solplan. All rights reserved.

9.5.7 Sand (Reese)

Reference: Reese, Cox & Koop 1974.

Table 9.19 — Parameters of Sand (Reese)

Parameter	Unit	Default	Description
φ'	°	30	effective friction angle
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'

k: enter 0 to have it estimated from φ'.

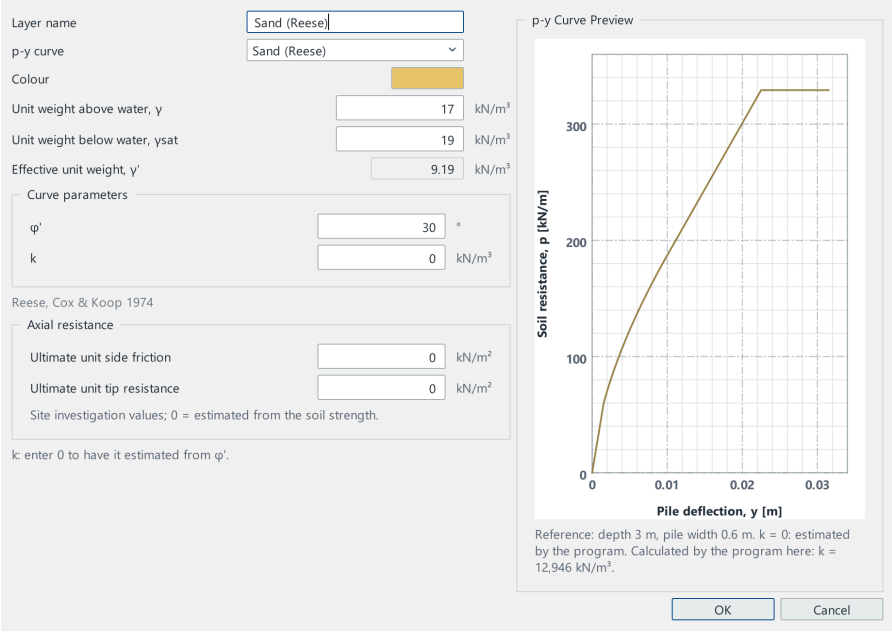


Figure 9.26 — Soil Layer Data, Sand (Reese)

9.5.8 API Sand (O'Neill)

Reference: O'Neill & Murchison 1983 / API RP-2A.

Table 9.20 — Parameters of API Sand (O'Neill)

Parameter	Unit	Default	Description
φ'	°	30	effective friction angle
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'

k: enter 0 to have it estimated from φ'.

© 2026 Solplan. All rights reserved.

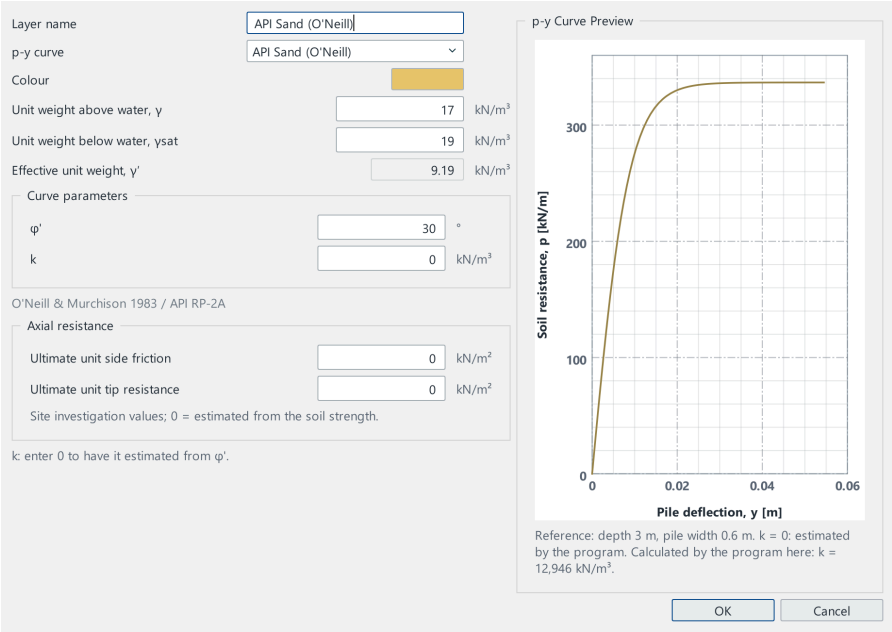


Figure 9.27 — Soil Layer Data, API Sand (O'Neill)

9.5.9 Liquefied Sand (Rollins)

Reference: Rollins et al. 2005.

The model has no parameters other than the unit weights.

For soil liquefied by earthquake shaking. Only the effective unit weight is used; no friction angle is needed. The model was calibrated for relative densities of about 45 to 55 percent.

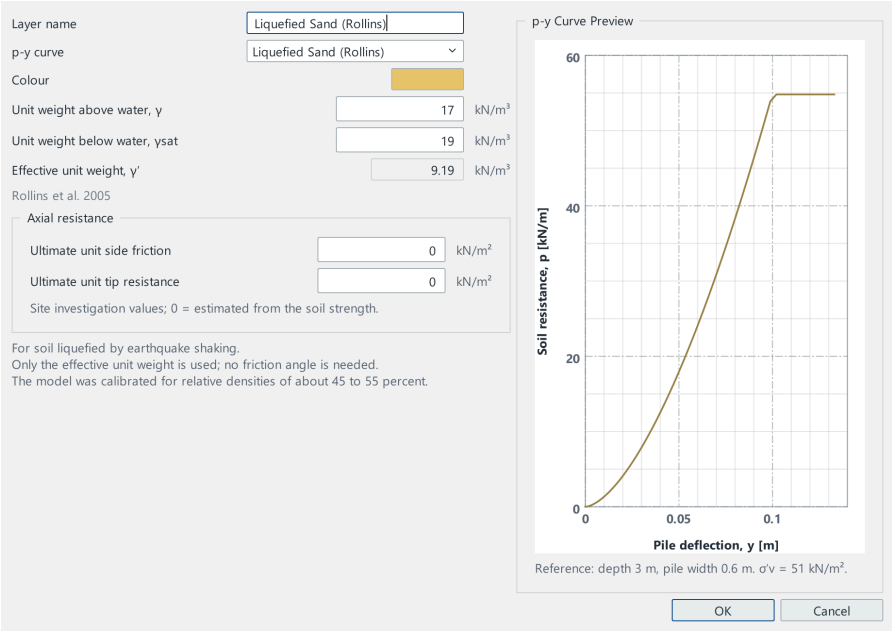


Figure 9.28 — Soil Layer Data, Liquefied Sand (Rollins)

9.5.10 Liquefied Sand Hybrid

Reference: Franke & Rollins 2013: lower of the dilative (Rollins et al. 2005) and residual (Matlock) curves.

Table 9.21 — Parameters of Liquefied Sand Hybrid

Parameter	Unit	Default	Description
(N1)60cs		10	corrected clean-sand SPT blow count; gives the residual strength and ε50 when those are 0

© 2026 Solplan. All rights reserved.

Parameter	Unit	Default	Description
Sur	kN/m²	0	residual shear strength of the liquefied sand; 0 = from (N1)60cs (33rd percentile; zero below 5 blows, which leaves the layer without lateral resistance)
ε50		0	strain at 50 percent of the residual strength; 0 = from (N1)60cs

Calibrated for sand with a relative density of about 45 to 55 percent, to depths of about 6 m (19.7 ft). Option 1: enter the blow count (N1)60cs and 0 for the residual strength; the residual strength and ε50 are then computed from the blow count. Option 2: enter the residual strength and ε50 directly. ε50 usually ranges from 0.0005 for stronger soil to 0.05 for weaker soil.

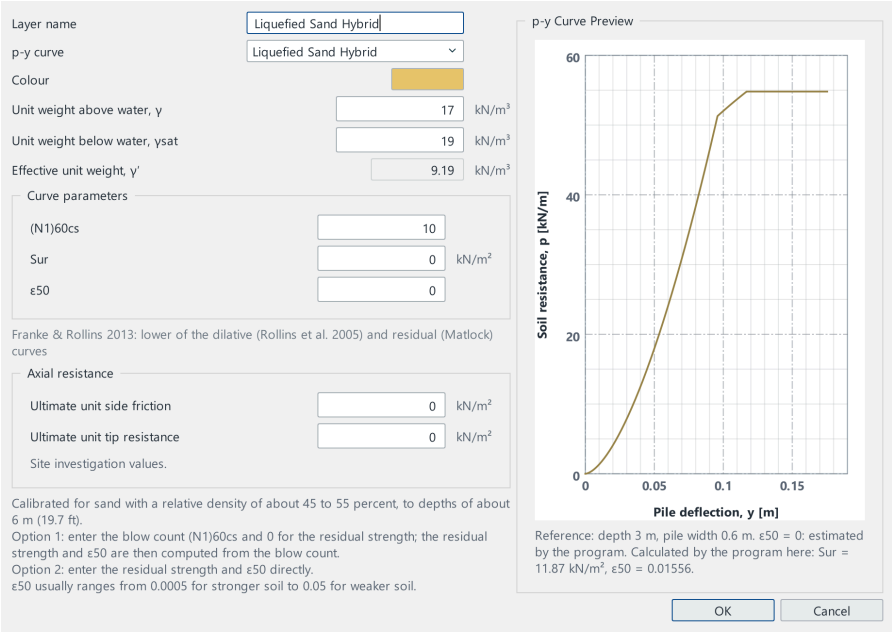


Figure 9.29 — Soil Layer Data, Liquefied Sand Hybrid

9.5.11 Silt / c-phi Soil (Cemented Sand)

Reference: Reese & Van Impe 2011.

Table 9.22 — Parameters of Silt / c-phi Soil (Cemented Sand)

Parameter	Unit	Default	Description
cu	kN/m²	50	undrained shear strength; about half the unconfined compressive strength qu
φ'	°	30	effective friction angle
ε50		0	strain at 50 percent of the maximum stress; 0 = estimated from cu
k	kN/m³	0	subgrade reaction modulus; Es = k·x; 0 = estimated from cu or φ'

Both the cohesion and the friction angle must be non-zero.

© 2026 Solplan. All rights reserved.

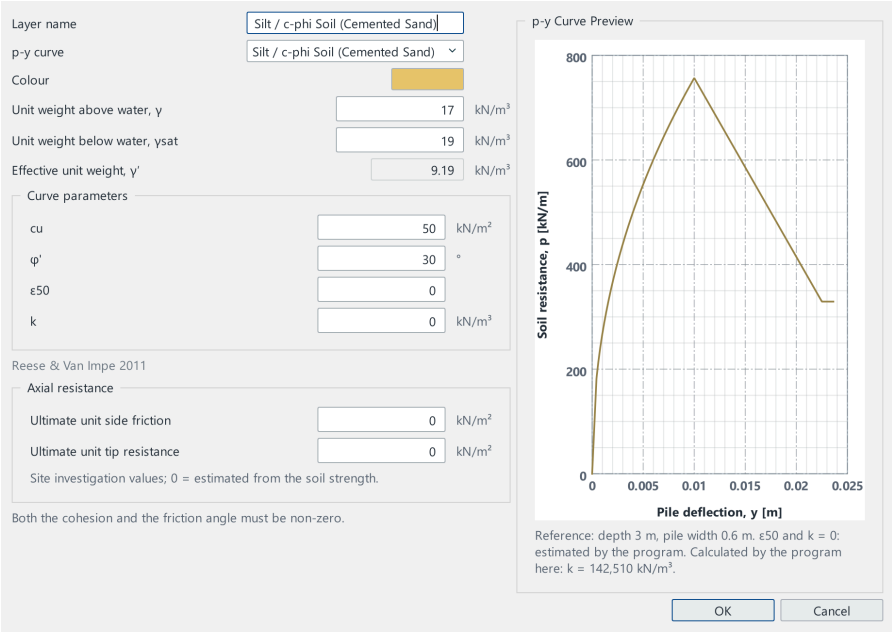


Figure 9.30 — Soil Layer Data, Silt / c-phi Soil (Cemented Sand)

9.5.12 Weak Rock

Reference: Reese 1997.

Table 9.23 — Parameters of Weak Rock

Parameter	Unit	Default	Description
qu	kN/m²	1,000	uniaxial compressive strength of the rock
Eir	kN/m²	500,000	initial rock mass modulus
krm		0.0005	strain factor for weak rock; 0 = linear p-y curve (no krm), and a pile tip in this layer then has no tip curve and cannot be analysed
RQD	%	50	rock quality designation

Intended for cohesive intermediate geomaterials such as claystone and siltstone, with a uniaxial compressive strength of about 500 to 5,000 kPa (75 to 750 psi).

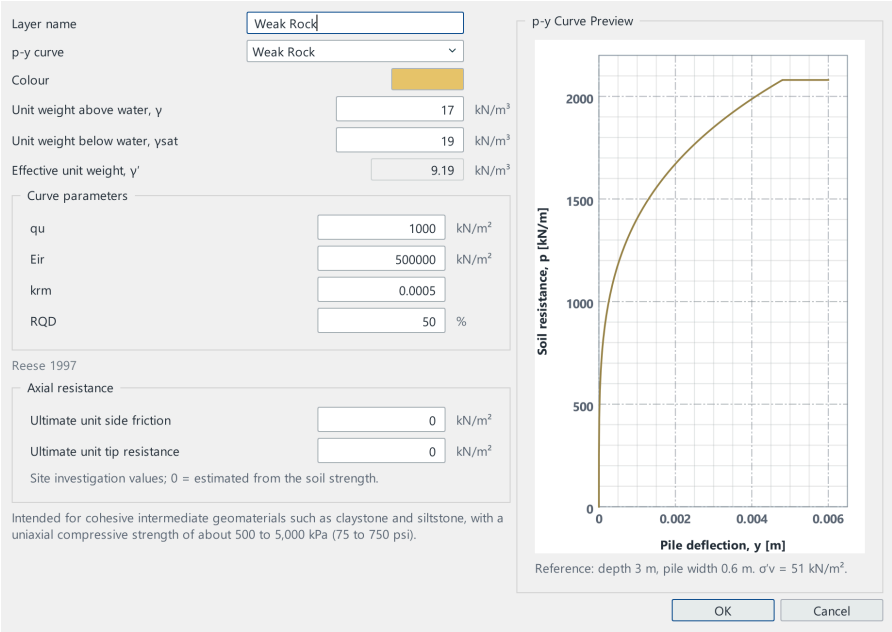


Figure 9.31 — Soil Layer Data, Weak Rock

© 2026 Solplan. All rights reserved.

9.5.13 Strong Rock (Vuggy Limestone)

Reference: Reese & Nyman 1978.

Table 9.24 — Parameters of Strong Rock (Vuggy Limestone)

Parameter	Unit	Default	Description
qu	kN/m²	1,000	uniaxial compressive strength of the rock

The effective unit weight of this layer sets the vertical effective stress in the layers below it. Intended for limestone with a uniaxial compressive strength of about 5,000 to 16,700 kPa (750 to 2,500 psi).

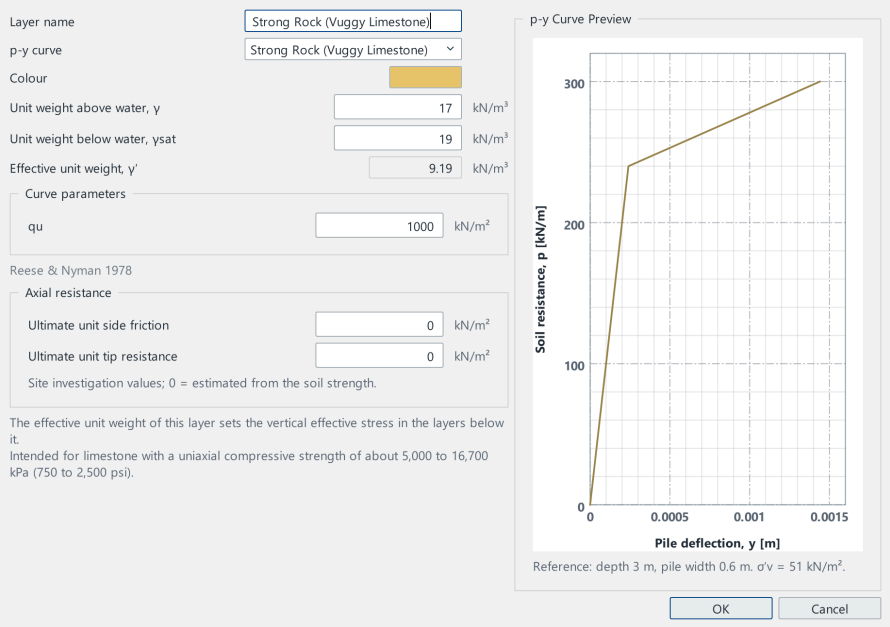


Figure 9.32 — Soil Layer Data, Strong Rock (Vuggy Limestone)

9.5.14 Piedmont Residual

Reference: Simpson & Brown 2006.

Table 9.25 — Parameters of Piedmont Residual

Parameter	Unit	Default	Description
Field test		1	field test that gives the initial p-y modulus Esi
N60		20	SPT blow count; Esi = 22 N60 lb/in³
qc	kN/m²	0	cone tip resistance; Esi = 0.8136 qc per inch
ED	kN/m²	0	dilatometer modulus; Esi = 0.076 ED per inch
EM	kN/m²	0	Menard pressuremeter modulus; Esi = 0.235 EM per inch

Only the value of the selected field test is used; the other test values are ignored.

© 2026 Solplan. All rights reserved.

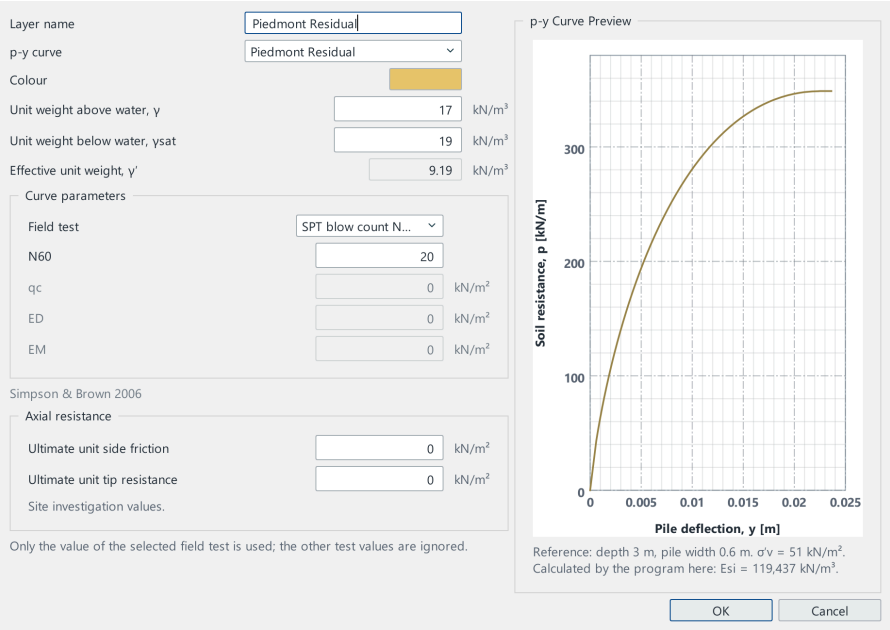


Figure 9.33 — Soil Layer Data, Piedmont Residual

9.5.15 Massive Rock

Reference: Liang, Yang & Nusairat 2009 (Hoek-Brown).

Table 9.26 — Parameters of Massive Rock

Parameter	Unit	Default	Description
qu	kN/m²	20,000	uniaxial compressive strength of the intact rock
mi		10	Hoek-Brown material index of the rock type (claystone 4 ... granite 32)
v		0.25	Poisson's ratio of the rock mass
GSI		50	geological strength index
Ei	kN/m²	5,000,000	intact rock modulus (option 1); used when the rock mass modulus is 0: Em = Ei/100 e^(GSI/21.7)
Em	kN/m²	0	rock mass modulus (option 2); 0 = from Ei and GSI
ε50		0	strain factor used only by the tip load-settlement curve of a driven pile ending in this layer; 0 = none, and such a pile cannot be analysed

Em: enter 0 to compute it from Ei and GSI; a non-zero Em is used as entered and Ei is ignored.

© 2026 Solplan. All rights reserved.

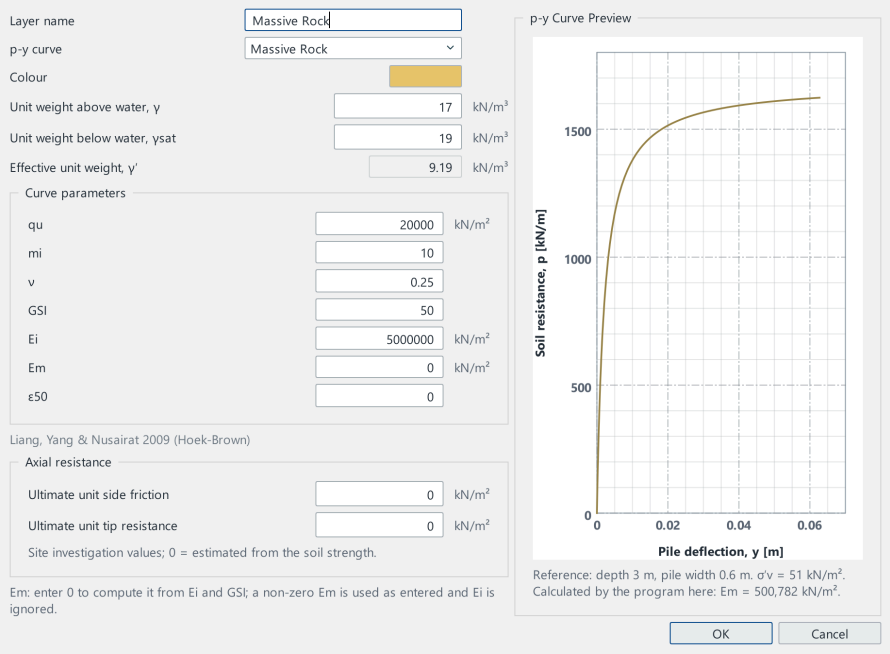


Figure 9.34 — Soil Layer Data, Massive Rock

9.5.16 Loess

Reference: Johnson, Parsons, Dapp & Brown 2006.

Table 9.27 — Parameters of Loess

Parameter	Unit	Default	Description
qc	kN/m²	3,000	cone tip resistance

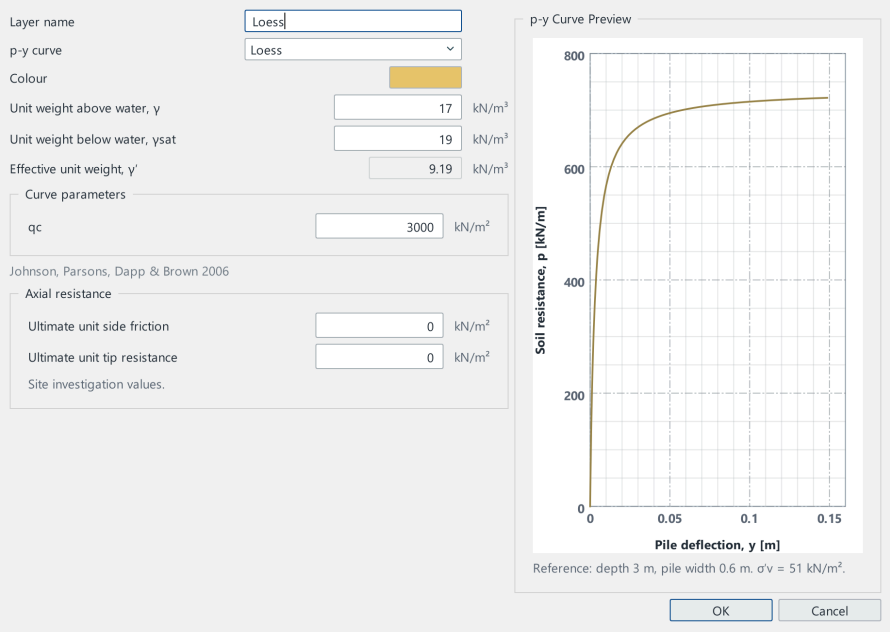


Figure 9.35 — Soil Layer Data, Loess

9.5.17 Elastic Subgrade

Reference: linear spring.

Table 9.28 — Parameters of Elastic Subgrade

Parameter	Unit	Default	Description

© 2026 Solplan. All rights reserved.

Parameter	Unit	Default	Description
Es	kN/m³	20,000	constant subgrade reaction modulus
σmax	kN/m²	0	maximum stress transfer; p is limited to b times this value; 0 = no limit on stress transfer

The effective unit weight of this layer sets the vertical effective stress in the layers below it. Maximum stress transfer: 0 means no limit.

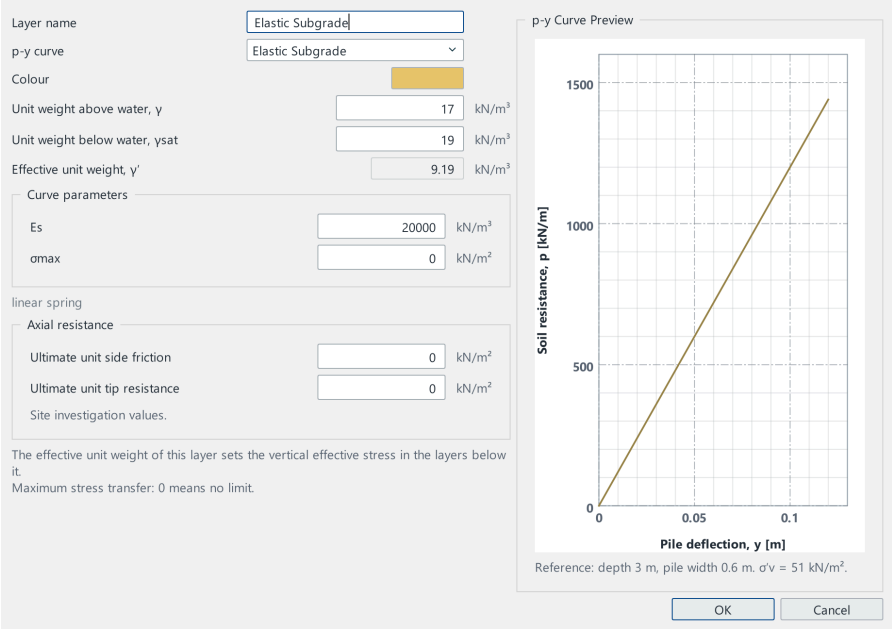


Figure 9.36 — Soil Layer Data, Elastic Subgrade

9.5.18 User-defined p-y

Reference: y-p table per depth, entered in the User Curves tab.

The model has no parameters other than the unit weights.

The effective unit weight of this layer sets the vertical effective stress in the layers below it. The lateral resistance of this layer comes from user-defined p-y curves. Type the p-y curves under Define > Soil > User p-y Curves, tick User-defined p-y curves in Analyze > Set Analysis Options and assign the set to the piles (Assign > Pile > p-y Set).

© 2026 Solplan. All rights reserved.

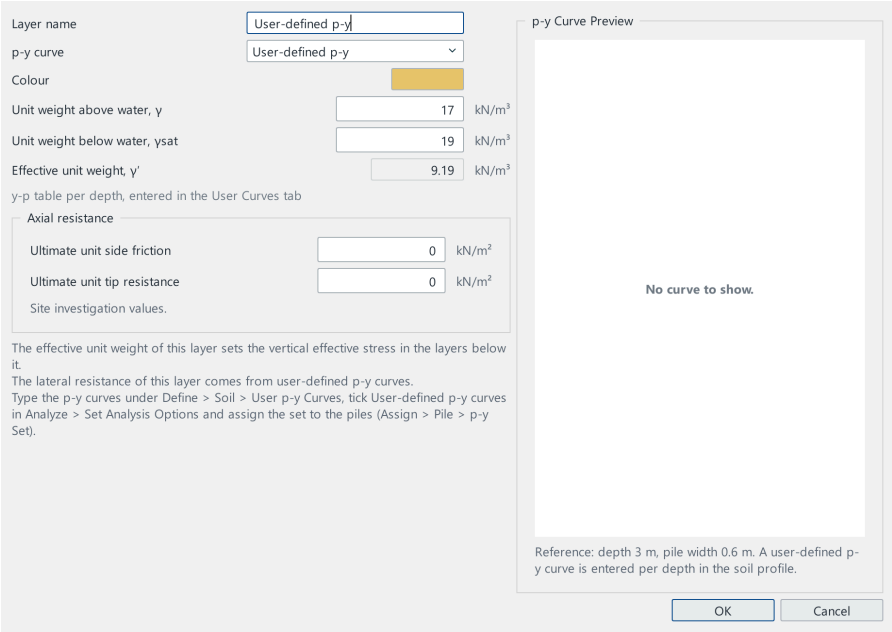


Figure 9.37 — Soil Layer Data, User-defined p-y

9.6 Soil profile

Define ▸ **Soil** ▸ **Soil Profile…** opens *Soil Profile Data* (Figure 9.38), which stacks the soil layers from the ground surface downward. The drawing on the right shows the stack to scale with the colours of the layers, the water table and the scoured zone. Every pile uses this profile. The ground surface is level, or it slopes when a shape is set with **Sloping…** (Section 9.6.1); a ground surface elevation assigned to a pile overrides both (Chapter 10).

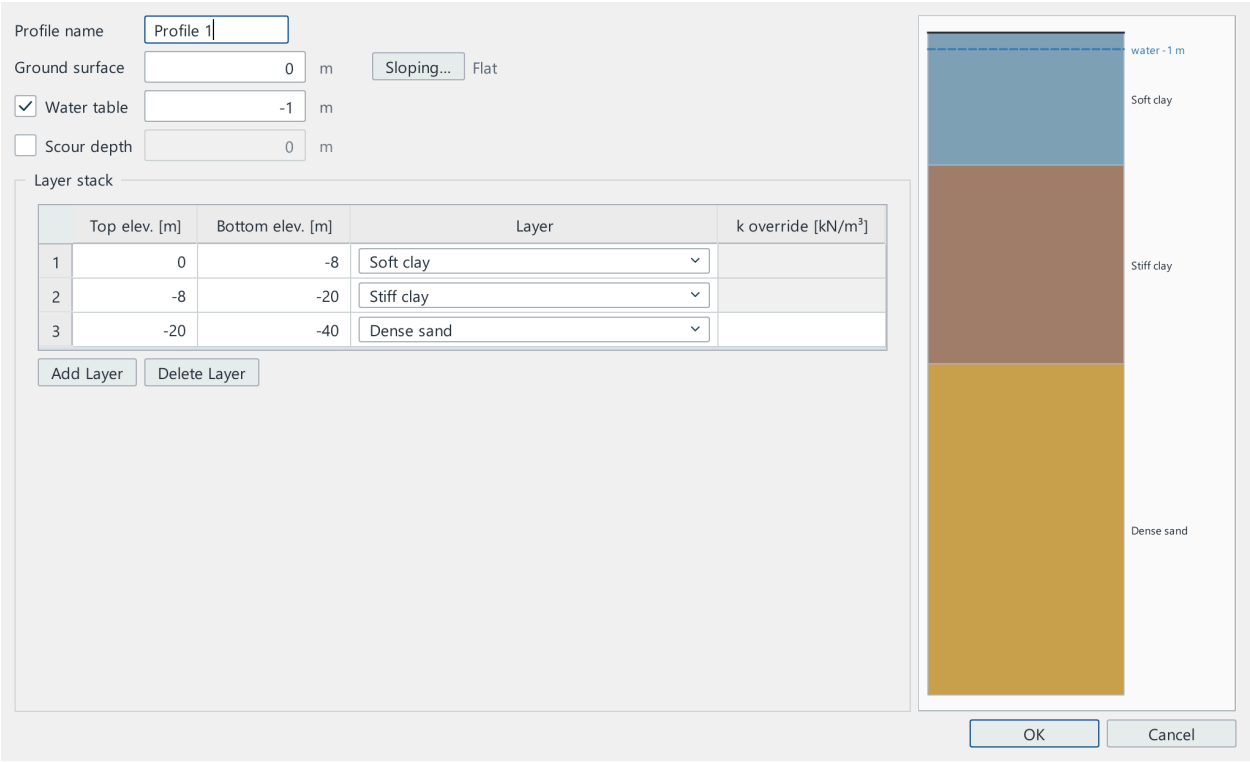


Figure 9.38 — Soil Profile Data

Table 9.29 — Soil Profile Data

Field	Unit	Description

© 2026 Solplan. All rights reserved.

Field	Unit	Description
Profile name		name of the profile
Ground surface	length	elevation of the ground surface in the global system
Water table	length	ticked: the elevation of the water table; below it the unit weight below water of each layer is used. Unticked: a profile without a water table, in which the unit weight above water of each layer is the effective unit weight at every depth.
Scour depth	length	ticked: the depth of soil removed by scour below the ground surface. The removed soil gives no lateral resistance; the effective ground surface, ground surface minus scour depth, is shown.
Layer stack		one row per layer: **Top elev.** and **Bottom elev.** of the layer, the soil **Layer** chosen from the soil layers, and **k override**, an optional p-y modulus *k* for this row. A blank override uses the value in Soil Layer Data; the override lets a shared layer use a different *k* here, and it is accepted only for models that use *k*. **Add Layer** adds a row under the last one; **Delete Layer** removes the selected row.
Sloping…		opens *Sloping Ground* (Section 9.6.1) to give the ground surface of this profile a slope; the line beside the button summarises the current shape

The profile needs at least one layer; every row needs its top above its bottom and a soil layer, and the layers must be contiguous: a gap between the bottom of one layer and the top of the next is reported with its elevations. The last layer should extend below the tips of the longest piles.

9.6.1 Sloping ground

Sloping… in *Soil Profile Data* opens *Sloping Ground* (Figure 9.39). The ground surface changes in one direction in plan: along a distance axis it is described by break points of distance and elevation, linear between two points. The soil layers keep their elevations, so on the lower side of a slope the upper layers are cut by the ground surface. The section on the right shows the ground, the layers, the water table and every pile, with the point where its axis meets the ground, before the shape is accepted.

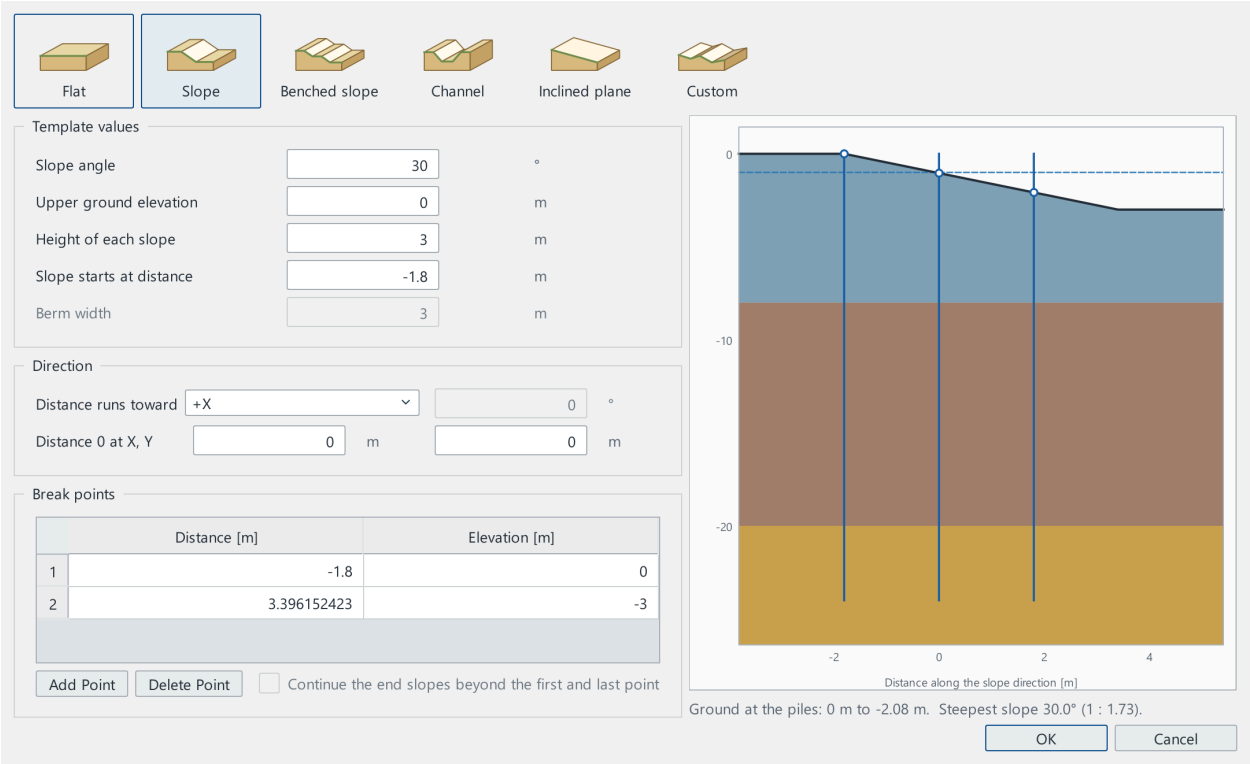


Figure 9.39 — Sloping Ground, template Slope

© 2026 Solplan. All rights reserved.

Table 9.30 — Sloping Ground

Field	Unit	Description
Templates		**Flat** (level ground at the ground surface of the profile), **Slope** (level, one slope down, level again), **Benched slope** (two slopes with a level berm between them), **Channel** (a slope down to a level bed, then up again), **Inclined plane** (one constant slope across the model) and **Custom** (the break points are typed). A template fills the table of points; the points can then be edited, which makes the shape Custom. Only the fields that the chosen template uses are active.
Slope angle	°	angle of each slope from the horizontal, 0.5° to 89.5°
Upper ground elevation	length	elevation of the ground before the first slope
Height of each slope	length	difference of elevation over one slope
Slope starts at distance	length	distance along the axis where the upper ground starts to fall; when the dialog opens it is the first row of piles
Berm width	length	width of the level part between two slopes (Benched slope) or of the bed (Channel)
Distance runs toward	°	direction in plan in which the distance increases: +X, +Y, −X, −Y, or Other angle with the plan angle counter-clockwise from +X. The templates put the upper ground at the start and fall toward this side.
Distance 0 at X, Y	length	point in plan where the distance is zero
Points	length	table of break points, **Distance** and **Elevation**, with the distances increasing; **Add Point** and **Delete Point** edit it
Continue the end slopes beyond the first and last point		unticked: the ground is level beyond the first and the last point; ticked: the end slopes continue

The line under the table summarises the shape, for example *Level ground at the soil profile ground surface*. The ground surface of each pile is the point where the axis of the pile meets the sloping surface, so a battered pile takes the ground where it actually enters the soil. Next to a steep slope, a head above the ground takes the first point below the head, and a buried head takes the nearest point above it. The analysis treats the shape exactly as a ground surface elevation assigned to each pile; such an assignment is an exception that overrides the shape for the selected piles (Chapter 10), and the piles that carry one are drawn with a ring. Scour is measured from the ground of each pile. The table *Ground Surface* of **Display ▸ Show Tables** and section F of the output report list the ground of every pile.

In the soil that remains above the tip of a pile, the layers above the ground of that pile give no resistance, and the curves of the layers below are computed from the ground of that pile: the effective stress, and the depth used by the curves for sand, weak rock and liquefied sand, start at the ground of the pile, not at the ground surface of the profile. Check Model reports an error when the distances do not increase, when a value is not a number, or when the lowest point of the ground, less the scour depth, is below the bottom of the profile; a shape with a single point is not used (the ground is level) and is reported as a warning.

9.7 User-defined curves

Three kinds of curves can be entered instead of the curves generated from the soil. Each is used only when the corresponding option of **Analyze ▸ Set Analysis Options…** is ticked, and only by the piles to which it is assigned; the other piles use the curves generated from the soil.

9.7.1 Axial (L-D) curves

Define ▸ Soil ▸ Axial (L-D) Curves… lists the axial load–displacement curves, with the name (Axial Curve) and the number of Points of each. An axial curve gives the load at the pile head against the axial displacement of the head, for example from a load test, and replaces the curve that the program integrates from the load-transfer curves along the pile. The form *Axial (L-D) Curve* (Figure 9.40) holds the **Curve name** and a table of points, **Displacement** and **Load**, with positive values for compression and negative values for tension; **Add Point** and **Delete Point** edit the table, and

© 2026 Solplan. All rights reserved.

the graph shows the curve. Piles without a curve use the load–displacement curve generated from the soil layers. The curve is assigned with **Assign ▸ Pile ▸ Axial (L-D) Curve…** and is used when the option *User-defined L-D curves* is ticked.

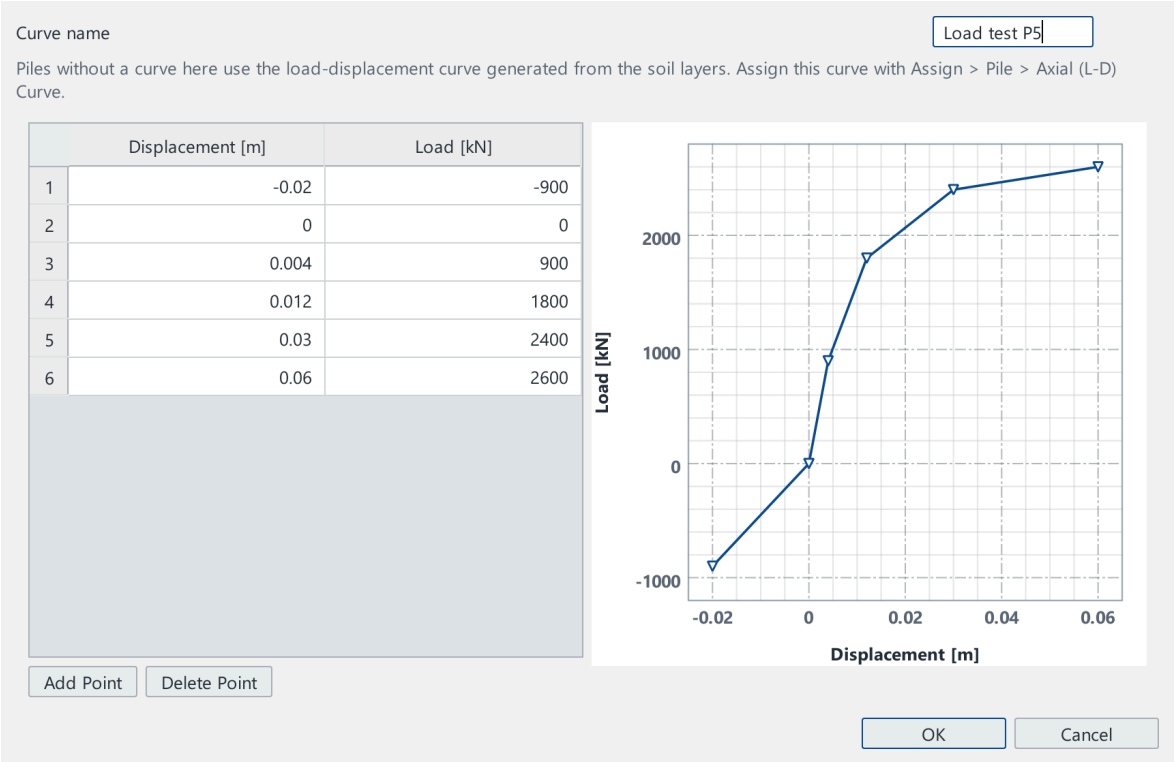


Figure 9.40 — Axial (L-D) Curve from a load test

9.7.2 Torsional (T-R) curves

Define ▸ Soil ▸ Torsional (T-R) Curves… lists the torque–rotation curves, with the name (Torsional Curve) and the number of Points of each. A torsional curve gives the torque at the pile head against the rotation of the head about the pile axis. The form *Torsional (T-R) Curve* holds the **Curve name** and the table of points, **Rotation** and **Torque**, edited with **Add Point** and **Delete Point**. Piles without a curve use the torque–rotation curve generated from the soil layers. The curve is assigned with **Assign ▸ Pile ▸ Torsional (T-R) Curve…** and is used when the option *User-defined T-R curves* is ticked.

