Embedded Pile Row in Plaxis 2D

Current way of modeling piles in 2D

Although piles are a real 3D element there still is a need to model piles in 2D. Reasons could be that interest is mostly on the global behaviour of a structure or to obtain some preliminary results on deformations or structural forces of the piles. Currently, when modeling a pile (row) in Plaxis 2D, users would have to make use of plate elements and/or node to node anchors. Both however have their own specific possibilities and limitations:

Plate elements i.c.w. interfaces

- Possibility to enter an axial stiffness;
- Interaction with soil due to interfaces, but soil cannot flow through the plates (discontinuous mesh);
- Possibility to enter a bending stiffness and to obtain structural forces in piles;
- When using interfaces unrealistic shear planes may be introduced.

Node to node anchors

- Possibility to enter an axial stiffness;
- No interaction with soil, soil can flow through the n2n anchors (continuous mesh);
- No possibility to enter a bending stiffness and to obtain structural forces in piles.

With the new structural element *Embedded pile row* the best of the above properties are combined:

- Possibility to enter an axial stiffness;
- Interaction with soil due to line to line interfaces and soil can “flow through” the embedded pile row (continuous mesh);
- Possibility to enter a bending stiffness and to obtain structural forces in piles;
- No unrealistic shear planes are introduced.

Principle of 2D embedded pile row

The “embedded pile row” element can be used to simulate a row of piles with a certain spacing perpendicular to the model area. The stiffness properties are entered per pile, the program calculates the smeared properties per meter width. Special feature of this structural element is that it is not directly coupled to the mesh. It is indirectly coupled via a line to line interface (consisting of spring elements and sliders).

The principle is shown in the figures below. Note that as a result of this implementation the mesh is continuous so soil can “flow through” the embedded pile row.

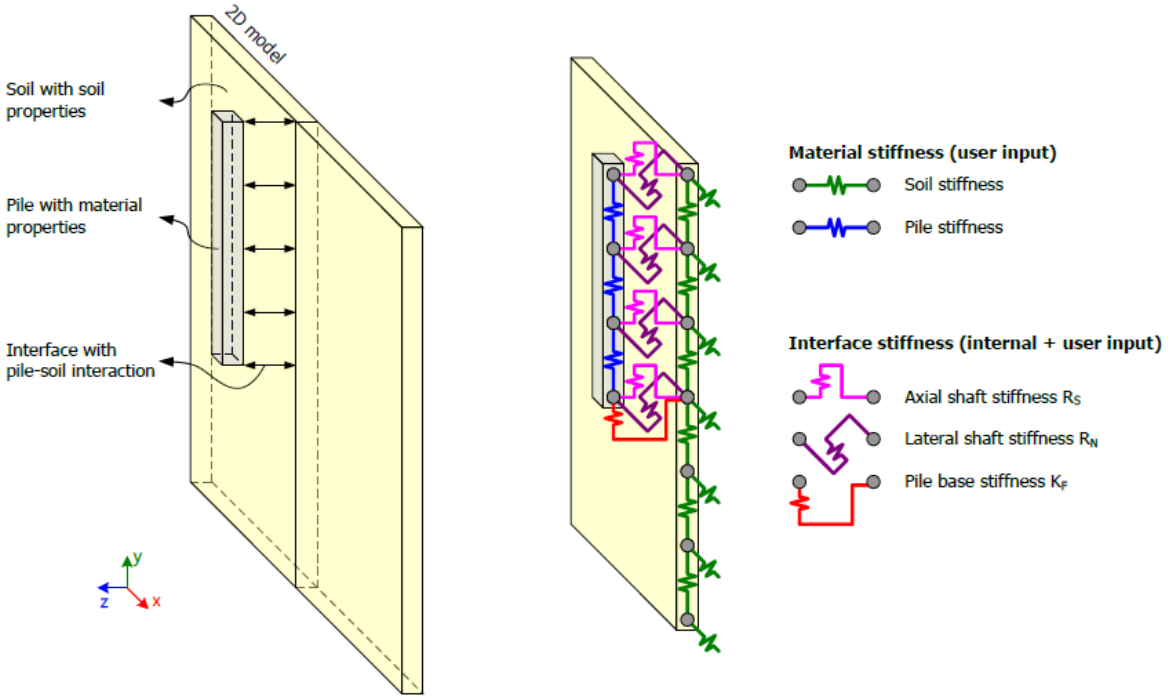
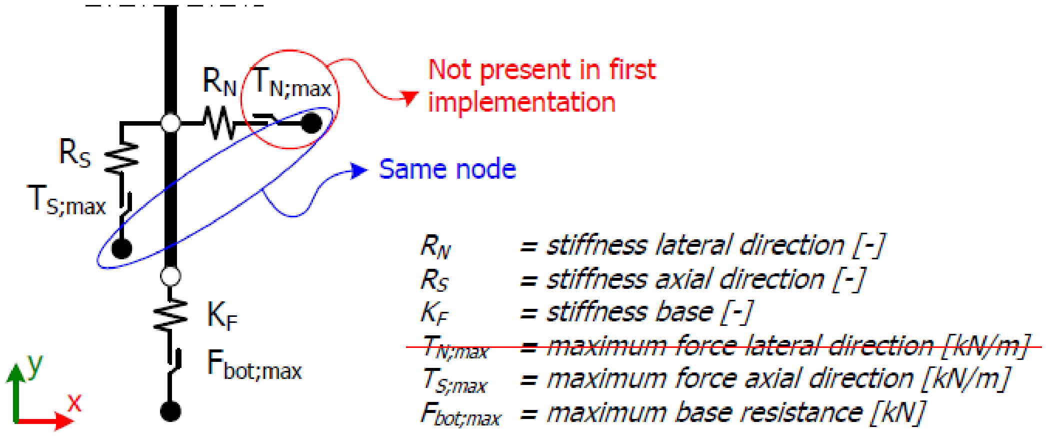


Figure 1. Principle of 2D embedded pile row

When looking in more detail at the way the interface works note that the axial springs also have sliders to be able to represent a maximum shaft and base force (which is a user input).



R_N = *stiffness lateral direction [-]*
R_S = *stiffness axial direction [-]*
K_F = *stiffness base [-]*
~~$T_{N;max}$ = *maximum force lateral direction [kN/m]*~~
$T_{S;max}$ = *maximum force axial direction [kN/m]*
$F_{bot;max}$ = *maximum base resistance [kN]*

Figure 2. Principle of interface

2D vs. 3D embedded pile behaviour

The embedded piles in 2D basically behave in the same manner as the 3D embedded piles: a structural line element coupled via springs and sliders to the mesh. The biggest difference, which also accounts for the 2D vs. 3D behaviour, is the stiffness of the line to line interface.

The stiffness of the springs in the 3D line to line interface is set to a high value such that elastic deformations are negligible but not so stiff that numerical problems arise. As a result of this choice all deformations of the pile are a result of elastic/plastic deformations of the soil itself and/or from plastic deformations in the line to line interface.

In a 2D model however this principle no longer works since the soil displacements are no longer a representation of reality but rather an average of the out of plane soil displacement. The latter can be shown by calculation (Sluis, 2012) but can also be explained by realizing that in the 2D model and an equivalent 3D model/reality the same amount of force per m1 is transmitted to the soil giving the same (average) deformations. The above is graphically explained in Figure 3 where a 3D slice is shown with a width equal to the spacing of the piles in the row.

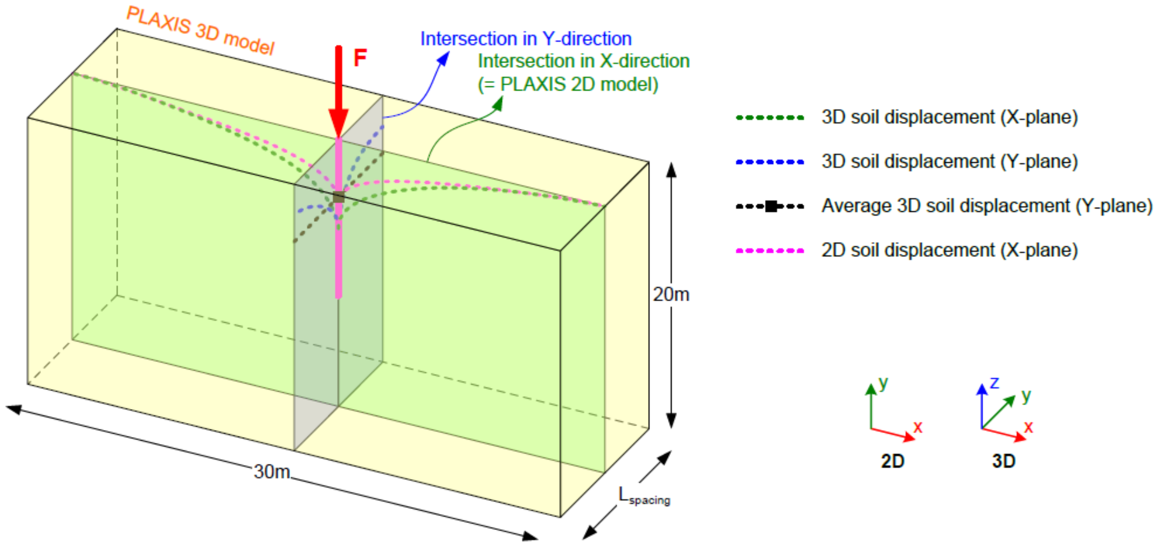


Figure 3. 2D vs. 3D embedded pile behaviour

Now in order to obtain a realistic load-displacement behaviour for our 2D embedded pile row we have to find suitable values for the line to line interface stiffness's since these determine the relative displacements between (average) soil displacement and embedded pile row.

Within a Master thesis project (Sluis, 2012) a set of formulas has been derived for the axial and lateral spring stiffness's. The formula for the axial stiffness is based upon the load-displacement curves for Bored piles as shown in the Dutch annex of the Eurocode. The formula for the lateral spring stiffness is based upon fitting with 3D calculations.

The derived formulas are used to determine a default value within Plaxis for the so called interface stiffness factors (ISF). The actual stiffness of the springs in the interface (shaft axial, base axial and shaft lateral) is calculated in the following way:

R = ISF * Gsoil / Ls

In which:
R = interface stiffness
ISF = interface stiffness factor
Gsoil = shear modulus of the soil
Ls = pile spacing in out of plane direction

Users should determine if the default setting for the ISF factors are reasonable for their situation (also see the boundary conditions for which the defaults are derived) or determine a new set of ISF values by validating with 3D calculations, measurements, codes of practice, etc.

The default settings were derived based on the following assumptions:

- Load-settlement curve according to Dutch annex of Eurocode for Bored piles;
- Foundation piles;
- Static loading;
- Relative stiff piles compared to the soil;
- Ls / Deq in the range of 2 to 8 (Ls = spacing, Deq = equivalent diameter)

The ISF factors determine the behaviour: for very high spring stiffness's the embedded pile row behaves (more or less) as a plate element (without interfaces), for very low spring stiffness's the embedded pile row behaves (more or less) as a node to node anchor.

In order to show the possibilities and limitations of the embedded pile row a case is discussed.

Case: embankment on soft soil with piled bridge abutment

Introduction

A typical soil profile in the Netherlands is the following: deep sand overlain with soft clay and/or peat layers. When constructing for example a new highway/railway crossing, new approach embankments need to be constructed. Since these embankments are constructed on top of this soft soil they give rise to (large) vertical and horizontal deformations. For this situation in general piled abutments are created inside the embankments to support the bridge deck and to minimize deformations of the bridge deck.

A typical situation is shown in the graph below.

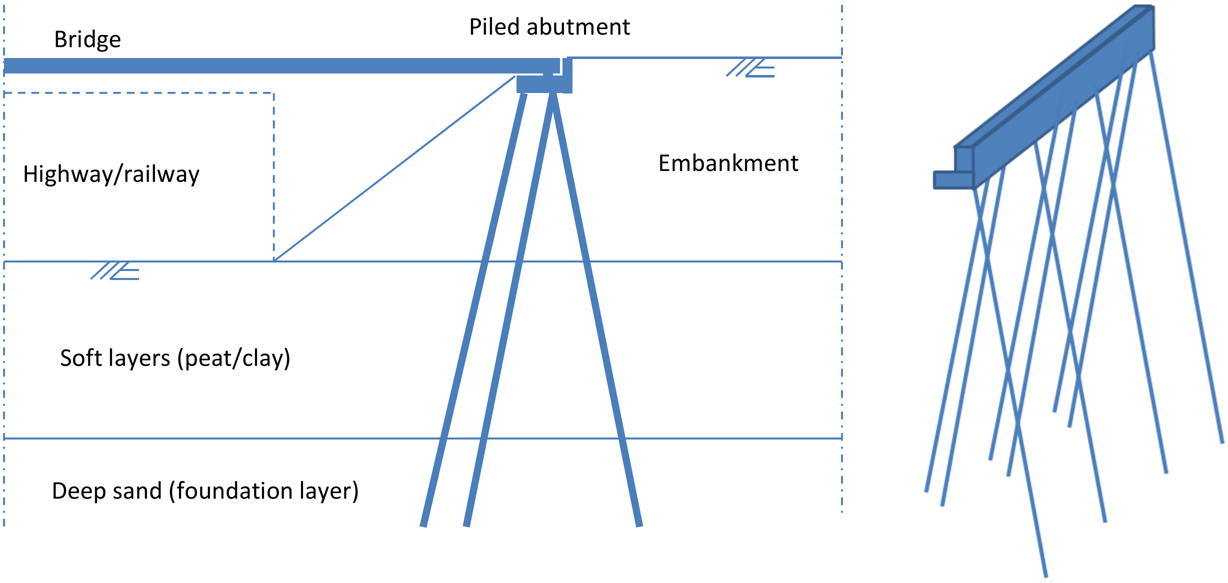


Figure 4. Highway crossing with piled abutment and 3D view of piled abutment

The pile group beneath the abutment typically has a number of vertical and/or inclined pile rows to give it sufficient strength and stiffness. As a result of the deformations of the embankment the pile group will be influenced, possibly giving rise to (large) structural forces in the piles and (additional) deformations of the abutment.

In these situations the contractor has a.o. the following questions:
- What kind of piles do I need to install?
- If concrete piles, what kind of reinforcement do I need?
- What is the total deformation of the bridge foundation?

The above questions need to be answered by the structural and geotechnical engineer. Due to the complicated soil-structure interaction a model is helpful to gain insight in this interaction.

Geometry

The geometry used in this case is shown in Figure 5 below. <u>All geometry lines (except the pile rows) are already provided in a Plaxis model</u>. Use the following steps to finalize the setup of your model:
- First read all paragraphs below (up to the paragraph on *results*);
- Create all soil material sets and apply the soil material sets to the relevant clusters;

- Then create the plate material set for the abutment and appoint the relevant material sets;
- Finally draw the pile rows and appoint the relevant material set here (by drawing the pile rows at the end there is no need to apply the soil material sets to lots of small clusters);
- Create and inspect the mesh;
- Continue to Calculations.