© 2026 Solplan. All rights reserved.

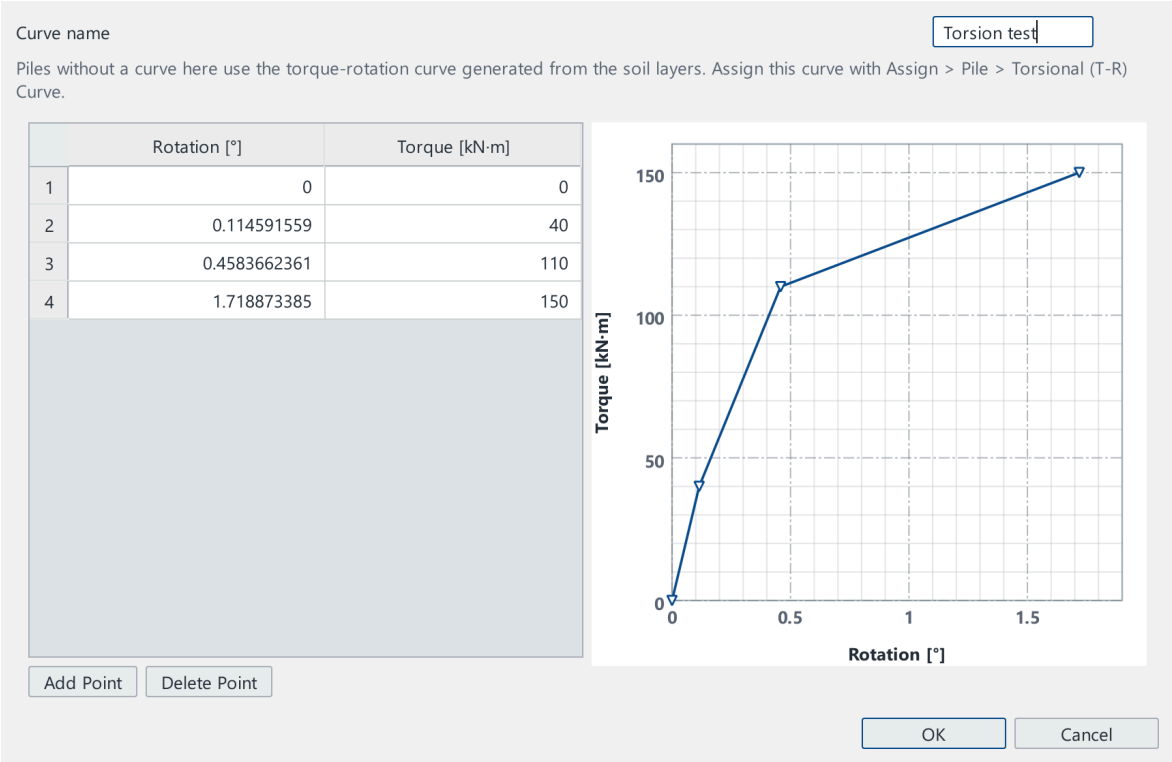


Figure 9.41 — Torsional (T-R) Curve

9.7.3 User p-y curves

Define ▸ Soil ▸ User p-y Curves… defines sets of p-y curves (Figure 9.42). The **p-y set** to edit is chosen in the list at the top; **Add Set** creates a set and **Delete Set** removes it. For each set:

- **Depth below origin**: the depths at which a curve is given, measured downward from the global elevation zero. Type a depth and click **Add Depth**; **Delete Depth** removes the selected depth.
- **Curve points**: the points of the curve at the selected depth, deflection **y** and soil resistance **p** per unit length of pile, edited with **Add Point** and **Delete Point**. The graph shows the curve.

Between two depths the curves are interpolated. A set must reach the pile tip. A set is assigned to piles with **Assign ▸ Pile ▸ p-y Set…** and is used when the option *User-defined p-y curves* is ticked; it then replaces the curves of the soil layers for those piles over the depth it covers. A soil layer whose p-y curve is *User-defined p-y* takes its resistance from the set of the pile (Section 9.5.18).

© 2026 Solplan. All rights reserved.

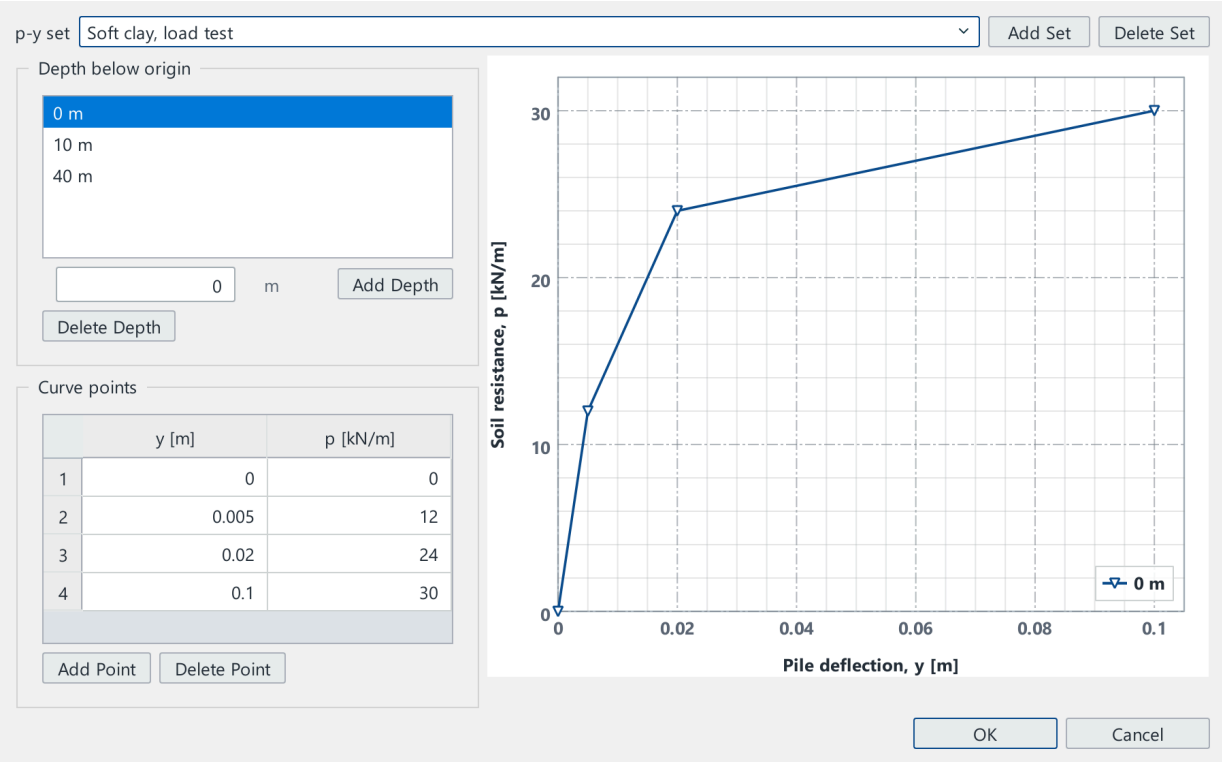


Figure 9.42 — Define User p-y Curves

9.8 Lateral group effect

Define ▸ **Soil** ▸ **Lateral Group Effect (p-y)…** defines sets of p-multipliers for closely spaced piles (Technical Reference, chapter “Group effects”). The list shows each Set with its Method and number of Rows. The set used by the whole model is chosen in the list under the table, either a set or *Ignore effects*, and confirmed with **Use selected set**; the same choice is offered in the analysis options, and a load case or a combination may use another set (Chapter 11).

© 2026 Solplan. All rights reserved.

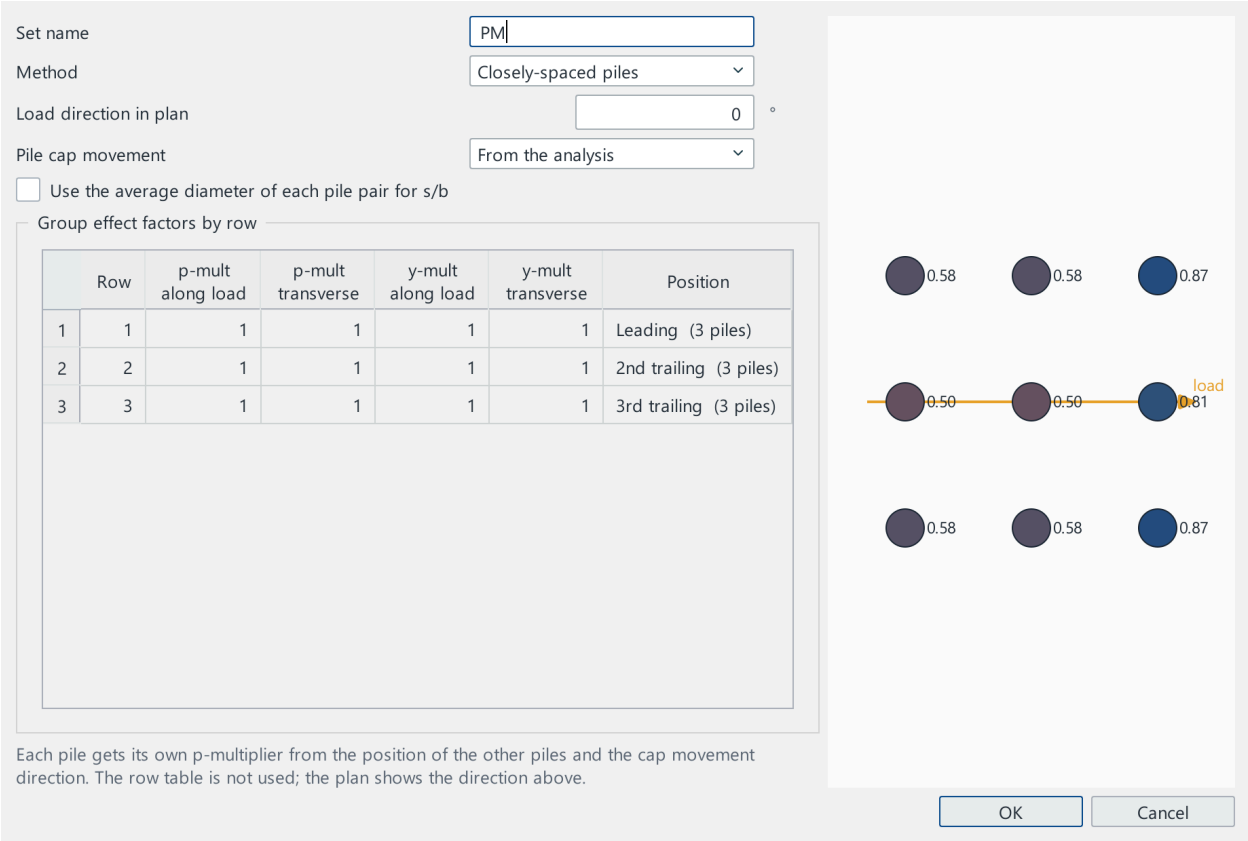


Figure 9.43 — Lateral Group Effect (p-y) with the method Closely-spaced piles

Table 9.31 — Lateral Group Effect (p-y)

Field	Description
Set name	name of the set
Method	*Manual*: the factors of each row are typed. *AASHTO LRFD Table 10.7.2.4-1*, *Reese Van Impe* and *User table*: factors by rows from a table against the spacing ratio *s*/*D*. *Closely-spaced piles*: a factor for each pile from the positions of all piles. *Per pile, x/y directions*: factors typed for each pile, for movement along X and along Y.
Load direction in plan	direction, measured from the X axis, that defines the leading and trailing rows of the row methods; for *Closely-spaced piles*, the reference direction of the cap movement
Pile cap movement	for *Closely-spaced piles*: *From the analysis*, the factors follow the direction in which the pile cap moves, updated until they settle; *Reference direction*, the load direction above is used as a fixed pile cap movement
Use the average diameter of each pile pair for s/b	off: every pair of piles uses the diameter of the first pile in the model
Group effect factors by row	one row per row of piles: **Row** number, **p-mult along load** and **p-mult transverse**, **y-mult along load** and **y-mult transverse**, and the **Position** of the row (leading, 2nd trailing, …) with its number of piles. **Compute from spacing** recalculates the p-multipliers from the reference table and the pile spacing; **Edit these factors manually** copies the factors to the Manual method, so that the analysis uses the edited values.

The plan on the right shows the piles with the factor of each pile and the load direction, so that the leading row can be checked. For *Closely-spaced piles* the row table is not used: each pile gets its own p-multiplier from the position of the other piles and the direction of the cap movement, as the note under the table states.

© 2026 Solplan. All rights reserved.

9.9 Lateral reduction sets

Define ▸ Soil ▸ Lateral Reduction Sets… lists sets of factors on the lateral soil resistance that vary along the pile, with the Set name, the number of Depths and their Range. The form *Lateral Reduction Set* holds the **Set name** and a table of **Depth**, **P-Factor** and **Y-Factor**, edited with **Add Depth** and **Delete Depth**. The soil resistance is multiplied by P-Factor / Y-Factor. Between two depths the factors are interpolated linearly; above the first depth and below the last there is no reduction. The depths are measured vertically below the global elevation zero; for piles with user-defined p-y curves they are measured along the pile from its head. A set is assigned with **Assign ▸ Pile ▸ Lateral Reduction Set…** and is used when the option *Lateral reduction along depth* is ticked. The factors act together with the group factors.

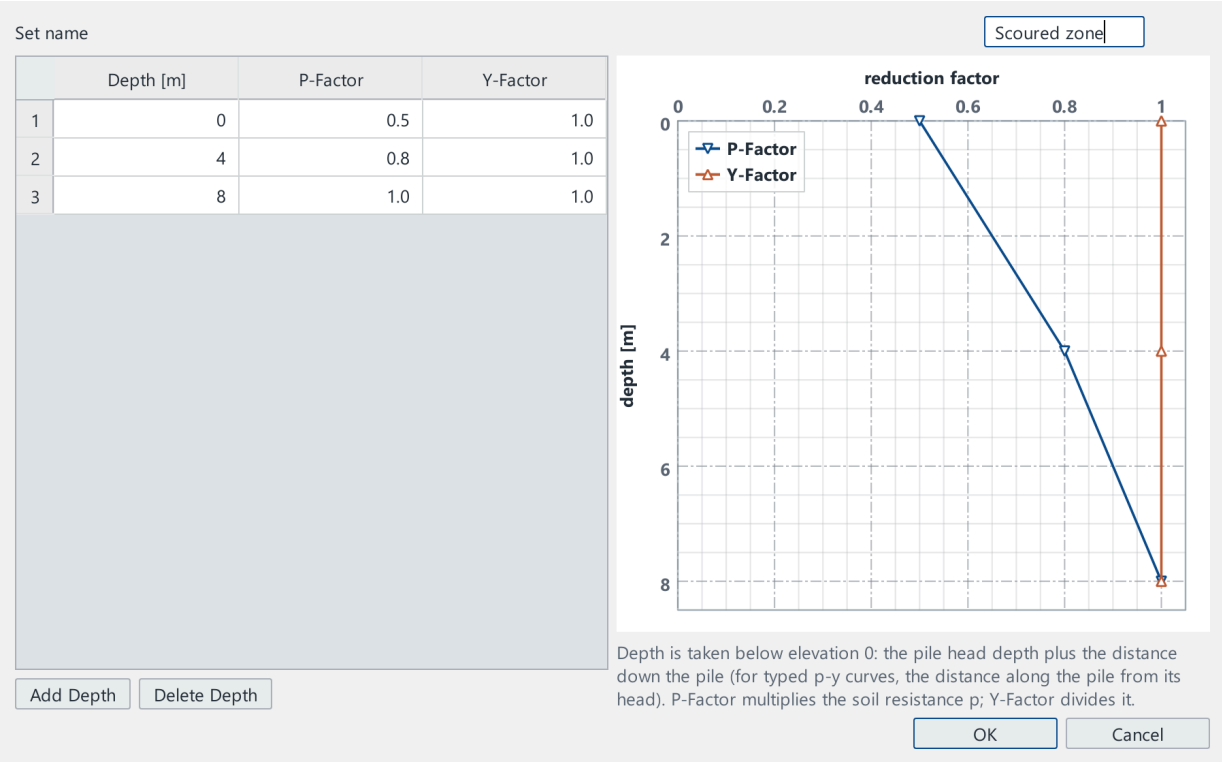


Figure 9.44 — Lateral Reduction Set

9.10 Printing depths for p-y curves

Define ▸ Soil ▸ Printing Depths for p-y Curves… lists the depths at which the p-y curves generated by the analysis are printed for verification (Figure 9.45). **Print internally generated p-y curves for verification** switches the printing on. Each row of the table gives the **Pile**, the **Depth below origin**, positive downward from the global origin ($z = 0$), and the **Displacements y, optional**, at which the curve is evaluated: up to 25 values, or empty for the 17 default points. The printout labels each curve with its depth below the ground line of that pile. **Add Row**, **Insert Row** and **Delete Row** edit the table; **Preview Curves** draws the curves. Printing does not change the analysis. The printed curves are shown with **Display ▸ Result Graphs and Response ▸ Show Printed p-y Curves…**.

© 2026 Solplan. All rights reserved.

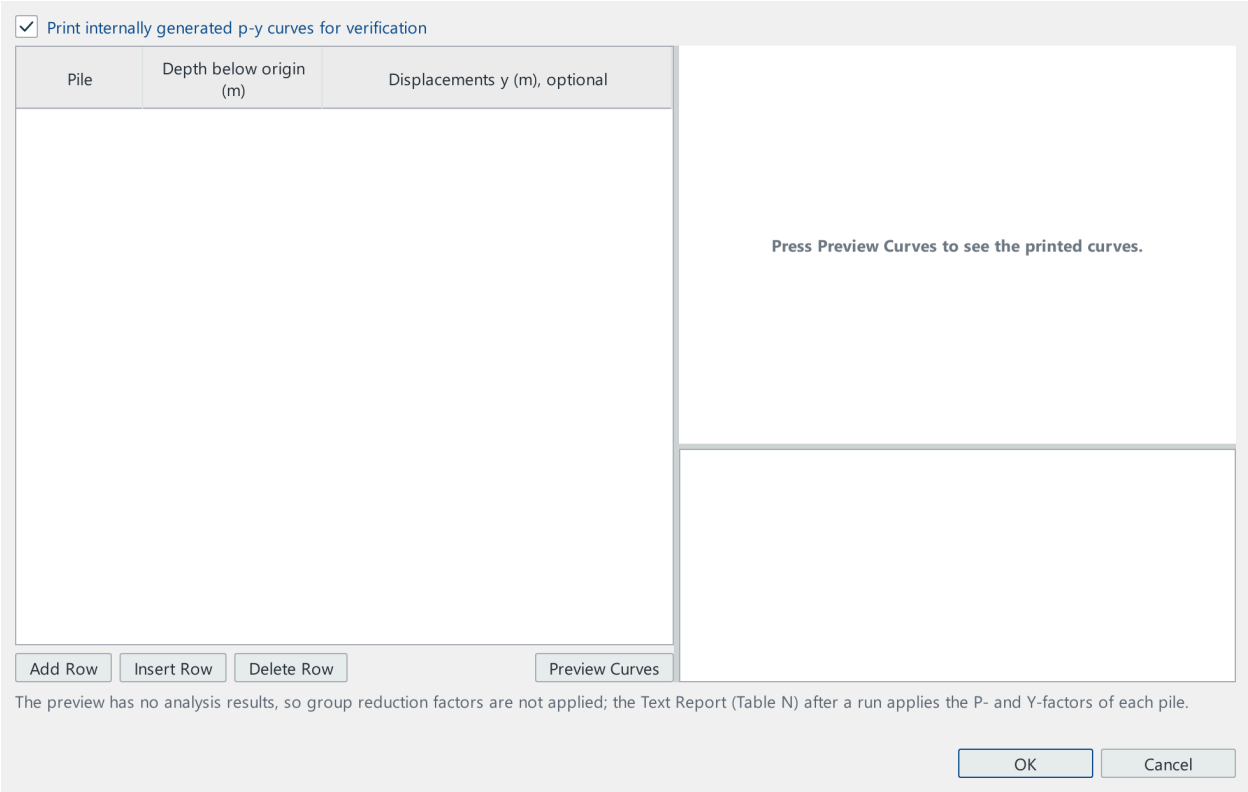


Figure 9.45 — Printing Depths for p-y Curves

9.11 Groups

Define ▸ Groups… lists named groups of objects, with the Group name and the number of Members. The form *Group* holds the **Group name** and shows the **Members**: the number of objects in the group and the number selected now. **Set to Current Selection** makes the group equal to the current selection, **Add Current Selection** adds the selected objects to it, and **Empty the Group** removes all members. A group is used to select its members again (**Select ▸ Coordinate Specification ▸ by Group…**) and to colour the piles; objects are also added to a group with **Select ▸ Assign to Group…**.

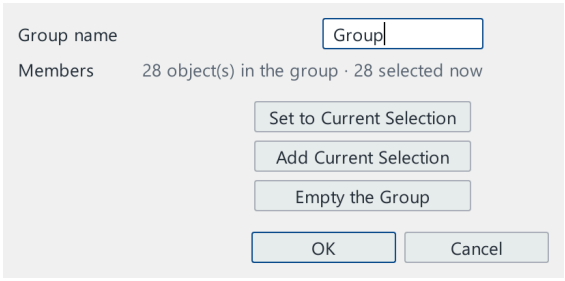


Figure 9.46 — Group

9.12 Model tables

Display ▸ Show Tables… lists the model in tables (Chapter 13). **File ▸ Export ▸ Model Tables (CSV)…** and **Model Tables (Excel)…** write them to files that can be edited in a spreadsheet, and **File ▸ Import ▸ Model Tables (CSV / Excel)…** reads them back. **File ▸ Import ▸ Solpile Model (.lpx)…** reads the sections and the soil of a single-pile model of Solpile.

© 2026 Solplan. All rights reserved.

10 Assigning properties

The **Assign** menu gives properties to the selected objects. Select the piles first (Section 8.3); the **Apply** button of a dialog that acts on piles is dimmed while no pile is selected. When the selected piles have different values, the dialog says so, and a value is changed only when its field is edited. Assigned properties are shown with **Display ▸ Show Assigns** and in the **Properties** panel. Loads are assigned with the commands of Chapter 11.

10.1 Pile properties

Each command of **Assign ▸ Pile** opens a small dialog with the field of that property, an **Apply** button and a **Close** button; the dialog stays open, so that the property can be applied to several selections in turn. The number of selected piles is shown in the dialog (Figure 10.1).



Figure 10.1 — Assign Pile Batter

Table 10.1 — Commands of Assign ▸ Pile

Command	Fields and effect
Section…	**Pile section**: the section of the whole pile
Section along Length…	segments of a pile with different sections, for example a cased upper part above an uncased lower part (Figure 10.2). For each segment: **Top** and **Bottom** elevations, the **Section**, and **E override**, a modulus of elasticity that replaces that of the material (empty: the material of the segment section). The column **Nonlinear EI from** is dimmed: the analysis uses the nonlinear *EI* of the segment's own section. **Add Segment** and **Delete Segment** edit the table; **Clear** removes every segment, so that the pile falls back to its single assigned section; **Use Values Shown** replaces the segments of every selected pile with those of the first selected pile shown in the table.
Type (driven / drilled)…	**Installation**: Driven pile or Drilled shaft. The type selects the axial load-transfer curves generated from the soil (Technical Reference, chapter "Soil response for axial loading"). New piles are driven; the shape of the section does not choose the installation, so bored piles must be assigned as drilled shafts.
Head Connection…	the connection of the pile head to the cap (Table 10.2)
Batter…	the inclination of the pile, entered as the **Angle from vertical**, the **Angle from horizontal** or the **Batter ratio 1 : m** (horizontal : vertical = 1 : *m*; enter *m* only, 0 selects a vertical pile), and the **Azimuth in plan** of the batter; the three inclination fields follow each other. The **Common batters** buttons set 1:3, 1:4, 1:6, 1:8, 1:12 and Vertical. The head stays in place and the tip moves.
Length…	**Pile length**: the length from the head to the tip, along the pile axis
Ground Surface Elevation Override…	**Assign an override** and the **Ground surface elevation**: a ground surface for the selected piles only, for example beside a scour hole or a local excavation; the soil layers keep their elevations, and the scour depth is subtracted from this level. It is an exception: the piles that carry it are drawn with a ring, and the ground between piles is not drawn through it. A slope of the whole site is entered once with **Sloping…** in the soil profile (Section 9.6.1). Unticked, the piles use the ground of the profile, level or sloping.

© 2026 Solplan. All rights reserved.

Command	Fields and effect
Axial Load Reduction Factor…	**Axial load reduction factor**: a factor on the axial load–displacement curve of the pile, 1 = no reduction, 0.8 = 80 %; used when the option *Reduction factors for axial loads* is ticked. Named lateral reduction sets act on the p-y curves and are assigned separately.
Lateral Reduction Set…	**Reduction set**: a set of factors on the lateral soil resistance along the depth, or <none> (Section 9.9); used when the option *Lateral reduction along depth* is ticked
p-y Set…	**p-y set**: a set of user-defined p-y curves (Section 9.7), or Curves of the soil layers
Axial (L-D) Curve…, Torsional (T-R) Curve…	**Axial curve** or **Torsional curve**: a user-defined curve of the pile head, or Generated from soil
Local Axes…	**Rotation about axis 1**: the rotation of the cross-section about the pile axis, which turns local axes 2 and 3
Segmentation…	**Number of increments** into which the pile is divided for the analysis (default 100). More increments give more accurate results near points of rapid change, at a higher cost.
Project Default Pile-Head Connection…	the dialog *Pile Head Connection*: the **Connection** used by every pile that has no connection of its own, with **Spring z-z** and **Spring y-y** for elastic connections; optionally clears the connections of individual piles

Table 10.2 — Pile-head connections

Connection	Meaning
Project	the project default connection
Fixed	the pile head rotates with the cap; it carries moments
Pinned	the pile head is free to rotate relative to the cap; no moment is transferred, and a pile pinned in both planes carries no torque
Elastic (rotational springs)	rotational springs between the pile head and the cap, with the stiffness *Spring z-z* and *Spring y-y* (moment per radian) in the two bending planes
Different per plane	Fixed, Pinned or Elastic separately in **Plane z-z** and **Plane y-y**

In *Assign Pile Head Connection* the **Connection** list starts with *Project (…)*, which shows the project default in brackets; the spring fields are dimmed unless the connection is elastic.

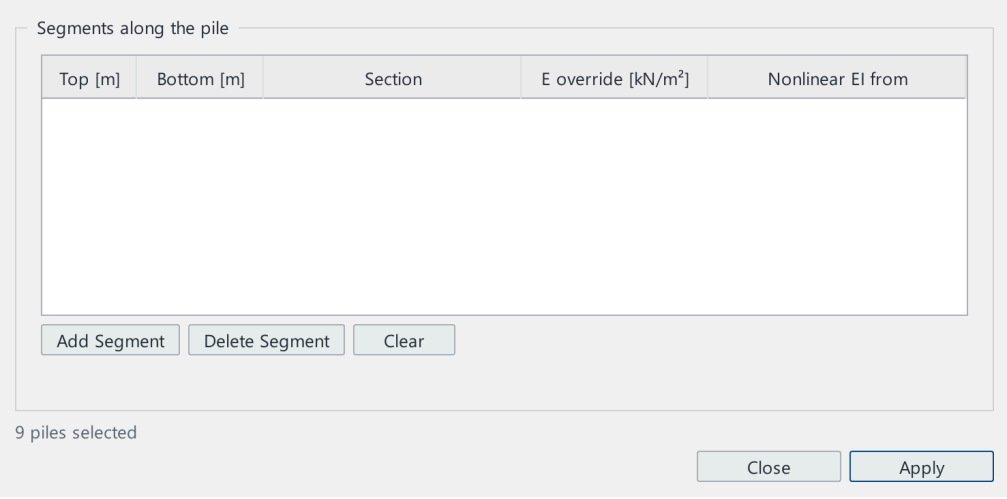


Figure 10.2 — Assign Section along Length

Assign ▸ Copy / Paste Assigns… copies chosen properties from one pile, chosen in **Copy assigns from**, to the selected piles (Figure 10.3). The **Assignments to paste** are ticked one by one, or all at once with **All** and **None**: Pile section, Pile type, Axial load reduction factor, Lateral reduction set, Section along length, Head connection, p-multiplier set, User p-y curve set, Axial curve, Torsional curve, Generated axial curve source, Generated torsional curve source, Ground elevation override, Local axes rotation, Segmentation, Batter (pile direction) and Pile length.

© 2026 Solplan. All rights reserved.

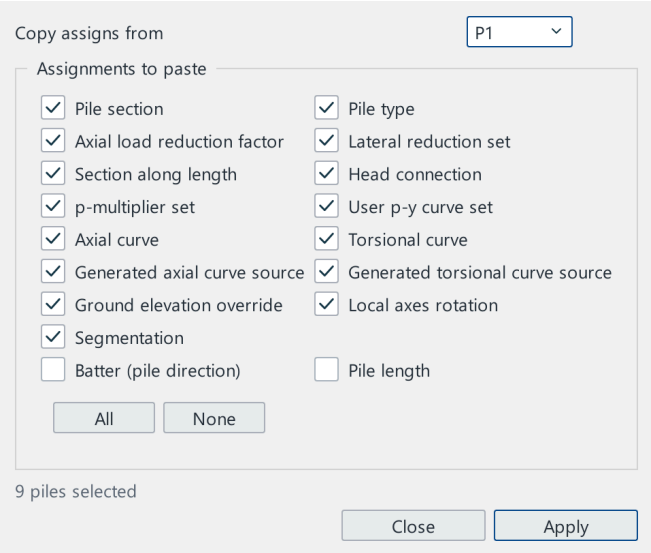


Figure 10.3 — Copy / Paste Assigns

10.2 Cap properties

Table 10.3 — Commands of Assign ▸ Cap

Command	Fields and effect
Cap Section…	**Cap section**: the section of the cap (thickness, material and mesh size, Section 9.4); its material and thickness are shown
Cap Elevation…	**Mid-plane elevation** of the cap; the top and bottom elevations are shown. The cap and its mesh move; the pile heads keep their coordinates.
Cap Behaviour…	**Cap behaviour**. Rigid: the cap moves as a rigid body with three translations and three rotations, the usual model for a thick concrete cap. Flexible: the cap is a mesh of plate elements and bends under load; it shows how a thin or long cap spreads the load unevenly to the piles (Technical Reference, chapter “Mechanics of the pile group”).
Embedment & Soil Resistance…	the soil resistance on an embedded cap (Table 10.4)
Generate Flexible Cap Mesh	creates the plate mesh of a flexible cap from the outline of the cap, the pile heads, the point loads and the mesh size of the cap section
Clear Legacy Cap Restraints…	shown only for models that carry restraints of the cap from an older file; removes them

Table 10.4 — Assign Cap Embedment

Field	Meaning
Include soil resistance on the pile cap	switches the resistance on; it does not move the cap or add soil (set the cap position with Cap Elevation and the ground with the soil profile)
Soil resistance model	From the soil layer data: the p-y curves of the layers in contact with the cap
Method of analysis	Applying an equivalent lateral load, or Applying equivalent lateral springs
Override the soil-contact elevations; Top elevation, Bottom elevation	the range of contact; by default from the pile-head level to the pile-head level plus the cap thickness
Equivalent diameter	width of the cap face for the p-y curves; 0 = the diagonal of the cap in plan
Spring increments	number of increments over the contact range for the equivalent springs (1 to 20)

NOTE After a change of the cap outline, the element size, the cap section or the piles, the stored mesh of a flexible cap may no longer match the model. Check Model warns in that case; generate the mesh again before the run.

© 2026 Solplan. All rights reserved.

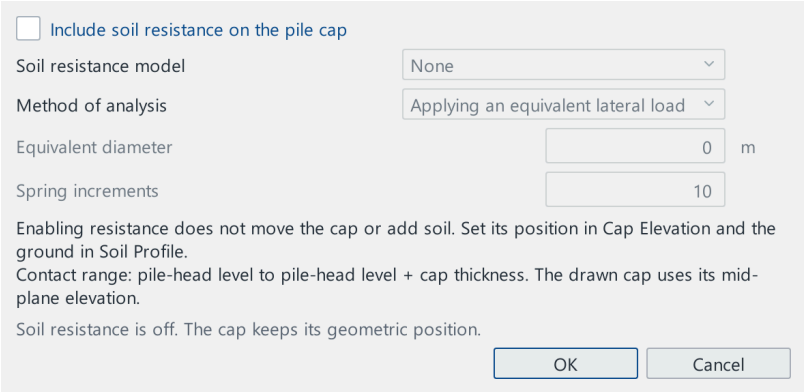


Figure 10.4 — Assign Cap Embedment

© 2026 Solplan. All rights reserved.

11 Loads, load cases and combinations

Loads are entered in three levels. A **load pattern** groups loads of one kind. A **load case** multiplies patterns by factors and is analysed as one nonlinear problem. A **load combination** combines load cases. The results are produced for load cases and combinations.

11.1 Load patterns

Define ▸ Load Patterns… lists the patterns with the Load Pattern name, its Type and its Self Weight multiplier. The form *Load Pattern* holds the **Load pattern name** and the **Type**: DEAD, SUPERDEAD, LIVE, WIND, QUAKE, SNOW, BERTHING, MOORING, CURRENT, WAVE or OTHER. SUPERDEAD is the superimposed dead load (finishes, utilities, wearing surface). The type is used by the design combinations (Section 11.4); it does not change the analysis of a load case. Loads are always assigned to a pattern.

NOTE The **Self weight multiplier** of a pattern adds the weight of the pile cap to that pattern: the weight per unit volume of the cap material × the cap thickness × its plan area, times the multiplier (1 = full weight). The weight of the piles is not included.

11.2 Assigning loads

Loads are assigned with the commands of the **Assign** menu to the selected objects, for the **Load pattern** chosen in the dialog (Figure 11.1). Under *Apply to the selected objects*, every load dialog offers **Replace** (the new load replaces the loads of the pattern on the selection), **Add** (it is added to them) and **Delete** (the loads of the pattern are removed, after confirmation if **Ask before deleting loads** is ticked). Values are in global axes unless stated otherwise.

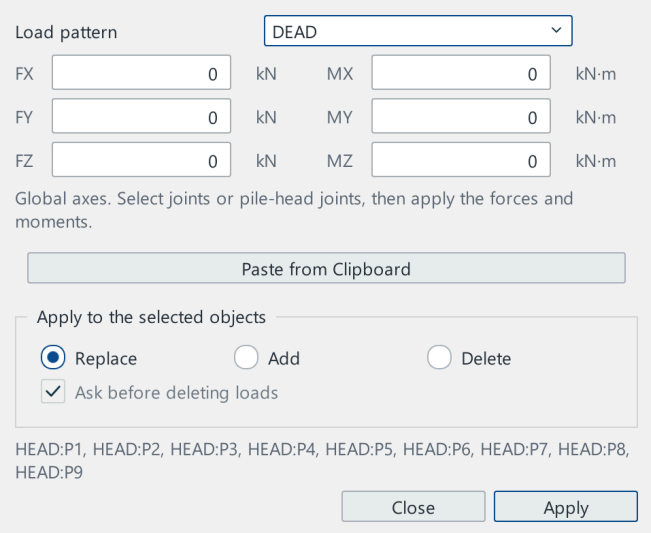


Figure 11.1 — Assign Joint Forces

The dialogs for point forces and moments have four helpers: **Point Coordinates…** types the point at which the load acts; **Pick Point** picks a joint, a pile head or a grid intersection on the canvas; **Equivalent Load…** moves a load given at another point to the selected point, with the moments that the move creates (exactly one target point must be selected); **Paste from Clipboard** reads six numbers per line in the current display units, FX FY FZ MX MY MZ. In *Assign Point Loads on Cap*, **Apply at** chooses Selected points / pile-head joints or the Cap reference point.

Table 11.1 — Load commands

Command	Fields and effect
Joint Loads ▸ Point Forces / Moments…	forces F_X, F_Y, F_Z and moments M_X, M_Y, M_Z at the selected joints (pile heads or reference points); they act on the cap at the position of the joint

© 2026 Solplan. All rights reserved.

Command	Fields and effect
Cap Loads ▸ Point Forces / Moments…	forces and moments on the cap, at the cap reference point or at the selected points
Cap Loads ▸ Distributed Load on Cap Plates…	forces per unit area F_X, F_Y, F_Z on every plate, or the chosen plates, of the flexible-cap mesh; F_Z points up, so a downward load is negative. Used only when the cap is flexible.
Pile Loads ▸ Distributed Lateral Load…	a load per unit length along the selected piles, in the **Direction** Global X, Global Y, Global Z, Local 2 or Local 3, between the **Elevation, top** and the **Elevation, bottom** (**Span the whole pile** fills them from the pile), varying linearly from the **Load at top** to the **Load at bottom**. For global loads on a battered pile, the part along the pile axis is dropped unless **Add the part along the pile axis as a force at the head** is ticked, in which case it is applied as one axial force at the pile head.
Pile Loads ▸ Point Force Along Pile…	a force **FX**, **FY**, **FZ** at an **Elevation** on the selected piles
Free-field Soil Movement…	a *Movement profile* of horizontal soil movement: **Depth** below the ground surface with the movements **UX** and **UY**, interpolated linearly between the points (**Add Point**, **Delete Point**), applied to the selected piles

In *Assign Distributed Loads on Cap Plates*, **Apply to** shows the plates concerned, and the loads are **FX per unit area**, **FY per unit area** and **FZ per unit area**. **Cap Behaviour…** and **Generate Cap Mesh** open the cap behaviour and generate the mesh from the dialog.

Display ▸ Show Loads… draws the loads on the model (Figure 11.2): **Show** chooses None (do not draw loads), All loads / all patterns, or one pattern; the *Load types* Point forces, Moments, Loads along piles, Cap area loads, Cap beam loads and Soil movement are switched one by one, or all with **Show All Types**; **Show load values** writes the values beside the arrows; **Arrow scale** and **Distributed arrows** set the length of the arrows and their number along a distributed load. A free-field soil movement is drawn with arrows at evenly spaced depths between the points of its profile and a line through their tips, from the effective ground surface that the analysis uses. The number of loads of each kind is counted.

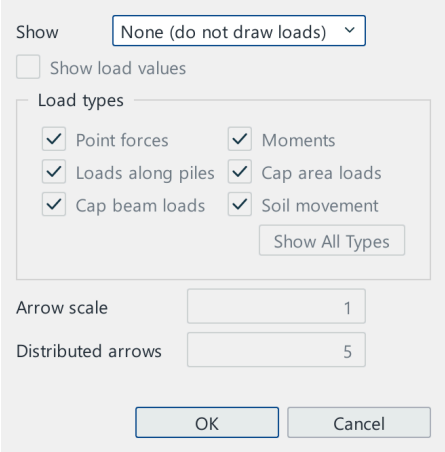


Figure 11.2 — Show Loads

11.3 Load cases

Define ▸ Load Cases… lists the load cases with the Load Case name, its Type, its Patterns and whether it is Run (Figure 11.3). Besides the usual buttons, the list has:

- **Add One per Pattern**: one nonlinear static case per load pattern, named after it;
- **Generate Load Levels…**: independent nonlinear static cases with increasing pattern factors, for the selected case and the number of levels asked for; each starts from zero (use Pushover for an incremental analysis);
- **Modify Selected…**: applies one setting to every selected case at once: follow the model default for every control, copy every control from the first selected case, run these cases, or do not run them;
- **Confirm Deletion**: ask before deleting a row.

© 2026 Solplan. All rights reserved.