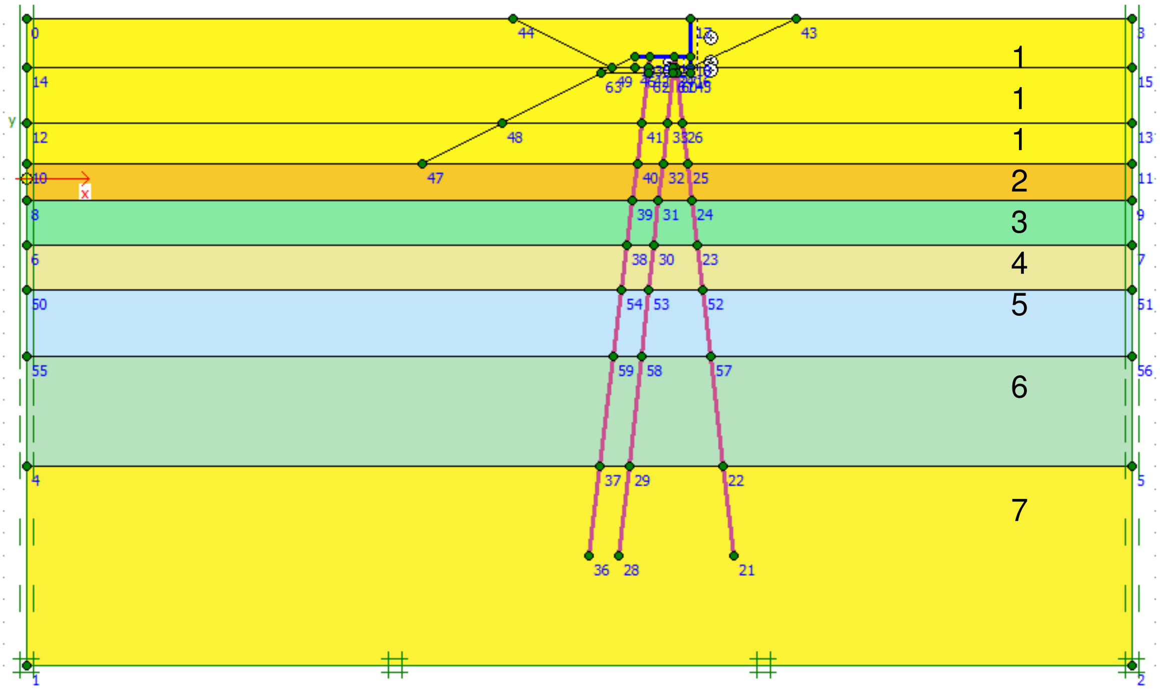


Figure 5. Input geometry

We recognize a total of 7 different soil layers:

Layer nr.	**layer**	**Top / bottom [m NAP]**	**Material model**
1	Sand, embankment	+7.2 / +0.7	HS
2	Sand, toplayer	+0.7 / -1	HS
3	Clay	-1 / -3	SSC
4	Peat	-3 / -5	SSC
5	Clay, sandy	-5 / -8	SSC
6	Sand, clayey	-8 / -13	SSC
7	Deep sand	-13 / -22	HS

Table 1. Soil layers in model

Soil and structural properties

The properties of the different soil and structural materials are shown in the tables below. Note that all parameters not addressed should be left to their default values. Insert the values for the different materials and assign the material sets to the relevant layers and structural elements.

Identification		**Sand, embankment / Sand, toplayer / Deep sand**	**Sand, emb., c=5 and no TC**
Material model		Hardening soil	Hardening soil
Drainage type		Drained	Drained
γ_unsat	kN/m^3	18	18
γ_sat	kN/m^3	20	20
E_50^ref	kN/m^2	3.00E+04	3.00E+04
E_oed^ref	kN/m^2	3.00E+04	3.00E+04
E_ur^ref	kN/m^2	9.00E+04	9.00E+04
power (m)		0.5	0.5
c_ref	kN/m^2	1	5
ϕ (phi)	°	30	30
ψ (psi)	°	0	0
Tension cut off		yes	no
k_x	m/day	0.1	0.1
k_y	m/day	0.1	0.1
R_inter		0.67	0.67
OCR		1	1
POP	kN/m^2	0	0

Table 2. HS material properties (used for sand layers)

Identification		**Clay**	**Clay, sandy**	**Peat**	**Sand, clayey**
Material model		Soft soil creep	Soft soil creep	Soft soil creep	Soft soil creep
Drainage type		Undrained (A)	Undrained (A)	Undrained (A)	Undrained (A)
γ_unsat	kN/m^3	14	18	11	17
γ_sat	kN/m^3	14	18	11	19
λ* (lambda*)		0.12	0.08	0.22	0.015
κ* (kappa*)		0.024	0.016	0.044	0.003
μ*(mu*)		6.00E-03	4.00E-03	1.10E-02	7.50E-04
c_ref	kN/m^2	5	5	2	1
ϕ (phi)	°	20	23	18	27
ψ (psi)	°	0	0	0	0
k_x	m/day	2.00E-04	2.00E-04	5.00E-04	0.1
k_y	m/day	1.00E-04	1.00E-04	1.00E-04	0.1
R_inter		0.5	0.5	0.5	0.67
OCR		1.8	1.6	2	1.5
POP	kN/m^2	0	0	0	0

Table 3. SSC material properties (used for soft clay and peat layers)

Identification		**Abutment**
Material type		Elastic
Isotropic		Yes
EA_1	kN/m	2.00E+07
EA_2	kN/m	2.00E+07
EI	kN m^2/m	1.67E+06
d	m	1
w	kN/m/m	25
ν (nu)		0

Table 4. Plate properties (used for abutment)

Identification		**Pile row ctc 2.4 m**
E	kN/m^2	2.00E+07
γ	kN/m^3	(25-10) = 15 *(*)*
Pile type		predefined: massive circular pile
D	m	0.54
L_spacing	m	2.4
Skin resistance		Linear
T_top, max	kN/m	10
T_bot, max	kN/m	100
F_max	kN	1000
Values ISF		Default

Table 5. Embedded pile row properties (used for pile rows)

(*) With this choice the total weight in the model is OK, however the normal forces in the pile are less realistic since the soil weight is not taken into account in the normal forces. For this case the differences are considered to be small.

Modeling of abutment
The way the abutment is modeled is shown in Figure 6. Note that the situation is shown for the final calculation phases.

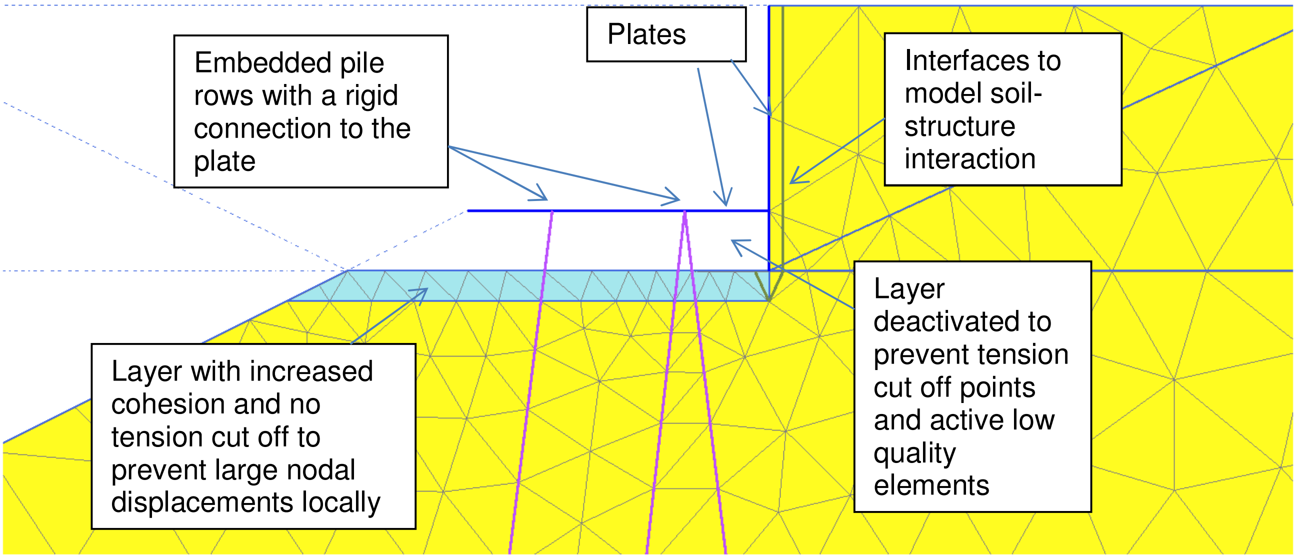


Figure 6. Detail of abutment

As a result of the large settlements of the embankment and the local interaction with the embedded pile rows, as well as the low effective stresses locally, large local nodal displacements may occur in the layer beneath the abutment. To prevent this behaviour we replace the material set in the layer beneath the abutment with a material set with improved properties during the calculation phases. The influence on results is expected to be small.

Input embedded pile row
Select the embedded pile row button from the toolbar and draw the element on the desired location (see below). Be sure to draw the element in one go from begin to end point.

Note that when you double click on the element you can select “embedded pile row” and then specify the top point of the embedded pile row and specify what the connection type here is. In this case the top point is the point with the largest y-coordinate and the connection type is rigid.

Note that the first point drawn of the embedded pile row element becomes (by default) the top point of the element.

The coordinates of the embedded pile rows are shown below:

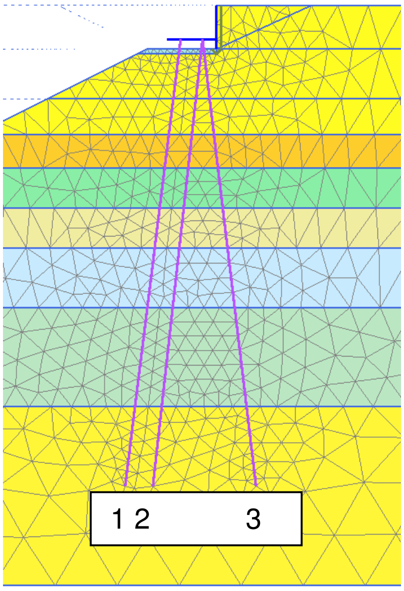


Figure 7. Numbering of embedded pile row

Pile row		Top	Bottom
1	X [m]	28.20	25.40
	Y [m]	5.50	-17.00
2	X [m]	29.30	26.80
	Y [m]	5.50	-17.00
3	X [m]	29.30	32.00
	Y [m]	5.50	-17.00

Table 6. Coordinates of embedded pile rows

Mesh

Mesh size: Medium.

Notice the automatic refinement around the structural elements (actually the line element size is set to 0.25 internally) giving a good mesh by default.

Use a point refinement once for each of the bottom points of the pile rows.

Waterlevel and Consolidation boundaries

Throughout all phases we use a general phreatic level at (Reference level) -0.5m.

We assume that consolidation is only possible through the upper and lower boundary. Therefore we use closed flow boundaries on the vertical boundaries of the model in all phases.

Phases

We use the phases, calculation types and time intervals as shown in Table 7.

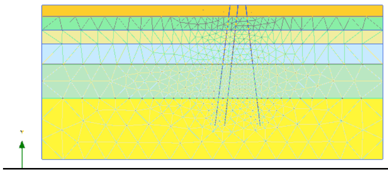
Note:

- Since these are all UM phases we cannot make use of the reset displacements option.
- To be able to compare results of the 2D model with a 3D model we only use an updated mesh and not updated pore water pressures since this feature is not implemented yet in 3D. An updated mesh is required to capture second order effects such as: additional bending moments due to eccentricity, reduced loading on piles due to soil layer thickness reduction, etc.

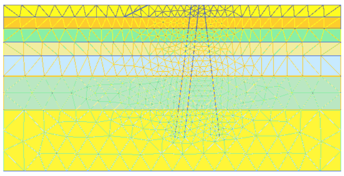
Identification	Phase no.	Start from	Calculation	Loading input	Pore pressure	Additional steps	Time interval
Initial phase	0	N/A	K0 procedure	Unassigned	Phreatic		0.00 day
emb. layer 1	1	0	Consolidation (UM)	Staged construction	Phreatic	250	2.00 day
cons. layer 1	2	1	Consolidation (UM)	Staged construction	Phreatic	250	30.00 day
emb. layer 2	3	2	Consolidation (UM)	Staged construction	Phreatic	250	2.00 day
cons. layer 2	4	3	Consolidation (UM)	Staged construction	Phreatic	250	60.00 day
emb. layer 3	5	4	Consolidation (UM)	Staged construction	Phreatic	250	2.00 day
cons. layer 3	6	5	Consolidation (UM)	Staged construction	Phreatic	250	90.00 day
Excavate for abutment	7	6	Consolidation (UM)	Staged construction	Phreatic	250	2.00 day
install abutment and piles + replace layer beneath abutment	8	7	Consolidation (UM)	Staged construction	Phreatic	250	10.00 day
remove sand beneath deck + sand fill behind abutment	9	8	Consolidation (UM)	Staged construction	Phreatic	250	5.00 day
consolidate < 5 kpa	10	9	Consolidation (UM)	Minimum pore pressure	Phreatic	1000	-

Table 7. Phases, calculation types and time intervals

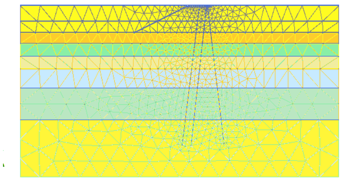
All phases used throughout the model are shown below.



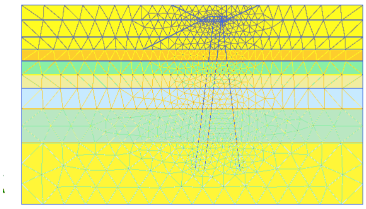
Initial phase



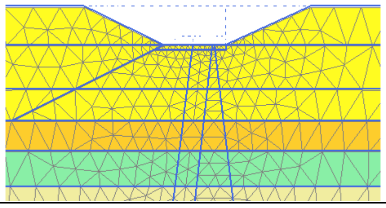
Phase 1 and 2



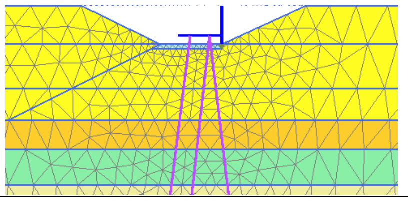
Phase 3 and 4



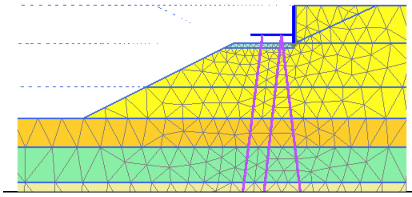
Phase 5 and 6



Phase 7



Phase 8



Phase 9 and 10

Table 8. Overview of phases

Results

Compare the results of the 2D model with the results of a comparable 3D model as shown in Annex A. Results of the structural forces from the 2D model are also shown together with the results of a comparable 3D model in annex A.

When looking at the deformations of the piles and abutment note the following: due to the use of updated mesh a reset displacements is not possible. As a result when activating the embedded piles they directly obtain the deformations of the soil. The total displacements of the piles and abutment relative to their initial (installation) position can be calculated as the sum of all the phase displacements.