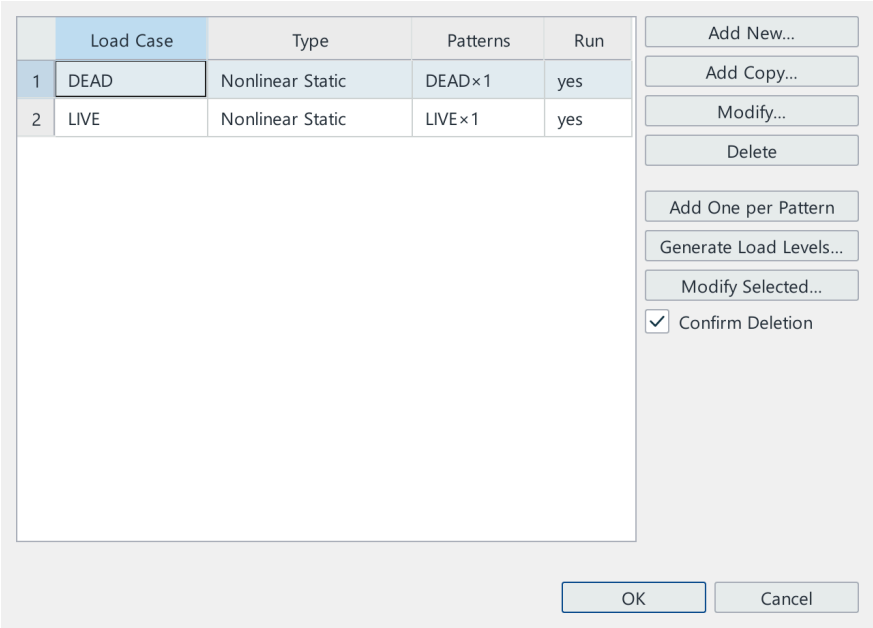


Figure 11.3 — Define Load Cases

The data of a load case are (Figure 11.4):

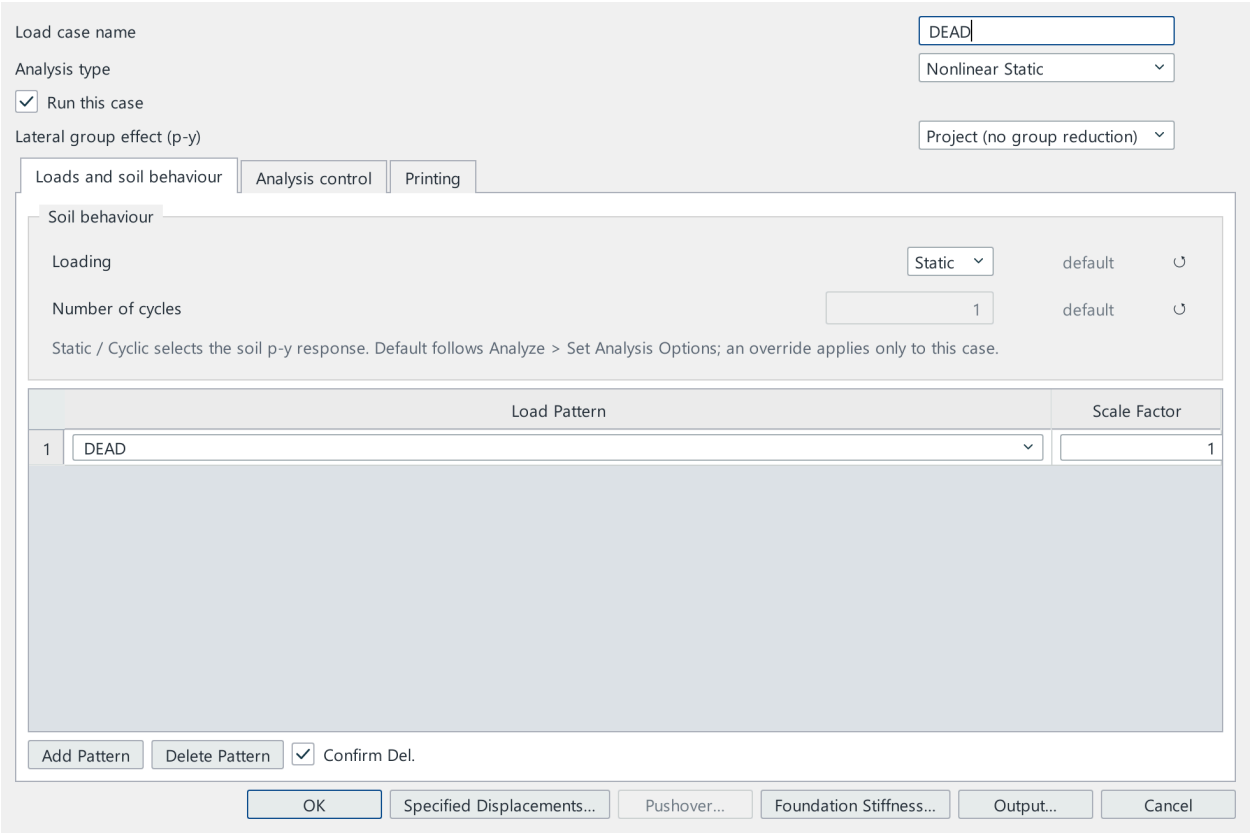


Figure 11.4 — Load Case Data, tab Loads and soil behaviour

Table 11.2 — Load Case Data

Field	Meaning
Load case name	name of the case

© 2026 Solplan. All rights reserved.

Field	Meaning
Analysis type	Nonlinear Static: the loads are applied and the nonlinear response of the group is found. Pushover: the loads are increased through a series of factors (Table 11.4). A case read from an older model as Linear Static or Nonlinear Staged is analysed as Nonlinear Static.
Run this case	unticked: the case stays in the model but is not run
Lateral group effect (p-y)	the group effect of the model, or another set for this case only, for example a table method whose leading row follows the load direction of this case
Tab Loads and soil behaviour	
Load Pattern; Scale Factor	the patterns of the case with their factors; **Add Pattern** and **Delete Pattern** edit the table, and **Confirm Del.** asks before a row is deleted
Loading; Number of cycles	Static or Cyclic soil curves; the number of cycles is part of the cyclic criteria of some models, not a load factor. The default follows Analyze ▸ Set Analysis Options; an override applies only to this case.
Initial conditions	shown only for a load case read from an older file that started from another case; every load case starts from zero. To start from the loads of another case, add both to a Linear Add combination, which is analysed as one load set.
Tab Analysis control	
Iterations, pile group; Iterations, single pile	limits on the iterations of the group (outer loop) and of one pile inside a group iteration (inner loop)
Tolerance, equilibrium	the relative change of the cap movement between iterations at which the group has converged
Tolerance, pile deflection	the change of the deflection of a pile at which a single-pile iteration has converged
Initial step; Minimum step	the first fraction of the load and the smallest step to which a step that does not converge is halved (Technical Reference, “Load steps”); 1 and 1 apply the load at once
Also stop on absolute cap movement; Absolute cap displacement, Absolute cap rotation	an additional convergence test on the absolute change of the cap movement
Include axial load variation along the piles	the axial thrust in the lateral analysis varies along each pile instead of being the head force
Tab Printing	
Printing code; Printing increment	what the output report lists for the case: the cap only, the cap and the pile heads, or the full length of every pile, printing every *N*th node

Each control field shows whether it follows the default of the model (**Analyze ▸ Set Analysis Options…**), marked *default,* or overrides it for this case; the reset button beside a field returns it to the default. Four buttons open further data of the case:

Table 11.3 — Further data of a load case

Button	Fields
Specified Displacements…	dialog *Specified Displacements or Forces*: **Prescribe cap degrees of freedom in this load case**; then, for each Degree of freedom of the cap, the Control (Force or Displacement) and the Value; and under *Point of application*, **Apply at a point other than the cap load reference point**, with its X, Y and Z
Pushover…	dialog *Pushover*, available when the Analysis type is Pushover (Figure 11.5): **Run this case as an incremental pushover**; the **Number of steps** (at most 25 load points), the **Ramp** (Arithmetic or Logarithmic), the **First factor** and the **Last factor**, **Generate Factors**, and the table of the **Load factor** of each step, edited with **Add Step** and **Delete Step**; the components held constant (Table 11.4)
Foundation Stiffness…	*Compute the 6×6 foundation stiffness matrix of this load case* (Table 11.5)

© 2026 Solplan. All rights reserved.

Button	Fields
Output…	dialog *Output Variables*: the quantities listed in the reports along the piles. **Follow the model default**, or a choice of: *Local displacement* Along axis 1, Along axis 2, Along axis 3 and Resultant in plane 2–3; *Global displacement* Along X, Along Y, Along Z and Horizontal resultant; *Forces and stiffness* Moment about axis 3, Moment about axis 2, Resultant moment, Shear along axis 2, Shear along axis 3, Resultant shear, Soil reaction along axis 2, Soil reaction along axis 3, Resultant soil reaction, Axial force, Total stress, Bending stiffness EI about axis 3, Bending stiffness EI about axis 2, Distributed load along axis 2, Distributed load along axis 3 and Soil modulus Es; and *Demand / capacity (combinations)* Bending moment and Shear force. **Select All** and **Select None** set every box; **Save as Default** makes the choice the model default for every case that has not overridden its output.

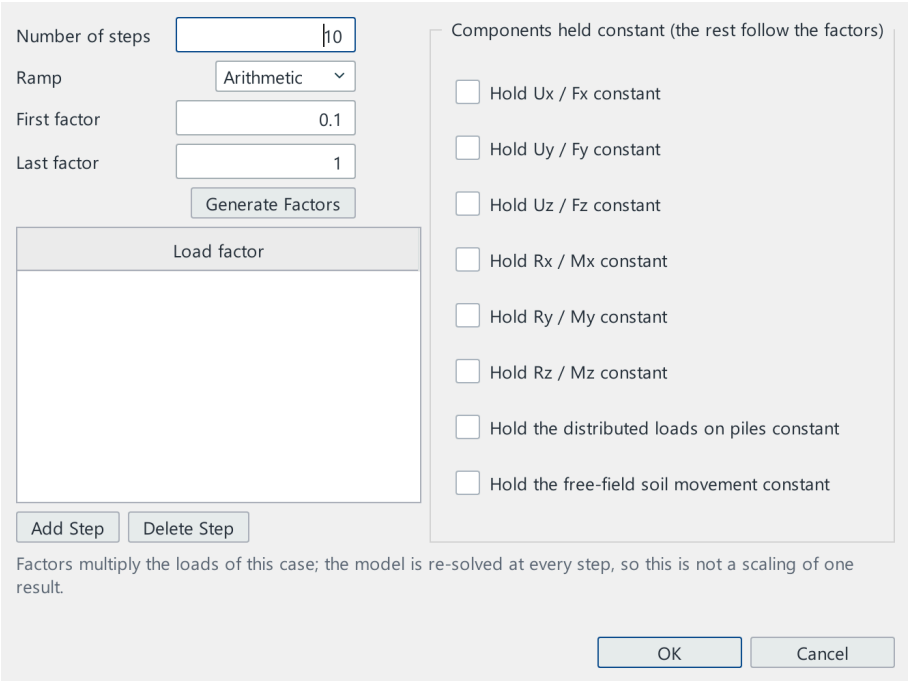


Figure 11.5 — Pushover

Table 11.4 — Pushover: components held constant

Option	Effect
Hold Ux / Fx constant; Hold Uy / Fy constant; Hold Uz / Fz constant; Hold Rx / Mx constant; Hold Ry / My constant; Hold Rz / Mz constant	the component keeps its value in every step; the others are multiplied by the load factors
Hold the distributed loads on piles constant	loads along the piles are not factored
Hold the free-field soil movement constant	soil movements are not factored

Table 11.5 — Foundation Stiffness

Field	Meaning
Compute the 6x6 foundation stiffness matrix of this load case	switches the computation on; the other fields are dimmed until it is ticked
Based on nonlinear soil response	each loaded degree of freedom of the cap is pushed alone at the load levels, and the stiffness is the inverse of the secant flexibility; **Diagonal terms only** drops the coupling terms between translation and rotation, an approximation
Based on equivalent elastic springs	the analysed state is replaced by elastic springs and condensed to the cap; *Consider axial force in piles* adds the second-order effect of the axial forces

© 2026 Solplan. All rights reserved.

Field	Meaning
Based on the applied load; Based on a target deflection at the pile cap	for equivalent springs: the state at the load of the case, or at a **Target** displacement or rotation of a chosen degree of freedom (Ux, Uy, Uz, Rx, Ry, Rz), reached within a **Tolerance** (%) and a number of **Maximum iterations** (analyses, the first one, with the load of the target direction removed, included); **Scale only the load associated with the deflection** scales only the load component of the target degree of freedom
Number of levels; Ramp; First level, Last level	up to 10 levels, as fractions of the load of the case (or of the target), Arithmetic or Logarithmic; **Generate Load Levels** fills the table of *Load / displacement levels* (Fraction of the case load), which can be edited with **Add Level** and **Delete Level**

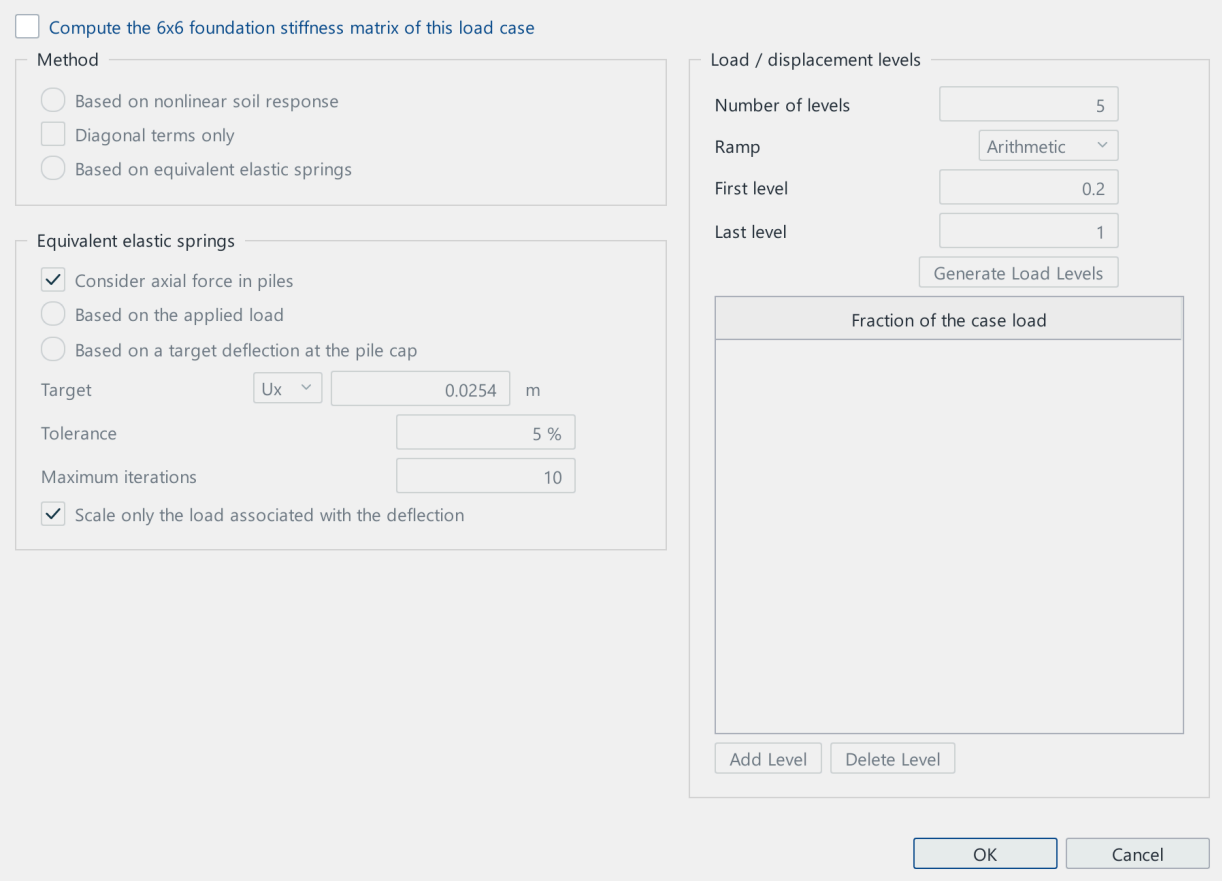


Figure 11.6 — Foundation Stiffness

Every load case starts from the unloaded state. To analyse the loads of two cases acting together, combine them in a Linear Add combination.

11.4 Load combinations

Define ▸ **Load Combinations…** combines load cases or other combinations with scale factors. The list shows each Combination with its Type and Members. Besides the usual buttons, it has:

- **Design Combo Options…**: the code and the parameters of the design combinations (Table 11.7);
- **Add Default Design Combos**: adds the strength combinations of the chosen code for the load patterns of the model;
- **Envelope All Cases**: one envelope combination over every load case in the model;
- **Load Factors…** reads a set of LRFD load factors from a CSV file, and **Save Factors…** writes the combinations out as a CSV of load factors;
- **Confirm Deletion**: ask before deleting a row.

© 2026 Solplan. All rights reserved.

The data of a combination are (Figure 11.7):

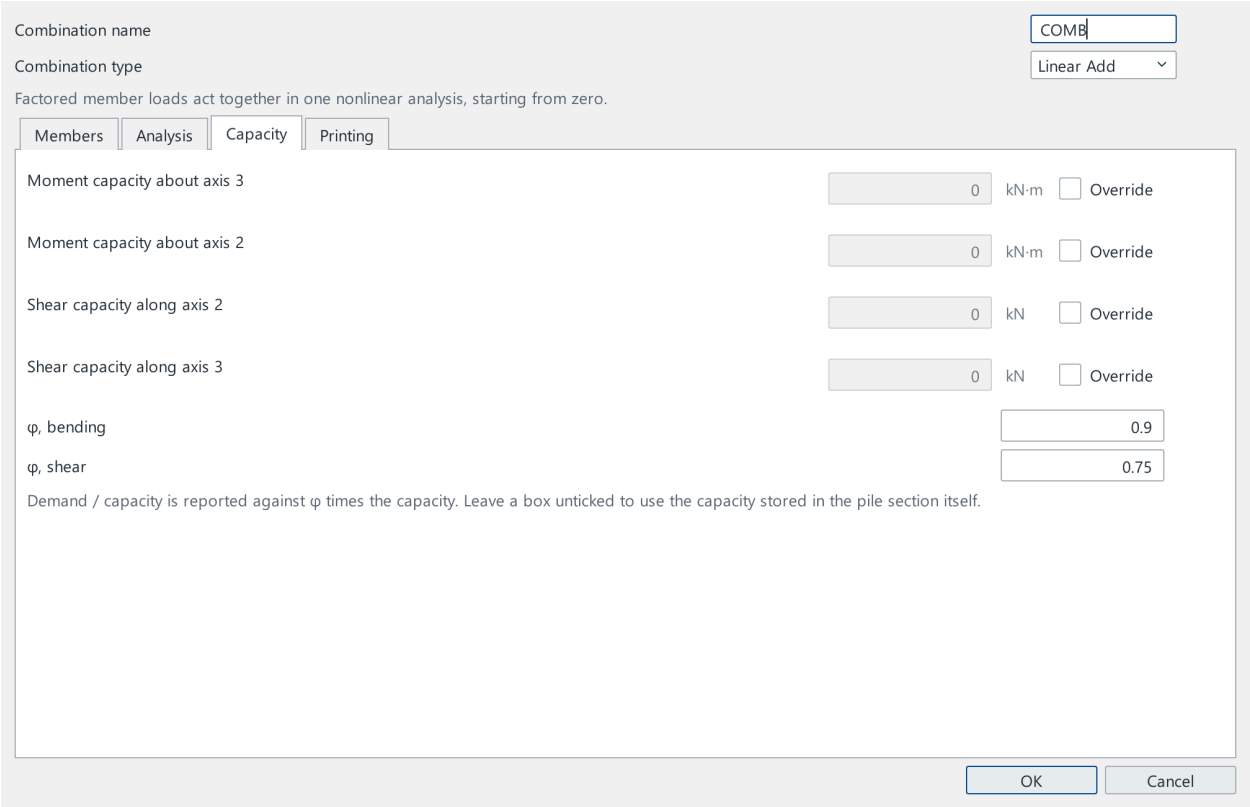


Figure 11.7 — Load Combination, tab Capacity

Table 11.6 — Load Combination

Field	Meaning
Combination name	name of the combination
Combination type	Linear Add: the factored member loads act together in one nonlinear analysis, starting from zero. Envelope: the largest and smallest value of every quantity among the members.
Tab Members	the members, Load Case or Combination, with their Scale Factor; **Add Case** and **Delete Case** edit the table, and **Confirm Del.** asks before a row is deleted
Tab Analysis	the same controls as the Analysis control tab of a load case: Loading, Number of cycles, Iterations, pile group, Iterations, single pile, Tolerance, equilibrium, Tolerance, pile deflection, Initial step, Minimum step, Also stop on absolute cap movement with the Absolute cap displacement and the Absolute cap rotation, and Include axial load variation along the piles; each follows the model default or overrides it for this combination
Tab Capacity	**Moment capacity about axis 3**, **Moment capacity about axis 2**, **Shear capacity along axis 2** and **Shear capacity along axis 3**, each with **Override**: ticked, the value replaces that of the sections for this combination; and the resistance factors **φ, bending** (default 0.9) and **φ, shear** (default 0.75), for the demand/capacity check of the combination
Tab Printing	**Give this combination its own printing options**, with the Printing code and the Printing increment as for a load case; **Output Variables…** chooses the quantities printed along the piles for this combination, including the demand / capacity ratios

NOTE Because the response of the soil is nonlinear, a Linear Add combination is analysed again with its factored loads; its result is not the sum of the results of its members. Combination types of older models that cannot be analysed (Absolute Add, SRSS, Range Add) are kept in the file; Check Model reports them when they are selected to run.

© 2026 Solplan. All rights reserved.

Table 11.7 — Design Combination Options

Field	Description
Code / template	ACI 318-19, ASCE 7-16 or AASHTO LRFD 9th: the strength combinations for DEAD, SUPERDEAD, LIVE, SNOW, WIND and QUAKE patterns; wind and earthquake patterns are separate alternatives. SUPERDEAD takes the DEAD factor under ACI 318-19 and ASCE 7-16; under AASHTO LRFD it is DW (1.50 with DC 1.25, 0.65 with DC 0.90, 1.00 in Service I and Extreme Event I)
QUAKE patterns contain	Total seismic effect E, or Horizontal seismic effect Eh; with Eh, the vertical earthquake effect 0.2 × S_{DS} × DEAD is added
SDS (g); Redundancy factor, rho	the design spectral acceleration and the redundancy factor of the seismic combinations
LL factor with EQ	the factor on live load with earthquake (AASHTO γ_{EQ}, project-specific)
Include project marine templates	optional example factors for berthing, mooring, current and wave; they are project factors, not code combinations, and must be reviewed

Use custom combinations for other load types or directional rules.

© 2026 Solplan. All rights reserved.

12 Analysis

12.1 Analysis options

Analyze ▸ Set Analysis Options… holds the settings that apply to the whole model and the defaults of the load cases and combinations (Figure 12.1). The defaults are used wherever a case or a combination shows *default*.

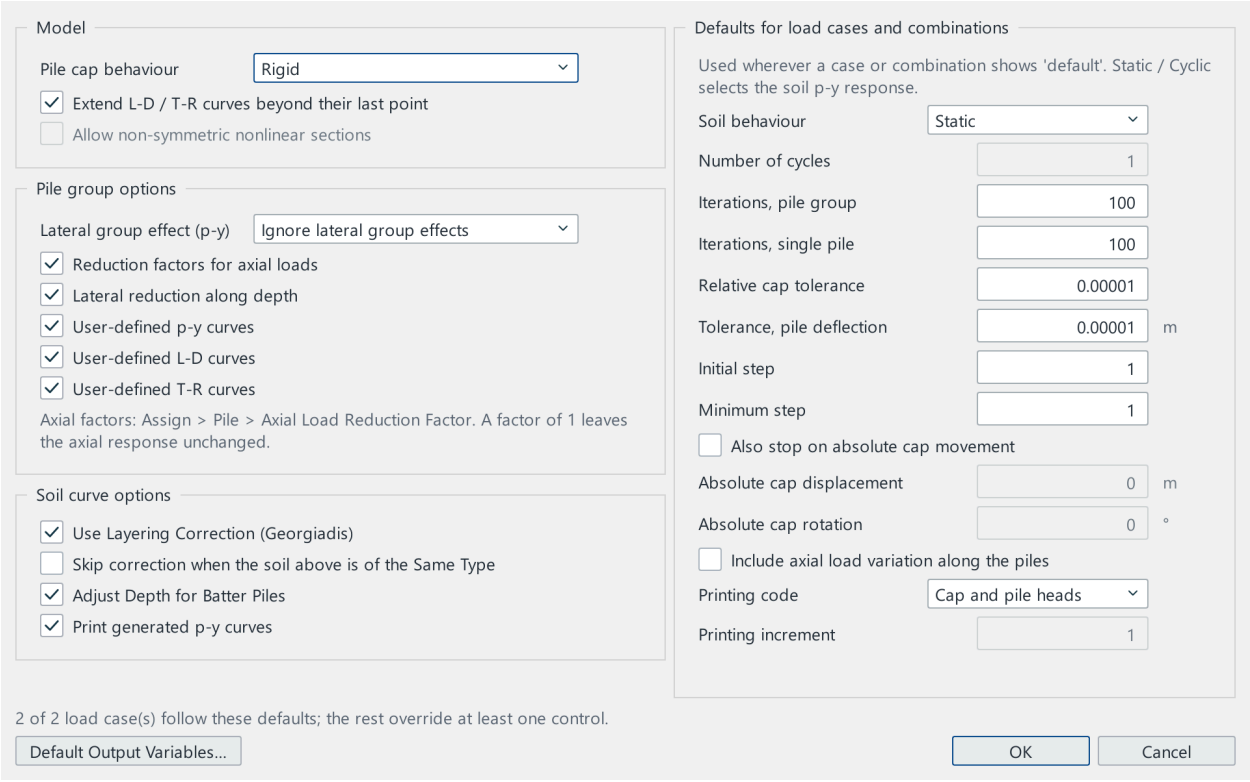


Figure 12.1 — Analysis Options

Table 12.1 — Analysis options

Option	Meaning
Model	
Pile cap behaviour	Rigid or Flexible (the same setting as Assign ▸ Cap ▸ Cap Behaviour)
Extend L-D / T-R curves beyond their last point	off: a pile whose axial head displacement passes the end of its curve carries no axial load, and the analysis reports an apparent failure in bearing; on: its axial force is held at the value of the last point
Allow non-symmetric nonlinear sections	a separate negative moment–curvature branch for sections with unsymmetric reinforcement; off, such a section stops the run. Dimmed when the model has no such section.
Pile group options	
Lateral group effect (p-y)	the set of p-multipliers of the model, or Ignore lateral group effects
Reduction factors for axial loads	on: the axial L-D loads are multiplied by the factor assigned to each pile with Assign ▸ Pile ▸ Axial Load Reduction Factor (1 = unchanged); off: those factors are ignored. Independent of the lateral group effects.
Lateral reduction along depth	on: the lateral reduction sets assigned to the piles are used
User-defined p-y curves, User-defined L-D curves, User-defined T-R curves	on: the user-defined curves assigned to the piles replace the curves generated from the soil for those piles
Soil curve options	
Use Layering Correction (Georgiadis)	corrects the p-y curves near a layer boundary by the equivalent-depth method

© 2026 Solplan. All rights reserved.

Option	Meaning
Skip correction when the soil above is of the Same Type	omits that correction below a layer of the same model
Adjust Depth for Batter Piles	measures the depth of the soil curves vertically for battered piles
Print generated p-y curves	prints the generated curves at the printing depths (Section 9.10)
Defaults for load cases and combinations	
Soil behaviour; Number of cycles	static or cyclic curves
Iterations, pile group; Iterations, single pile	iteration limits of the group (outer loop) and of one pile inside a group iteration (inner loop), default 100 each
Relative cap tolerance	relative change of the cap movement between iterations, $\Delta U/U$ (default 0.00001)
Tolerance, pile deflection	convergence of the single-pile analyses (default 0.00001 m)
Initial step; Minimum step	load stepping (default 1 and 1: the load is applied at once)
Also stop on absolute cap movement	an additional convergence test, with the Absolute cap displacement and the Absolute cap rotation
Include axial load variation along the piles	see Table 11.2
Printing code; Printing increment	see Table 11.2
Default Output Variables…	the default columns printed along each pile in the text report: the dialog *Output Variables*, with the same choices as for a load case (Table 11.3) and **Select All** and **Select None**

The defaults are adequate for most models. A tighter tolerance makes the results more precise at the cost of more iterations; a larger iteration limit, or load steps smaller than 1, can help a heavily loaded group to converge.

12.2 Load cases to run

Analyze ▸ Set Load Cases to Run… chooses the cases and combinations that the next run analyses (Figure 12.2). **Run all load cases** runs every case, including cases added later; otherwise the cases are ticked one by one. Combinations are selected separately in the list below. **Select All** and **Select None** set every box of a list.

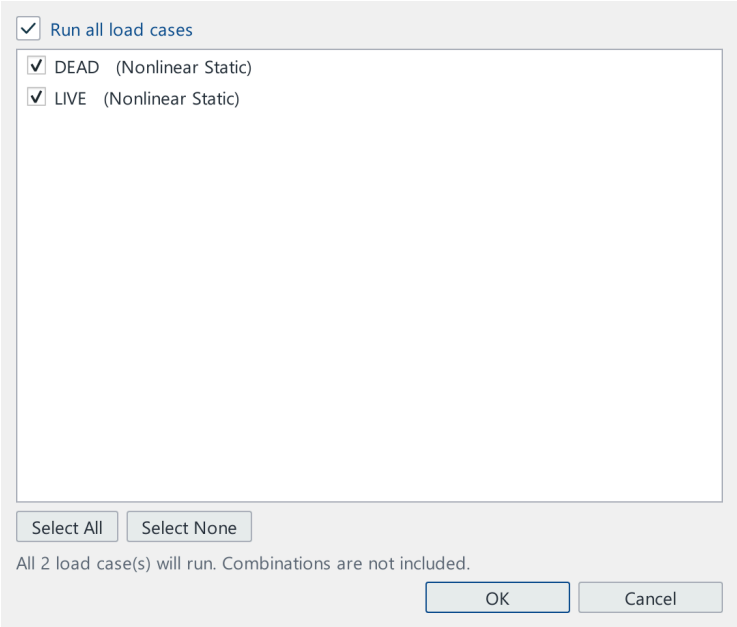


Figure 12.2 — Load Cases to Run

© 2026 Solplan. All rights reserved.

12.3 Check Model

Analyze ▸ Check Model… (F6) examines the model before a run and lists its findings in the **Command** panel. A finding has a code, a level and the objects it concerns:

- **Error**: the model cannot be analysed as it is, for example a pile without a section, a pile that does not reach the soil, or a load on an object that no longer exists;
- **Warning**: the model can be analysed, but the result may not be what is intended, for example a flexible-cap mesh that no longer matches the cap;
- **Information**: a remark about the model, for example a field that the analysis does not use.

A click on a finding selects the objects concerned. The run starts only when the model has no errors. Check Model is also run automatically at the start of every run.

The values are checked as the analysis reads them. A field that the analysis does not read in the current setting, shown dimmed in its form, is not checked: for example the SPT blow count of a Piedmont residual soil whose test is a cone, or a field of a hybrid model that the other fields replace. Such a value cannot block a run. A value that is not a number in a field that the analysis reads, for example in a file or a table pasted from a spreadsheet, is an error.

12.4 Running the analysis

Analyze ▸ Run Analysis (F5) analyses the chosen cases and combinations. The progress is shown on the status bar, and the run can be cancelled. At the end, a summary lists one line per case: whether it converged, the movement of the cap and the largest pile force. The notes of each case are in the **Command** panel. The canvas then shows the bending moment diagram of the case with the largest moment, and the view is fitted to the whole model; with vertical loads only, and so no moment, it shows the deformed shape instead.

NOTE A case that does not converge is reported with its last iterate and a note that explains the cause, for example a load beyond the axial capacity of a pile. A case that converged only for part of its load reports the fraction reached. Results of a case that did not converge are not a solution and should not be used for design.

Any change of the model after a run clears its results, so that results always belong to the model as it is shown.

12.5 Multi-file analysis

Analyze ▸ Multi-File Analysis (Batch Run)… analyses a list of model files one after the other and writes the text reports next to each file (Figure 12.3). The list is built with **Add Files…** and **Remove Selected**, emptied with **Clear List**, saved and reopened with **Save List…** and **Open List…**, and checked with **Check Files**, which reports files that are missing or cannot be read. **Replace** changes the text typed in the first box to the text of the second box in every path of the list, for example after the files were moved. **Run All Analyses** analyses every file; **Run Moment-Curv.** only computes the moment–curvature curves of the nonlinear sections; **Stop Batch** stops the batch: the load case being analysed stops at its current iteration, no report is written for that file, and the remaining files are not run. The messages of the run are listed in the window; **Save Messages…**, **Open Messages…** and **Clear Messages** keep, reload and clear them.

© 2026 Solplan. All rights reserved.

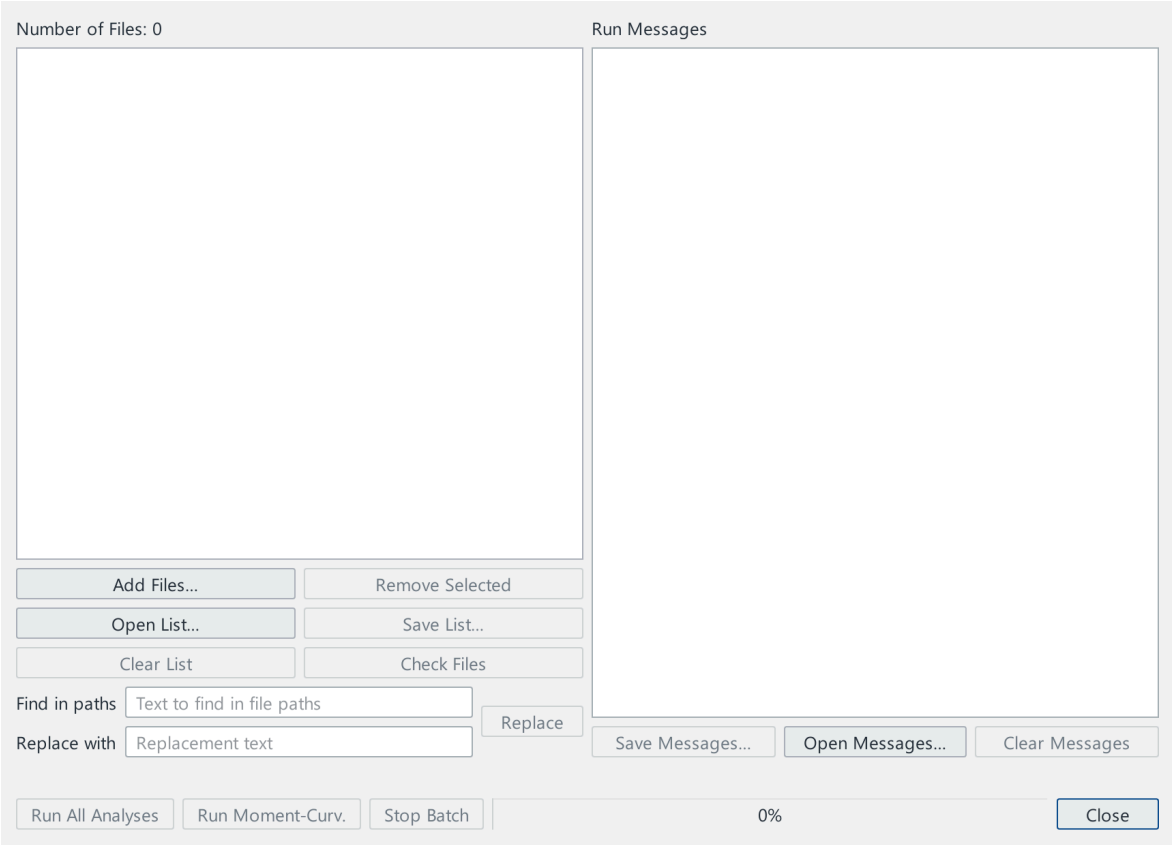


Figure 12.3 — Multi-File Analysis (Batch Run)

12.6 Command Line and the MCP server

Every model can also be built, changed, analysed and reported with short text commands. The same commands work in two places: the **Command Line** of the Solgroup window, which acts on the open project, and the **MCP server**, through which an AI assistant works on a project of its own in the background while the window is in use.

Window ▸ Command, or the **Command** button in the status bar, shows the Command Line in the panel at the bottom, beside *Messages*. A command is typed after the `>` prompt and run with Enter; the arrow keys recall earlier commands, and the answer is written above the prompt. Every change made by a command is one step of **Edit ▸ Undo**; a command that is refused leaves the model as it was, with the reason. `run` starts the same analysis as **Analyze ▸ Run Analysis**.

© 2026 Solplan. All rights reserved.

Table 12.2 — Commands

Command	Use
help [command]	the list of commands, or the details of one
new [template] [param=value…]	a new model from a layout template, for example `new rectangular nx=3 ny=3 sx=1.8`; `templates` lists the templates and their parameters
open, save, info, units	open or save a project, its summary, and the display unit system
tables, table <name>	the model and result tables of **Display ▸ Show Tables**, and one of them
set <table> <row> <column> <value>	one cell of a model table, with the same limits as its form, for example `set Piles 1 Length 20`
import <file>	read tables exported to CSV or Excel and edited
list, add, edit, delete	sections, piles, soil layers, profile rows, load patterns, loads on the cap, load cases (`add cases auto` makes one per pattern) and combinations, for example `add loads DEAD fz=-2700 fx=300`
types, fields	the p-y models, section shapes and templates, and their parameters with units, defaults and limits
check, run, results, pile <label>	Check Model, the analysis, the cap displacement and pile-head extremes of each case, and the forces at one pile head
export <file>	a project copy (.ggp), every table with the results (.xlsx), or the three text reports (.txt)
undo, redo	undo or redo the last change

Numbers are read and written in the display unit system of the project, as in the tables; angles are in degrees.

Without the window, `solgroup-cli.exe --script commands.txt` runs a file of commands, one per line (a line starting with # is a note); it stops at the first command that fails, or goes on with `--keep-going`, and ends with exit code 0 only when every command succeeded. `solgroup-cli.exe --shell`, optionally followed by a project file, opens an interactive command prompt in the console; `quit` leaves it.

The sign-in of the program is also available in the console: `solgroup-cli.exe --login` asks for the e-mail and password of the solplan.id account (`--email` gives the e-mail, `--google` signs in with Google in the browser), `--licence` shows the account and its licences, and `--logout` signs out on this computer. The installer offers **Add the 'solgroup' command-line tool to PATH**; with it, the console tool is started as `solgroup` from any folder, for example `solgroup --script commands.txt`.

The MCP server starts with `solgroup-cli.exe --mcp` from the installation folder (or `python -m geogroup --mcp` from the source). An assistant that supports the Model Context Protocol registers it as a local server, for example `claude mcp add solgroup -- "C:\Program Files\Solplan\Solgroup\solgroup-cli.exe" --mcp`. The server offers tools for the project, the model tables and collections, Check Model, the run, the results and the export, and one tool that runs any text command. Its analyses use the licence of the signed-in account in the same way as the program.

© 2026 Solplan. All rights reserved.

13 Results

The results of the last run are kept in the open model until the model is changed. They are shown in the units of the model, through the commands of the **Display** menu: on the model, in graphs, in tables and in text reports.

13.1 Results on the model

Table 13.1 — Display ▸ Results in Canvas

Command	Fields and content
Show Deformed Shape…	the deflected piles and cap of the **Load case**; the **Scale factor** magnifies the displacements for visibility and changes no result (**Auto** suggests a scale that makes the largest displacement about 8 % of the model size). **Show Undeformed Shape** returns to the model.
Force Diagrams in Canvas…	tab *Result*: diagrams along every pile, or with **Selected piles only**, of the **Load case / combination**; the **Component** is M3 or M2 bending moment, Resultant Moment, V2 or V3 shear force, Resultant Shear, N axial force along pile, N axial force at pile head, T torsion at pile head, Soil Reaction y dir, Soil Reaction z dir or Resultant Soil Reaction. Tab *Appearance*: **Diagram scale**, **Diagram style** (Ordinates, Solid fill or Outline), **Diagram colour** (By component or a fixed colour) and **Show minimum and maximum values**. The axial force along the pile exists when **Analyze ▸ Analysis Options… Include axial load variation along the piles** was ticked for the run; without it the dialog says so.
Show Contour Plot…	window *Heat Map* (Figure 13.1). Tab *Result*: the **Load case / combination** and the **Result component** (U2 or U3 lateral deflection, Resultant Deflection, M3 or M2 bending moment, Resultant Moment, V2 or V3 shear force, Resultant Shear, soil reactions, Total Stress, Bending Stiffness, N axial force along pile, the axial force or torsion at the pile head, or a cap component for a flexible cap); **Show colour scale**; **All piles** or **Selected piles**; **Only piles containing the minimum or maximum**; **Show heat map in canvas**. Tab *Colour scale*: **Colour palette** (Spectrum, Sequential or Diverging), **Minimum** and **Maximum** (**Reset to Extremes** restores the minimum and maximum of the quantity over the piles), the number of **Colour bands** and the resulting **Increment**, and **Show cap mesh edges** (for cap results).

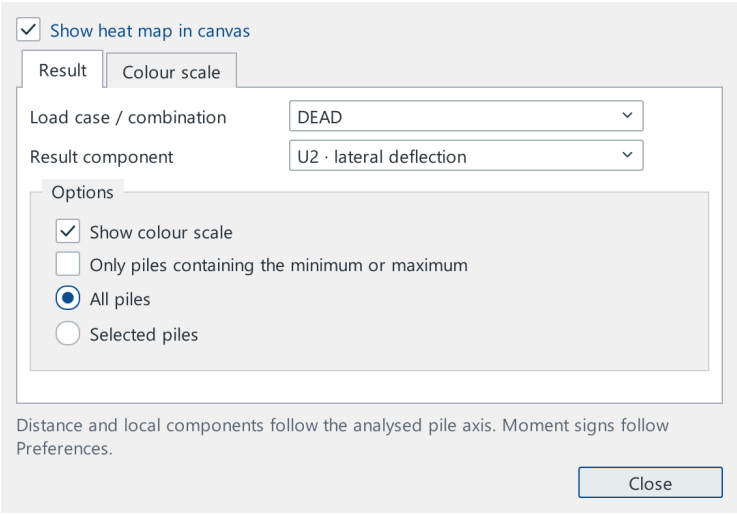


Figure 13.1 — Heat Map, Result tab

13.2 Result graphs and cap response

Display ▸ Result Graphs and Response ▸ Show Forces Along Piles… opens the result graphs (Figure 13.2). The **Load case / combination**, the **Pile** and the **Quantity** are chosen at the top; **Resultant of planes 2 and 3** plots the SRSS resultant, always positive, instead of the two local bending planes separately. **All load cases and**

© 2026 Solplan. All rights reserved.

combinations overlays the chosen pile in every load case and combination that has results, one curve each, to compare them on one graph. Each page has a graph and a table of values that can be copied:

Table 13.2 — Pages of the result graphs

Page	Content
Pile profile	the Quantity against the distance from the pile head, drawn upright like the pile: the distance on the vertical axis, increasing downward, and the value on the horizontal axis at the top. Lateral Deflection, Bending Moment, Shear Force, Soil Reaction p, Soil Modulus Es, Soil Reaction by Plane, Bending Stiffness EI, Demand / Capacity or Total Stress; in the two bending planes separately or as their resultant
Profile values	the same quantity in a table at every node of the pile: the Distance and the value in each plane, for example Lateral Deflection · plane 2 and Lateral Deflection · plane 3
Pile heads	a pile-head **Component** for every pile, as a graph and a table: N axial compression, T torsion, V2 and V3 shear, M2 and M3 bending, Ux, Uy, Uz global displacement and Rx, Ry, Rz global rotation
Cap plates	for a flexible cap, the **Component** at the Gauss points of the chosen **Plate** (Gauss point and value, for example M11 · cap bending in kN·m/m): M11 and M22 cap bending, M12 cap twisting, V13 and V23 cap transverse shear, S11, S22 and S12 stresses at the top and at the bottom face
Combined D/M/V	deflection, moment and shear of one pile side by side, for Plane 2: D2 / M3 / V2, Plane 3: D3 / M2 / V3, or the Resultants

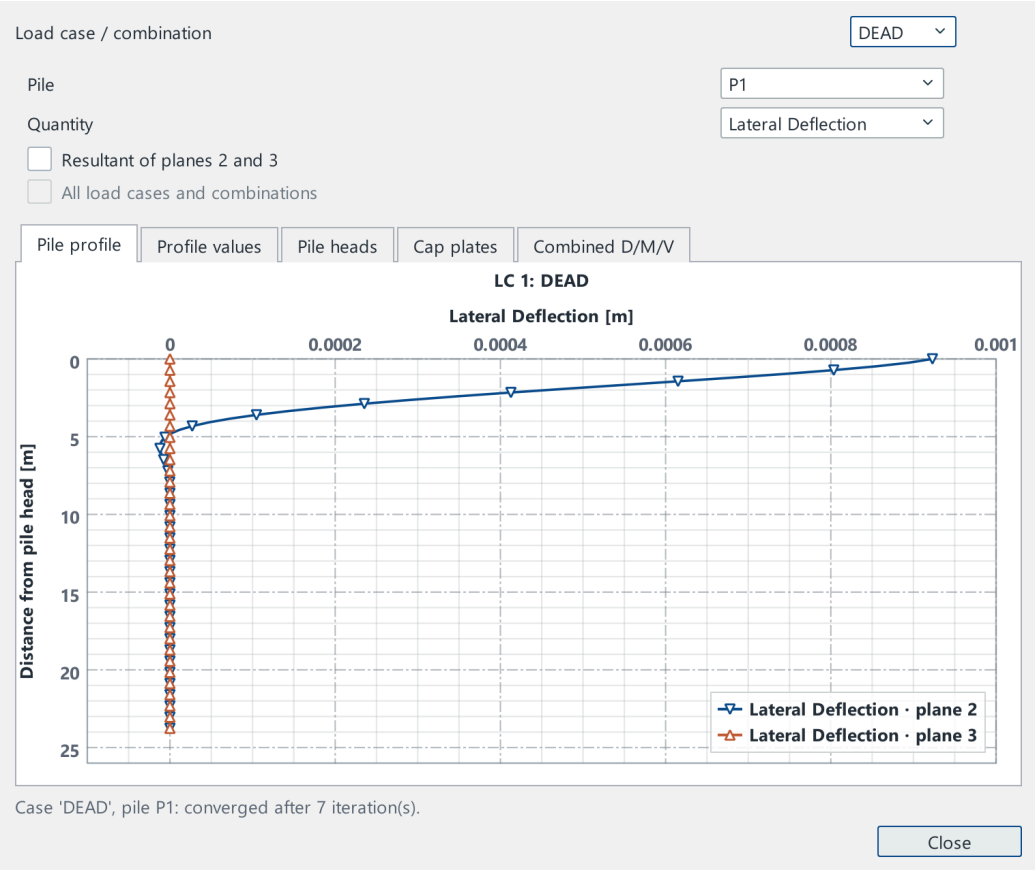


Figure 13.2 — Result Graphs: lateral deflection along a pile

© 2026 Solplan. All rights reserved.