Variations

To gain more insight on the possibilities and limitations of the embedded pile row you could try to make the following variations with the 2D model:

Use of plate elements instead of embedded pile row

Replace the embedded pile row with plate elements with an interface to see the differences and the added possibilities of the embedded pile row compared with plates. Make an appropriate choice for the R_inter of the interface (or use the default values presented in the tables). Results of this calculation have been presented in Annex B.

Variation of ISF

Vary the ISF factor of the embedded pile row with a factor of 10 up and down to see the influence on results.

Stiffer axial pile behaviour (displacement pile)

In Plaxis we cannot model the installation effect of piles. As a result piles behave as a bored pile. In general the installation effect (soil displacements) will result in a stiffer axial behaviour of the pile. To simulate this stiffer behaviour some tricks may be used in practice. For demonstration purposes we now start here with the use of the embedded piles as if they were bored piles (although in practice mostly driven piles are used in this situation). As a variation we may add a fixed end anchor at the bottom of the piles to see how this influences results.

Resume

From the comparison made in this case between the 2D and 3D model it is has become clear that the new structural element *Embedded Pile Row* is able to represent pile behaviour in a 2D model in a better qualitative and quantitative manner compared with the current possibilities (plates and n2n anchor).

With results so far it seems that the default settings already may give good results. Nevertheless users should ensure themselves that, when the defaults are being used, these values are valid for their situation. Otherwise a new set of ISF values should be derived by validating behaviour with 3D calculations, measurements, codes of practice, etc.

References

Brinkgreve, R.B.J., Engin, E. and Dao, T.P.T. (2012). Possibilities and limitations of embedded pile elements for lateral loading. ISSMGE – TC211 Int. Symp. on Ground Improvements IS-GI, May-June 2012, Brussels (Belgium).

Sluis, J. (2012), Validation of Embedded Pile Row in Plaxis 2D, M.Sc. thesis Delft University of Technology, Delft (The Netherlands).

Annex A: 3D model

A 3D model has also been made of the case presented here to be able to compare 2D and 3D results of the embedded piles. The results are presented in this annex.

Model set up

The 3D model set up is the same as the 2D model (i.e. same geometry, material parameters, boundary conditions, etc.), only differences being:

- use of 10 noded tetrahydral elements (2nd order) in 3D instead of the 15 noded triangular elements (4th order) in 2D;
- use of the "real" pile positions, see graph below.

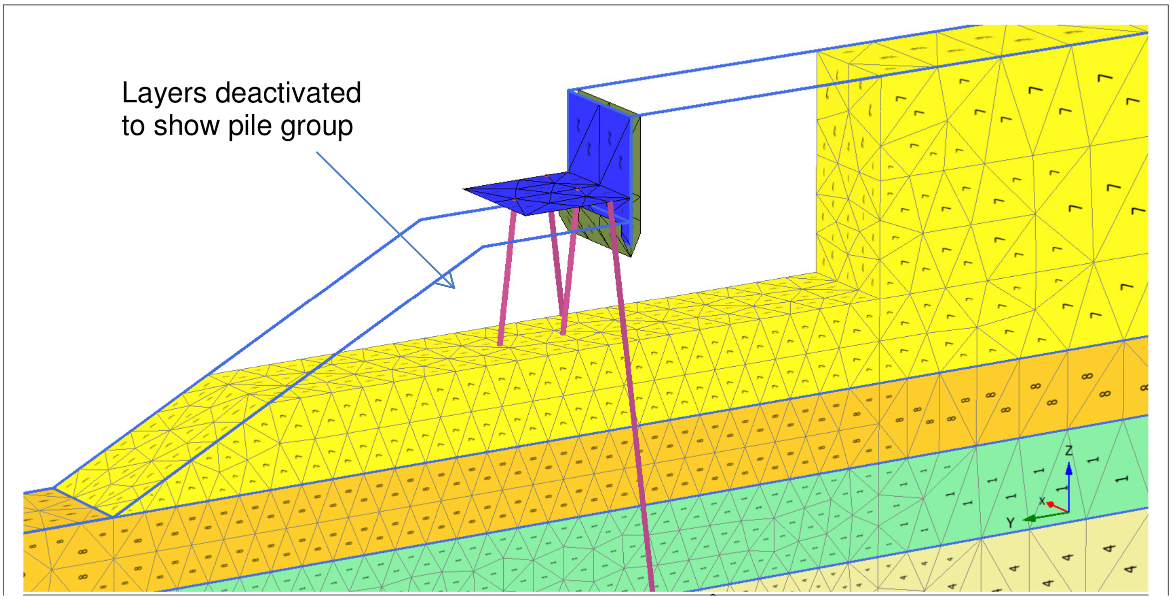


Figure A.1. Set up of abutment in 3D model

3D embedded piles

A 3D embedded pile is a pile composed of beam elements that can be placed in arbitrary direction in the sub-soil and that interacts with the sub-soil by means of special interface elements. The interaction may involve a skin resistance as well as a foot resistance. Although an embedded pile does not occupy volume, a particular volume around the pile (elastic zone) is assumed in which plastic soil behaviour is excluded. The size of this zone is based on the (equivalent) pile diameter according to the corresponding embedded pile material data set. This makes the pile almost behave like a volume pile. However, installation effects of piles are not taken into account and the pile-soil interaction is modeled at the centre rather than at the circumference.

In Plaxis 3D it is possible to place an embedded pile at a boundary of the model. However it should then be realized that in this case basically half a pile is modeled. So stiffness, weight and strength properties should be divided by two.

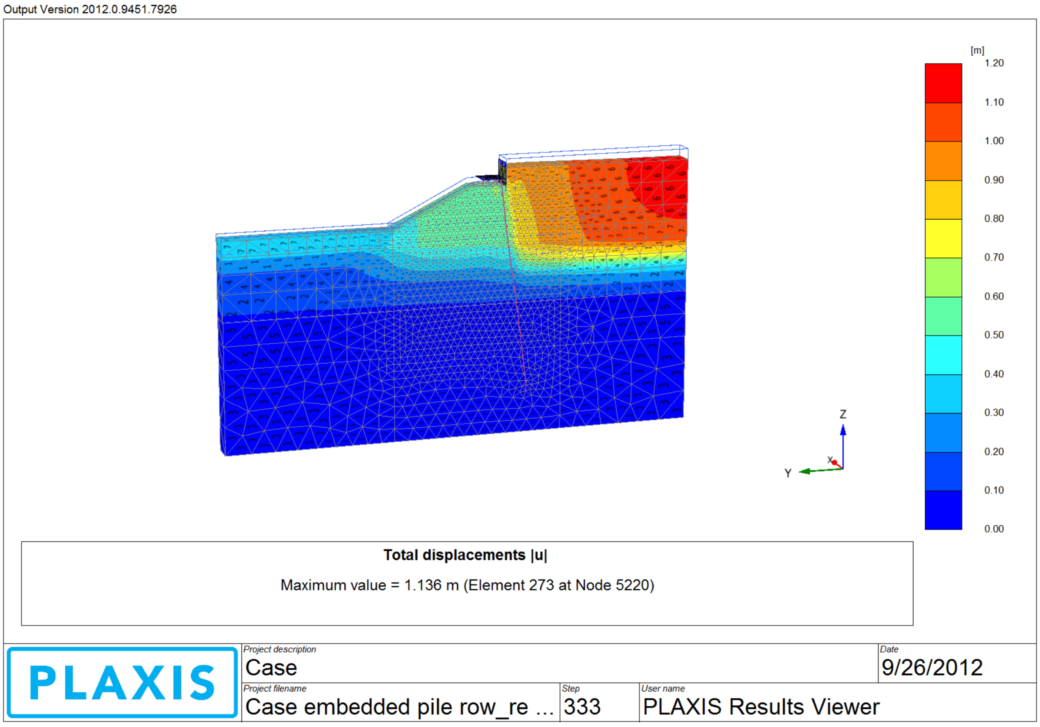
Lateral loading of embedded piles

In (Brinkgreve et al, 2012) the lateral loading capabilities for rough piles of the PLAXIS embedded pile element are validated. Although the embedded pile was primarily developed to describe the axial loading behaviour of foundation piles, the results of this research show that it has lateral loading capabilities as well. Distinction should be made between serviceability states (relatively small differential displacements between the pile and the soil) and ultimate limit states (large differential displacements and failure).