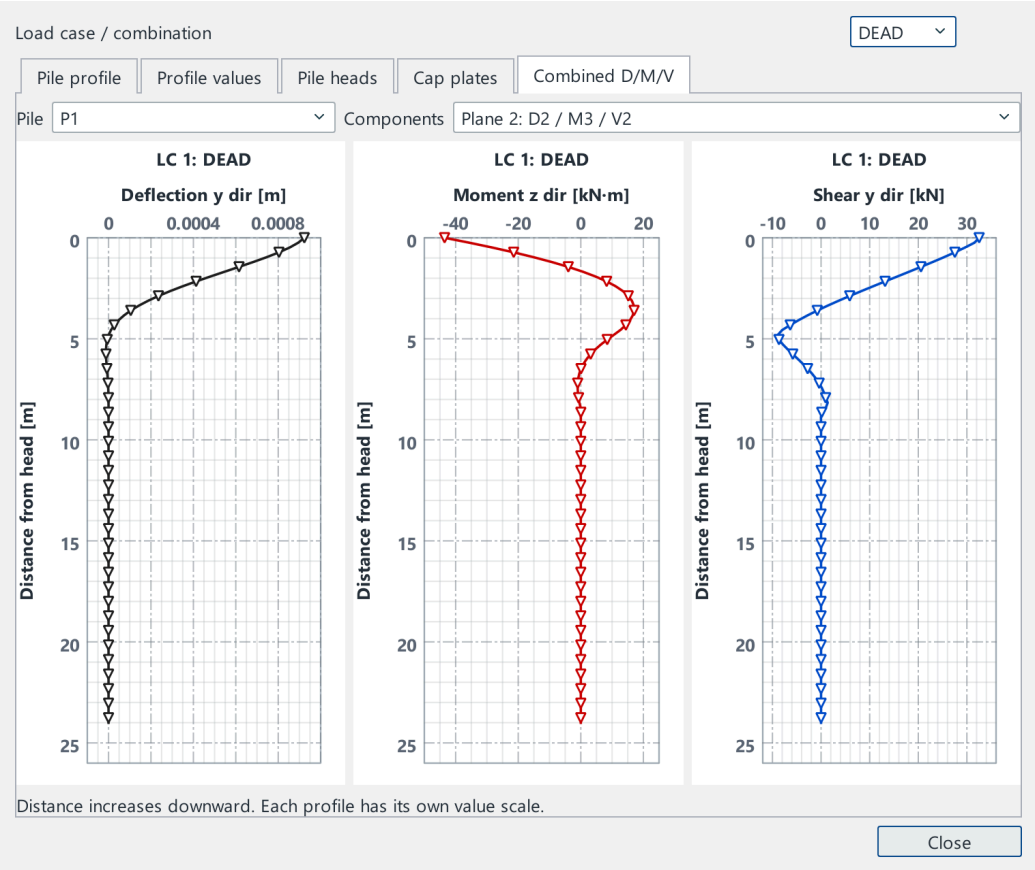


Figure 13.3 — Result Graphs: Combined D/M/V

Table 13.3 — Other commands of Display ▸ Result Graphs and Response

Command	Content
Show Cap Response…	tabs: **Displacement** (Ux, Uy, Uz, Rx, Ry, Rz of the cap for each Load Case, with the number of Iterations and the Status); **Foundation** (the stiffness matrix **K** = $\mathbf{S}^{-1}$, the flexibility, or a graph of a diagonal term against the displacement, for cases with a foundation stiffness analysis); **Pushover** (a displacement against a load or the load factor); **Envelope** (the extremes of the envelope combinations: for each Envelope, Row and Item, the cap displacements Ux, Uy, Uz, Rx, Ry, Rz and the pile-head Axial, Shear-2, Shear-3, Moment-2, Moment-3 and Torsion, with the case that gives each); **Reduction** (the P-Factor and Y-Factor of every Pile); **Passive** (the soil resistance on an embedded cap for each Load Case: Force X, Force Y, Elevation, Width and Method). **Copy** copies the matrix as plain numbers followed by a table of their units, or every converged pushover step as text (factor, load and cap displacement).
Special Plots…	compares solved load states without combining them: a quantity along the piles (U2 or U3 lateral deflection, moments, shears, soil reactions, Total Stress, Bending Stiffness, the distributed loads, and with axial load variation the axial force and the global displacements) for several cases or combinations and several piles in one graph; the axial load of every pile against its head moment (*Pile Axial Load vs. Top Moment*); or the movement of the cap against the load acting on it, one point per case (*Pile Cap Movement vs. Acting Load*, with the *Horizontal* and *Vertical* components chosen)
Show Printed p-y Curves…	window *Printed p-y Curves*: the p-y curves printed at the printing depths (Section 9.10)
Export Graphs to Excel…	window *Export to Excel*: choose the case and the plots (**Select All**, **Unselect All**), **Add markers in plots**, and **Generate Excel File…** writes the data and the charts to a workbook; **Close Window** closes it

© 2026 Solplan. All rights reserved.

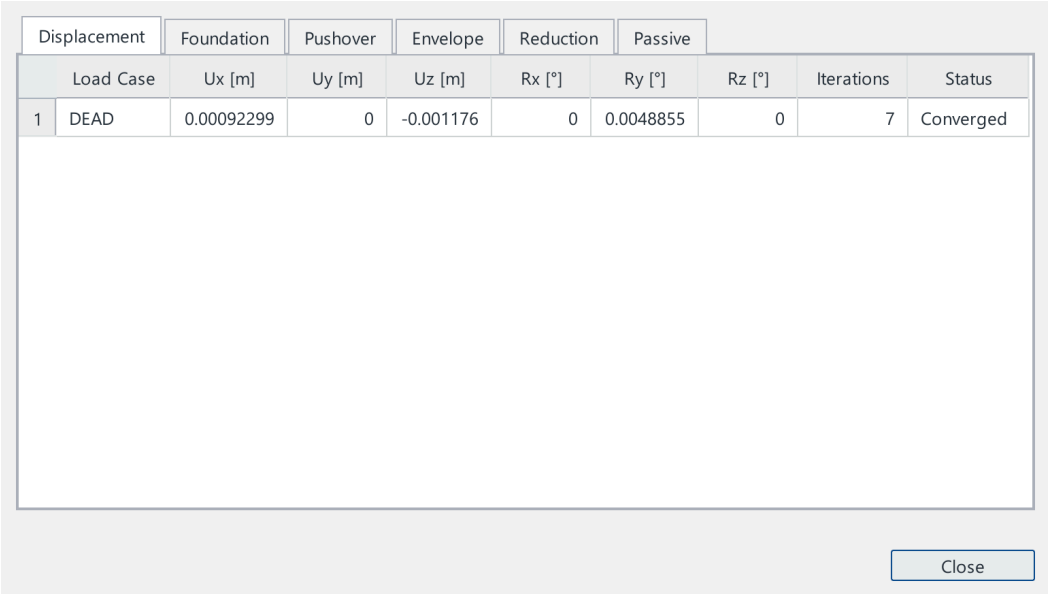


Figure 13.4 — Show Cap Response, Displacement tab

13.2.1 Results of special analyses

The pages of **Show Cap Response…** other than Displacement are filled by the analyses that produce them. The figures below were made with the 3 × 3 group of the examples, run with the corresponding options.

Foundation. For a case with a foundation stiffness analysis, the page shows the matrix of a chosen *Load Level*, as the *Stiffness* $K = S^{-1}$ or the *Flexibility* $u = S \cdot F$, with rows for the response and columns for the load, and a note on the load of the level and on the degrees of freedom that were not swept (Figure 13.5). *Kii vs displacement (graph)* draws each diagonal term against the movement of the cap, which shows how the stiffness falls with the load (Figure 13.6). **Copy** copies the matrix with its units for use in a structural program.

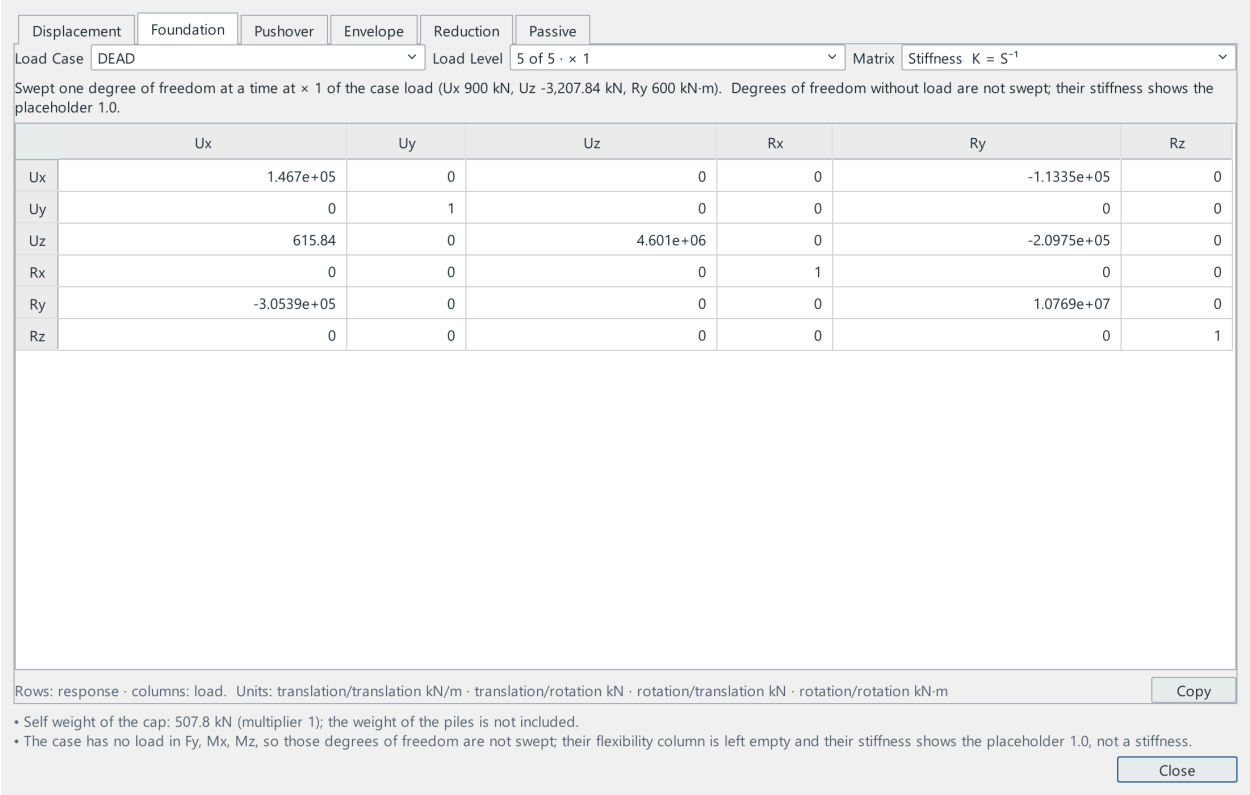


Figure 13.5 — Show Cap Response, Foundation: stiffness matrix at the full load

© 2026 Solplan. All rights reserved.

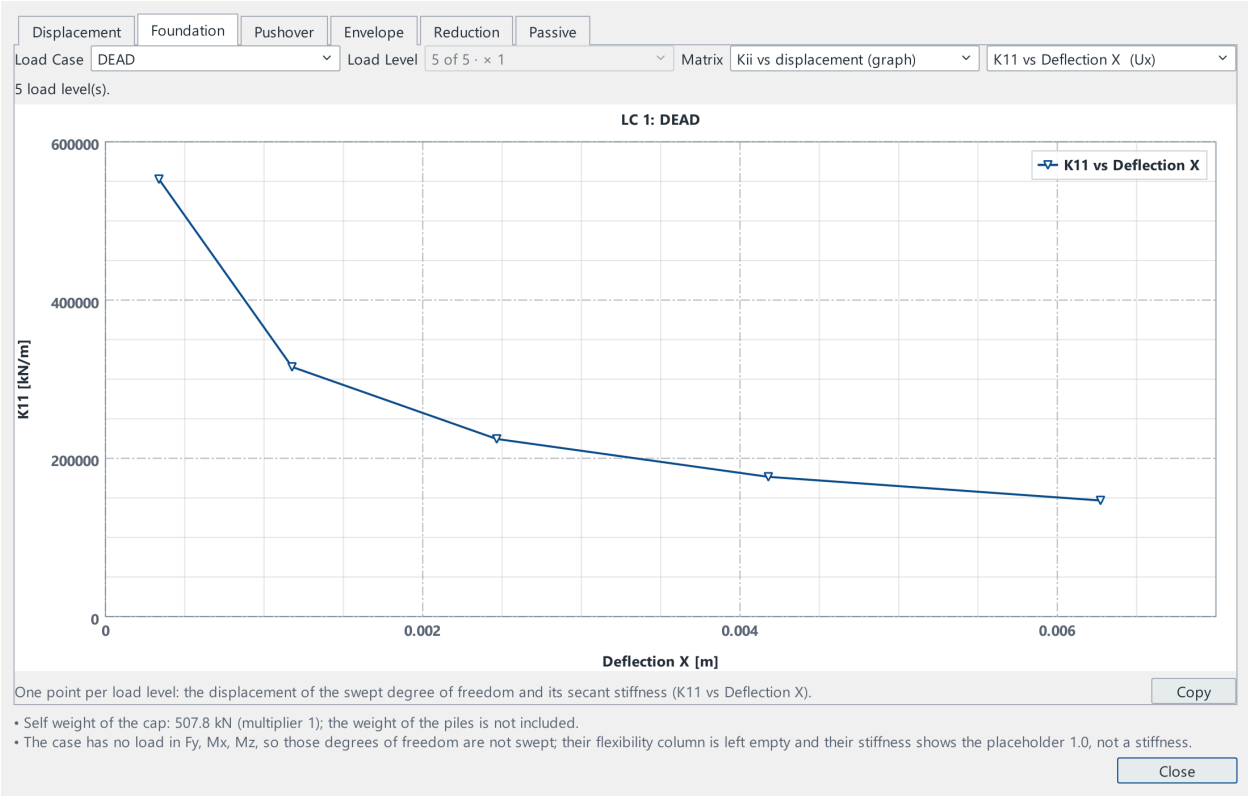


Figure 13.6 — Show Cap Response, Foundation: diagonal stiffness against the movement of the cap

Pushover. For a pushover case, the page plots a displacement of the cap (X) against a load component or the load factor (Y) over the converged steps, and states the number of steps and the components held constant (Figure 13.7).

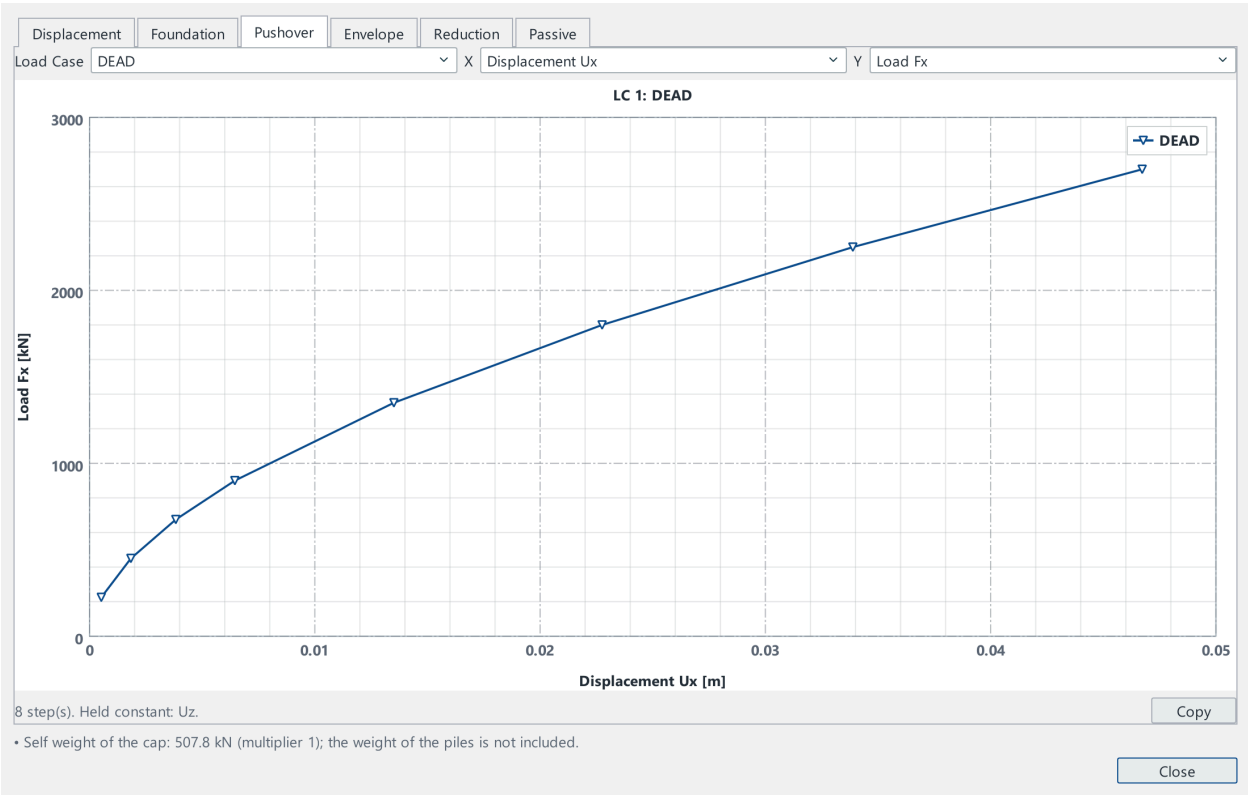
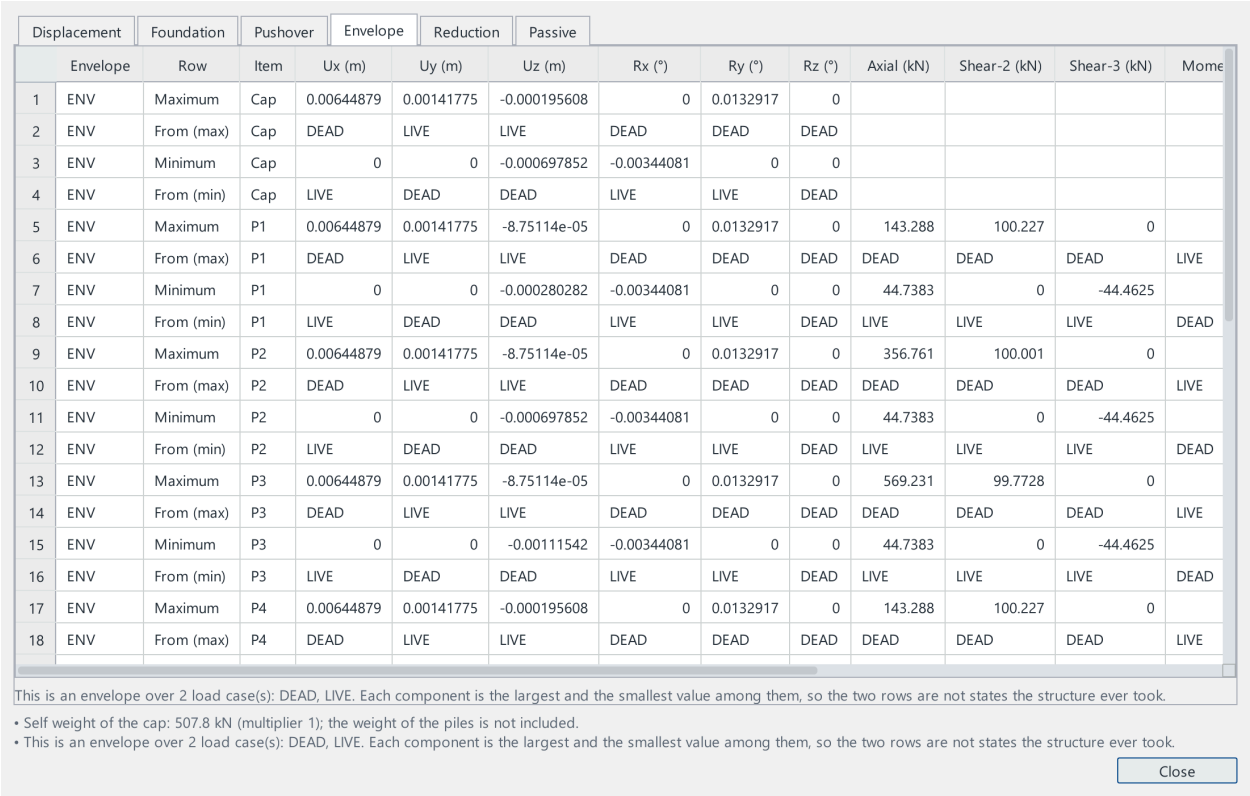


Figure 13.7 — Show Cap Response, Pushover

© 2026 Solplan. All rights reserved.

Envelope. For an envelope combination, the page lists for each component the largest and the smallest value among the members and the member that gives it: the displacements of the cap and the forces at the head of each pile (Figure 13.8).

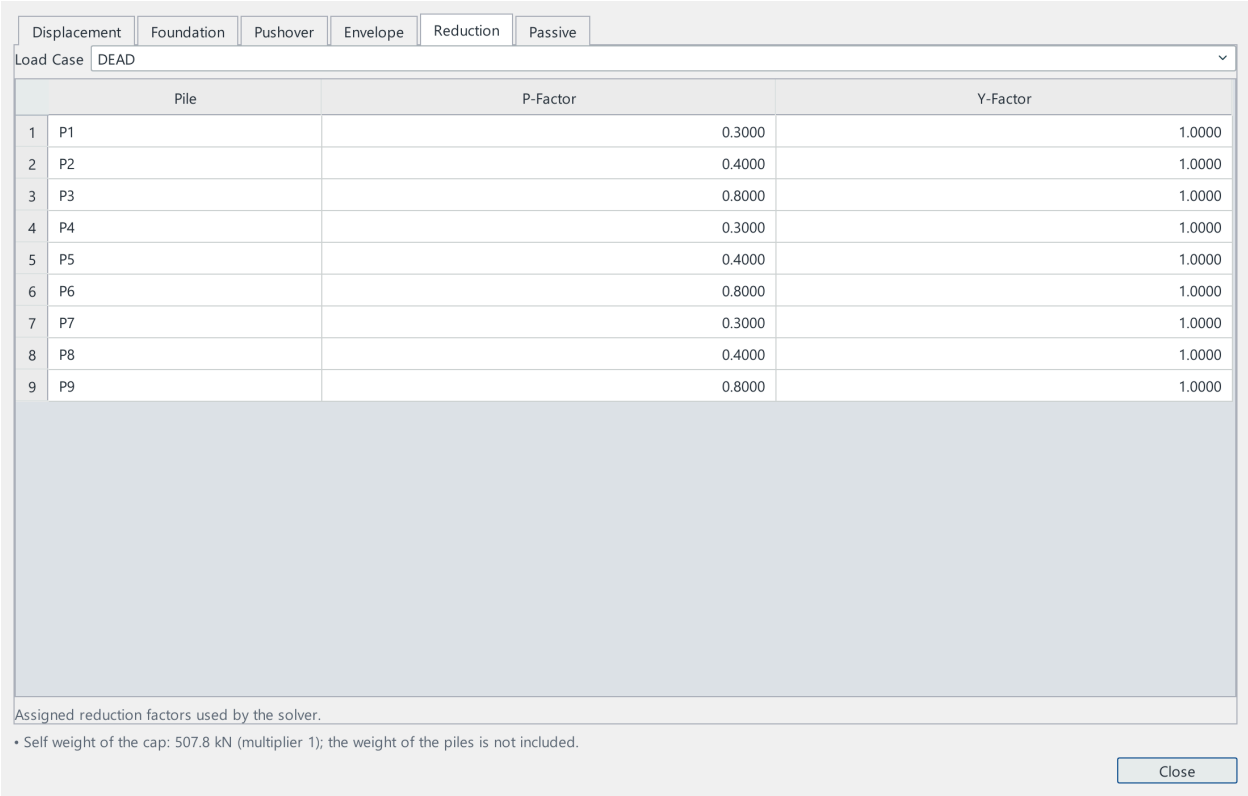


	Envelope	Row	Item	Ux (m)	Uy (m)	Uz (m)	Rx (°)	Ry (°)	Rz (°)	Axial (kN)	Shear-2 (kN)	Shear-3 (kN)	Mome
1	ENV	Maximum	Cap	0.00644879	0.00141775	-0.000195608	0	0.0132917	0				
2	ENV	From (max)	Cap	DEAD	LIVE	LIVE	DEAD	DEAD	DEAD				
3	ENV	Minimum	Cap	0	0	-0.000697852	-0.00344081	0	0				
4	ENV	From (min)	Cap	LIVE	DEAD	DEAD	LIVE	LIVE	DEAD				
5	ENV	Maximum	P1	0.00644879	0.00141775	-8.75114e-05	0	0.0132917	0	143.288	100.227	0	
6	ENV	From (max)	P1	DEAD	LIVE	LIVE	DEAD	DEAD	DEAD	DEAD	DEAD	DEAD	LIVE
7	ENV	Minimum	P1	0	0	-0.000280282	-0.00344081	0	0	44.7383	0	-44.4625	
8	ENV	From (min)	P1	LIVE	DEAD	DEAD	LIVE	LIVE	DEAD	LIVE	LIVE	LIVE	DEAD
9	ENV	Maximum	P2	0.00644879	0.00141775	-8.75114e-05	0	0.0132917	0	356.761	100.001	0	
10	ENV	From (max)	P2	DEAD	LIVE	LIVE	DEAD	DEAD	DEAD	DEAD	DEAD	DEAD	LIVE
11	ENV	Minimum	P2	0	0	-0.000697852	-0.00344081	0	0	44.7383	0	-44.4625	
12	ENV	From (min)	P2	LIVE	DEAD	DEAD	LIVE	LIVE	DEAD	LIVE	LIVE	LIVE	DEAD
13	ENV	Maximum	P3	0.00644879	0.00141775	-8.75114e-05	0	0.0132917	0	569.231	99.7728	0	
14	ENV	From (max)	P3	DEAD	LIVE	LIVE	DEAD	DEAD	DEAD	DEAD	DEAD	DEAD	LIVE
15	ENV	Minimum	P3	0	0	-0.00111542	-0.00344081	0	0	44.7383	0	-44.4625	
16	ENV	From (min)	P3	LIVE	DEAD	DEAD	LIVE	LIVE	DEAD	LIVE	LIVE	LIVE	DEAD
17	ENV	Maximum	P4	0.00644879	0.00141775	-0.000195608	0	0.0132917	0	143.288	100.227	0	
18	ENV	From (max)	P4	DEAD	LIVE	LIVE	DEAD	DEAD	DEAD	DEAD	DEAD	DEAD	LIVE

Figure 13.8 — Show Cap Response, Envelope

Reduction. When a set of group factors is used, the page lists the p-factor and the y-factor of every pile for the chosen case, after the iteration on the direction of movement where the method uses it (Figure 13.9).

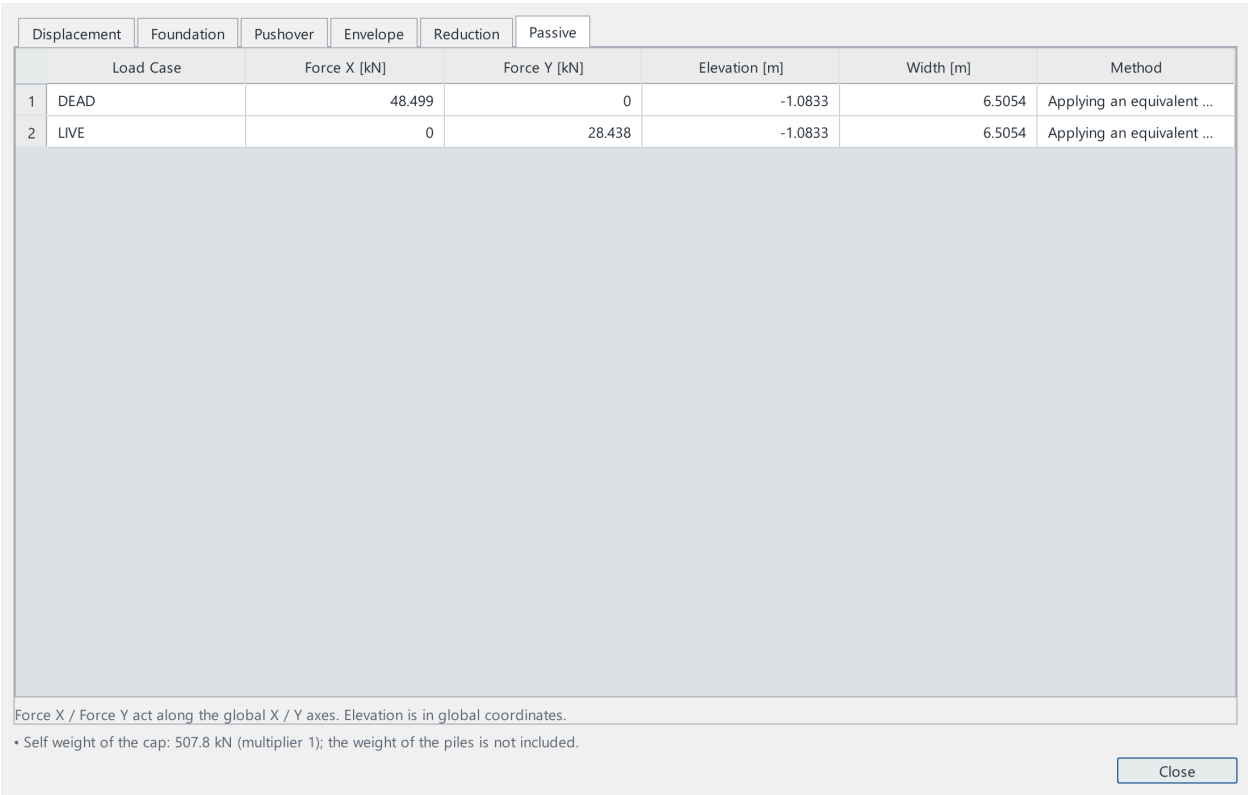
© 2026 Solplan. All rights reserved.



	Pile	P-Factor	Y-Factor
1	P1	0.3000	1.0000
2	P2	0.4000	1.0000
3	P3	0.8000	1.0000
4	P4	0.3000	1.0000
5	P5	0.4000	1.0000
6	P6	0.8000	1.0000
7	P7	0.3000	1.0000
8	P8	0.4000	1.0000
9	P9	0.8000	1.0000

Figure 13.9 — Show Cap Response, Reduction

Passive. For an embedded cap with soil resistance, the page lists for each case the force of the soil on the cap in X and Y, the elevation at which it acts, the width of the face and the method (Figure 13.10).



	Load Case	Force X [kN]	Force Y [kN]	Elevation [m]	Width [m]	Method
1	DEAD	48.499	0	-1.0833	6.5054	Applying an equivalent ...
2	LIVE	0	28.438	-1.0833	6.5054	Applying an equivalent ...

Figure 13.10 — Show Cap Response, Passive

© 2026 Solplan. All rights reserved.

Special Plots. The window compares cases and piles in one graph (Figure 13.11): select the quantity at the top, the cases or combinations on the left and the piles on the right. A quantity along the piles is drawn like the pile profile, with the distance from the pile head on the vertical axis, increasing downward; Pile Axial Load vs. Top Moment and Pile Cap Movement vs. Acting Load have the load on the vertical axis.

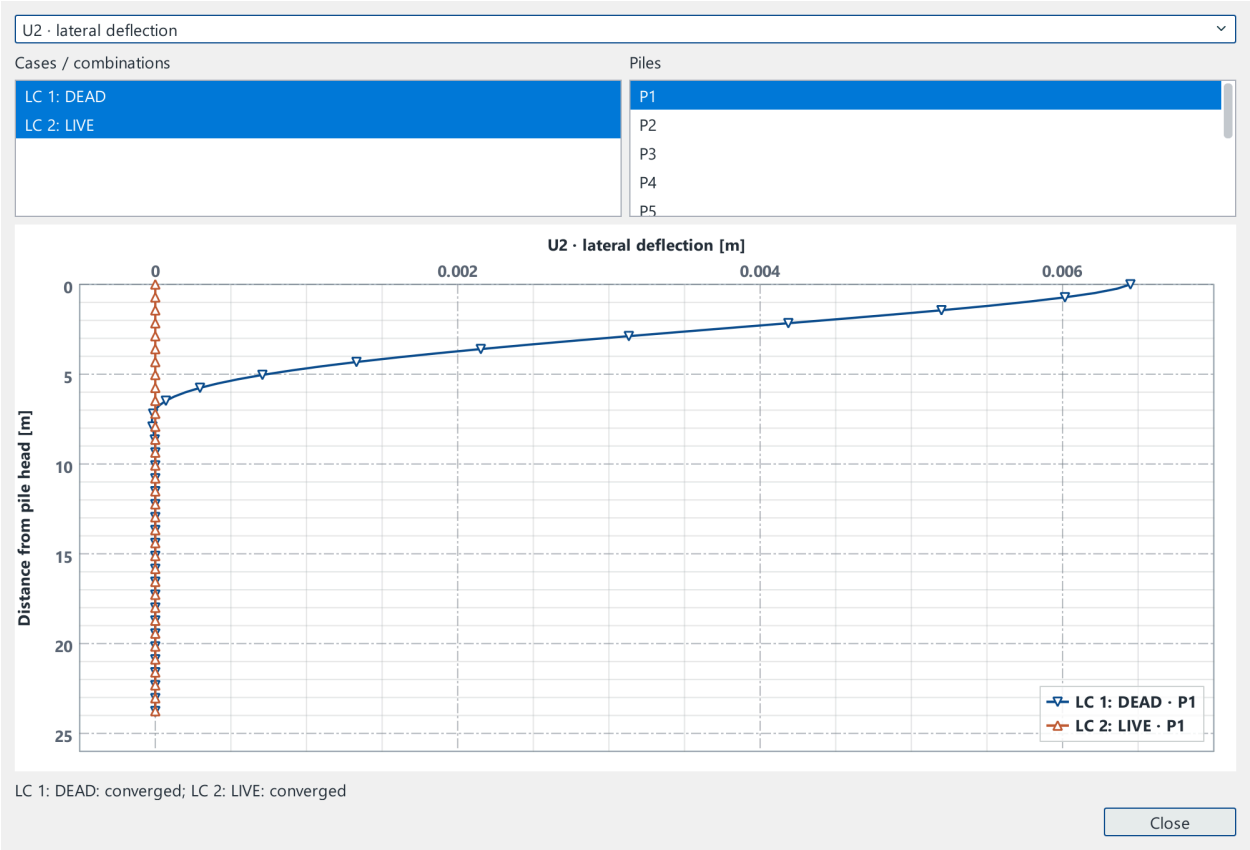


Figure 13.11 — Special Plots: lateral deflection U2 of pile P1 for two cases (LIVE acts in Y, so its U2 is zero)

13.2.2 Quantities along the piles

The figures below show each quantity of the page *Pile profile* for pile P1 of the model of Chapter 6 under the combination COMB2. The vertical axis is the distance from the pile head along the pile, increasing downward; the value is read on the axis at the top, and the legend names the plane of each curve.

Lateral Deflection: the movement of the pile axis normal to the pile, in the directions of local axes 2 (plane 2) and 3 (plane 3), measured from the undeformed position of the pile. With *Resultant of planes 2 and 3* the square root of the sum of their squares is shown instead.

© 2026 Solplan. All rights reserved.

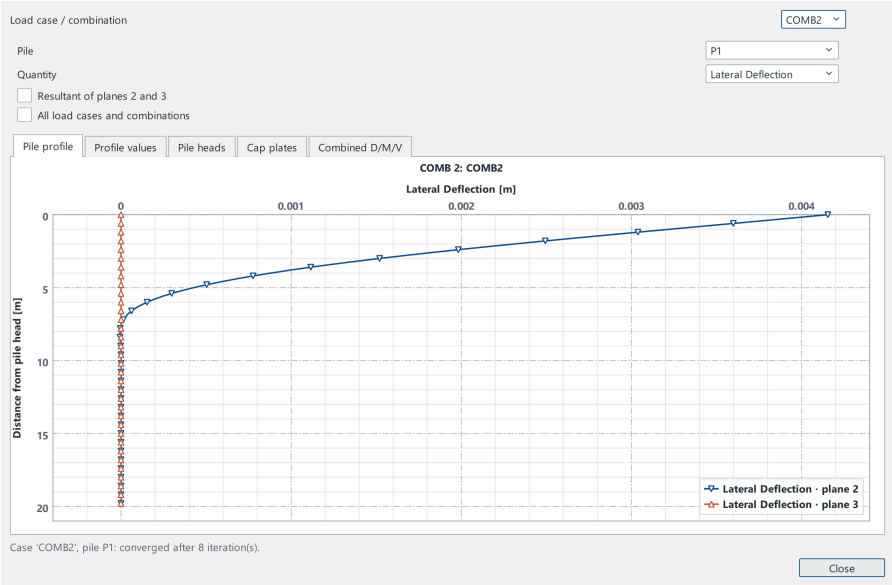


Figure 13.12 — Result Graphs, Lateral Deflection

Bending Moment: the moment $M3$ that bends the pile in plane 2 and the moment $M2$ that bends it in plane 3. The side on which a positive moment puts the pile in tension is set by the moment sign convention of the preferences. For fixed heads the largest moment is usually at the head; a second extreme of the opposite sign follows a few diameters below.

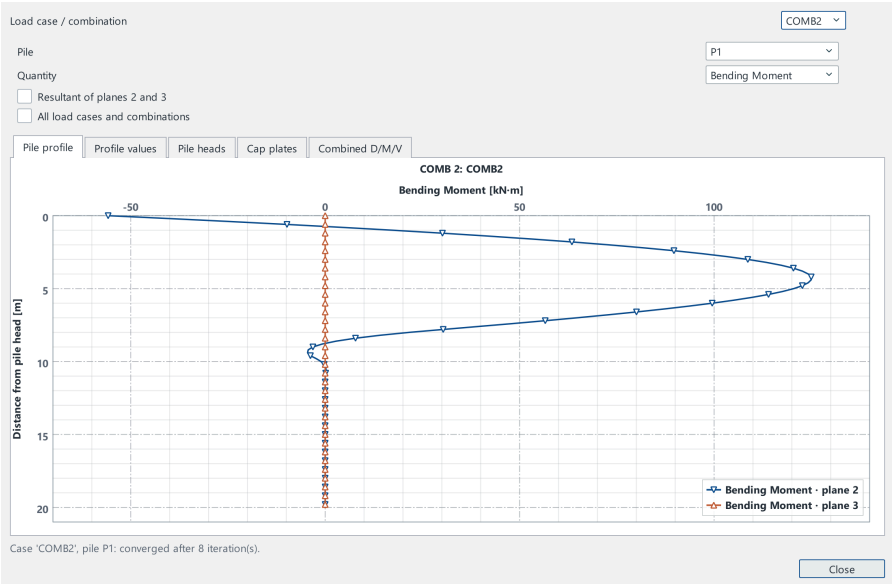


Figure 13.13 — Result Graphs, Bending Moment

Shear Force: the shear $V2$ along local axis 2 and $V3$ along axis 3. At the head it equals the lateral force that the cap transmits to the pile; below, it changes as the soil reaction is integrated along the pile.

© 2026 Solplan. All rights reserved.

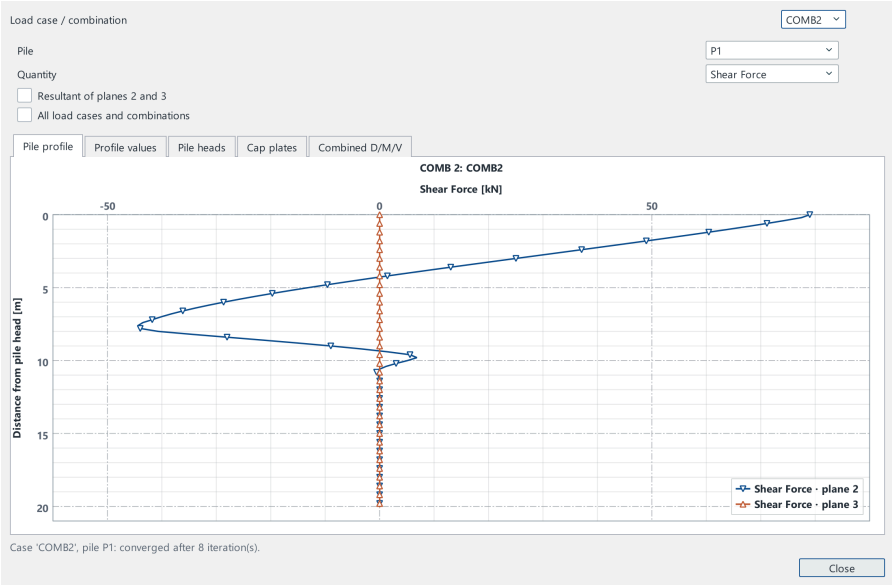


Figure 13.14 — Result Graphs, Shear Force

Soil Reaction p: the resultant soil reaction per unit length of the pile, the force that the soil exerts on the pile, opposing its movement relative to the soil. It is largest a short distance below the ground surface and changes sign where the deflection changes sign.