1. Considering serviceability states it can be concluded that the embedded pile behaves quite realistic and similar to a ‘classical’ finite element model in which the pile is modeled using volume elements with or without surrounding interfaces. Displacements and bending moments of the embedded pile are similar to those observed for the volume pile model. This conclusion applies to piles subjected to a lateral force at the top, as well as piles subjected to lateral soil movement.
2. Considering ultimate limit states it can be concluded that the embedded pile generally over-estimates the lateral loading capacity, at least when it is used in a ‘normal’ way, i.e. without defining a cylinder equal to the elastic zone around the embedded beam. In order for users to improve the behaviour of the embedded pile for ultimate limit states, a local refinement around the embedded pile could be applied. Moreover, the developers could consider improving the embedded pile element by extending the line-to-volume and point-to-volume interface elements (as part of the embedded pile formulation) with elastoplastic springs in lateral direction.

Model results

To be able to compare results please note the different set of axis in 2D and 3D. In 2D horizontal deformations are in x-direction, whereas they are in y-direction in this 3D model. Also note the legend settings (min/max and amount of intervals), be sure they are the same for both models when comparing. Note that you can adjust the legend settings by double clicking the legend in Output.



Total displacements |u|

Maximum value = 1.136 m (Element 273 at Node 5220)

Project description: Case

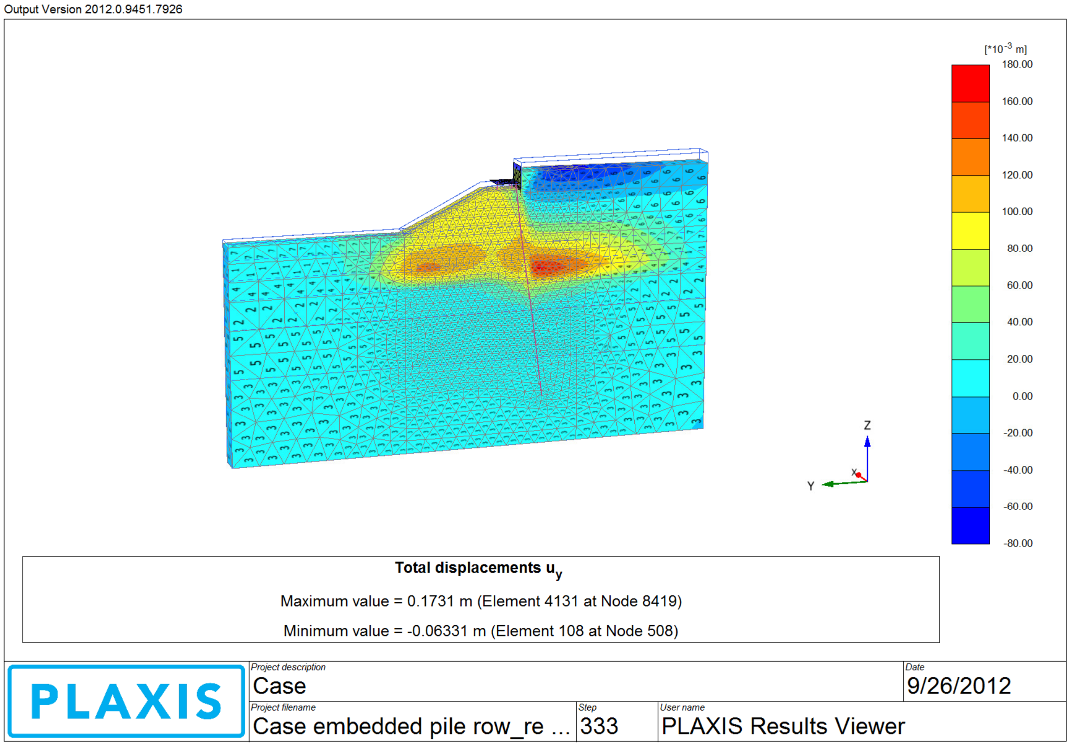
Date: 9/26/2012

Project filename: Case embedded pile row_re ...

Step: 333

User name: PLAXIS Results Viewer

Figure A.2. Total displacements



Output Version 2012.0.9451.7926

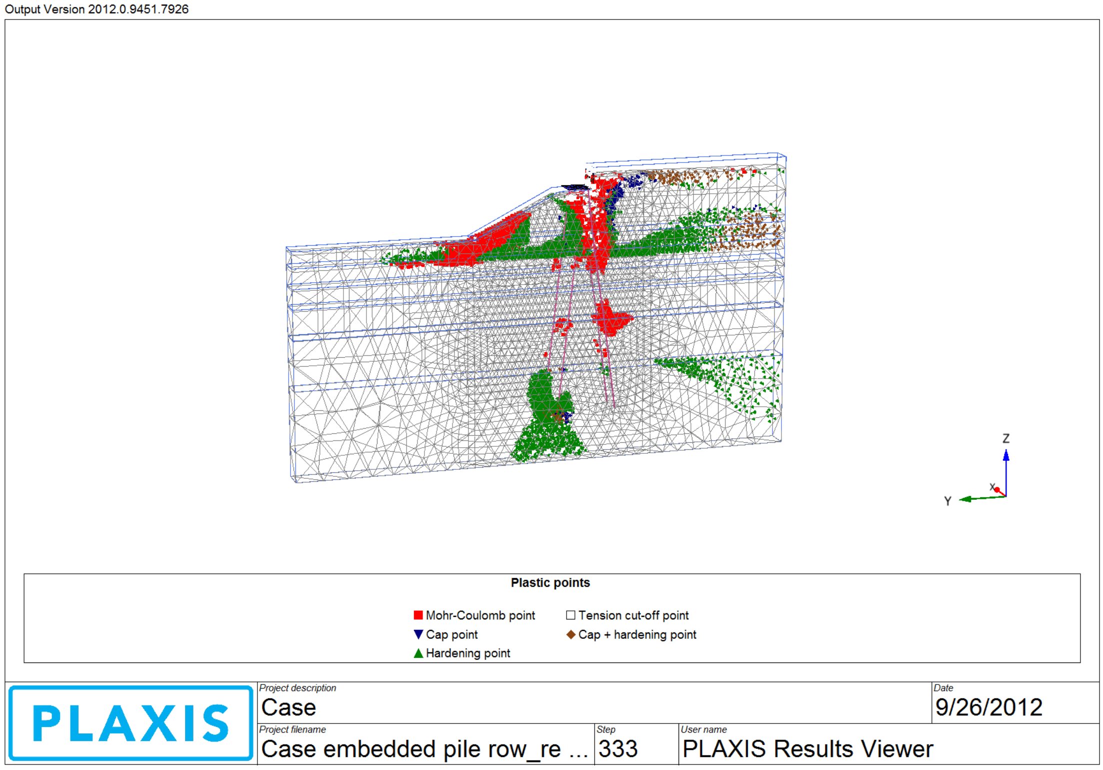
Total displacements u_y

Maximum value = 0.1731 m (Element 4131 at Node 8419)

Minimum value = -0.06331 m (Element 108 at Node 508)

Project description: Case
Date: 9/26/2012
Project filename: Case embedded pile row_re ...
Step: 333
User name: PLAXIS Results Viewer