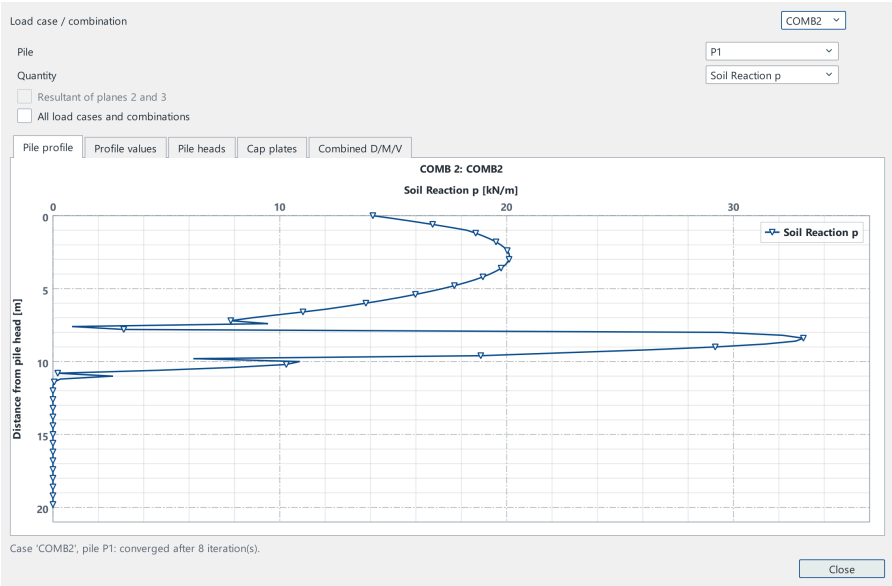


Figure 13.15 — Result Graphs, Soil Reaction p

Soil Modulus Es: the secant modulus of the p-y curve at each node, the soil reaction divided by the deflection, $E_s = p/y$. It shows how far the soil near the surface has softened: where the curve has reached its ultimate resistance the modulus is small; deeper, where the deflection is small, it approaches the initial stiffness of the curve.

© 2026 Solplan. All rights reserved.

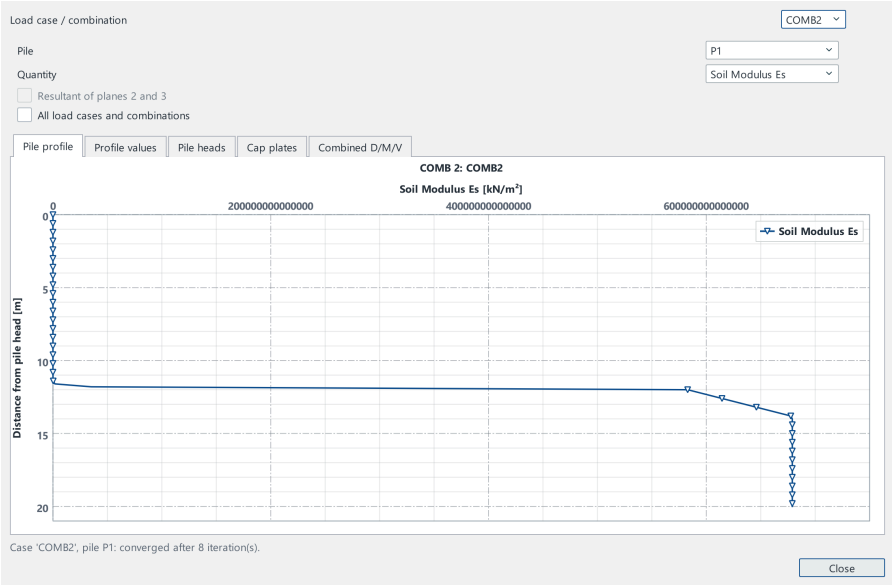


Figure 13.16 — Result Graphs, Soil Modulus Es

Soil Reaction by Plane: the soil reaction resolved into the two bending planes of the pile, to be compared with the deflection of each plane.

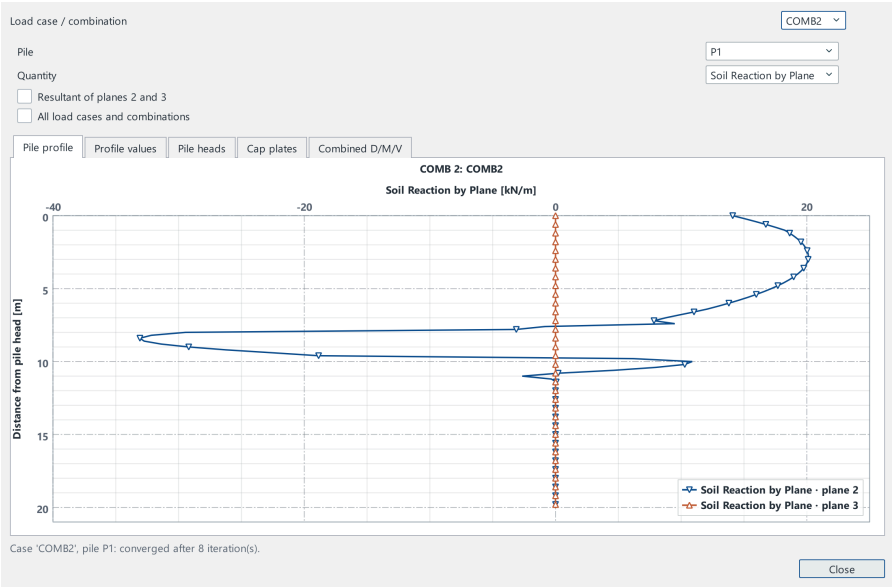


Figure 13.17 — Result Graphs, Soil Reaction by Plane

Bending Stiffness EI: the bending stiffness of the section at each node in each plane. For an elastic section it is constant; for a nonlinear section it is the secant stiffness M/φ of the moment–curvature curve at the moment of the node, and drops where the section cracks or yields. In this example the moments stay below the cracking moment and the stiffness is that of the uncracked section along the whole pile.

© 2026 Solplan. All rights reserved.

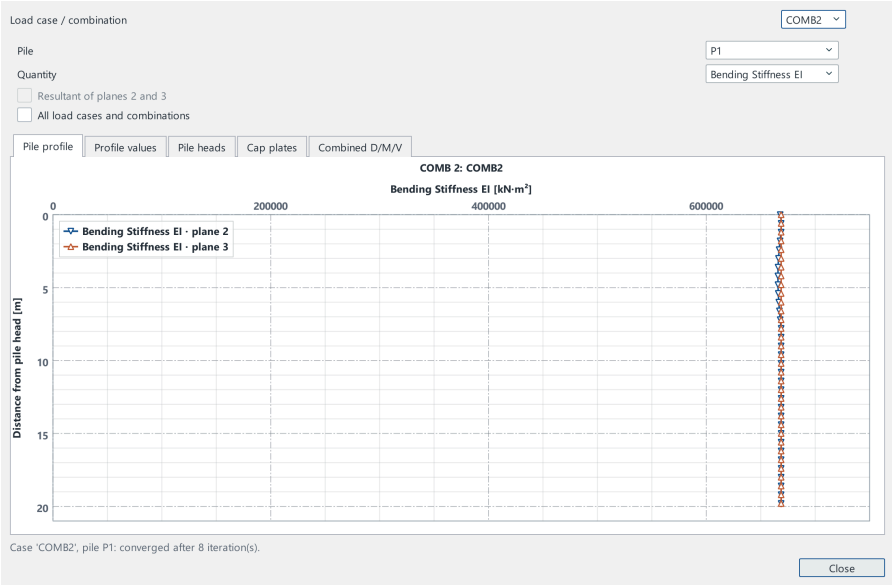


Figure 13.18 — Result Graphs, Bending Stiffness EI

Demand / Capacity: for a load combination, the ratio of the bending moment to the factored moment capacity of the section, $\sqrt{((M3/\varphi_{\mathrm{b}}M_{\mathrm{n33}})^2 + (M2/\varphi_{\mathrm{b}}M_{\mathrm{n22}})^2)}$, and the same ratio for the shear forces with φ_{v}. The capacities are those of the Design (LRFD) tab of the section, or the overrides of the combination. Here the moment capacity of the section is 1,003 kN·m, computed from its moment–curvature curve, and the largest ratio, at the head, is well below one.

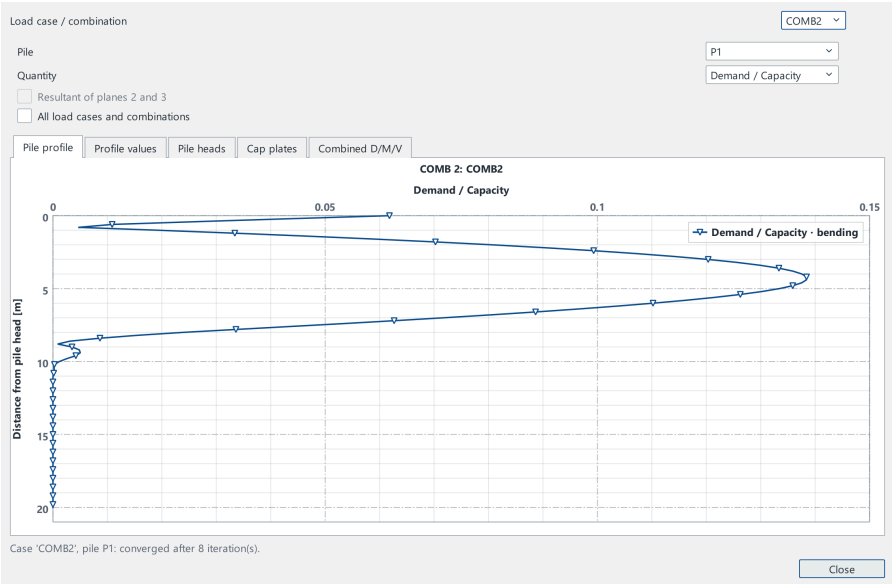


Figure 13.19 — Result Graphs, Demand / Capacity

Total Stress: the total normal stress, the axial stress plus the bending stress at the extreme fibre, for elastic sections; for nonlinear sections the stress is not computed and the graph is empty.

© 2026 Solplan. All rights reserved.

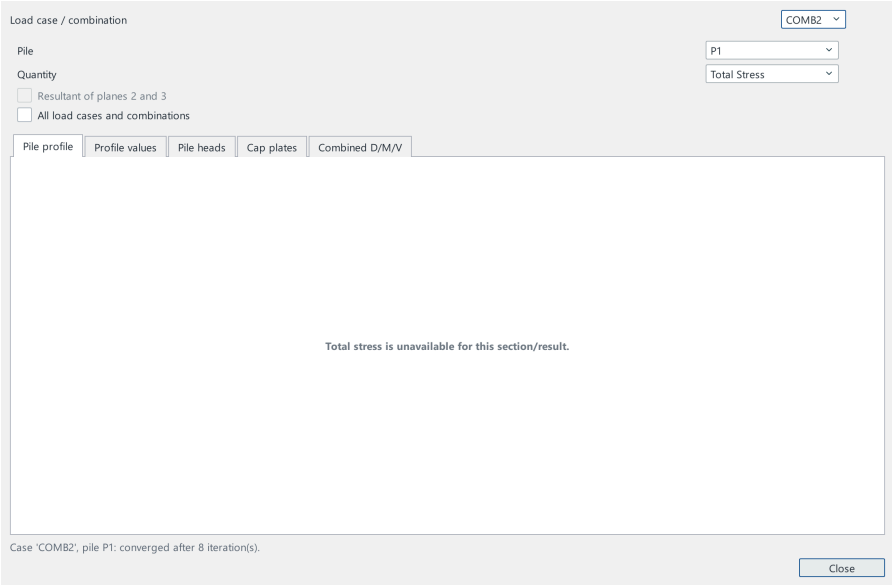


Figure 13.20 — Result Graphs, Total Stress

13.3 Tables and curves

Display ▸ Show Tables… lists the model and the results in tables (Figure 13.21). The table is chosen in the tree on the left, where the tables are grouped by subject; the box above the table filters its rows, and a right-click on a column heading filters or sorts the rows by that column. The cells can be copied (Ctrl+C), and **Export This Table…** and **Export All…** write one table or all of them. A table of more than 1 000 rows shows its first 1 000 rows; the export contains every row, and the status line states how many rows were written. Model tables can be edited in place where a cell is editable; a value that is not accepted is restored and the reason is shown under the table. A cell, and a table imported from CSV or Excel, accepts only the values that the corresponding form accepts, for example a Poisson's ratio from 0 to 0.499, a unit weight, modulus or mesh size not below zero, or an axial reduction factor above 0 and up to 1. The table *Ground Surface* lists the ground surface of every pile, from the profile, its sloping shape or the pile's own elevation (Section 9.6.1).

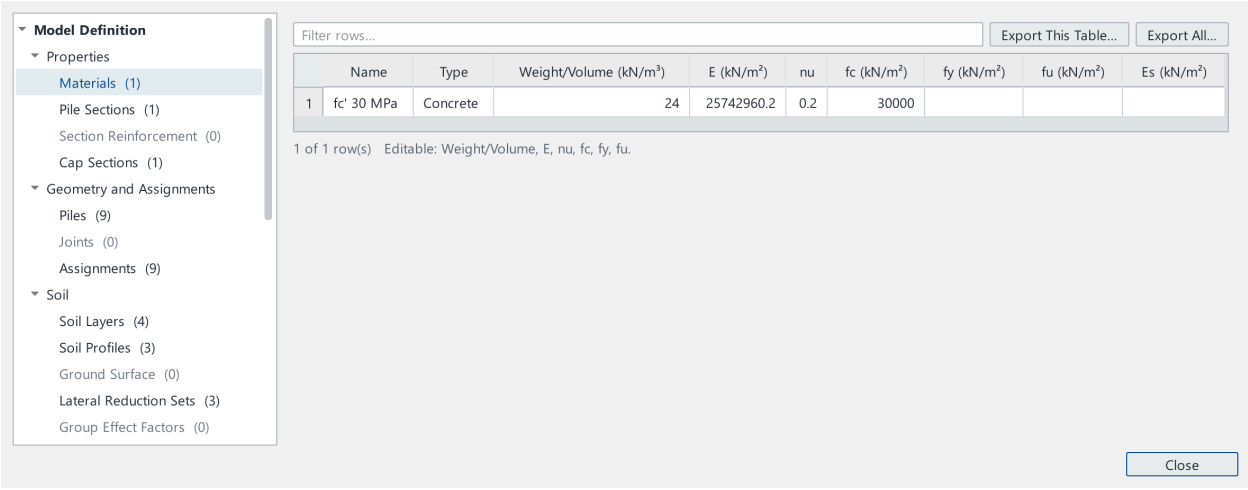


Figure 13.21 — Show Tables

Display ▸ Show Soil Curves… draws the curves that the analysis uses, on three tabs: p-y (lateral), L-D (pile head) and T-R (pile head) (Figure 13.22). With a **Pile** chosen it shows the curves that pile actually uses; for the p-y curves, **Apply p-multiplier, y-multiplier and lateral reduction** shows the curve after the group and reduction factors of the chosen **Load case** (p-multipliers are bound to a load case, because the leading row follows the load direction). With <none> it shows the plain layer curves of a **Soil profile** for a **Pile section**, of which only the diameter is used,

© 2026 Solplan. All rights reserved.

at a **Number of curves** between **From depth** and **To depth**. **Display ▸ User-Defined Curves…** draws the user-defined p-y, Axial L-D and Torsional T-R curves, for a chosen set or curve and depth, on a Graph tab and a Values tab (Depth, y, p).

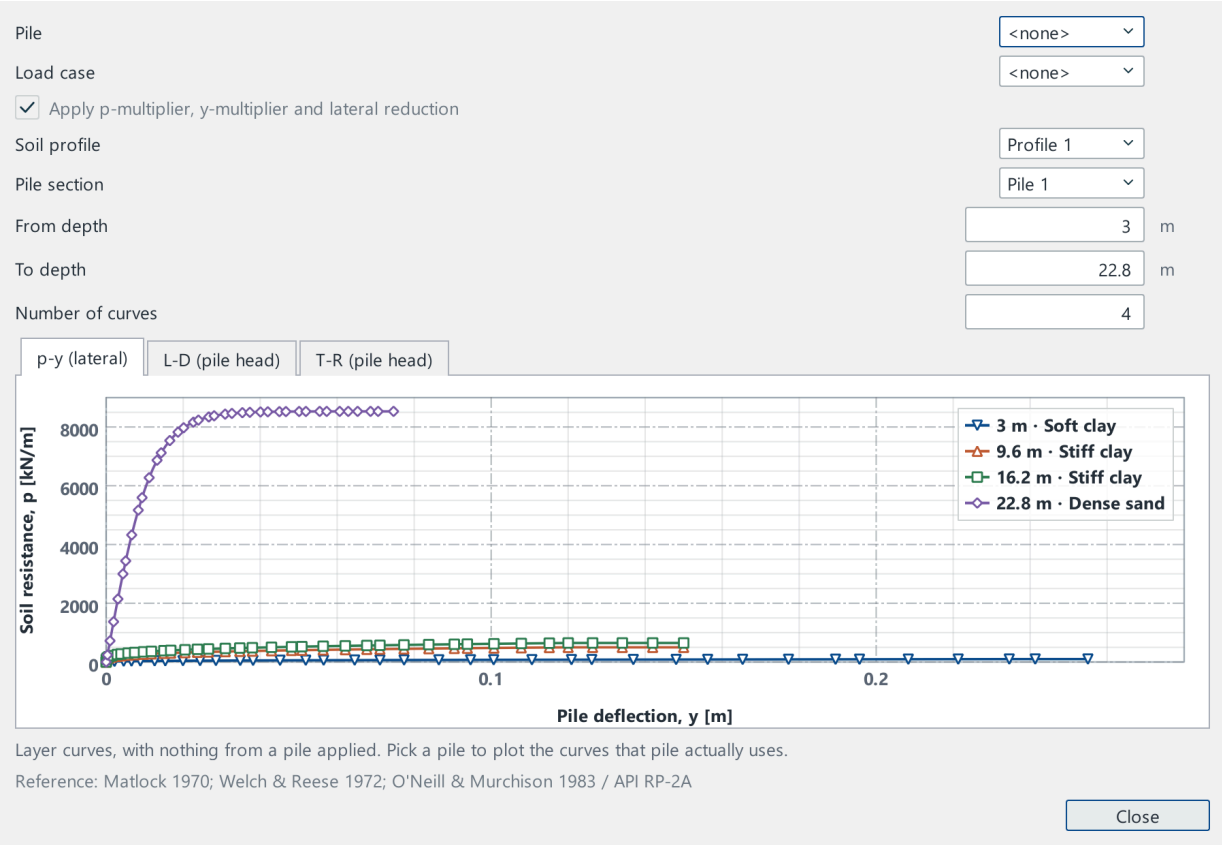


Figure 13.22 — Show Soil Curves, p-y tab

Display ▸ Show Moment-Curvature… draws the moment–curvature relation of the nonlinear sections, with its values.

13.4 Text reports

Display ▸ Show Text Report shows three plain-text reports, on three tabs (Figure 13.23):

- **Input Echo**: the complete input of the model, as read by the analysis;
- **Output Results**: the results of every case in full, pile by pile and node by node, as set by the printing code and the output variables;
- **Summary Results**: the cap displacement and the pile-head forces of every case, with the extreme values.

The report shown can be copied to the clipboard (**Copy**), saved as a text file (**Save…**), printed (**Print…**), or saved with the other two side by side (**Save All…**). **File ▸ Export ▸ Text Reports (.txt)…** writes the three reports to files.

13.4.1 Contents of the text reports

The three reports are plain text in a fixed-width layout, with the units of the model in the column heads. Each begins with a heading that names the program, the model, the report and the units. Their sections are:

Table 13.4 — Sections of the text reports

Report	Section	Content
Input Echo	MODEL INPUT	title, three-dimensional model, rigid or flexible cap, and the analysis options in force (batter, generated axial and torsional curves, layering correction)

© 2026 Solplan. All rights reserved.

Report	Section	Content
	PILE CAP; CAP SECTIONS	extent, thickness and elevation of the cap, the load reference point, restrained degrees of freedom, passive resistance, and the cap sections
	LOADING AND SOLUTION CONTROL	load patterns, joint and cap loads, load cases and combinations, the equivalent concentrated load of each case at the reference point, and the solution controls (iteration limits, tolerances, load steps)
	PILE LAYOUT; PILE HEAD CONNECTION	coordinates, batter, length and assignments of every pile, and the head connections
	MATERIALS AND PILE SECTIONS	materials and the properties of each section
	SOIL PROFILE; LAYER …	ground surface, water table, scour, and the parameters of every layer
	AXIAL LOAD-SETTLEMENT CURVES; TORQUE-TWIST CURVES; MOMENT-CURVATURE DATA	the user-defined curves and the moment–curvature data of the nonlinear sections, where present
	MULTIPLIERS AND REDUCTION FACTORS	group factors, axial and lateral reduction factors
Output Results	RESULTS BY LOAD CASE	for each case and combination: its type and status (converged, with the number of iterations), the equivalent load and the displacement of the cap at the reference point
	PILE RESPONSE › PILE …	for each pile: the head displacements in global axes, the head displacements and forces in the pile axes, and, as set by the printing code and the output variables, the values at the nodes along the pile
Summary Results	RESULTS BY LOAD CASE	for each case: the cap response, the head displacements and forces of all piles with their minimum and maximum, and the minimum and maximum values along every pile with their depths
	OVERVIEW OF ALL CASES AND COMBINATIONS	the displacement of the cap and the extreme pile-head forces of every case on one page

For the case of Example 1 (Section 16.1) the cap response of the summary reads:

```
  FORCE X  FORCE Y  FORCE Z  MOMENT X  MOMENT Y  MOMENT Z
       kN       kN       kN      kN·m      kN·m      kN·m
  -------  -------  -------  --------  --------  --------
   300.00   0.0000  -5400.0    0.0000    450.00    0.0000

[ DISPLACEMENT OF PILE CAP AT REFERENCE POINT ]

     DISP. X  DISP. Y     DISP. Z  ROT. X     ROT. Y  ROT. Z
           m        m           m       °          °       °
  ----------  -------  ----------  ------  ---------  ------
  9.2299E-04   0.0000  -0.0011760  0.0000  0.0048855  0.0000
```

and the pile-head forces, in the local axes of each pile, with their extremes:

© 2026 Solplan. All rights reserved.

```
[ PILE HEAD FORCES (LOCAL AXES) ]

     PILE   AXIAL  SHEAR 2  SHEAR 3  TORSION  MOMENT 2  MOMENT 3  STRESS
                kN       kN       kN     kN·m      kN·m      kN·m   kN/m²
  -------  ------  -------  -------  -------  --------  --------  ------
       P1  522.05   33.350   0.0000   0.0000    0.0000    43.536  3899.4
       P2  600.00   33.333   0.0000   0.0000    0.0000    43.537  4175.1
       P3  677.95   33.317   0.0000   0.0000    0.0000    43.539  4450.9
       P4  522.05   33.350   0.0000   0.0000    0.0000    43.536  3899.4
       P5  600.00   33.333   0.0000   0.0000    0.0000    43.537  4175.1
       P6  677.95   33.317   0.0000   0.0000    0.0000    43.539  4450.9
       P7  522.05   33.350   0.0000   0.0000    0.0000    43.536  3899.4
       P8  600.00   33.333   0.0000   0.0000    0.0000    43.537  4175.1
       P9  677.95   33.317   0.0000   0.0000    0.0000    43.539  4450.9

  MINIMUM  522.05   33.317   0.0000   0.0000    0.0000    43.536  3899.4
  AT PILE      P1       P3       P1       P1        P1        P1      P1
  MAXIMUM  677.95   33.350   0.0000   0.0000    0.0000    43.539  4450.9
  AT PILE      P3       P1       P1       P1        P1        P3      P3
```

The axial force is positive in compression. Shear 2 and moment 3 belong to bending in the plane of local axis 2; for the vertical piles of this group, plane 2 contains the global X axis, along which the load acts. The column STRESS is the total normal stress at the head, the axial stress plus the bending stress at the extreme fibre ($P/A + M/S$); it is given for elastic sections only. The reports can be read into a spreadsheet as fixed-width text.

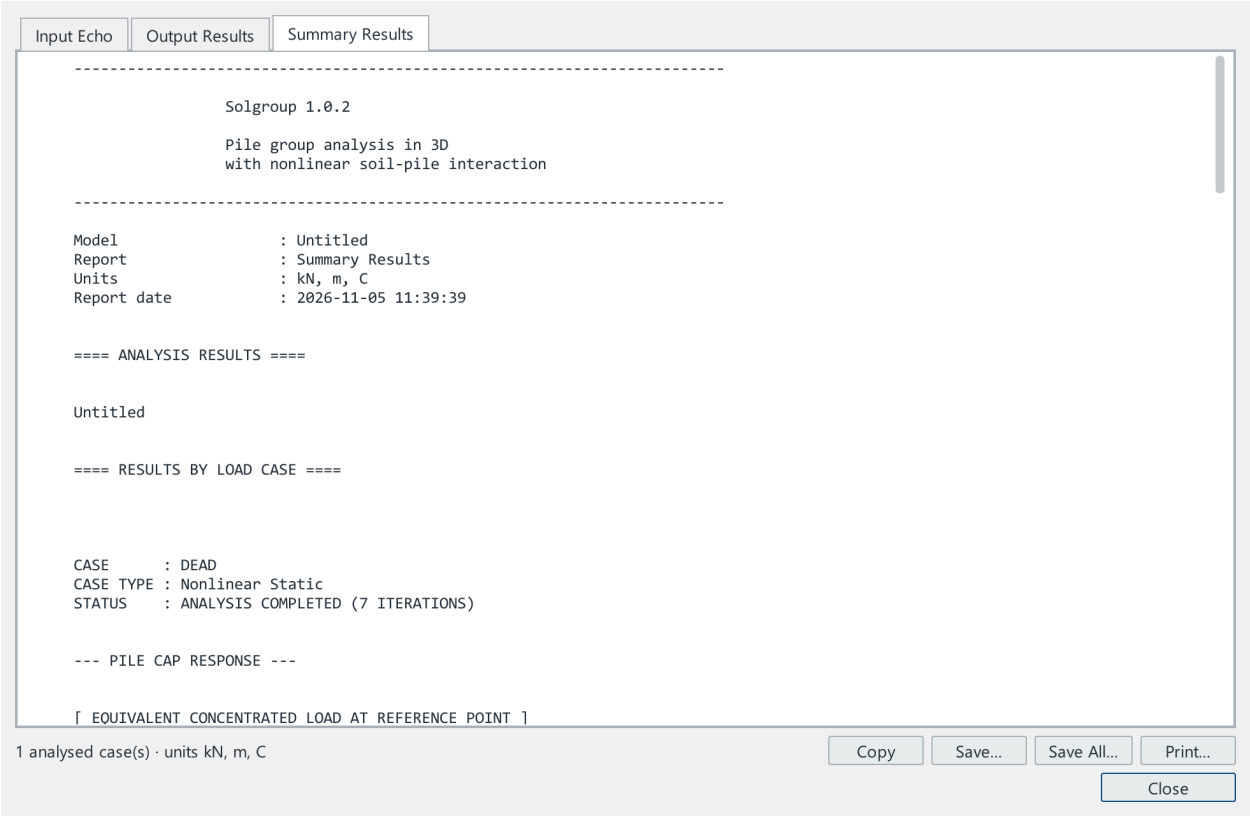


Figure 13.23 — Show Text Report, Summary Results

13.5 Checking the results

Before results are used for design, check at least the following:

1. Every case converged (summary of the run, **Command** panel), for the full load.
2. The sum of the pile-head forces balances the applied loads (summary results).
3. The deflections are within the range of the p-y curves used, and no pile has exceeded its axial capacity.

© 2026 Solplan. All rights reserved.

4. The group effects and the cyclic or static soil behaviour match the design situation.
5. For a flexible cap, the mesh matches the model (no warning from Check Model) and is fine enough: a finer mesh should not change the pile-head forces noticeably.
6. The results are not sensitive, beyond what the design can accept, to the soil parameters that are uncertain.

© 2026 Solplan. All rights reserved.

14 Reports and exports

14.1 Report Builder

File ▸ Export ▸ Report Builder (Word · PDF · Excel)… (also in the **Display** menu) builds a report of the model and its results (Figure 14.1). Choose the content, check it in the preview and export it:

Table 14.1 — Content of the Report Builder

Group	Items
Report Sections	Analysis Options; Materials & Sections; Piles & Joints; Soil; Loading; Check Model; Results — Cap Response & Pile Heads; Results — Special Analyses; Results — Flexible Cap; Results — Run Notes
Result Charts	Deflection along piles; Bending moment along piles; Shear along piles; Soil reaction along piles; Pushover curves; Foundation stiffness curves. Each selected chart is drawn for every converged load case and combination with a response (envelopes excepted, as they are not states of the structure), case by case, up to 60 charts; the cases left out by that limit are named at the end of the chapter
Model Views	3D model view; Plan view (X-Y)
Appendix	Input Echo (text); Output Results (text)

An item without content in the model is dimmed, with the reason in its tooltip, for example *No load case ran a pushover or foundation stiffness analysis*. The report is exported with one of three buttons:

- **Export Word…** (.docx) for a report that will be edited further;
- **Export PDF…** for a finished report, with a table of contents and page numbers;
- **Export Excel…** (.xlsx) with one worksheet per table, for further calculation.

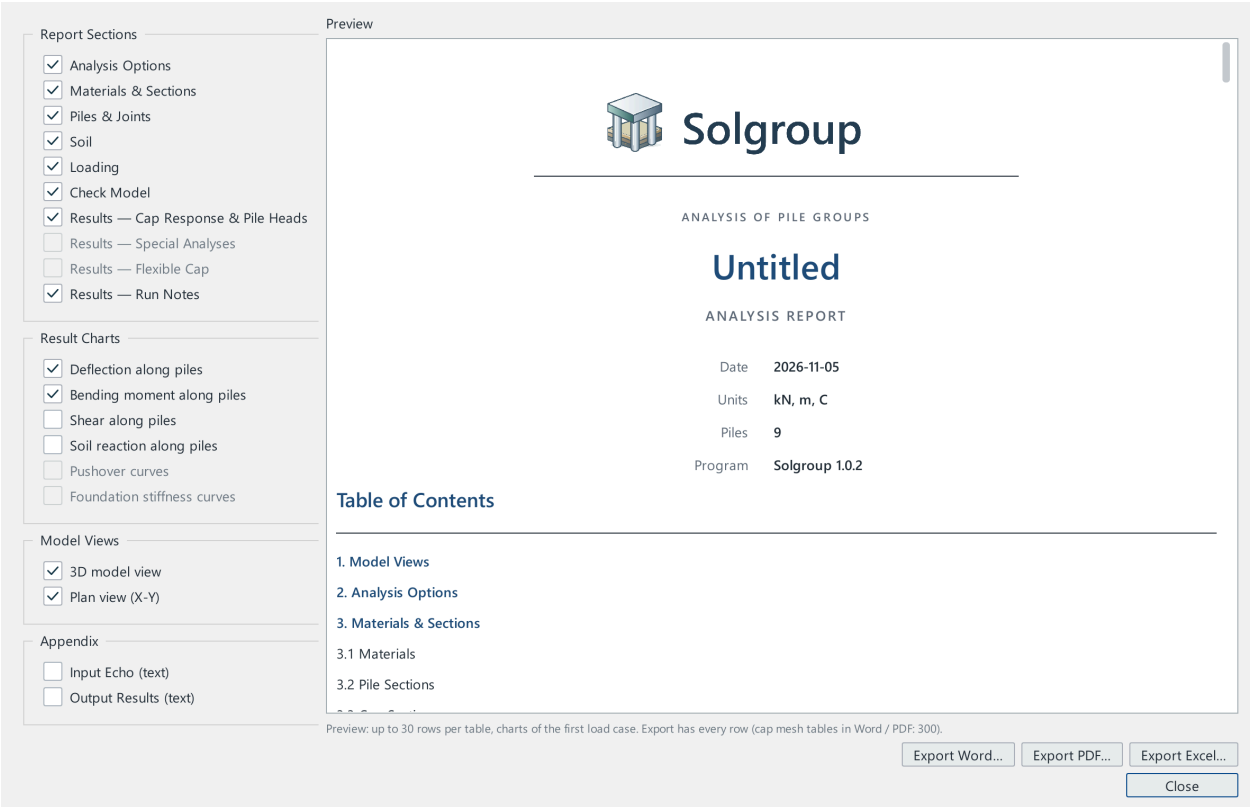


Figure 14.1 — Report Builder

© 2026 Solplan. All rights reserved.

NOTE The mesh of a large flexible cap can have tens of thousands of plates. In Word and PDF reports its tables are limited to their first rows, with a sentence that says so; the Excel report contains every row.

14.2 Other exports

Table 14.2 — Exports

Command	Content
File ▸ Export ▸ Model Tables (CSV)…, Model Tables (Excel)…	the model in tables, which can be edited and imported again
File ▸ Export ▸ Text Reports (.txt)…	the input echo, output and summary reports
File ▸ Export ▸ View Bitmap…	an image (PNG or BMP) of the active view
File ▸ Print Graphics… (Ctrl+P)	a print of the active view
Display ▸ Result Graphs and Response ▸ Export Graphs to Excel…	the data of result graphs
Copy commands in tables and graphs	values copied to the clipboard

With a trial licence, results can be viewed and models saved, but the commands above do not export. With an academic licence, exported reports and images carry the watermark of the licence.

14.3 Contents of a report

Table 14.3 lists the sections of a report of Example 1 (Section 16.1) with every item of the Report Builder selected, and the tables and columns of each section; the PDF has 15 pages (Figure 14.2). A section appears only when it has content: Results — Special Analyses and Results — Flexible Cap are left out here because the example has neither. Wide tables are split into panels in Word and PDF and kept whole in Excel. The result charts are drawn for every converged load case and combination with a response, one curve per pile; the case LIVE of the example carries no load and has no chart.

Table 14.3 — Sections, tables and columns of a report

Section	Content
Project Information	*Project Information*: Item, Value
Model Views	figures: Model view; Plan view (X-Y)
Analysis Options	*Analysis Options*: Option, Value
Materials & Sections	*Materials*: Name, Type, Weight/Volume (kN/m³), E (kN/m²), nu, fc (kN/m²), fy (kN/m²), fu (kN/m²), Es (kN/m²) *Pile Sections*: Name, Shape, Behaviour, Material, A (m²), I22 (m⁴), I33 (m⁴), J (m⁴), Perimeter (m), p-y Width (m) *Cap Sections*: Name, Thickness (m), Material, Rigid, Mesh Size (m)
Piles & Joints	*Piles*: ID, Label, X (m), Y (m), Z Head (m), Z Tip (m), Length (m), Batter from Vert (°), Batter Ratio 1:m, Azimuth (°) *Assignments*: Pile, Section, Pile Type, Head Condition, p-y Set, L-D Curve, T-R Curve, Axial Reduction, Lateral Reduction Set, Segments, Ground Elev Override (m), Local Axis Rotation (°)
Soil	*Soil Layers*: Name, p-y Model, Gamma Unsat (kN/m³), Gamma Sat (kN/m³), Side Friction (kN/m²), Tip Resistance (kN/m²), Params *Soil Profiles*: Profile, Top Elev (m), Bottom Elev (m), Layer, k Override (kN/m³), Ground Elev (m), Water Elev (m), Scour (m)
Loading	*Load Patterns*: Name, Type, Self Weight Multiplier *Joint Loads*: Target, Pattern, Fx (kN), Fy (kN), Fz (kN), Mx (kN·m), My (kN·m), Mz (kN·m) *Load Cases*: Name, Type, Initial Condition, Patterns, p-Mult Set, Steps, Soil Behaviour, Cycles, Tol Equilibrium, Tol Pile Deflection (m), Iter Group, Iter Pile, Processors, Printing, Output Variables, Overrides
Check Model	*Check Model*: Level, Code, Message

© 2026 Solplan. All rights reserved.

Section	Content
Results — Cap Response & Pile Heads	*Cap Response*: Load Case, Ux (m), Uy (m), Uz (m), Rx (°), Ry (°), Rz (°), Converged, Iterations, Message *Pile Head Results*: Load Case, Pile, Converged, Ux (m), Uy (m), Uz (m), Rx (°), Ry (°), Rz (°), Axial (kN), Shear-2 (kN), Shear-3 (kN), Moment-2 (kN·m), Moment-3 (kN·m), Torsion (kN·m)
Results — Charts	figures: Resultant Deflection vs Distance — DEAD; Resultant Moment vs Distance — DEAD; Resultant Shear vs Distance — DEAD; Resultant Soil Reaction vs Distance — DEAD
Results — Run Notes	*Run Notes*: Load case, Note
Appendix — Input Echo	the input echo of the text reports (Section 13.4), in a fixed-width font
Appendix — Output Results	the output report of the text reports, in a fixed-width font

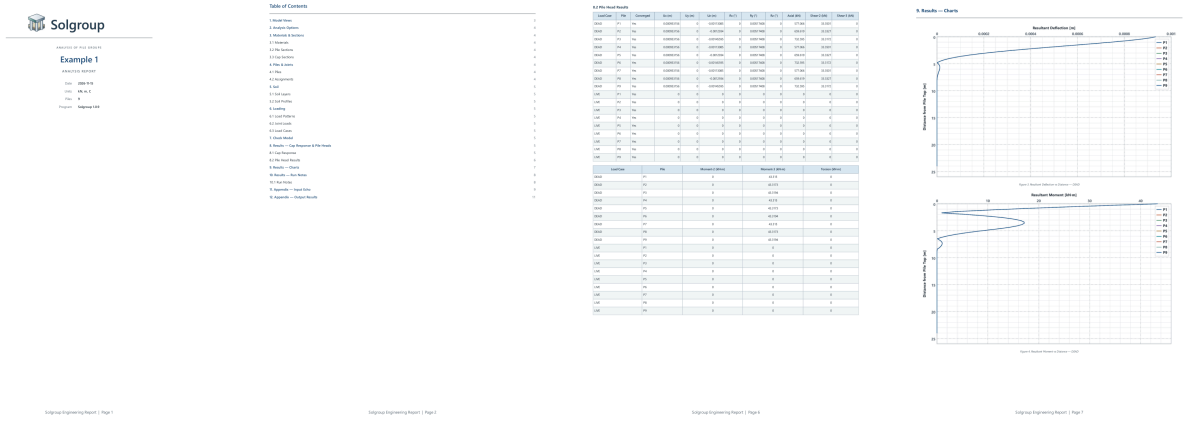


Figure 14.2 — Pages of the PDF report of Example 1: cover, table of contents, pile head results and result charts

© 2026 Solplan. All rights reserved.

15 Options, tools and help

15.1 Preferences

Options ▸ **Preferences…** holds the settings of the program (Figure 15.1):

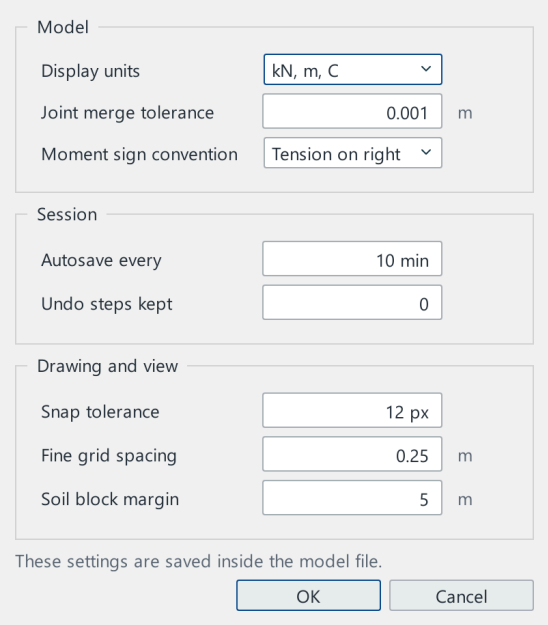


Figure 15.1 — Preferences

Table 15.1 — Preferences

Setting	Meaning
Display units	the unit system for input and display (Chapter 5): kN, m; kgf, m; tonf, m; kip, in; kip, ft; or N, mm. The model is stored in kN and m. Switching between SI and English units clears the results, which are computed again by the next run.
Joint merge tolerance	distance within which two joints are merged
Moment sign convention	which side of the pile a positive moment puts in tension (tension on right or on left). It flips the sign of the moment diagrams in graphs, contour plots and exported graphs; tables and text reports keep the sign of the analysis.
Autosave every	interval at which a recovery copy (<name>.autosave.ggp next to the model) is written while there are unsaved changes; it is offered when the model is opened again. 0 turns autosave off.
Undo steps kept	0 means unlimited; a new limit applies from the next model opened or created
Snap tolerance	distance in pixels within which the cursor snaps to a point
Fine grid spacing	spacing of the fine grid of the snap
Soil block margin	how far the drawn soil block extends past the model

15.2 Other options

Table 15.2 — Commands of the Options menu

Command	Use
Rebuild Pile Layout from a Template…	replaces the piles of the model by a layout from a template, with the same fields as the New Model window; the soil, the sections and the load cases are kept

© 2026 Solplan. All rights reserved.

Command	Use
Colours…	the colours of the drawing, each chosen with its colour button: Pile; Pile, selected; Pile, hovered; Pile cap; Pile / cap without a section; Pile cap edge; Grid line; Grid bubble; Water table; Ground surface; Scour line; Object label; Global axis X; Global axis Y; Global axis Z; Joint; Reference point / line; Load arrow; Moment arrow; Free-field soil movement; Deformed shape. **Reset to Defaults** restores them.
Set Display Options…	see Table 8.8
Include Case / Combination Numbers on Graphs	names the cases on graphs with their number, as LC *n* or COMB *n*
Show Vertical Lines from Pile Heads	draws from each pile head a vertical line of the length of the pile, a reference for battered piles
Reset Toolbar Layout	restores the toolbars and panels to their default positions

15.3 Tools, Help and Window

- **Tools ▸ Model Statistics…** counts the objects and definitions of the model; **Open Model Folder** opens the folder of the model file.
- **Help ▸ Licence…** shows the account and the licence in use, and offers offline borrowing, returning a borrowed licence, managing the account and signing out (Section 2.3). **About Solgroup…** shows the version and the copyright notice (Figure 15.2).
- The **Window** menu shows or hides the panels (Model Explorer, Properties, Workflow, Messages) and the toolbars.



Figure 15.2 — About Solgroup

15.4 Keyboard shortcuts

Table 15.3 — Keyboard shortcuts

Key	Command
Ctrl+N, Ctrl+O, Ctrl+S, Ctrl+Shift+S	New Model, Open, Save, Save As
Ctrl+P	Print Graphics
Ctrl+Z, Ctrl+Y	Undo, Redo
Ctrl+X, Ctrl+C, Ctrl+V, Del	Cut, Copy, Paste, Delete
Ctrl+R	Replicate
Ctrl+A, Ctrl+Shift+A	Select All, Select None
Ctrl+E	Set Display Options
Ctrl+T	Show Tables
Home, Z, F2	Restore Full View, Zoom to Selection, Refresh Window
R, F, Q, C	Reshape, Draw Pile, Quick Draw Pile, Draw Poly Cap
F5, F6	Run Analysis, Check Model

© 2026 Solplan. All rights reserved.

Key	Command
Esc	cancel the current tool (drawing, line or polygon selection, zoom, rotate, reshape) and return to selection; with nothing to cancel, clear the selection
Backspace	while drawing, remove the last clicked point

© 2026 Solplan. All rights reserved.

16 Worked examples

This chapter follows eighteen analyses of pile groups and six analyses of pile sections from the input to the result. All numbers were computed with Solgroup 1.0.10; each model can be rebuilt from the steps given.

Unless an example states otherwise, its model has the soil profile and water table, the pile section *Pile 1* and the cap section *CAP-1000* of Example 1 (Section 16.1). A model started with **File ▸ New Model…** has one soil layer and no sections, so these are defined first, as Example 1 describes.

The loads follow one convention, as in Getting started (Chapter 6). The pattern DEAD carries only the self weight of the cap (self weight multiplier 1), computed by Solgroup; nothing is typed into it. The loads brought to the cap by the structure above it are typed into a pattern *SDL* of type SUPERDEAD (self weight multiplier 0), and the load case DEAD applies both patterns, DEAD and SDL, with a factor of 1 (**Define ▸ Load Cases…**, **Modify…**, **Add Pattern**). Environmental loads that are compared or combined (berthing, mooring, current, waves, an impact) have patterns of their own. Unless an example says so, "the load" of an example is the load in SDL, and the results are those of the load case DEAD, which includes the weight of the cap.

16.1 Example 1: rectangular group under combined load

16.1.1 Model

1. **File ▸ New Model…**, template **Rectangular Pile Group**: 3 × 3 piles at a spacing of 1.8 m in X and Y, pile length 24 m, pile heads at elevation 0, vertical piles, a cap created automatically with an edge offset of 0.5 m.
2. The new model is completed as in Getting started (Chapter 6): the soil profile with the ground surface at elevation 0, soft clay (Soft Clay (Matlock), 15 / 16 kN/m^3, c_u = 25 kPa, ε_{50} = 0.02) to −8 m, stiff clay (Stiff Clay without Free Water, 17.5 / 19 kN/m^3, c_u = 120 kPa, ε_{50} = 0.005) to −20 m and dense sand (API Sand (O'Neill), 18 / 20.5 kN/m^3, φ = 38°, k = 61 000 kN/m^3) to −40 m, with the water table at −1 m; the pile section *Pile 1*, an Elastic section (user-defined EI) of 0.6 m diameter in the concrete of 30 MPa, assigned to every pile; and the cap section *CAP-1000*, 1 m thick, assigned to the cap.
3. The cap is rigid (**Assign ▸ Cap ▸ Cap Behaviour…**) and the pile heads are fixed to it (project default).
4. One load on the cap (**Assign ▸ Cap Loads ▸ Point Forces / Moments…**), in the pattern SDL: F_x = 300 kN, F_z = −5400 kN (downward) and M_y = 450 kN·m. The load case DEAD applies the patterns DEAD (the self weight of the cap) and SDL, each with a factor of 1.
5. **Analyze ▸ Check Model…** reports no error; **Analyze ▸ Run Analysis**.

After the run, **Display ▸ Results in Canvas ▸ Force Diagrams in Canvas…** draws the bending moment on the piles in the model view, with the values at the head and at the largest moment below it (Figure 16.1).

© 2026 Solplan. All rights reserved.

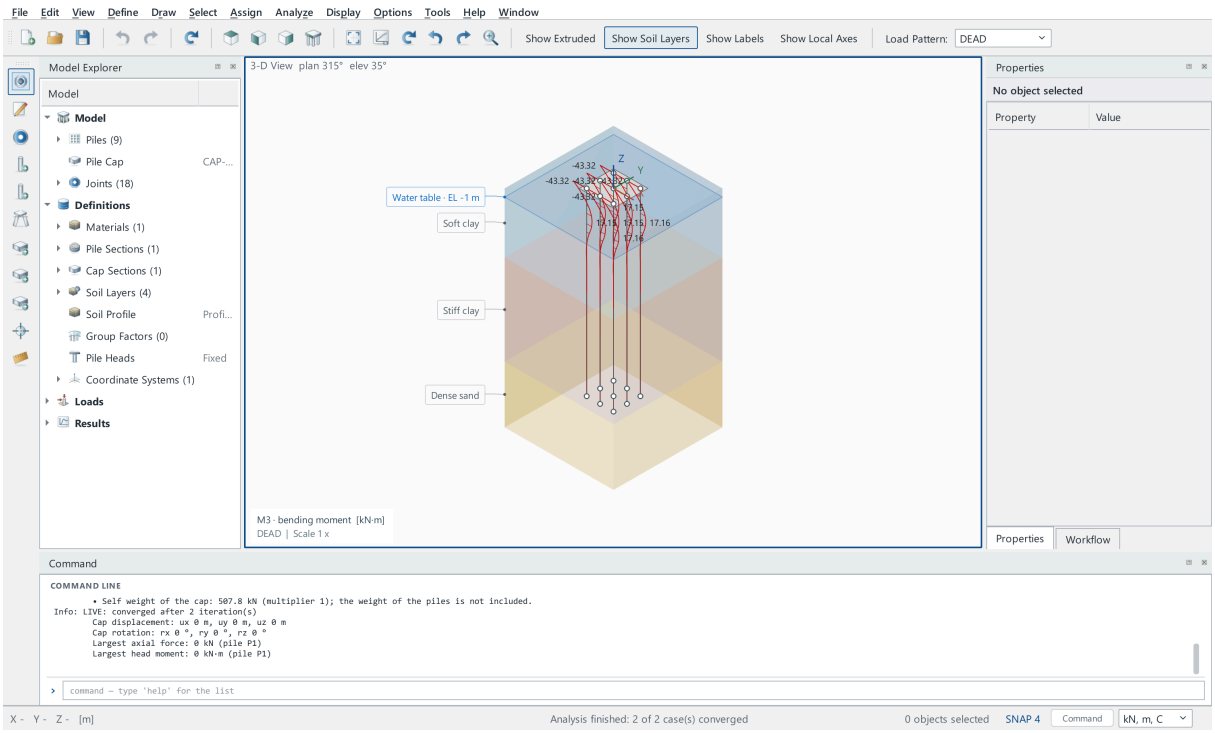


Figure 16.1 — Example 1 — bending moment *M*3 of the piles, case DEAD

16.1.2 Results

The case converged in 7 iterations. The cap moves 0.933 mm in X and −1.293 mm vertically, and rotates 0.0903 mrad about Y. Table 16.1 lists the forces at the head of each pile. The vertical load and the moment are shared between the rows: the moment adds compression to the piles on one side of the cap and relieves those on the other, while the lateral load is shared almost equally.

Table 16.1 — Pile head forces of the example

Pile	X (m)	Y (m)	Axial (kN)	Shear (kN)	Head moment (kN·m)	Largest moment below the head (kN·m)
P1	−1.8	−1.8	577.1	33.35	43.32	17.15
P2	0.0	−1.8	659.6	33.33	43.32	17.15
P3	1.8	−1.8	732.6	33.32	43.32	17.16
P4	−1.8	0.0	577.1	33.35	43.32	17.15
P5	0.0	0.0	659.6	33.33	43.32	17.15
P6	1.8	0.0	732.6	33.32	43.32	17.16
P7	−1.8	1.8	577.1	33.35	43.32	17.15
P8	0.0	1.8	659.6	33.33	43.32	17.15
P9	1.8	1.8	732.6	33.32	43.32	17.16

The sum of the axial forces is 5907.8 kN and the sum of the shear forces 300.0 kN, which equal the applied vertical and lateral loads. Figure 16.2 shows the deflection and the bending moment along the piles of the first row. The moment is largest at the fixed head, changes sign a few metres below it, and reaches a smaller maximum of opposite sign; the deflection dies out within the upper five metres of soft clay.

© 2026 Solplan. All rights reserved.

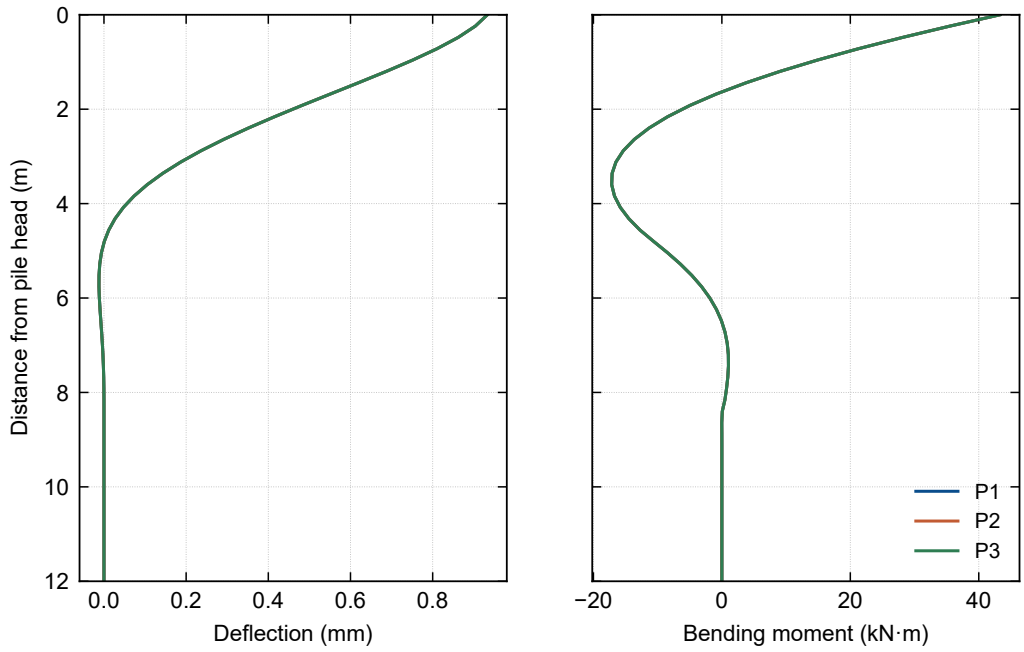


Figure 16.2 — Deflection and bending moment along piles P1 to P3

16.2 Example 2: jetty bents with battered piles

A jetty deck on bents is loaded horizontally across the bents, for example by a berthing ship. Battered piles carry much of the horizontal load by axial force, which makes the foundation stiffer.

16.2.1 Model

1. **File ▸ New Model…**, template **Trestle / Bent (jetty)** with its defaults: four bents at 6 m, four piles per bent at 3 m, the outer pile pair of each bent battered at 9.46° (1:6) away from the bent, piles 24 m long, and a deck (cap) over all bents. The bents lie along Y and their piles across X. The pile heads and the deck are at +4.0 m, the seabed (ground surface) is at 0.0 m and the water at +3.0 m, so the piles stand free through the water before they enter the soil.
2. The deck is rigid and the pile heads are fixed.
3. **Analyze ▸ Analysis Options…**: **Include axial load variation along the piles** is ticked, so that the axial force along each pile comes from its t-z analysis and can be drawn.
4. Load on the deck in the pattern SDL: F_x = 800 kN across the bents and F_z = −3000 kN from the surfacing, services and equipment on the deck. The deck is the cap itself, so its own weight comes from the self weight multiplier of DEAD and is not typed.
5. For comparison the same model is built with all piles vertical (outer batter 0).

© 2026 Solplan. All rights reserved.

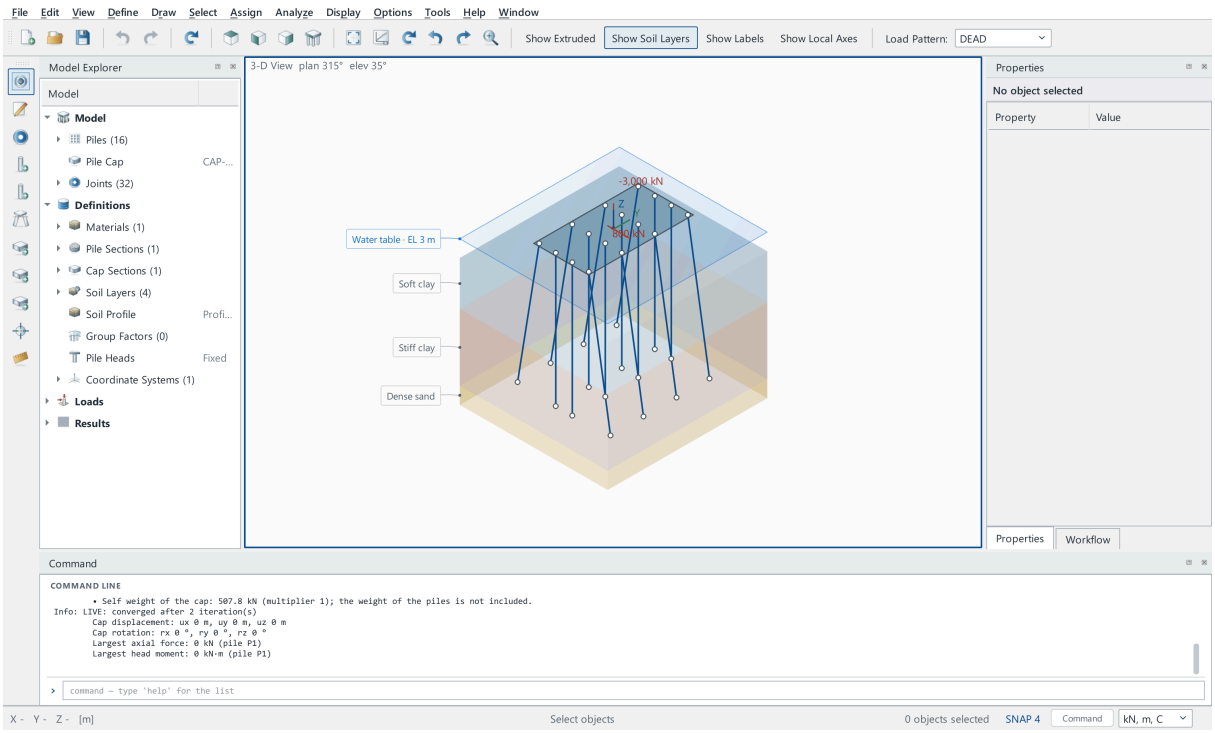


Figure 16.3 — Example 2 — the jetty model with its load on the deck

16.2.2 Results

With the battered piles the deck moves 8.644 mm in X; with all piles vertical it moves 13.454 mm, so that the batter reduces the horizontal movement by 36 %. Table 16.2 lists the head forces of the piles of the first bent.

Table 16.2 — Head forces of the first bent (piles P1 to P4, from −X to +X)

Pile	Axial, battered (kN)	Axial, vertical (kN)	Shear, battered (kN)	Shear, vertical (kN)	Moment, battered (kN·m)	Moment, vertical (kN·m)
P1	325.6	395.0	40.7	50.1	158.6	193.7
P2	644.0	446.6	39.8	50.0	155.7	193.7
P3	355.5	498.3	40.1	50.0	156.0	193.8
P4	576.9	550.0	39.2	49.9	153.2	193.7

With vertical piles the axial forces vary linearly across the bent, from the rotation of the deck, and every pile takes the same shear. With battered piles the pile battered in the direction of the load (P4) gains compression and the pile battered against it (P1) loses compression: the horizontal load is partly carried as a pair of axial forces, and the shears and moments of the vertical piles decrease. The batter here is mild; steeper batters carry more of the load axially.

Display ▸ Results in Canvas ▸ Show Contour Plot… with **N · axial force along pile** colours each pile along its length by its axial force (Figure 16.4): the force is largest at the head and decreases with depth as the shaft friction carries it into the soil. The same component in **Force Diagrams in Canvas…** draws it as a diagram on every pile.

© 2026 Solplan. All rights reserved.

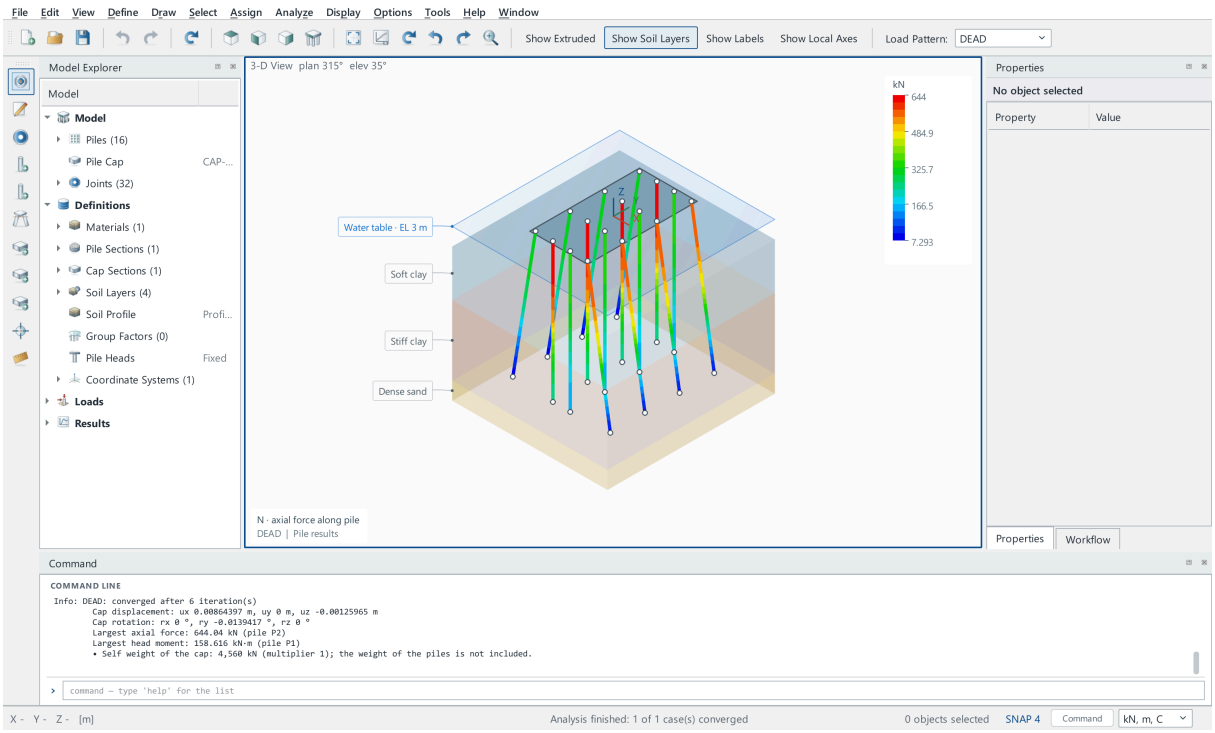


Figure 16.4 — Example 2 — axial force along the piles, battered piles

16.3 Example 3: rigid and flexible caps under two columns

A cap that is thin compared with the spacing of the piles bends under load, and the piles near the loads receive more than their share. This example carries two columns on a square group and compares a rigid cap with flexible caps of decreasing thickness.

16.3.1 Model

1. Template **Rectangular Pile Group**: 3 × 3 piles at 2.4 m, otherwise as in example 1. The cap is 5.8 m square.
2. Two columns stand on the middle line of the cap, 2.4 m apart, each halfway between two piles. Each column is a reference point (**Draw ▸ Draw Reference Point**, then the exact position in the Properties panel): *C1* at $X = -1.2$ m and *C2* at $X = +1.2$ m, $Y = 0$.
3. Each column carries $F_z = -2700$ kN in the pattern SDL: the reference point is picked in the Model Explorer, and **Assign ▸ Cap Loads ▸ Point Forces / Moments…** applies the force at **Selected points / pile-head joints**. The pattern DEAD carries the self weight of the cap; the load case DEAD applies both.
4. First the cap is rigid. Then **Rigid cap** is unticked in the cap section (**Define ▸ Section Properties ▸ Cap Sections…**); the plate mesh is generated at once, 168 plates. Along X the plates are a little narrower, so that a node falls under each column as well as under every pile head. The analysis is repeated with the cap section 1.0, 0.5 and 0.3 m thick; the mesh follows each change of thickness by itself.

16.3.2 Results

Table 16.3 — Axial forces in the piles under two columns

Cap	Corner pile P1 (kN)	Edge pile P2 (kN)	Centre pile P5 (kN)	Settlement (mm)
Rigid	689.7	689.7	689.7	−1.36
Flexible, 1.0 m	573.7	687.3	929.0	−1.92
Flexible, 0.5 m	265.3	595.9	1543.8	−3.36
Flexible, 0.3 m	106.6	456.6	2081.1	−5.11

© 2026 Solplan. All rights reserved.

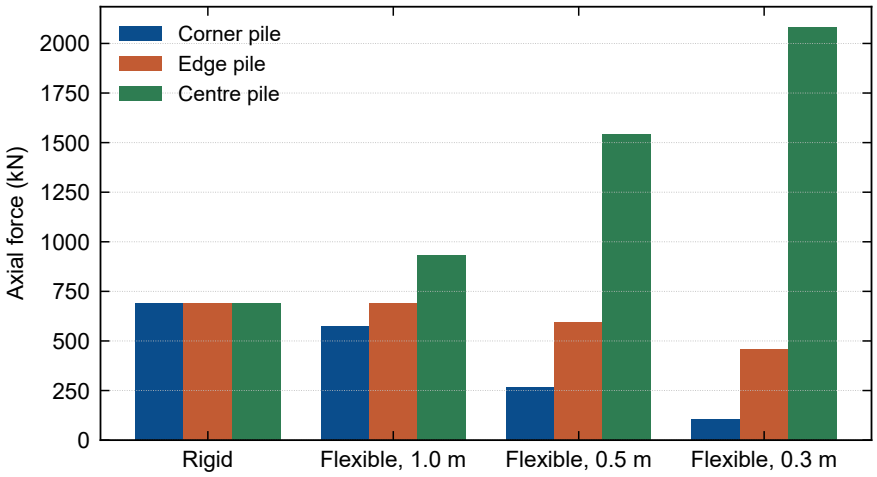


Figure 16.5 — Axial forces of corner, edge and centre piles for a rigid cap and for flexible caps

The rigid cap moves as one block: the two columns are placed symmetrically, so the nine piles share the column loads and the weight of the cap equally. A flexible cap of the same thickness already gives the centre pile, between the two columns, more than the corner piles; as the cap becomes thinner the load concentrates in the line of piles under the columns and the corner piles carry little. The settlement reported for a flexible cap is that of the cap reference point at its centre. In each case the sum of the axial forces equals the column loads plus the weight of the cap. A rigid cap is a good model only when the cap is stiff compared with the piles; the flexible cap shows when it is not.

The bending of a thin cap also turns the pile heads, so the piles carry head moments without any horizontal load: see **Display ▸ Show Tables ▸ Pile Head Results**, columns *Moment-2* and *Moment-3*.

The contours of the plate moments (**Display ▸ Results in Canvas ▸ Show Contour Plot…**, *M*11 or *M*22), best read in plan (**View ▸ XY**), show where the cap bends most: under the two columns, with moments of the other sign over the piles beside them (Figure 16.6).

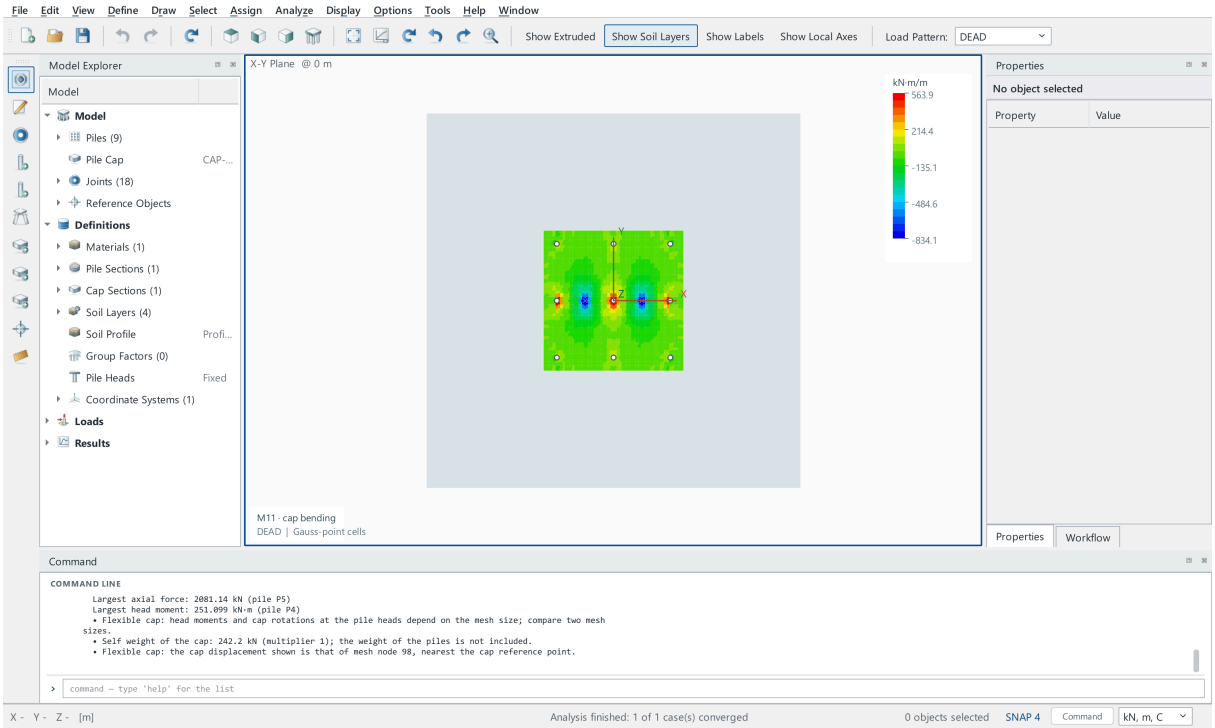


Figure 16.6 — Example 3 — plate moment *M*11 of the 0.3 m cap under the two columns, in plan

© 2026 Solplan. All rights reserved.

16.4 Example 4: group effect of closely spaced piles

The group of Example 1 (3 × 3 piles of 0.6 m diameter at 1.8 m, three diameters) carries F_x = 900 kN and F_z = −2 700 kN at the cap. It is analysed three times: without group reduction; with a set of the method *AASHTO LRFD Table 10.7.2.4-1*, load direction 0° (**Define** ▸ **Soil** ▸ **Lateral Group Effect (p-y)…**, **Compute from spacing**), which gives the p-multipliers 0.8, 0.4 and 0.3 to the leading, second and third rows at s/D = 3; and with a set of the method *Closely-spaced piles*, which computes a factor for every pile from the positions of the others. The set is selected for the model in **Analyze** ▸ **Set Analysis Options…**.

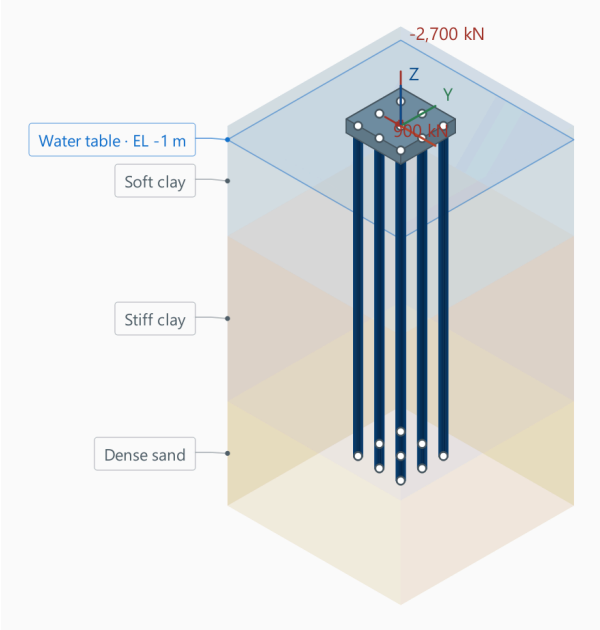


Figure 16.7 — Example 4 — the model in the 3D view

Table 16.4 — Example 4 — cap displacement and mean pile-head shear of each row

Group effect	Cap Ux (mm)	Leading row (kN)	Second row (kN)	Third row (kN)	Largest head moment (kN·m)
None	6.30	100	100	100	191
AASHTO, by rows	14.98	139	88	74	325
Closely-spaced piles	10.79	120	90	90	265

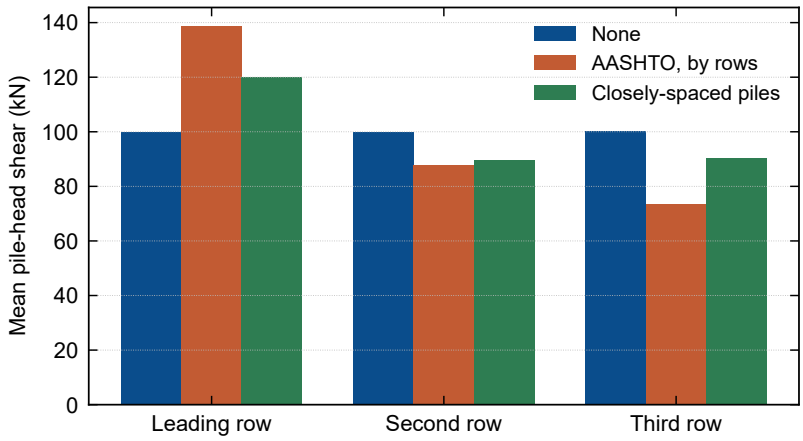


Figure 16.8 — Example 4 — mean pile-head shear of each row

© 2026 Solplan. All rights reserved.

Without reduction the three rows share the load equally. The group factors make the soil softer, so the cap moves more, and they make the rows unequal: the leading row, which pushes into undisturbed soil, takes the largest share, and the pile moments grow accordingly. The two methods differ in degree: the row table of AASHTO applies a single factor to a whole row, while the method of closely-spaced piles also distinguishes the corner and the middle piles of a row and follows the direction in which the cap actually moves. In every case the shears add up to the applied 900 kN. Group factors should be used whenever the spacing is less than about five diameters in the direction of loading.

16.5 Example 5: connection of the pile heads

Examples 5 to 11 each start again from the group and load of Example 4 (F_x = 900 kN, F_z = −2 700 kN) without group reduction and with fixed pile heads, and change only what they describe; a model carried on from the previous example keeps its group effect, head connection or extra load.

The same group and load are analysed with three pile-head connections (**Assign ▸ Pile ▸ Project Default Pile-Head Connection…**): fixed, pinned, and elastic with rotational springs of 20,000 kN·m/rad in both planes.

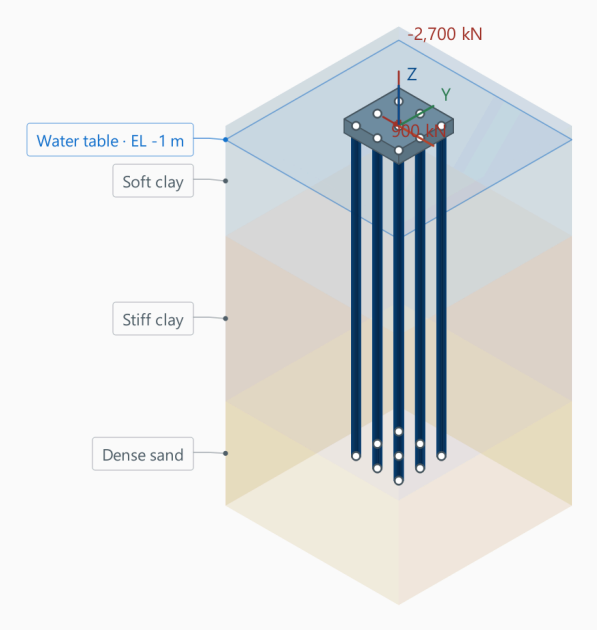


Figure 16.9 — Example 5 — the model in the 3D view

Table 16.5 — Example 5 — effect of the head connection

Connection	Cap Ux (mm)	Head moment (kN·m)	Largest moment (kN·m)	at depth (m)
Fixed	6.30	191	191	0.0
Elastic	15.74	72	139	4.1
Pinned	22.27	0	194	3.6

© 2026 Solplan. All rights reserved.

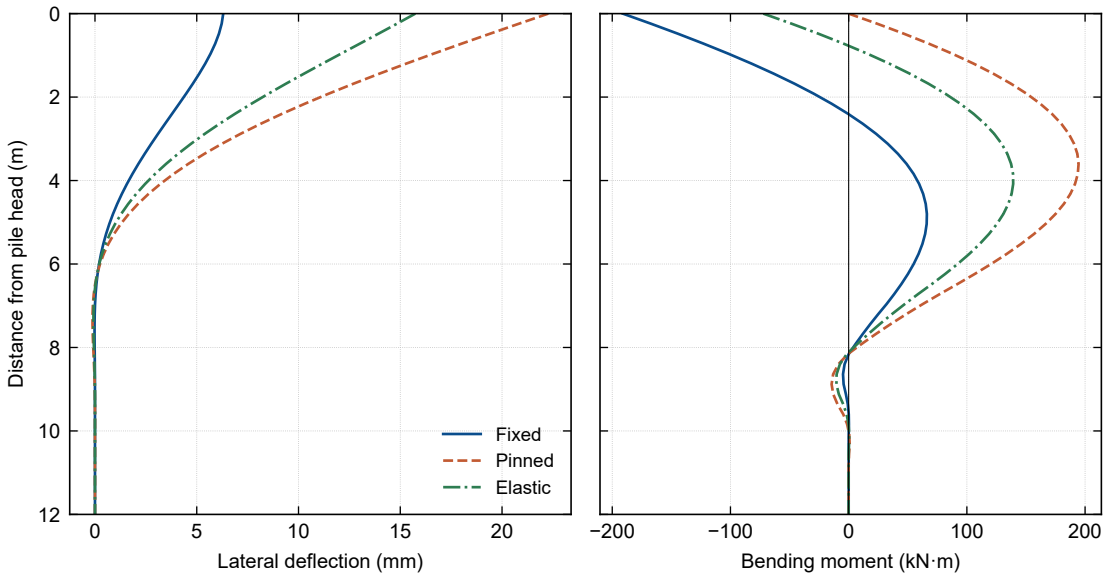


Figure 16.10 — Example 5 — deflection and moment of the centre pile for the three connections

A fixed head restrains the rotation of the pile, so the largest moment is at the head and the cap moves least. A pinned head carries no moment: the pile rotates freely at the cap, the cap moves several times more, and the largest moment moves down into the soil, where it is of similar size. The elastic connection lies between the two. The connection of real piles is rarely fully fixed; a range of connections should be analysed when the cap detail is uncertain, and the pile designed for the largest head moment and the largest moment in the soil that result.

16.5.1 A prescribed settlement

A load case can prescribe a movement of the cap instead of a force. How much vertical load settles this group by 10 mm, while the frame above keeps the cap level? In **Define ▸ Load Cases…** the case DEAD is copied (**Add Copy…**, name *SETTLE*); its loads stay, so the 900 kN horizontal load still acts. In **Specified Displacements…** the box *Prescribe cap degrees of freedom in this load case* is ticked, Uz is set to *Displacement*, −0.01 m, and Ry to *Displacement*, 0: a prescribed zero holds that rotation. The other degrees of freedom stay force controlled. A case like this cannot be part of a combination, because its load at a prescribed degree of freedom is an outcome of the analysis (Section 11.3).

After the run, **Display ▸ Show Tables…**, table *Cap Reactions*, has one row for each case that prescribes a movement: 10 mm of settlement takes 27,076.9 kN down at the cap, and holding the cap level against the horizontal load takes a moment of −1,835.4 kN·m. At this level of settlement the group gives about 2,708 kN per millimetre, ten times the column load of 2 700 kN for the 10 mm.

16.6 Example 6: stiffness of the foundation for the structural model

The load case of the group is given F_x = 900 kN, F_z = −2 700 kN and M_y = 600 kN·m, and **Foundation Stiffness…** is requested in the load case with the method *Based on nonlinear soil response* at five levels, 0.2 to 1.0 of the case load. Each loaded degree of freedom of the cap is pushed alone at each level and the stiffness is the inverse of the secant flexibility. Only the loaded degrees of freedom (Ux, Uz and Ry here) can be swept.

Table 16.6 — Example 6 — diagonal terms of the foundation stiffness

Level	K11, Ux (kN/m)	K33, Uz (kN/m)	K55, Ry (kN·m/rad)	K15, Ux–Ry (kN/rad)
0.2	552,763	4,601,049	11,285,922	−290,679
0.4	315,535	4,601,049	11,039,203	−204,286
0.6	224,447	4,601,049	10,918,646	−147,737
0.8	176,587	4,601,049	10,826,963	−134,337

© 2026 Solplan. All rights reserved.

Level	K11, Ux (kN/m)	K33, Uz (kN/m)	K55, Ry (kN·m/rad)	K15, Ux–Ry (kN/rad)
1.0	146,699	4,601,049	10,768,534	−113,345

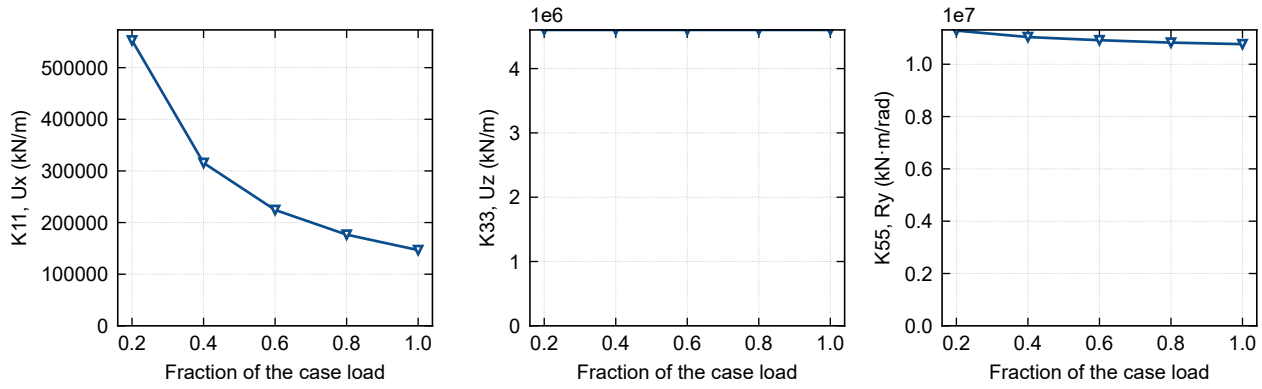


Figure 16.11 — Example 6 — diagonal stiffness terms against the load level

The horizontal stiffness falls steeply with the load, because the p-y curves of the soft clay near the surface are strongly nonlinear: at the full load it is about a quarter of its value at 0.2 of the load. The vertical stiffness does not change over this range, because the axial load of each pile stays well below its capacity, and the rotational stiffness, which comes mostly from the axial response of the outer piles, changes little. The coupling term K15 between horizontal translation and rotation is not small and should be kept in the structural model. The matrix of the level that matches the load of the structural analysis is copied from **Show Cap Response…**, tab Foundation (**Copy**); the analysis is iterated until the loads and the stiffness agree (Technical Reference, “Foundation stiffness and pushover”).

16.7 Example 7: pushover of the group

The load case is changed to the analysis type *Pushover* with the load factors 0.25, 0.5, 0.75, 1, 1.5, 2, 2.5 and 3 (**Pushover…** in the load case), and *Hold Uz / Fz constant*, so that the vertical load stays at 2 700 kN while the horizontal load grows from 225 to 2,700 kN.

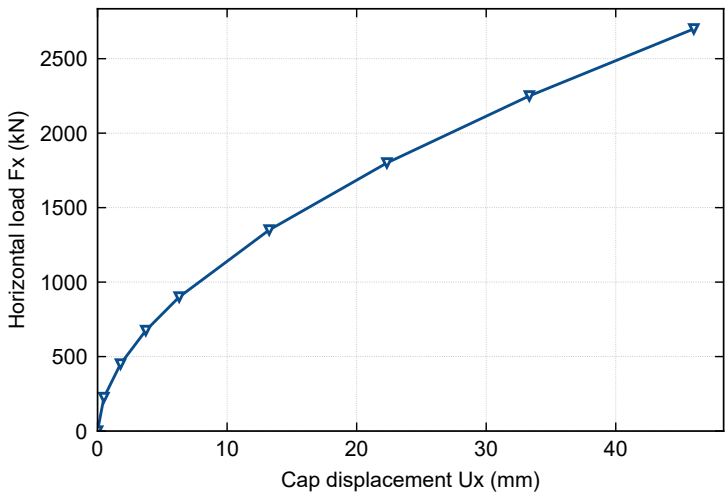


Figure 16.12 — Example 7 — horizontal load against cap displacement

At the design load of 900 kN the cap moves 6.3 mm; at twice that load it moves 22.3 mm and at three times 46.0 mm. The curve shows how quickly the group softens beyond the design load and is used to judge the reserve of the foundation, for example for an accidental load or an earthquake. Every step starts from the previous one; the path is shown in **Show Cap Response…**, tab Pushover, and in **Special Plots…**.

© 2026 Solplan. All rights reserved.

16.8 Example 8: piles in moving soil

Next to an embankment or in a lateral spread the soil moves past the piles. The group carries only its vertical load of 2 700 kN, and a free-field soil movement is assigned to every pile (**Assign ▸ Free-field Soil Movement…**): 100 mm in X at the ground surface, decreasing linearly to zero at 8 m depth.