Figure A.3. Horizontal displacements



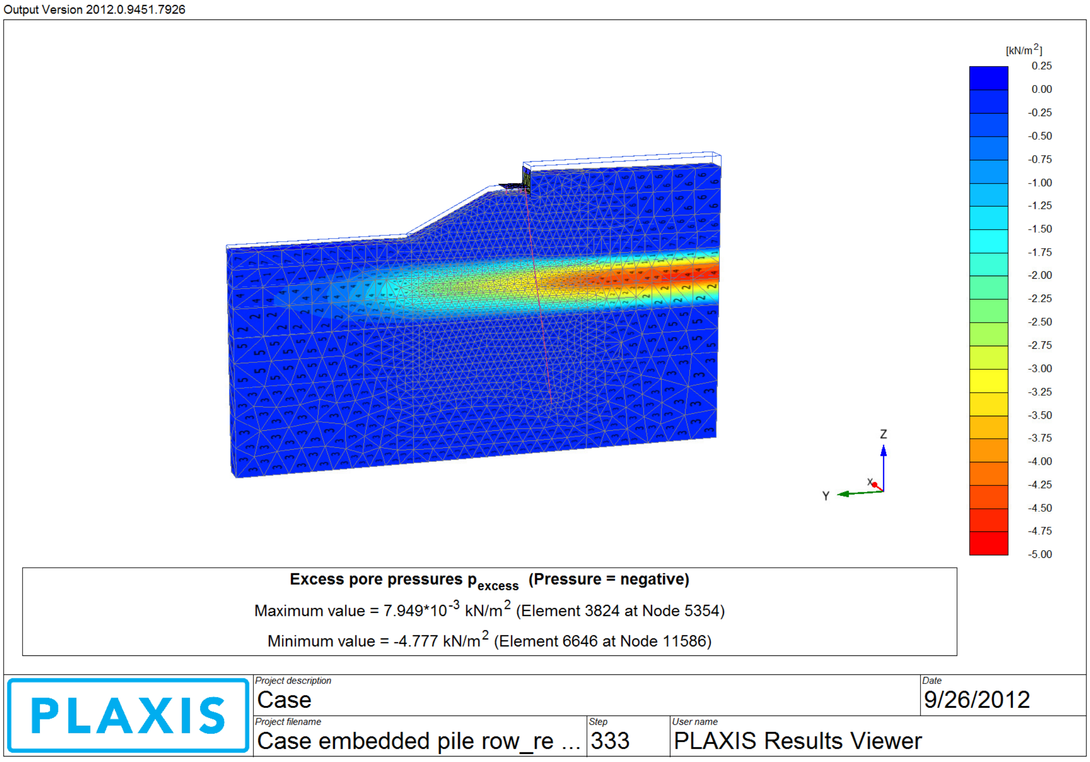
Output Version 2012.0.9451.7926

Plastic points

🟥 Mohr-Coulomb point
▼ Cap point
▲ Hardening point
☐ Tension cut-off point
◆ Cap + hardening point

Project description: Case
Date: 9/26/2012
Project filename: Case embedded pile row_re ...
Step: 333
User name: PLAXIS Results Viewer

Figure A.4. Plastic points


Output Version 2012.0.9451.7926

Excess pore pressures p_{excess} (Pressure = negative)

Maximum value = 7.949*10^{-3} kN/m^2 (Element 3824 at Node 5354)

Minimum value = -4.777 kN/m^2 (Element 6646 at Node 11586)

Project description: Case

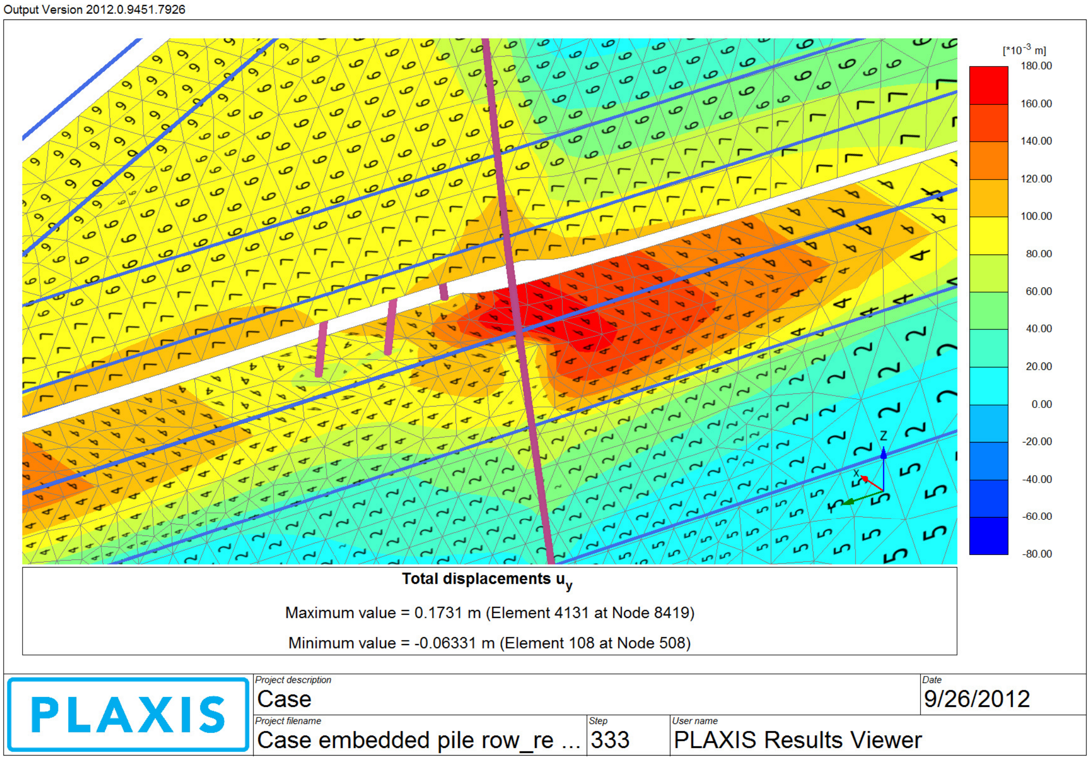
Date: 9/26/2012

Project filename: Case embedded pile row_re ...

Step: 333

User name: PLAXIS Results Viewer

Figure A.5. Residual Excess pore water pressures


Output Version 2012.0.9451.7926

Total displacements u_y

Maximum value = 0.1731 m (Element 4131 at Node 8419)

Minimum value = -0.06331 m (Element 108 at Node 508)

Project description: Case

Date: 9/26/2012

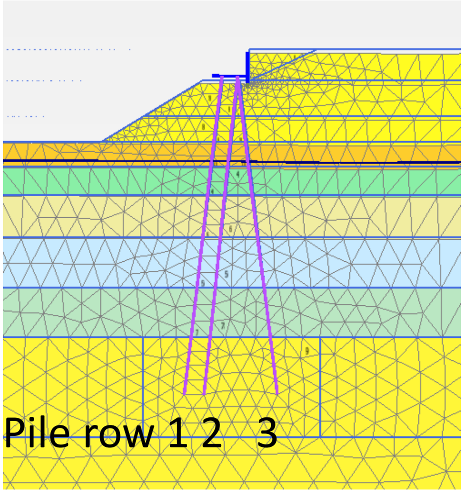
Project filename: Case embedded pile row_re ...

Step: 333

User name: PLAXIS Results Viewer

Figure A.6. Detail of horizontal displacements in soft layers

In the following figures the structural forces are compared. The piles are numbered in the following order:



Note: the 2D structural forces have been multiplied with the ctc distance whereas the values of the “half piles” from the 3D are multiplied with a factor of 2.