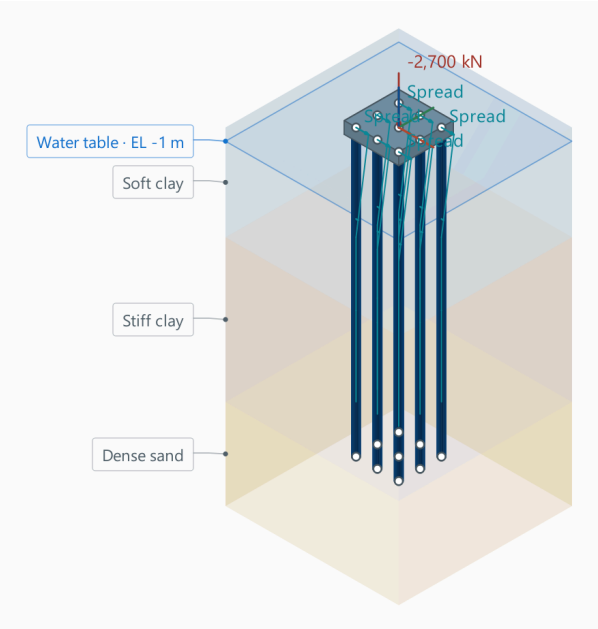


Figure 16.13 — Example 8 — the model in the 3D view; the profile of the soil movement is plotted in Figure 16.14

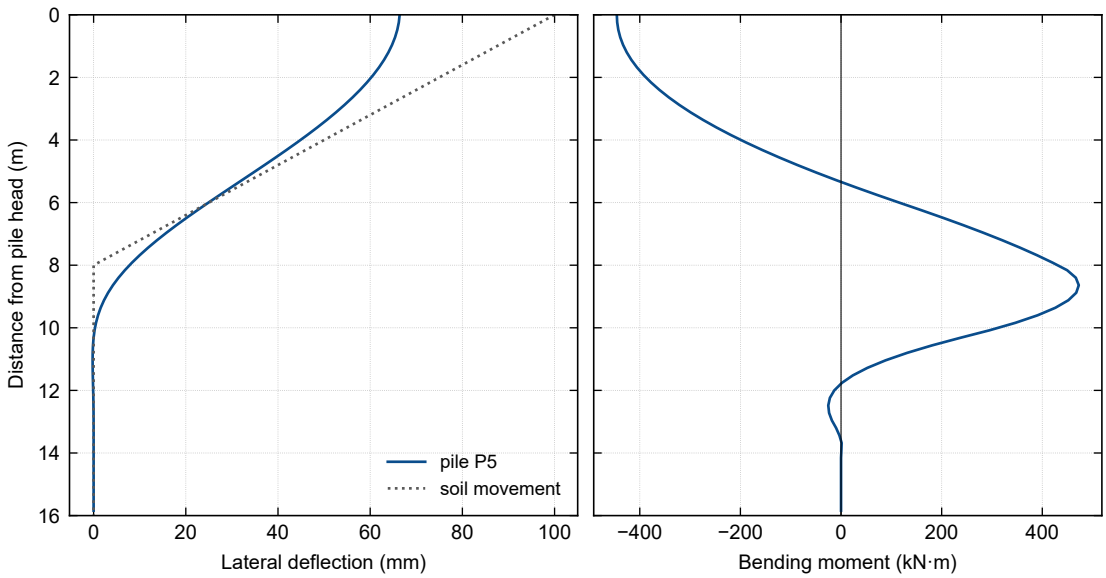


Figure 16.14 — Example 8 — soil movement, deflection and moment of the centre pile

Although no horizontal load acts on the cap, the moving soil drags the group: the cap moves 66.4 mm. The sum of the pile-head shears is zero, as it must be with no horizontal load on the cap; the moving soil loads the upper part of each pile and the stable soil below holds the lower part. The largest moment, 474 kN·m at 8.6 m below the head, occurs just below the moving zone, where the soil changes from driving the pile to resisting it, and the head moment is 448 kN·m. These moments are larger than those of the 900 kN cap load of Example 4: soil movement is often the governing action for piles near embankments.

© 2026 Solplan. All rights reserved.

16.9 Example 9: embedded pile cap

The pile heads are placed at elevation −1.5 m, so that the 1 m thick cap lies in the soil, and the group carries F_x = 900 kN and F_z = −2 700 kN. The analysis is made without and with the soil resistance on the cap (**Assign ▸ Cap ▸ Embedment & Soil Resistance…**, model *From the soil layer data*, method *Applying an equivalent lateral load*).

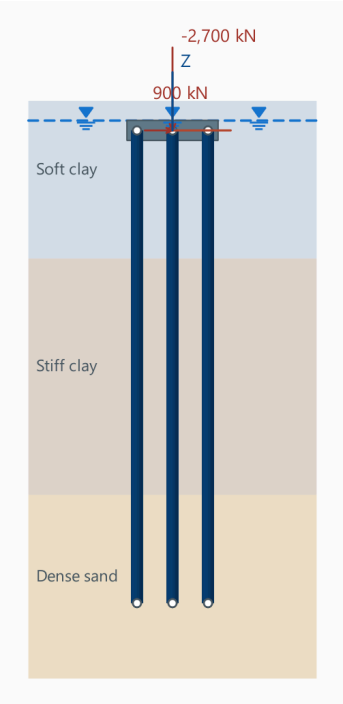


Figure 16.15 — Example 9 — the model in the X-Z elevation

Table 16.7 — Example 9 — effect of the soil resistance on the cap

Cap resistance	Cap Ux (mm)	Sum of pile shears (kN)	Resistance on the cap (kN)
Off	4.44	900	0
On	4.00	852	48

The soil in front of the cap takes 48 kN, and the piles carry the rest; the shears and the resistance add up to the applied load. The effect is modest here because the soil around the cap is soft clay; in stiff soil or compacted fill it can be much larger. The resistance should be counted only where the soil is certain to stay in contact with the cap.

16.10 Example 10: torsion from the position of a load

A ship that berths against a dolphin does not only push it. Its hull also rubs along the fender, and that friction acts at the edge of the cap, so it twists the dolphin. Nothing has to be typed for the twist: it comes from where the load acts.

16.10.1 Model

1. Template **Radial Batter Group**: eight steel pipes of 914 × 16 mm (Steel 355, elastic) on a ring of 2.5 m radius, battered 10° outward, 30 m long, heads at +4 m, and a round cap with an edge offset of 0.9 m. The same ring can be built from one battered pile with **Edit ▸ Replicate**, tab Radial, 45° per copy, seven copies (each copy turns its batter with it), and **Draw ▸ Quick Draw Cap** with the cap shape Circle in **Set Default Draw Properties**.
2. The seabed of Example 19: soft clay from 0 to −6 m over medium sand, water at +3 m.
3. Pattern SDL: F_z = −1,500 kN at the middle of the cap, the fender frame and the walkway on the dolphin.
4. A reference point *Fender* on the edge of the cap at (3.4, 0, +4) m. Pattern BERTH (type BERTHING), at the selected point: F_x = −600 kN towards the centre, and the friction of the hull, 0.2 times the push, F_y = 120 kN along the fender.

© 2026 Solplan. All rights reserved.

5. Combination *D+B* = DEAD + BERTH, Linear Add.

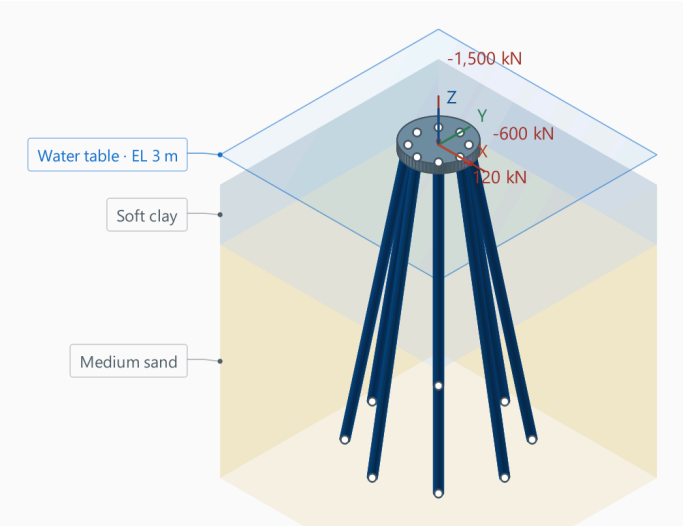


Figure 16.16 — Example 10 — the ring of battered piles under a round cap

16.10.2 Results

The push points at the centre of the ring and does not twist the cap; the friction acts 3.4 m from it, a torque of 408 kN·m from the position alone. Under D+B the cap turns 0.016° about Z. The battered ring takes the push along its piles: P5, on the far side and leaning away from the ship, is pressed with 510.4 kN, and P1, at the fender, with 87.2 kN. The piles also twist: P1 carries 12.6 kN·m of torsion at its head, and the eight heads together 101 kN·m. The rest of the torque is carried by the pile shears around the ring, times its radius. The torsional response of a pile comes from its torque–rotation curve (**Show Soil Curves…**, tab T-R; Technical Reference, “Torsional response”).

16.11 Example 11: effect of scour

The group of Example 4, without group reduction, is analysed with no scour and with 2 m of soil removed by scour (**Define ▸ Soil ▸ Soil Profile…**, *Scour depth*).

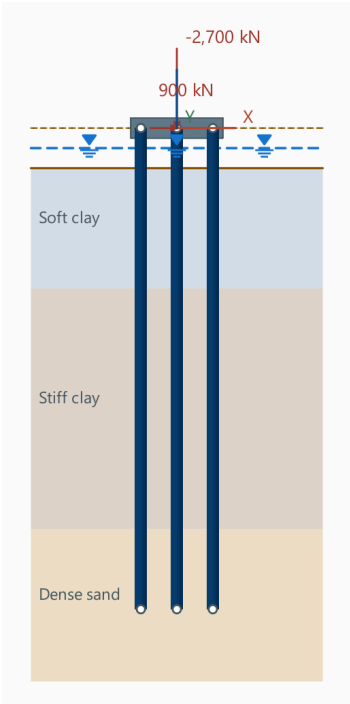


Figure 16.17 — Example 11 — the model in the X-Z elevation; the scour depth is a property of the soil profile

© 2026 Solplan. All rights reserved.

Table 16.8 — Example 11 — effect of 2 m of scour

Scour depth	Cap Ux (mm)	Largest moment (kN·m)
0	6.30	191
2 m	17.53	309

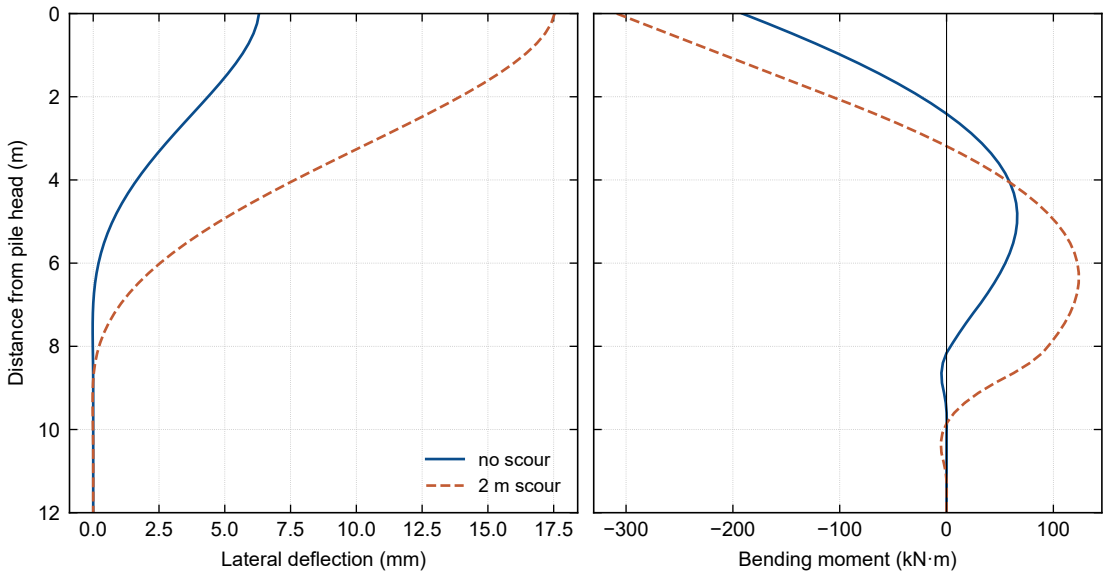


Figure 16.18 — Example 11 — deflection and moment of the centre pile with and without scour

Removing the top 2 m of soil, the part that resists the lateral load most, almost triples the displacement and increases the moments by more than half. Scour of the soil near the surface, by water, excavation or erosion, must be included where it can occur.

16.11.1 An impact on one pile

With 4 m of scour the piles stand free at the top, in a river. A drifting barge can strike one of them, and that load goes on one pile at its own elevation, not on the cap: a pattern *IMPACT* (type OTHER), pile P1 selected, **Assign ▸ Pile Loads ▸ Point Force Along Pile…**, F_x = 400 kN at elevation −1.5 m, and the combination *D+I* = DEAD + IMPACT.

Under D+I the struck pile P1 has a head shear of −218.3 kN: it pushes the cap, instead of the cap pushing it, while its neighbour P2 takes 154.4 kN from the cap. The head moment of P1 drops to 230.4 kN·m against 664.5 kN·m in P2, and its largest moment, 558.2 kN·m, is no longer at the head but at the impact, 1.5 m below it (**Show Forces Along Piles…**, case D+I, pile P1). A pile that can be struck is designed for that point as well as for its head.

16.12 Example 12: cracking of a concrete pile

The group is built with bored piles of 0.8 m diameter at 2.4 m spacing, with 16 bars of US #8 in a reinforcing steel *Rebar 420* added in **Define ▸ Section Properties ▸ Materials…** (f_y = 420 MPa, E = 200 GPa; the other entries of the round concrete section keep their defaults), and loaded with F_z = −2 700 kN at the middle of the cap and a horizontal load from 300 to 1 500 kN in X that comes through a bearing on the edge of the cap: a reference point *Bearing* at (0, 2.4) m, over the last row of piles, and the force applied at the selected point. Pushed off the middle line, the cap twists as well as slides. The section is first elastic with the gross *EI* of the concrete, then nonlinear, with moment–curvature curves at the thrusts 0, 1 000 and 2 000 kN.

© 2026 Solplan. All rights reserved.

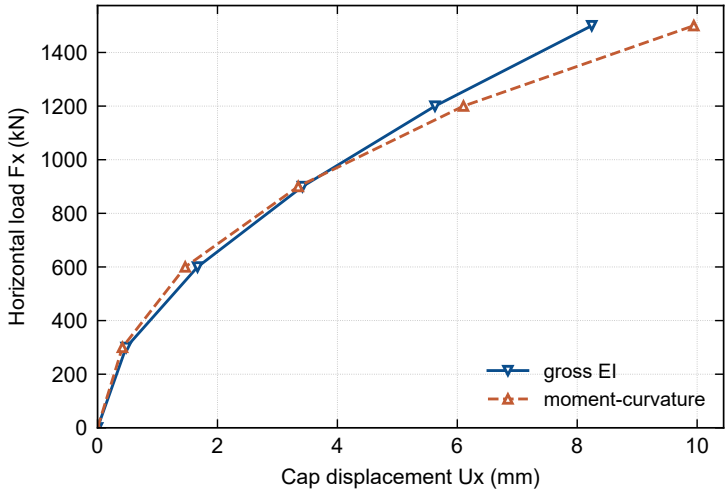


Figure 16.19 — Example 12 — elastic and nonlinear sections

At small loads the nonlinear section is slightly stiffer than the gross concrete section, because the reinforcement adds to the stiffness of the uncracked section: at 300 kN the cap moves 0.41 mm against 0.48 mm. Once the head moments exceed the cracking moment the section softens, and at 1 500 kN the cap moves 9.94 mm against 8.24 mm. The cracked piles also attract less moment at the head (542 against 683 kN·m). Because of the twist the piles do not share alike: with the gross section at 1 500 kN, P8 in the row under the bearing has a head moment of 682.7 kN·m and P2 in the far row 217.4 kN·m. For reinforced-concrete piles under significant lateral load the nonlinear section should be used, both for the displacements and for the distribution of the moments.

16.13 Example 13: moment–curvature of a bored pile

The nonlinear section of Example 12, a bored pile of 0.8 m diameter with 16 bars of US #8 (25.4 mm, steel *Rebar 420*) in concrete of 30 MPa, is examined in **Pile Section Data**, tab Nonlinear, with the thrusts 0, 2 000 and 4 000 kN entered manually; **Show moment-curvature…** draws the curves (Figure 16.20).

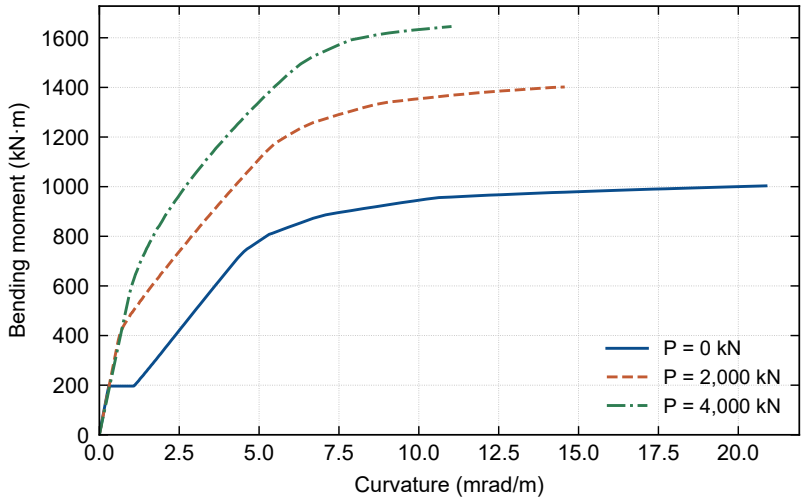


Figure 16.20 — Example 13 — moment–curvature of the bored pile at three thrusts

Table 16.9 — Example 13 — peak moment and initial stiffness

Axial thrust (kN)	Peak moment (kN·m)	Curvature at the peak (rad/m)	Initial *EI* (kN·m²)
0	1,003	0.0209	682,132
2,000	1,402	0.0146	646,284

© 2026 Solplan. All rights reserved.

Axial thrust (kN)	Peak moment (kN·m)	Curvature at the peak (rad/m)	Initial *EI* (kN·m²)
4,000	1,645	0.0110	605,946

Axial compression delays the cracking of the concrete and holds the tension reinforcement in a more effective position, so the peak moment rises with the thrust over this range, while the curvature at the peak, and with it the ductility, falls. The initial stiffness is that of the uncracked transformed section; it decreases slightly as the thrust grows, by about a tenth between 0 and 4 000 kN, because the concrete becomes softer at higher compressive stress. Because the curve of each node is interpolated between the thrusts of the list, the list must span the axial forces of the piles; a pile in tension needs a negative thrust in the list.

16.14 Example 14: prestressed concrete pile

A square prestressed pile of 400 mm side with chamfered corners, eight 1/2″ strands of grade 250 ksi, a prestress force of 1 000 kN before losses and a loss of 15 % is examined at the thrusts 0, 1 000 and 2 000 kN (Figure 16.21).

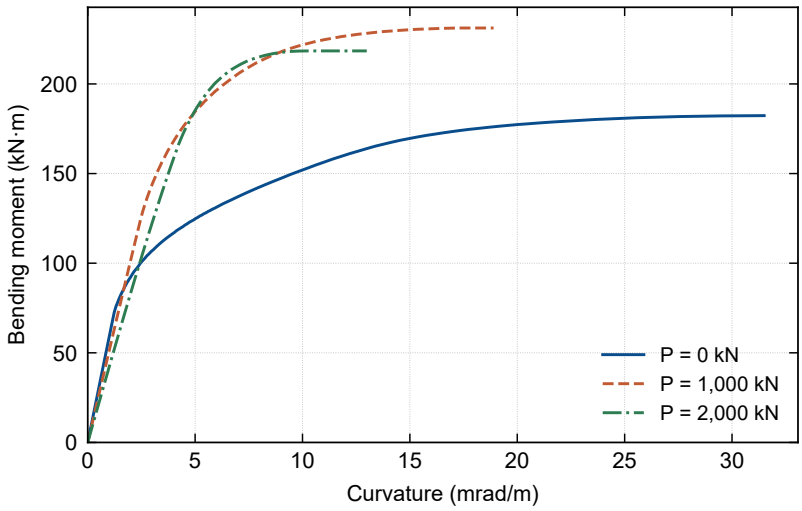


Figure 16.21 — Example 14 — moment–curvature of the prestressed pile

Table 16.10 — Example 14 — peak moment of the prestressed pile

Axial thrust (kN)	Peak moment (kN·m)	Curvature at the peak (rad/m)
0	182	0.0315
1,000	231	0.0173
2,000	218	0.0106

The prestress already compresses the section, so a moderate external thrust raises the capacity, but a larger thrust brings the concrete closer to crushing and the capacity falls again: at 2 000 kN the peak moment is lower than at 1 000 kN. A prestressed pile therefore has a narrow band of axial force in which its bending capacity is greatest, and the list of thrusts should include the extremes of the axial forces in the piles.

16.15 Example 15: steel pipe pile and a check by hand

A steel pipe of 508 mm outside diameter and 12 mm wall, yield strength 345 MPa, is examined without axial load (Figure 16.22). The plastic moment of a pipe follows from its plastic section modulus

$$Z = \frac{D^3 - (D - 2t)^3}{6}, \qquad M_p = F_y Z \tag{16.1}$$

© 2026 Solplan. All rights reserved.

which gives Z = 2,953×10^{-6} m^3 and M_p = 1,019 kN·m. The moment–curvature curve of Solgroup reaches 1,009 kN·m, 99.1 % of the plastic moment: the curve ends at a finite curvature, where the outermost fibres have strained far into the plastic range but the fibres near the neutral axis are still elastic, so the full plastic moment is approached but not reached. The agreement checks the input of the section; a large difference would point to a wrong yield strength or wall thickness.

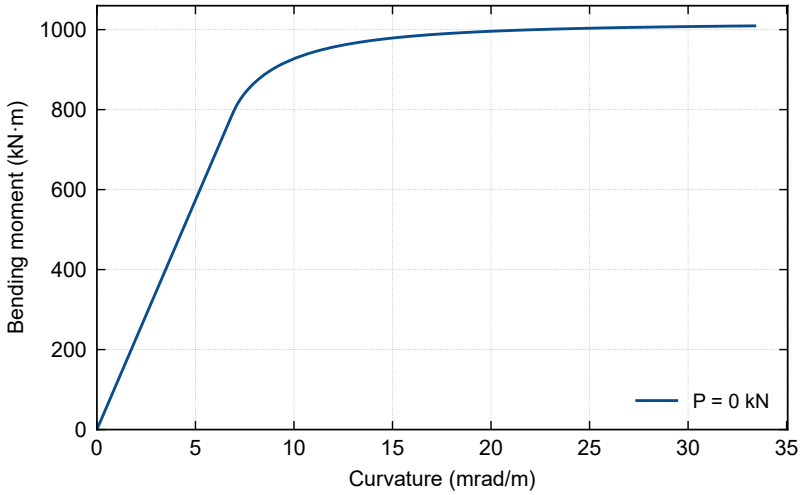


Figure 16.22 — Example 15 — moment–curvature of the steel pipe

16.16 Example 16: unsymmetrical group of bored piles

Four bored piles of 1.0 m diameter and 20 m length carry a column. The piles are drawn by hand on a grid (**Draw ▸ Quick Draw Pile**), or entered with **File ▸ New Model…**, template *Custom from Table*; the section is an elastic circular section of 1.0 m, and the cap is generated around the piles (**Draw ▸ Quick Draw Cap**) with an edge offset of 0.8 m. Two layouts are compared: a square of 3.0 m side, and a trapezoid in which the fourth pile is moved 3.0 m further in X, to (4.5, 1.5) (**Edit ▸ Move**), after which the cap is drawn again.

The column stands at the centre of the square, (0, 0). It is marked by a reference point, *Column*, and its load is applied at that point (**Selected points / pile-head joints**), in the pattern SDL: F_x = 500 kN, F_y = 300 kN and F_z = −6 000 kN. A pipe support is bolted to the head of P2 only: a joint load on the joint *P2 · Head* (**Assign ▸ Joint Loads ▸ Point Forces / Moments…**), F_y = 40 kN and F_z = −150 kN.

© 2026 Solplan. All rights reserved.

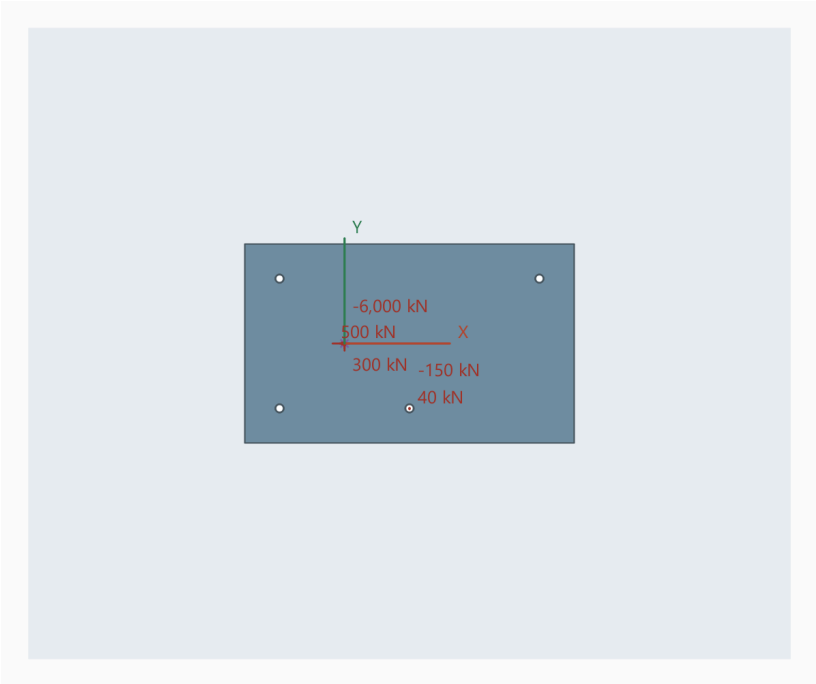


Figure 16.23 — Example 16 — the model in the plan, with the column load at its reference point and the pipe load on the head of P2

For the square the column lies at the centroid of the piles. When P4 is moved and the cap is drawn again, the middle of the cap moves to X = 1.50 m, but the column load stays where the column is, on its reference point at X = 0: a load applied with nothing selected would have moved with the middle of the cap. The centroid of the piles is now at X = 0.75 m, so the vertical load bends the group about the Y axis and F_y twists it.

Table 16.11 — Example 16 — cap movement and pile-head forces of the trapezoidal group

Pile	X (m)	Y (m)	Axial force (kN)	Resultant shear (kN)	Resultant head moment (kN·m)
P1	−1.5	−1.5	1,832	151.6	413
P2	1.5	−1.5	1,470	146.4	398
P3	−1.5	1.5	2,185	158.7	440
P4	4.5	1.5	1,502	150.2	415

The cap of the trapezoid moves 2.06 mm in X and 1.81 mm in Y and turns −0.098 mrad about Z. Because the column stays near P1 and P3, the outlying pile P4, now far from it, carries only 1,502 kN against 2,008 kN in the square, and P3, at the other end of its row and close to the column, takes the most, 2,185 kN. The twist adds to the shear of the piles on one side and subtracts from the other. Even the square group has unequal axial forces under a load with no moment: the fixed heads carry bending moments, which the cap balances by adding compression to the piles at the front and taking it from those at the back, and P2 carries the pipe as well.

Put a column load where the column is. With nothing selected a cap load goes to the middle of the cap, and that middle moves every time the cap is drawn again.

16.17 Example 17: strength increasing with depth

Five piles of 0.6 m diameter and 24 m length stand in a diamond, one at the centre and four at 2.0 m from it on the X and Y axes, and carry F_x = 400 kN and F_z = −2 500 kN. The upper 16 m are soft clay whose undrained shear strength grows from about 20 kPa near the surface to 50 kPa at 16 m; stiff clay follows below. The soil is described in two ways in **Define ▸ Soil ▸ Soil Layers…** and **Soil Profiles…**:

- four sublayers of 4 m, soft clay with c_u = 20, 30, 40 and 50 kPa;
- a single layer of 16 m with the average strength, c_u = 35 kPa.

© 2026 Solplan. All rights reserved.

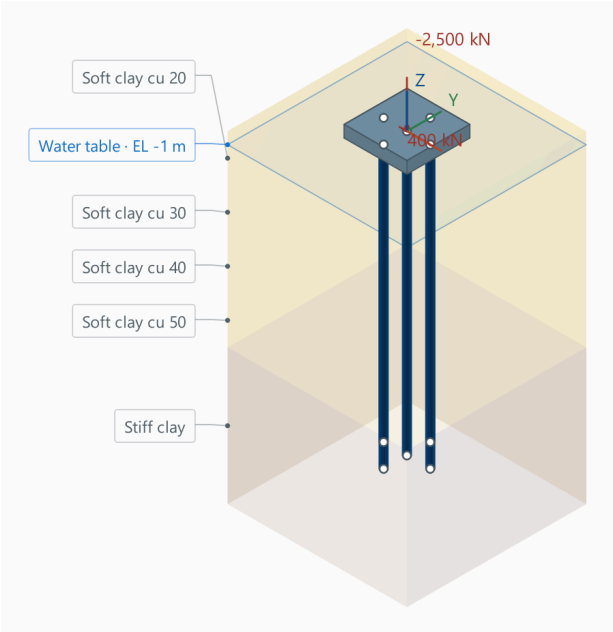


Figure 16.24 — Example 17 — the model in the 3D view

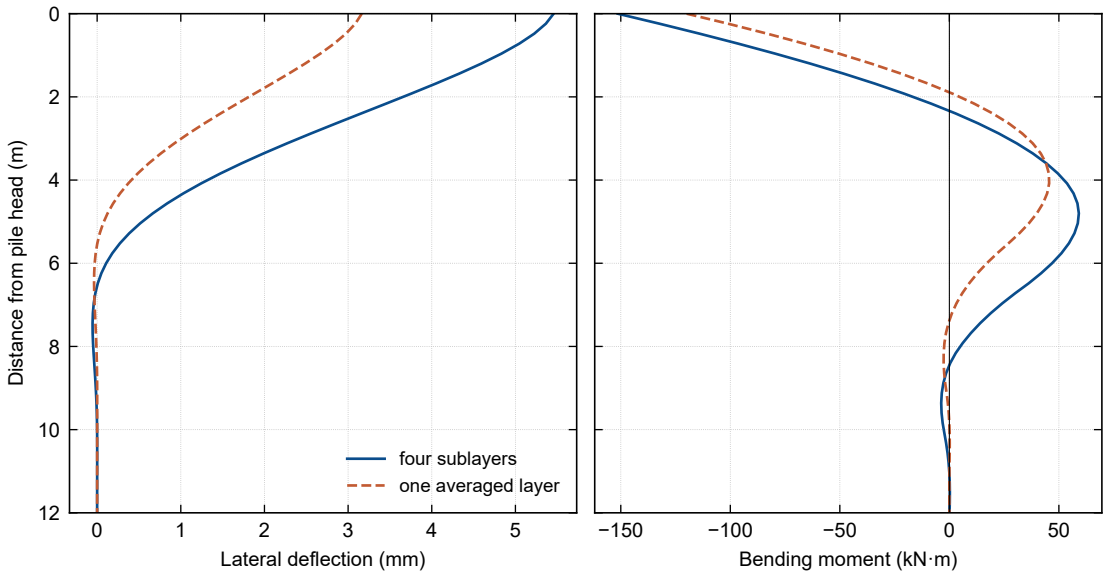


Figure 16.25 — Example 17 — deflection and moment of the centre pile for the two descriptions of the soft clay

Table 16.12 — Example 17 — response of the diamond group

Soft clay	Cap Ux (mm)	Cap Ry (mrad)	Largest moment (kN·m)	Front pile axial (kN)	Rear pile axial (kN)
Four sublayers	5.46	0.274	151	814	436
One averaged layer	3.16	0.231	120	782	483

The averaged layer moves the cap only 3.16 mm against 5.46 mm, and gives a head moment of 120 instead of 151 kN·m. The lateral response of a pile is governed by the soil within a few diameters of the ground surface, here the upper 3 to 5 m, where the real strength is well below the average of the layer. The axial forces barely change, because the vertical load is carried along the whole length. Divide a layer whose strength varies with depth into sublayers that follow the variation, at least over the upper ten diameters.

© 2026 Solplan. All rights reserved.

16.18 Example 18: ring of piles loaded in two directions

Eight piles on a ring of 3.0 m radius (template *Circular (Single Ring)*, start angle 0°, so that pile P1 lies on the X axis) carry a vertical load of 8 000 kN, a horizontal load of 600 kN and an overturning moment of 2 400 kN·m about the horizontal axis perpendicular to the load. The horizontal load and the moment are applied first along X (0°) and then at 22.5° from X, which points between two piles; the components are entered as F_x, F_y, M_x and M_y of one cap load.

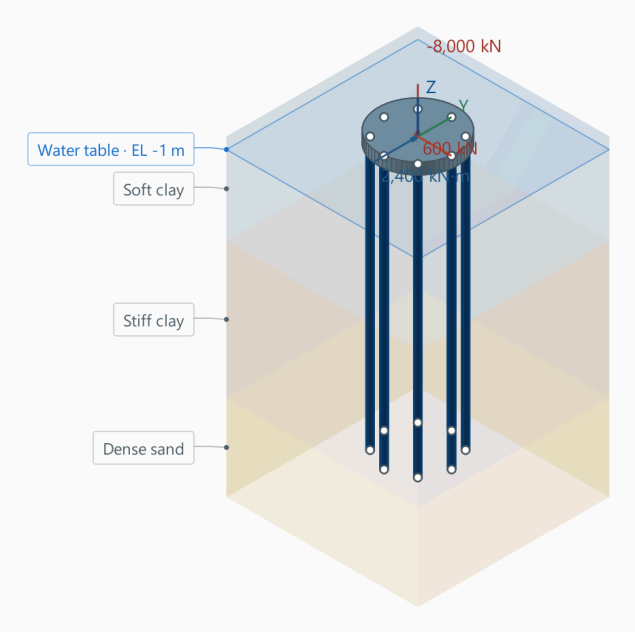


Figure 16.26 — Example 18 — the model in the 3D view

Table 16.13 — Example 18 — ring of eight piles

Direction of load	Horizontal cap movement (mm)	Largest axial force (kN)	Smallest axial force (kN)	Largest shear (kN)	Largest head moment (kN·m)
0° (towards P1)	4.02	1,401	830	75.2	129
22.5° (between P1 and P2)	4.02	1,380	851	75.2	129

The ring is laterally isotropic: the cap moves the same 4.02 mm in either direction, and the shears and head moments do not change. The axial forces do. The increase of the axial force can be checked by hand. The moment of inertia of the ring about any horizontal axis through its centre is $I = \Sigma r^2 = nR^2/2 = 36$ m^2, and the moment that the axial forces must carry is the applied moment plus the eight head moments, which the fixed heads transfer to the cap ($\Sigma M_{\text{head}} = 1{,}030$ kN·m):

$$\Delta N = \frac{(M + \Sigma M_{head})\, R \cos\alpha}{I} \tag{16.2}$$

with α the angle between the load and the nearest pile. For 0° this gives 285.8 kN, for 22.5° 264.1 kN; the program gives 1,401.4 − 1 000 and 1,379.6 − 1 000 kN. The direction that points at a pile governs the axial design of a ring; for a ring with few piles the difference grows, and both directions should be analysed.

16.19 Example 19: breasting dolphin with battered piles

A breasting dolphin stands on six steel pipe piles of 914 mm diameter and 16 mm wall (steel of 355 MPa, an elastic section of shape *Steel pipe*) in two rows of three, 3.0 m apart in X and 2.5 m in Y. The pile heads are 4.0 m above the seabed; 6 m of soft clay overlie medium sand (the layers *Soft clay* and *Medium sand* of the jetty tutorial, Section 7.3), with the water 3 m above the seabed. A berthing load of 600 kN acts in +X with a dead load of 1 500 kN. Three arrangements are compared, the batter being set with **Assign ▸ Pile ▸ Batter…**:

© 2026 Solplan. All rights reserved.

- all six piles vertical;
- the front row (on the side towards which the load pushes) battered 10°, with its tips outward in +X;
- both rows battered 10° outward, the rear row with its tips in −X.

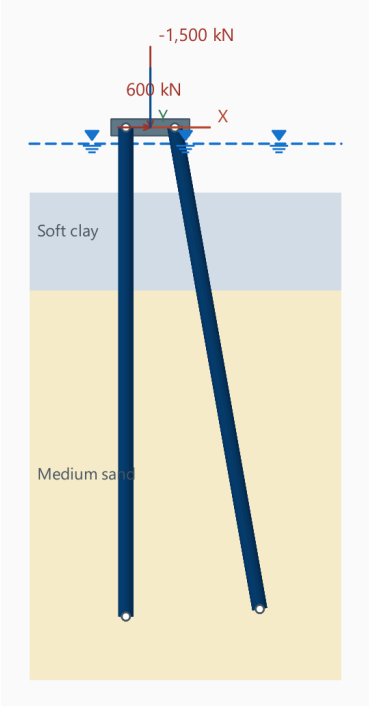


Figure 16.27 — Example 19 — the model in the X-Z elevation, with the front row battered

Table 16.14 — Example 19 — dolphin under the berthing load

Piles	Cap Ux (mm)	Front row axial (kN)	Rear row axial (kN)	Sum of shears (kN)	Largest head moment (kN·m)
All vertical	11.03	662	99	600	422
Front row battered	3.64	545	233	318	226
Both rows battered	3.94	596	176	387	301

With vertical piles the whole berthing load is carried in bending: the six shears add up to 600 kN, the cap moves 11.0 mm and the rear row is lifted (99 kN, a negative axial force is tension). Battering the front row lets its axial force carry part of the load: three piles of 545 kN at 10° give 3 × 545 × sin 10° = 284 kN in X, so the shears, which in a battered pile act perpendicular to its axis, carry the remainder. The displacement falls to 3.6 mm and the moments by more than a third. Battering the rear row as well reduces the displacement further, to 3.9 mm, but the moments rise again: under the small dead load the rear piles stay in compression (176 kN), and a compressed pile whose tip points away from the load pushes the cap the same way as the load. A rear batter pays only when the rear piles go into tension; check it with the smallest vertical load that can act with the berthing load.

16.20 Example 20: jetty on a sloping seabed

A jetty deck stands on six rows of two steel pipe piles (812 × 16 mm, 40 m long, heads at +3.0 m), 5.0 m apart in X and 6.0 m in Y, elastic sections of shape Steel pipe in the steel of Example 19. The seabed slopes from −2 m under the landward row to −12 m under the seaward row. One soil profile describes the layers of Example 19 by their elevations, soft clay from −2 to −8 m over medium sand, with the water at +1 m; the sloping seabed is entered with **Sloping…** in *Soil Profile Data* (Section 9.6.1): template **Inclined plane**, slope angle 21.8° (2 m in 5 m), upper ground elevation −2 m, starting at row 1. Every pile takes the seabed where its axis meets it, −2, −4, …, −12 m from row 1 to

© 2026 Solplan. All rights reserved.

row 6. The soil above the seabed of a pile is not counted, and the layers below keep their elevations. For comparison the seabed is also taken flat at the mean level of −7 m, assigned to every pile with **Assign ▸ Pile ▸ Ground Surface Elevation Override…**. The deck carries F_x = 1 200 kN and F_z = −9 000 kN.

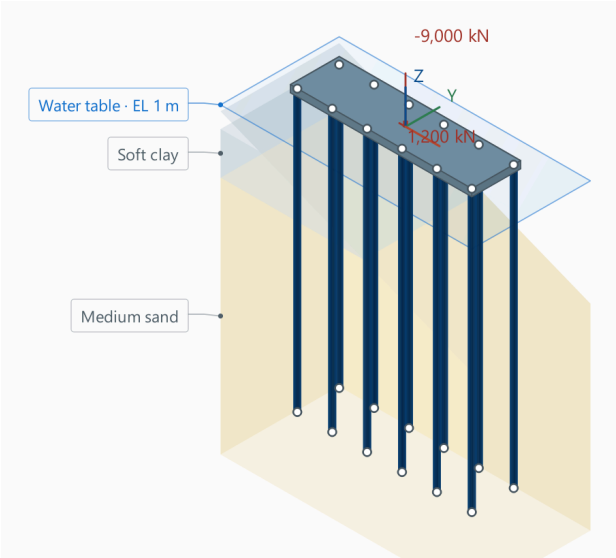


Figure 16.28 — Example 20 — the model in the 3D view, with the sloping seabed

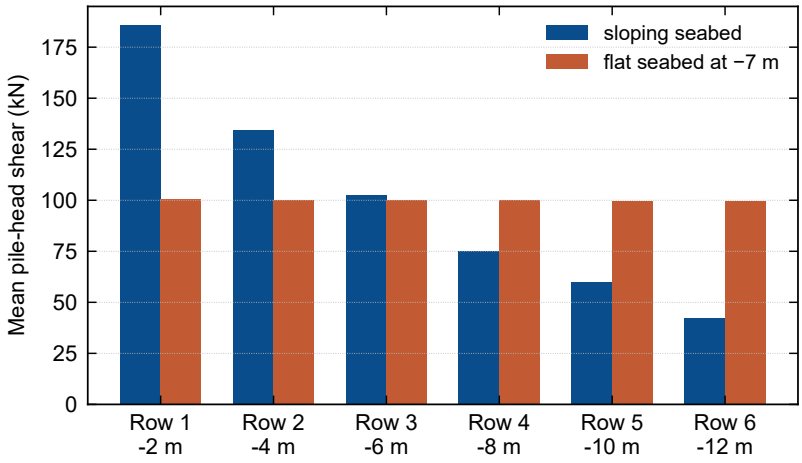


Figure 16.29 — Example 20 — mean pile-head shear of each row on the sloping and on a flat seabed

Table 16.15 — Example 20 — pile-head forces of each row on the sloping seabed

Row	Seabed (m)	Shear (kN)	Head moment (kN·m)	Axial force (kN)
1 (landward)	−2	186	1,030	992
2	−4	134	851	1048
3	−6	103	721	1094
4	−8	75	590	1133
5	−10	60	512	1186
6 (seaward)	−12	42	408	1230

On the flat seabed every row takes about the same shear, 100 kN. On the slope the short landward piles are much stiffer and attract the load: row 1 takes 186 kN, more than four times the 42 kN of row 6, and its head moment of 1,030 kN·m exceeds the 732 kN·m of the flat model by about two fifths. The total movement of the deck changes

© 2026 Solplan. All rights reserved.

little (38.6 against 41.0 mm), so a check of the deck displacement alone would not reveal the difference: a sloping seabed must be modelled pile by pile.

16.20.1 Waves on every row, and the stiffness of the sand

The sloping model is given a scour of 1.5 m (**Define ▸ Soil ▸ Soil Profile…**) and two further checks. Waves load the piles themselves: a pattern *WAVE* (type WAVE), all piles selected, **Assign ▸ Pile Loads ▸ Distributed Lateral Load…**, Global X, from the water surface at +1 m to −2 m, above even the shallowest seabed, 4 kN/m at the top and 1 kN/m at the bottom, the same 7.5 kN on each of the twelve piles. The deck shares the 90 kN out by stiffness: the short, stiff first row takes 9.8 kN at its head, more than its own wave, while the long sixth row has −5.4 kN: the deck holds it back and hands its wave to the stiff rows. The head moments are 68.7 and 5.0 kN·m.

The k of the sand comes from a table, and a cone test might show a softer sand. The soil profile can override k for one layer without a new layer: column *k override* of the sand row, 12,200 kN/m^3, half the table value. It is read only by layers whose p-y curves take a k, as this sand does. Under the case DEAD the deck then moves 49.7 mm against 49.4 mm, and the first row bends with 1,103.7 against 1,101.1 kN·m: hardly any change. The k only sets how steeply the sand curves start; at this load they are already far along, so the strength of the sand governs, not its first slope.

16.21 Example 21: rectangular section bending about both axes

A rectangular concrete pile of 400 × 500 mm (*Rectangular concrete*, **Width (axis 2), b** = 0.40 m and **Depth (axis 3), h** = 0.50 m) has 10 bars of US #8 in the pattern *Rectangular*, three bars on each short face and four on each long face, concrete of 30 MPa and reinforcement of 420 MPa. A rectangular section bends differently in its two planes; Solgroup computes the curve for the second plane from the same section turned through 90°, so the section is entered only once (Figure 16.30).

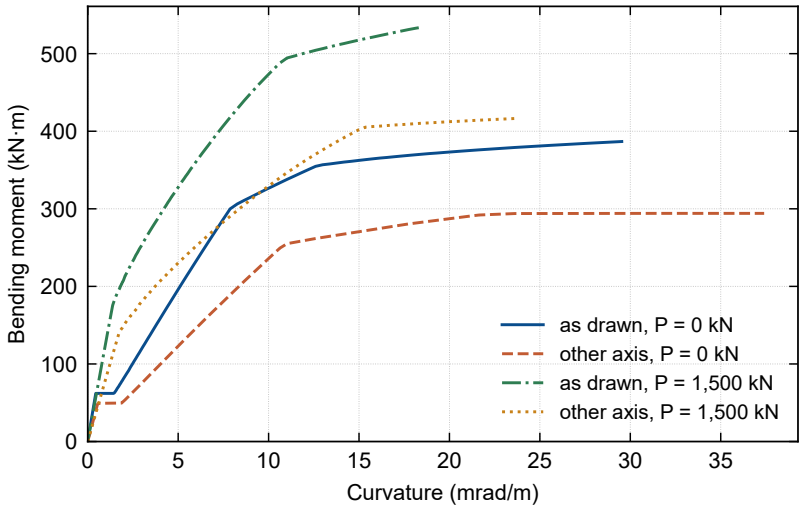


Figure 16.30 — Example 21 — moment–curvature of the rectangular section in its two planes, without axial load and at 1 500 kN

Table 16.16 — Example 21 — peak moment and initial stiffness in the two planes

Axial thrust (kN)	Plane	Peak moment (kN·m)	Curvature at the peak (rad/m)	Initial *EI* (kN·m²)
0	bending with I_{33} (depth 0.50 m)	387	0.0295	142,542
0	bending with I_{22} (depth 0.40 m)	294	0.0276	90,596
1 500	bending with I_{33}	534	0.0185	128,363
1 500	bending with I_{22}	416	0.0236	81,470

© 2026 Solplan. All rights reserved.

The gross moments of inertia are $bh^3/12 = 4.167\times10^{-3}$ and 2.667×10^{-3} m^4, a ratio of 1.56; the initial stiffnesses of the two curves without axial load stand in nearly the same ratio, which checks the orientation of the section. The bars do not scale the same way, so the ratio of the peak moments is smaller, $387/294 \approx 1.3$. When a rectangular pile is rotated in plan, use **Assign ▸ Pile ▸ Local Axes…** to turn its local axes rather than swapping width and depth.

16.22 Example 22: bored pile with a permanent casing

The bored pile of Example 13 (0.8 m diameter, 16 bars of US #8) is compared with the same pile left in a steel casing of 12 mm wall (*Round concrete with permanent casing*, casing steel of 355 MPa), both at an axial thrust of 1 000 kN (Figure 16.31).

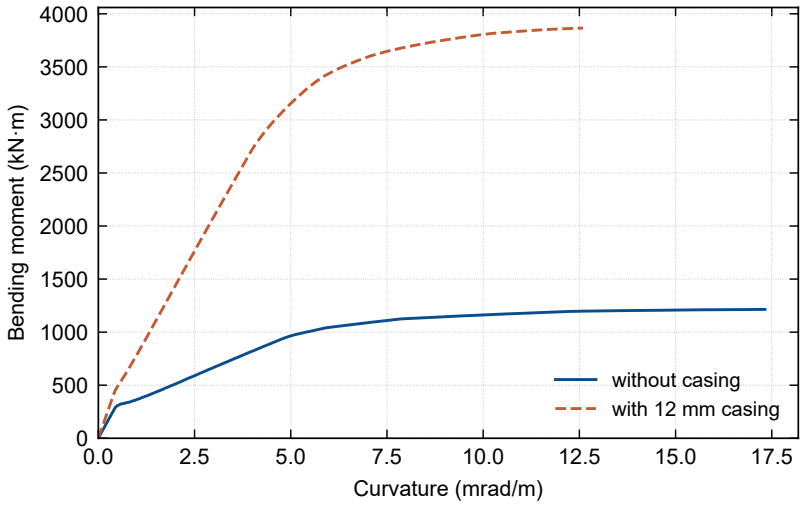


Figure 16.31 — Example 22 — moment–curvature with and without the casing

The casing triples the bending capacity, from 1,214 to 3,866 kN·m, and raises the initial stiffness 1.57 times. A rough check is the plastic moment of the casing alone, $F_y(D^3 - (D - 2t)^3)/6 = 2{,}645$ kN·m, added to the capacity of the uncased pile: the sum, 3,860 kN·m, is close to the curve. Over the length where the casing is not left in place, use the uncased section with **Assign ▸ Pile ▸ Section along Length…**.

16.23 Example 23: unsymmetrical reinforcement

A square concrete section of 500 mm, in the concrete and reinforcement of Example 21, has four bars of US #10 along one face and two bars of US #6 at the corners of the opposite face, entered with the pattern *Manual*: bar centres 75 mm from the faces, in the table X and Y from the section centre, the four #10 at X = −0.175, −0.0583, 0.0583 and 0.175 m with Y = −0.175 m, the two #6 at X = ±0.175 m with Y = +0.175 m. Its positive and negative bending capacities differ, and the program computes a separate branch for each direction (Figure 16.32). Such a section can be analysed only with **Allow non-symmetric nonlinear sections** ticked in **Analyze ▸ Set Analysis Options…**; without it, Check Model stops the run and names the section.

© 2026 Solplan. All rights reserved.

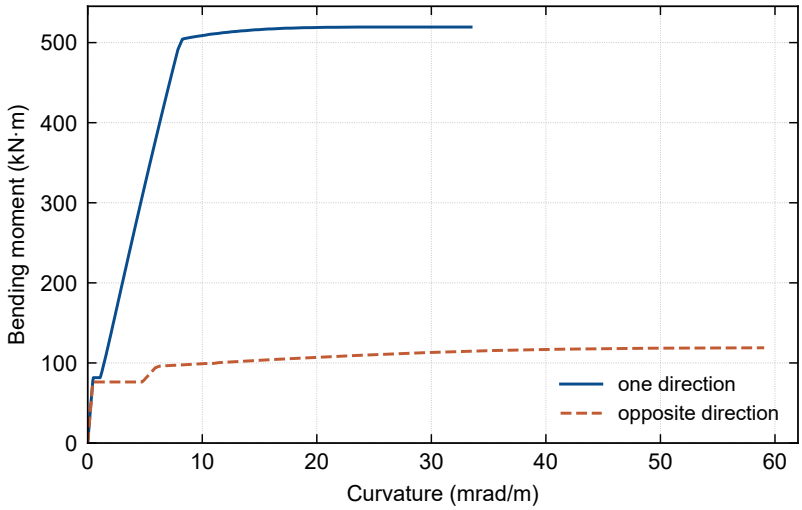


Figure 16.32 — Example 23 — moment–curvature of the unsymmetrically reinforced section in the two directions of bending

With the four large bars in tension the section carries 519 kN·m; with the two small bars in tension only 119 kN·m, 4.4 times smaller. The initial stiffness is the same in both directions (179,028 kN·m^2), since the uncracked section does not depend on which face is in tension. In the opposite direction the curve flattens as soon as the concrete cracks: the two small bars can hardly carry the cracking moment, the behaviour of a face with less than the minimum reinforcement. A pile with such a section must be oriented with **Local Axes…** so that its strong face meets the governing load, and both directions of the load should be analysed.

16.24 Example 24: editing the model in a spreadsheet

Large changes are often quicker in a spreadsheet than in the dialogs. The model of Example 1 is written with **File ▸ Export ▸ Model Tables (Excel)…** to one workbook of 25 sheets, one per table of **Show Tables**, with the units of the display in the column headings: the input tables (Materials, Pile Sections, Section Reinforcement, Cap Sections, Piles, Joints, Assignments, Soil Layers, Soil Profiles, Lateral Reduction Sets, Group Effect Factors, Load Patterns, Joint Loads, Load Cases, Load Combinations) and the result tables.

In the workbook three changes are made:

1. in the sheet *Piles*, whose columns are ID, Label, X (m), Y (m), Z Head (m), Z Tip (m), Length (m), Batter from Vert (°), Batter Ratio 1:m and Azimuth (°), the length of all nine piles is changed from 24 to 30 m;
2. a tenth row is added with the ID P99;
3. in the sheet *Joint Loads* the force Fx of the cap load of pattern SDL is changed from 300 to 600 kN.

The workbook is saved and read back with **File ▸ Import ▸ Model Tables (CSV / Excel)…**. The program reports:

```
Materials: 0 value(s) updated, 1 row(s) matched
Pile Sections: 0 value(s) updated, 1 row(s) matched
Cap Sections: 0 value(s) updated, 1 row(s) matched
Piles: 9 value(s) updated, 9 row(s) matched
   not found: P99
Assignments: 0 value(s) updated, 9 row(s) matched
Soil Layers: 0 value(s) updated, 3 row(s) matched
Load Patterns: 0 value(s) updated, 3 row(s) matched
Joint Loads: 1 value(s) updated, 1 row(s) matched
```

The rows are matched by their key column, not by their order, so a sheet sorted in the spreadsheet still lands on the right objects (Table 16.17). A row whose key is not in the model is reported as *not found* and ignored: the import changes existing objects but does not create new ones, so the pile P99 is not added. New piles are drawn or generated in the program, and their properties then edited in the workbook. Columns that cannot be edited in Show Tables,

© 2026 Solplan. All rights reserved.

such as Z Tip, and the columns of the result tables, are ignored without failing the import; a value that is not accepted, for example a negative length, is listed below the table with its line in the file. Sheets that are read only in Show Tables (Section Reinforcement, Soil Profiles, Load Cases and the result tables) and empty sheets are passed over without a line in the report. The whole import is one step of **Edit ▸ Undo**.

Table 16.17 — Key columns by which the rows of each model table are matched

Table	Key column(s)	Table	Key column(s)
Materials	Name	Soil Layers	Name
Pile Sections	Name	Lateral Reduction Sets	Set, Depth
Cap Sections	Name	Group Effect Factors	Set, Row
Piles	ID	Load Patterns	Name
Joints	ID	Joint Loads	Target, Pattern
Assignments	Pile	Load Combinations	Name

After the import the model is analysed again. With the doubled horizontal load the cap moves 3.13 mm instead of 0.93 mm and the largest head moment rises from 43 to 111 kN·m, more than in proportion to the load because the soil response is nonlinear; the longer piles change the lateral response little, since the upper part of a long pile governs it.

16.25 What to try next

- In example 1, select a lateral group-effect set by the AASHTO method in the analysis options, and compare the shear forces of the leading and trailing rows.
- Add a **Pushover** case to example 1 to follow the response of the group to several times its load.
- In example 2, compute the foundation stiffness of the deck (**Foundation Stiffness…** of the load case) with and without the battered piles.

© 2026 Solplan. All rights reserved.

17 Troubleshooting

This chapter shows what Solgroup reports when a model cannot be analysed or an analysis does not converge, and how to find the cause. The messages quoted are those of Solgroup 1.0.10 for small variations of the group of Example 1. Two places hold the information: **Analyze ▸ Check Model** lists the findings about the input, and, after a run, the **Command** panel lists one entry per load case with the notes of the analysis. A click on a finding selects the objects it concerns.

17.1 Errors found by Check Model

Check Model runs before every analysis, and a model with an error is not analysed. Each finding has a level, a code and a message that names the objects concerned. Two common errors:

Table 17.1 — Examples of errors found by Check Model

Code	Message	Cause and remedy
SEC-001	*Pile P1 has no pile section assigned.*	The pile has no section, for example after the section it used was renamed in another file. Select the piles and assign a section with Assign ▸ Pile ▸ Section.
SOIL-019	*The tip of pile P1 (7.00 m) is at or above the ground surface of its soil profile (0.00 m); the pile must reach into the soil.*	The pile does not reach the soil, for example because the pile heads were placed above the ground with too short a length, or the ground surface of the profile is wrong. Correct the length or the elevation of the piles, or the ground surface of the profile.

Warnings and information do not stop the run but should be read. For example, *LOAD-007: Load pattern 'LIVE' carries no load* points to a pattern that was meant to be loaded; *LOAD-015* states that the self weight multiplier of a pattern has no effect. The complete list of findings is in Appendix A.

17.2 Bearing failure under vertical load

The vertical load of the group of Example 1 is raised to 60 000 kN, far beyond the axial capacity of nine 0.6 m piles 24 m long. The run does not converge, and the Command panel reports for the case:

```
DEAD: NOT converged
  The group stiffness matrix is singular
  • Possible bearing failure: the axial head displacement of some piles is beyond the upper end of
    their load-displacement curve.
  • To hold these piles at the end of their curve instead, switch on 'Extend L-D / T-R curves beyond
    their last point' in Analysis Options.
  • No resistance left (bearing), pile P1: the axial head displacement 0.14256 m is past the
    0.0653617 m end of its curve, so it carries no axial load at all.
  …
```

The axial load–displacement curve of each pile ends at the displacement at which its full capacity is mobilised. A pile pushed beyond that point carries no axial load, so the group loses its vertical stiffness and the equations become singular. The remedies are, in order: check the load and its units; check the side friction and the tip resistance of the soil layers; lengthen the piles or add piles. The option *Extend L-D / T-R curves beyond their last point* holds the axial force of such piles at its value at the end of the curve, so that the analysis can continue; the result then shows a group that is at its axial capacity, which must not be used as a design state.

17.3 Failure under a very large horizontal load

With a horizontal load of 20 000 kN on the same group, the run does not converge either. The piles cannot carry the horizontal load by bending, the cap rotates strongly, and the overturning pushes the piles on one side down and pulls the piles on the other side up:

© 2026 Solplan. All rights reserved.

```
DEAD: NOT converged
  The group stiffness matrix is singular
  • Possible bearing failure: …
  • No resistance left (bearing), pile P2: the axial head displacement 0.132124 m is past the 0.0653617
m
    end of its curve, so it carries no axial load at all.
  …
  • Possible pull out failure: the axial head displacement of some piles is beyond the lower end of
    their load-displacement curve.
  • No resistance left (pull out), pile P1: …
```

A failure of this kind is a real failure of the foundation, not a numerical difficulty: the group cannot carry the load. The pushover of Example 7 shows how the displacement grows as the load approaches such a state.

17.4 An analysis that does not converge

Where the cause is not a failure of the soil or of a pile, a load case may still stop at the iteration limit. In that case:

1. Read the notes of the case in the Command panel; they name the piles whose iteration did not close and, where it applies, the reason.
2. Check the input that drives the nonlinearity: a nonlinear section whose thrust list does not span the axial forces of the piles, a p-y model with a parameter far outside its usual range, or a pile tip in a layer without a tip curve.
3. Apply the load in steps: set an *Initial step* and a *Minimum step* smaller than 1 in the Analysis control of the case (Table 11.2). A case that converges only for part of its load reports the fraction reached.
4. Raise the iteration limits (*Iterations, pile group* and *Iterations, single pile*) moderately; a case that needs many more iterations than the default is usually close to a failure.
5. Run a pushover with increasing load factors (Example 7) to find the load at which the response becomes unstable.

NOTE The results of a case that did not converge are its last iterate, not a solution. They are shown so that the cause can be found, and must not be used for design.

© 2026 Solplan. All rights reserved.

Appendix A Check Model messages

The table lists every finding of **Analyze ▸ Check Model**, by code. *Error* stops the run; *Warning* and *Info* do not, but should be read before the results are used. The sign ... marks the name of an object or a value that the message fills in. When a code has several variants of its message, the first three are shown.

Code	Level	Message (… = name or value filled in)
AXIAL-002	Error	Axial curve '… ' assigned to pile … is not in the catalog.
AXIAL-003	Error	Torsional curve '… ' assigned to pile … is not in the catalog.
AXIAL-006	Error	…
AXIAL-007	Error	…
AXIAL-008	Warning	The … curve '… ' needs at least two points; pile(s) … use the curve generated from the soil instead.
CAP-000	Warning	The pile cap was deleted, but the analysis still ties all pile heads to one rigid cap.
CAP-001	Error	The pile cap polygon has fewer than three vertices.
CAP-002	Error	The pile cap has no cap section.
CAP-003	Error	Cap section '… ' is not in the catalog.
CAP-004	Warning	The pile cap does not cover pile heads … . They are still tied to the cap: a rigid cap ignores its outline (an embedded or flexible cap uses it). A pile on its own needs a separate model.
CAP-005	Warning	The pile cap vertices are not coplanar.
CAP-006	Warning	Head of pile … at elevation … does not match the cap elevation of … .
CAP-007	Info	The cap is set to Flexible but carries no plate mesh yet; one is generated for the analysis from the cap outline and its section (Assign > Cap > Generate Flexible Cap Mesh keeps it in the model).
CAP-008	Error	… flexible-cap plate(s) point at a node that is not in the mesh (first: element …). … flexible-cap beam(s) point at a node that is not in the mesh (first: beam …).
CAP-009	Error	Flexible-cap plates use plate section number(s) … , which the mesh does not define.
CAP-010	Error	The cap is marked as embedded but no soil resistance model is chosen.
CAP-011	Error	An embedded cap needs an equivalent diameter, or plan dimensions to take it from.
CAP-012	Error	The user HFD curve needs both k50 and an ultimate resistance above zero.
CAP-013	Error	The HFD curve needs a maximum deflection above zero.
CAP-015	Error	The embedded part of the cap has its top (… , …) at or below its bottom (… , …).
CAP-016	Error	The springs along the embedded cap need 1 to … increments.
CAP-017	Error	The pile cap thickness is zero or negative.
CAP-018	Error	…
CAP-019	Error	Cap restraints (…) are not used. Prescribe a zero displacement in the load case instead, then clear them (Assign > Cap > Clear Legacy Cap Restraints).
CAP-020	Error	… pile head(s) are neither at a node nor inside a plate of the flexible-cap mesh (…); regenerate the mesh after moving piles.
CAP-021	Error	Plate section … needs a positive modulus and thickness.
CAP-022	Warning	Plate section … has Poisson's ratio … , above 0.5, the limit for an isotropic material; the plate is analysed with it, but check the value.
CAP-023	Info	… pile head(s) lie inside a flexible-cap plate rather than at a node (…); the pile is tied to the plate's four nodes through its shape functions.
CAP-024	Error	Plate section … has Poisson's ratio … ; the plate's bending and shear stiffness need a ratio between -1 and 1, so the flexible cap cannot be analysed.
CAP-025	Warning	No pile head of the flexible cap carries torsion and the cap has no beams: a moment about the vertical axis cannot be carried, and the cap rotations about it are not meaningful. Fix one pile head, or add cap beams.
CAP-026	Warning	The cap is embedded, but a flexible cap is analysed without the passive soil resistance on its sides yet; the embedded-cap settings are not used.

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
CAP-027	Error	The flexible cap has no stored mesh, and the one generated from its outline with an element size of … would have up to … plates; at most … are allowed. Use a larger element size in the cap section.
CAP-028	Warning	The pile heads are at different elevations (… to …); a flexible cap ignores the differences (a rigid cap uses them). Put the heads at one elevation, or use a rigid cap.
CAP-029	Error	Flexible-cap beams use pile section(s) … , which the catalog does not define; the analysis stops at them.
CAP-030	Error	The flexible cap has no stored mesh and its cap section '… ' is not in the catalog, so its plates cannot be generated. Assign a cap section (Assign > Cap > Cap Section). The flexible cap has no stored mesh and its cap section '… ' names no material, so its plates cannot be generated. Choose a material in Define > Section Properties > Cap Sections.
CAP-031	Error	… flexible-cap node(s) are not connected to any pile (first: node …).
CAP-032	Error	… flexible-cap plate(s) are not a convex four-sided shape: the nodes are out of order around the plate, a corner points inward or two corners coincide (first: element …). Its stiffness would be wrong.
CAP-033	Error	… flexible-cap beam(s) have no flexible length: the two nodes are at the same place in plan, or the end zones cover the whole beam (first: beam …).
CAP-034	Warning	The flexible-cap mesh keeps the thickness and material it was generated with from cap section '… ', which has changed since: the analysis uses the mesh values. Regenerate the mesh (Assign > Cap > Generate Flexible Cap Mesh) to apply the cap section. The flexible-cap mesh no longer matches the element size (…), cap outline, piles or loads: the analysis uses the stored mesh. Regenerate the mesh (Assign > Cap > Generate Flexible Cap Mesh) to apply them. The flexible cap uses the plates stored with its imported mesh (thickness …); cap section '… ' (thickness …) is not used by the analysis. Regenerate the mesh (Assign > Cap > Generate Flexible Cap Mesh) to use the cap section.
CAP-035	Warning	The pile cap sits below the ground surface (pile heads …), but soil resistance on the cap is off, so its sides carry no soil. To include it, use Assign > Cap > Embedment & Soil Resistance.
CASE-001	Warning	Load case '… ' contains no load pattern.
CASE-002	Error	Load case '… ' refers to load pattern '… ', which does not exist.
CASE-003	Error	Initial condition '… ' of load case '… ' does not exist.
CASE-004	Error	Load case '… ' has an unreasonable tolerance or iteration count.
CASE-005	Warning	The model has no load case.
CASE-006	Error	Load case '… ': the minimum step is larger than the initial step, so the solver can never reduce it.
CASE-007	Error	Load case '… ' asks for absolute cap criteria but leaves both the displacement and the rotation at zero.
CASE-008	Warning	Load case '… ' is cyclic with … cycle(s); it is still analysed with the cyclic p-y curves… , not the static ones. Check the number of cycles.
CASE-009	Error	Load case '… ' prints the full pile length with a printing increment below one.
CASE-010	Warning	Load case '… ' prints the full length of every pile but has every output variable switched off, so the table along the piles is empty.
CASE-011	Error	Pushover of load case '… ' holds every component constant, so nothing is pushed; untick at least one.
CASE-012	Error	Pushover of load case '… ' needs at least one load factor.
CASE-013	Warning	Pushover of load case '… ' asks for a logarithmic ramp from or to zero; the factors fall back to arithmetic.
CASE-014	Warning	Load case '… ' carries pushover settings but its analysis type is … .
CASE-015	Error	Load case '… ' asks for a foundation stiffness matrix but lists no load level.
CASE-016	Warning	Load case '… ' opens the specified-displacement table but every degree of freedom is still force controlled.
CASE-017	Warning	Load case '… ' prescribes zero displacement for … ; that is a restraint, not a load.
CASE-018	Warning	Load case '… ' asks for BOTH a foundation stiffness matrix and a pushover analysis; a load case runs only one of them, so it is run as the foundation stiffness analysis with the pushover left out.
CASE-019	Warning	Load case '… ' lists … foundation stiffness levels; … the Load Case dialog accepts at most 10, so editing this load case there requires removing some.

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
CASE-020	Warning	Pushover of load case '… ' lists … load factors; … the Load Case dialog accepts at most 25, so editing this load case there requires removing some.
CASE-023	Error	Load case '… ': the foundation stiffness target is the … , but the loads give no … at the cap origin, so no equivalent stiffness can be found. Load that direction or choose another target.
CASE-024	Warning	Load case '… ' is Linear Static; the analysis has no linear option, so it is solved with the same nonlinear soil-pile analysis as every load case.
CASE-025	Warning	Load case '… ' is Nonlinear Staged; the analysis has no staged option, so it is solved in one nonlinear analysis at full load (load steps: Incremental / Pushover).
CASE-026	Warning	Load case '… ' is of type Pushover but its incremental analysis is not set up (Define > Load Cases > Pushover), so it is solved as one nonlinear analysis.
CASE-027	Warning	Load case '… ' continues from '… '; every load case starts from zero, so the initial condition is not used.
CASE-028	Warning	'Use EI from the previous iteration' is not used by the analysis (load case(s) …).
CASE-029	Warning	The specified displacements of load case(s) … carry their own point of application, but an embedded or flexible cap is analysed about the cap reference point, where they are prescribed instead.
CASE-030	Warning	Load case '… ' asks for a foundation stiffness matrix but lists no nonzero load level, so the matrix is not computed; only the load case itself is solved… .
CASE-031	Warning	Pushover of load case '… ' has no nonzero load factor, so the pushover analysis is not run; only the load case itself is solved.
CASE-032	Error	Load case(s) … prescribe displacements of the pile cap, but the cap is analysed as a flexible cap, which has no single displacement to prescribe, so Run stops for them. Enter the load as forces, or set the cap behaviour to Rigid.
CASE-033	Warning	Load case(s) … ask for a pushover or foundation stiffness analysis, which is not available for a flexible cap yet; only the load case itself is solved.
CASE-034	Warning	Load case '… ': …
CASE-035	Error	Pushover of load case '… ' asks for … load steps; at most … are allowed. Lower the number of steps or list the load factors.
CASE-036	Warning	Load case '… ' asks for … , which a flexible cap does not compute yet; only the load case is solved.
CAT-001	Error	The name '… ' is used by … entries of the … catalog; every name must be unique, because the model refers to objects by name and only one of the entries would be used. Rename the extra ones in Define.
CHK-000	Error	Check Model could not finish its … checks (… : …). Correct the values reported here, or re-enter the … data.
COMB-001	Error	Combination '… ' is empty.
COMB-002	Error	Combination '… ' refers to '… ', which is neither a load case nor a combination.
COMB-003	Error	Circular reference between combinations: …
COMB-004	Warning	Combination '… ' of type … contains nonlinear load cases (…). Superposition of results is not valid for a nonlinear analysis.
COMB-005	Error	Combination '… ' (row …) uses load case '… ', which prescribes cap displacements/loads; such a case cannot be combined. Remove it from the combination.
COMB-007	Error	Combination '… ' of type … cannot be analysed: a combination is either re-analysed with its factored loads (Linear Add) or picks extremes (Envelope). Change its type or remove it from Analyze > Set Load Cases to Run.
COMB-008	Error	Combination '… ' has an unreasonable tolerance or iteration count.
COMB-009	Error	Combination '… ': the minimum step is larger than the initial step, so the solver can never reduce it.
COMB-010	Error	Combination '… ' asks for absolute cap criteria but leaves both the displacement and the rotation at zero.
COMB-011	Warning	Combination '… ' is cyclic with … cycle(s); it is still analysed with the cyclic p-y curves… , not the static ones. Check the number of cycles.
COMB-012	Warning	Load case '… ' holds pile-cap … at zero displacement and so does every other case of combination '… ' (row …); the combination keeps that restraint.

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
COMB-013	Warning	Combination '… ' does not apply the p-multiplier sets named by its load cases (…) to every pile: pile(s) … keep their own set from an older project and the other piles use … .
COMB-014	Error	Combination '… ' keeps the pile-cap restraint (…) of its load cases, but the cap is analysed as a flexible cap, which has no single displacement to hold, so Run stops for it. Set the cap behaviour to Rigid.
COMB-015	Warning	Combination '… ' has a … resistance factor of … ; no … demand/capacity ratio is computed for it. Combination '… ' has a … resistance factor of … , above 1.0; its demand/capacity ratios are computed against that factor times the capacity.
COMB-016	Warning	Combination '… ' enters a negative … ; it is read as zero, which means not entered.
COMB-017	Warning	Combination '… ': …
GEO-000	Error	The model has no piles.
GEO-001	Error	Pile … has zero or negative length.
GEO-002	Error	Pile … has no direction.
GEO-003	Warning	Pile … batter is … ° from vertical, beyond the usual limit of … °.
GEO-004	Error	Pile … points upward.
GEO-005	Error	Pile heads … and … coincide within the merge tolerance of … .
GEO-006	Warning	Piles … and … intersect each other. The analysis treats each pile on its own; check the geometry if the crossing was not intended.
GEO-007	Warning	Minimum spacing is … = … D, below … D. Group effects are significant; check the p-multipliers.
GEO-008	Warning	All piles are vertical and in a single line. This group has no overturning resistance across that line; consider battered piles.
GEO-009	Error	The model needs at least two piles.
GEO-011	Error	Pile … starts on the axis of pile … (divided pile), so it would hang from the cap as a separate pile. Join them (Edit > Join); to vary the section, use Assign > Pile > Section along Length.
GEO-012	Warning	The embedment / cut-off elevation of pile(s) … is not used: the analysis takes each pile over its whole length, from the head at the cap to the tip.
GEO-013	Error	A coordinate of … is more than … from the origin, beyond any site. Check the coordinates and their units.
HEAD-001	Error	… is negative (…).
HEAD-002	Warning	… has translational or torsional springs; the analysis connects a pile to the cap with two rotational springs only, so those springs are not used.
HEAD-003	Warning	… is Free; a pile under a cap is never free, so it is analysed as Pinned. … has a Free plane; a pile under a cap is never free, so that plane is analysed as Pinned.
HEAD-004	Error	… is Specified Rotation, which the analysis does not support; use Fixed, Pinned or Elastic (Assign > Pile > Project Default Pile-Head Connection, or Assign > Pile > Head Connection for single piles).
LOAD-001	Error	A joint load refers to load pattern '… ', which does not exist.
LOAD-002	Error	A joint load refers to joint '… ', which does not exist.
LOAD-003	Error	A distributed load refers to pile '… ', which does not exist.
LOAD-004	Error	A distributed load has an inverted elevation range.
LOAD-005	Error	A pile point load refers to pile '… ', which does not exist.
LOAD-006	Warning	Soil movement '… ' has fewer than two depths, so it is not applied; a movement profile needs at least two depths.
LOAD-007	Warning	Load case '… ' carries no load: its load patterns are empty.
LOAD-008	Warning	A distributed load on cap plates exists, but the cap has no plate mesh; the load is not applied.
LOAD-009	Warning	A distributed load on cap plates names plate set '… '; plate sets are not supported yet, so the load is not applied.
LOAD-010	Error	A distributed load on cap plates covers plates … ..... , none of which exists (the mesh has …).
LOAD-011	Warning	A distributed load on cap beams exists, but the flexible-cap mesh has no beam; the load is not applied.

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
LOAD-012	Warning	A distributed load on cap beams names beam set '… '; beam sets are not supported yet, so the load is not applied.
LOAD-013	Error	A distributed load on cap beams covers beams … ..... , none of which exists (the mesh has …).
LOAD-014	Error	A point load on pile … has an elevation outside the pile.
LOAD-015	Info	The self weight multiplier of load pattern(s) … adds nothing: cap section '… ' has no material with a weight per unit volume.
LOAD-016	Error	A point load on pile … has an axial component, but the pile's axial response comes from … , which cannot carry an interior axial force. Remove the axial component or use the pile's generated curves.
LOAD-017	Error	A … refers to load pattern '… ', which does not exist, so it is not applied.
LOAD-018	Warning	Soil movement '… ' is not assigned to any existing pile, so it is not applied. Soil movement '… ' lists pile(s) … , which do not exist; they are ignored.
LOAD-019	Error	A distributed load on pile … lies outside the pile.
LOAD-020	Warning	A distributed load on pile … extends beyond the pile; the part outside is not applied.
LOAD-021	Error	A point load on pile … lies too close to the pile head for the analysis mesh (it would need … segments; at most 4,000). Apply it at the head as a joint load, or move it further down the pile.
MAT-001	Warning	Material '… ': … = … MPa is outside … -… MPa for … . Check the units of the value.
NUM-001	Error	… has no value. Enter a valid value. … more values are not finite numbers.
OPT-001	Info	… is off (Analyze > Set Analysis Options), so the values assigned to pile(s) … are kept but not used.
OPT-002	Error	The source dimension of the model (…) must be 2 or 3 (a project file value; it has no screen).
PM-001	Error	p-multiplier set '… ' … but is not in the catalog.
PM-002	Warning	Set '… ' row … : … p-multiplier … is greater than 1.0. Set '… ' pile … : … -direction p-multiplier … is greater than 1.0. Set '… ', table row s/D … : the p-multiplier of the … (…) is greater than 1.0.
PM-003	Error	Set '… ' row … : … p-multiplier is negative. Set '… ', table row s/D … : the p-multiplier of the … is negative. Set '… ' pile … : … -direction p-multiplier is negative.
PM-004	Warning	Set '… ' uses method … but its table is empty; fill it from the reference or choose AASHTO.
PM-006	Error	Set '… ' row … : … y-multiplier is zero or negative; it must be positive. Set '… ' pile … : … -direction y-multiplier is zero or negative; it must be positive.
PM-007	Error	… pile(s) are assigned p-multiplier set '… ', which is not in the catalog.
PM-008	Warning	Some piles still carry their own p-multiplier set from an older project; it takes precedence over the group effect of the model, which applies to every pile, except that one analysis applies only one closely-spaced or per-pile set (see PM-010).
PM-009	Info	Set '… ' measures s/b with the diameter of the first pile (… , …), but the piles have … diameters, so the factors depend on the pile order. Tick 'Use the average diameter of each pile pair' to avoid it.
PM-010	Warning	One analysis applies only one closely-spaced or per-pile set: … . In load cases without their own set, set '… ' (of the first such pile) is applied to all these piles. Keep one such set.
PM-011	Warning	Set '… ' has no entry for row … of the … pile rows along its load direction (… deg): pile(s) … are analysed with a p-multiplier and a y-multiplier of 1.0.
PM-012	Warning	Set '… ' has a table row whose spacing ratio s/D is zero or negative. Set '… ' lists the same spacing ratio s/D in more than one table row; the factors jump at that ratio instead of changing gradually. Set '… ' lists row … more than once; only the last entry of each row is used.
RED-001	Error	Pile … has an axial load reduction factor of zero or less.
RED-002	Warning	Pile … has an axial load reduction factor above 1.0, which increases the load instead.
RED-003	Error	Pile … refers to lateral reduction set '… ', which is not in the catalog.
RED-004	Warning	Lateral reduction set '… ' has fewer than two depths, so it is not applied; a reduction set needs at least two depths.
RED-005	Error	Lateral reduction set '… ' repeats a depth.
RED-006	Error	Lateral reduction set '… ' has … at depth … .

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
RUN-001	Error	The run list contains '… ', which does not exist.
SEC-001	Error	Pile … has no pile section assigned.
SEC-002	Error	Pile section '… ' assigned to pile … is not in the catalog.
SEC-003	Error	Pile section '… ' has no material.
SEC-004	Error	Material '… ' used by pile section '… ' is not in the catalog.
SEC-005	Warning	Pile … is divided into only … increments; 50 is a reasonable minimum.
SEC-006	Error	A section segment on pile … has an inverted elevation range.
SEC-007	Error	Section '… ': the area is zero or negative.… Section '… ': the area is zero or negative. Enter the area in the section's dimensions. Pile section '… ' has a zero or negative area.
SEC-008	Error	Pile section '… ' is nonlinear but has no reinforcement material, so its moment-curvature would use a default strength that is nowhere on screen.
SEC-009	Error	Pile section '… ': …
SEC-011	Error	Section '… ': the torsional stiffness GJ is zero or negative.… … the torsional stiffness GJ is zero or negative.
SEC-012	Error	Section '… ': the modulus of elasticity is zero or negative.…
SEC-013	Error	Section '… ': the moment of inertia … is zero or negative. … the moment of inertia … is zero or negative.
SEC-014	Error	Section '… ': the equivalent diameter is zero or negative.… Section '… ': the equivalent diameter is zero or negative. … the equivalent diameter is zero or negative.
SEC-015	Error	Sections along pile … do not cover the whole pile: the first section must … .
SEC-016	Error	Section '… ': moment-curvature curve … (axial thrust …) must start at zero moment and zero curvature.
SEC-018	Warning	Section '… ' is nonlinear and tapered (only elastic sections taper): it uses its head values over the whole pile, with axial and torsional stiffness from the gross section instead of the automatic values. To vary it: Assign > Pile > Section along Length.
SEC-019	Warning	The material override of pile(s) … is not used: the analysis takes the material of each pile's section.
SEC-020	Warning	Section '… ' has no axial thrust levels for its M-φ curves, so the analysis rebuilds the curve at every axial force (slow, may not converge). Enter them in Define > Section Properties > Pile Sections.
SEC-021	Error	Pile section '… ': … Its moment-curvature cannot be computed, so the analysis does not run rather than fall back to the gross stiffness.
SEC-022	Error	Section '… ' uses a specified moment capacity, but its capacity about axis … is zero or negative. Enter a positive moment capacity for both axes.
SEC-023	Warning	Pile section '… ' uses … , an option the section dialog no longer offers. It was kept from the file and still changes the moment-curvature curve, but the dialog does not show it.
SEC-024	Error	Pile section '… ': …
SEC-025	Error	Pile section '… ': …
SEC-026	Warning	Pile section '… ' is a … shape but its material '… ' is … ; choose a … material (Define > Section Properties > Pile Sections > Material).
SEC-027	Error	Pile section '… ': … Choose one of the listed cores in Define > Section Properties > Pile Sections.
SEC-028	Error	Pile … is divided into … increments; the analysis handles at most … (Assign > Pile > Segmentation accepts 10 to 200).
SEC-029	Warning	A row of the sections along pile(s) … lies inside another row, between … : the nodes there are analysed with the last row that starts above them ('… '), not with the row that spans them. Make each row start where the previous one ends.
SEC-030	Error	A row of the sections along pile(s) … overrides the modulus of elasticity with a negative value (…). Enter a positive modulus, or leave it empty to use the material's modulus.
SEC-031	Error	A row of the sections along pile(s) … names section '… ', which is not in the catalog. The analysis stops at that row; choose an existing pile section for it.

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
SEC-032	Error	Pile section '… ' names '… ' as its … material, which is not in the Materials catalog, so its moment-curvature would use strengths that are not on screen. Choose an existing material in Define > Section Properties > Pile Sections.
SEC-033	Error	Section '… ': no moment-curvature curve can be computed at axial load… (…); the section cannot carry that load. Remove the axial load from the section's list of axial loads, or change the section.
SEC-034	Error	Sections along pile(s) … : … .
SEC-035	Warning	Pile … is divided into … increments, more than the 200 the Segmentation form accepts; the analysis uses all of them.
SET-001	Error	The unit system '… ' of this model is not one this version knows, so the calculation rules of the file's unit family cannot be chosen. Choose a unit system in Preferences.
SET-002	Warning	The moment sign convention '… ' in Preferences is not one this version knows, so the moment diagrams use 'Tension on right'. Choose a convention in Preferences.
SOIL-001	Error	The model has no soil profile (Define > Soil > Soil Profile).
SOIL-002	Error	Soil profile '… ' assigned to pile … is not in the catalog.
SOIL-003	Error	Soil profile '… ' has no layers.
SOIL-004	Error	The tip of pile … (…) is below the bottom of soil profile '… ' (…); extend the profile to the pile tip.
SOIL-005	Error	Profile '… ' has a gap with no layer between elevation … and … .
SOIL-006	Error	Layers '… ' and '… ' in profile '… ' overlap.
SOIL-007	Error	Layer '… ' used by profile '… ' is not in the Soil Layers catalog.
SOIL-009	Error	Layer '… ' in profile '… ' has an inverted elevation range.
SOIL-011	Error	Scour depth of profile '… ' is negative.
SOIL-012	Error	Scour of profile '… ' removes the entire soil profile. The lowest point of the sloping ground of profile '… '… is below the bottom of the profile.
SOIL-013	Error	Layer '… ': the … unit weight is zero or negative.
SOIL-014	Warning	Layer '… ': saturated unit weight is less than the unsaturated value.
SOIL-015	Error	Layer '… ' uses user-defined p-y curves, but pile(s) … have no p-y set (tick User-defined p-y curves in Analyze > Set Analysis Options, then Assign > Pile > p-y Set).
SOIL-016	Error	Layer '… ': the uniaxial compressive strength of the weak rock is zero or negative. Enter a positive strength.
SOIL-017	Error	Layer '… ': the friction angle of the cemented silt is zero or negative. Enter a positive friction angle.
SOIL-018	Error	The … of pile … is above the top of the soil profile (…); lower it to the profile or below.
SOIL-019	Error	The tip of pile … (…) is at or above … (…); the pile must reach into the soil.
SOIL-020	Error	Layer '… ': the friction angle of the API sand is zero or negative. Enter a positive friction angle.
SOIL-021	Warning	Pile(s) … use soil profile '… ' instead of the project profile '… ' (kept from an older version) and are analysed with it. To vary the ground per pile, use Assign > Pile > Ground Surface Elevation.
SOIL-022	Warning	Soil profile '… ' has no layer; pile(s) … take their lateral resistance from their user-defined p-y set. Without a layer there is no generated axial curve either, so check the axial load-displacement curve of those piles.
SOIL-030	Error	Layer '… ' (loess): … '… ' asks for … cycles of loading, but the loess model covers 1 to 10 cycles. Set the number of cycles to 10 or fewer.
SOIL-031	Error	Layer '… ' (massive rock): Ei and Em are both zero or negative. Enter the intact-rock modulus Ei (Em then follows from Ei and GSI) or the rock-mass modulus Em.
SOIL-032	Warning	Layer '… ' (loess) has a zero or negative cone tip resistance qc, so it gives no lateral resistance. Layer '… ' (massive rock) has a zero or negative uniaxial compressive strength, so it gives no lateral resistance. Layer '… ' (liquefied sand hybrid): (N1)60cs below 5 gives a zero residual strength, so the layer gives no lateral resistance (enter the residual strength to use option 2).
SOIL-033	Error	Layer '… ' (Piedmont residual) has no valid field test selected (SPT, CPT, dilatometer or pressuremeter).

© 2026 Solplan. All rights reserved.

Code	Level	Message (… = name or value filled in)
SOIL-034	Warning	The … of pile(s) … reaches … below elevation 0, but the pile tip is at … , so every load case stops at Run. Extend the curves to the pile tip (Define > Soil > User p-y Curves). The … of pile(s) … holds a single curve; a set needs two curves to interpolate between, so every load case stops at Run. Add a second curve deeper than the pile tip (Define > Soil > User p-y Curves).
SOIL-035	Warning	Pile(s) … : … Until then every load case stops at Run.
SOIL-036	Error	The ground surface of soil profile '… '… (…), used by pile(s) … , is above the top of its first layer (…); raise the layer or lower the ground surface so that no part of the ground is without a layer.
SOIL-037	Error	Layer '… ' (…): … cannot be negative.
SOIL-038	Error	Layer '… ' (…): … , so the layer cannot be analysed.
SOIL-039	Warning	Layer '… ' (…): … ; the analysis uses the value as entered.
SOIL-040	Error	Pile(s) … are assigned p-y set '… ', which is not in the catalog; without it they take the p-y curves of their soil layers instead (Assign > Pile > p-y Set).
SOIL-041	Error	The sloping ground of profile '… ' has a value that is not a finite number. The sloping ground of profile '… ': each point must be farther along than the one before it.
SOIL-042	Warning	The sloping ground of profile '… ' has a single point and is not used; add a second point or choose Flat.

© 2026 Solplan. All rights reserved.