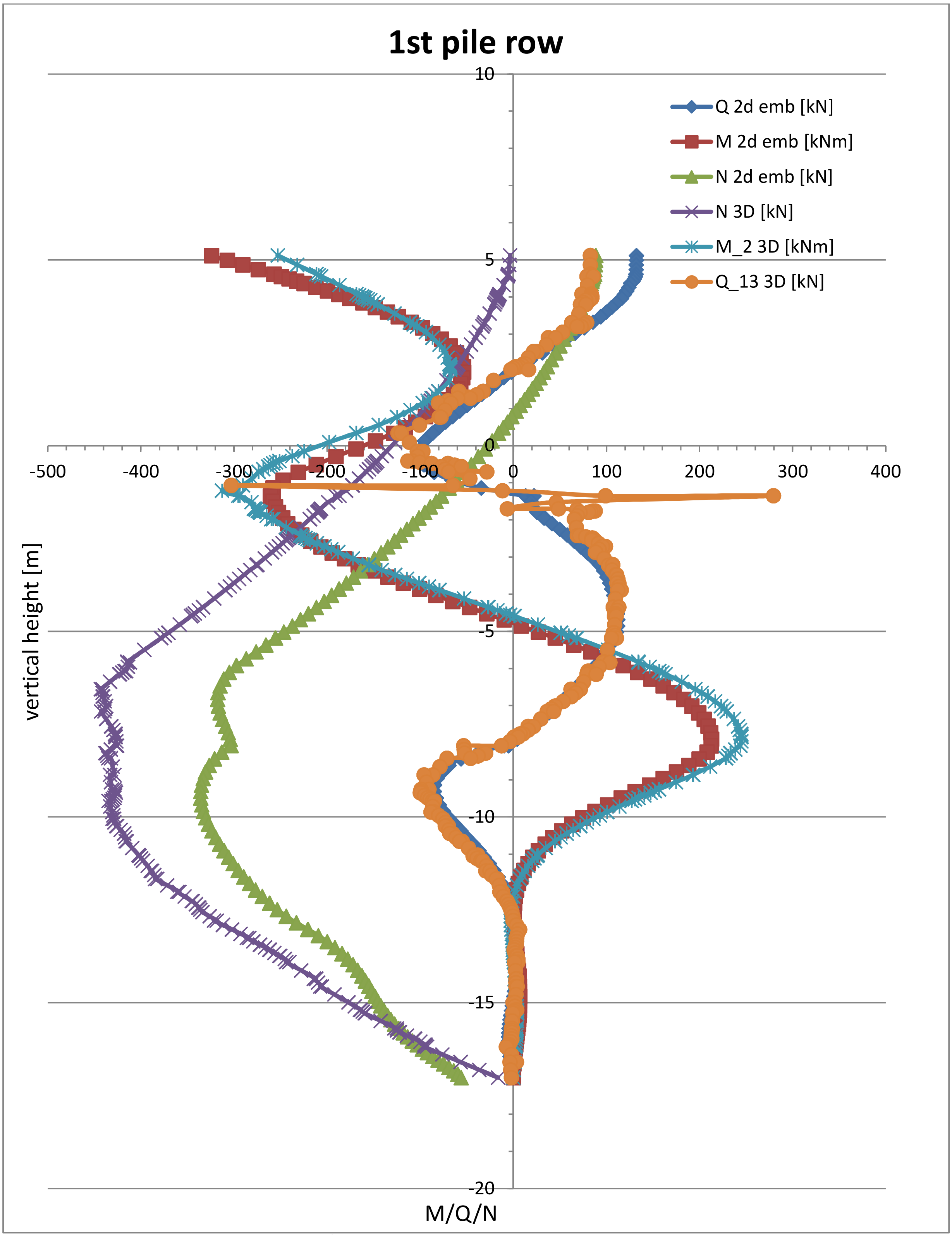


Figure A.7. Structural forces of 1st pile row

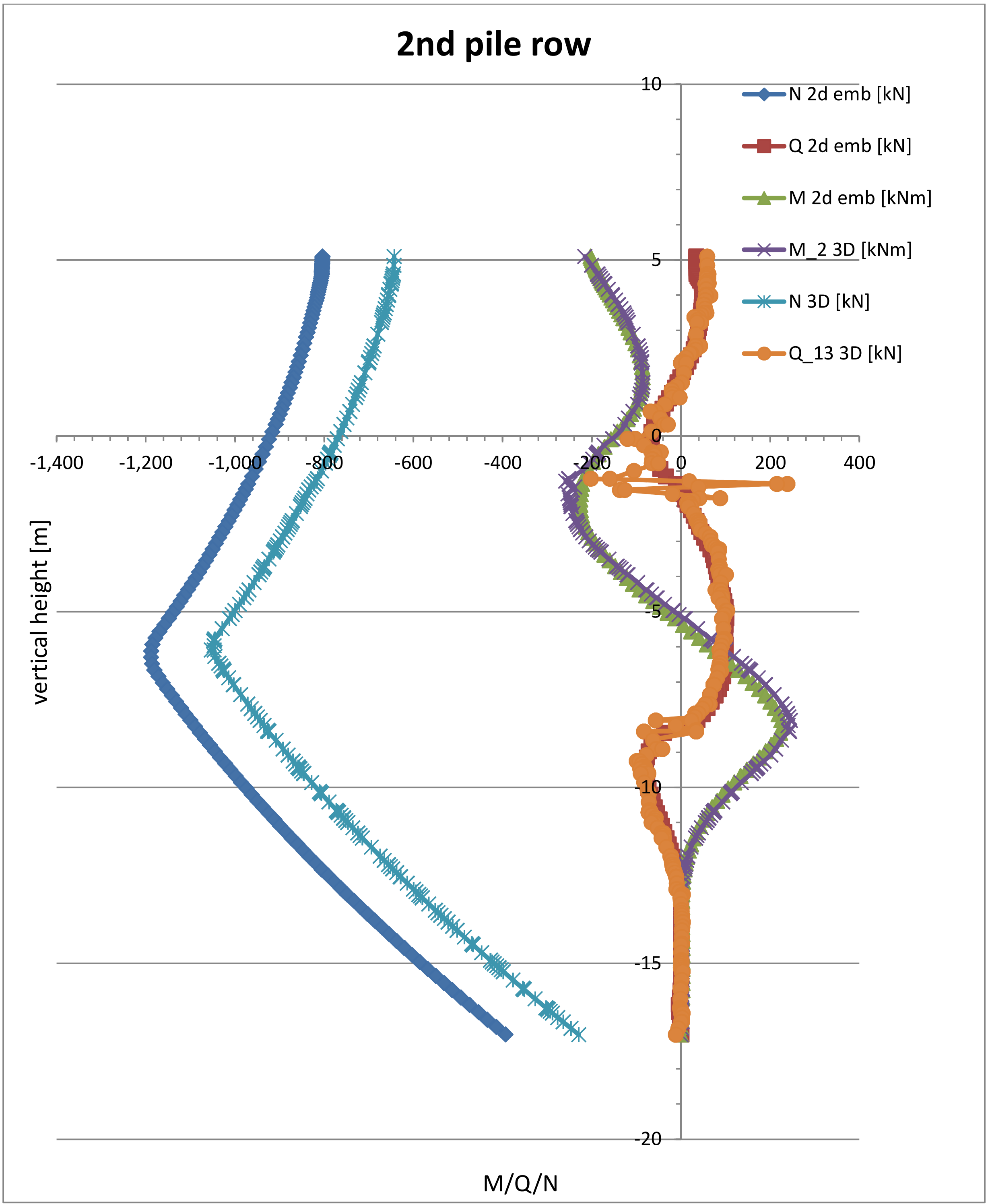


Figure A.8. Structural forces of 2nd pile row

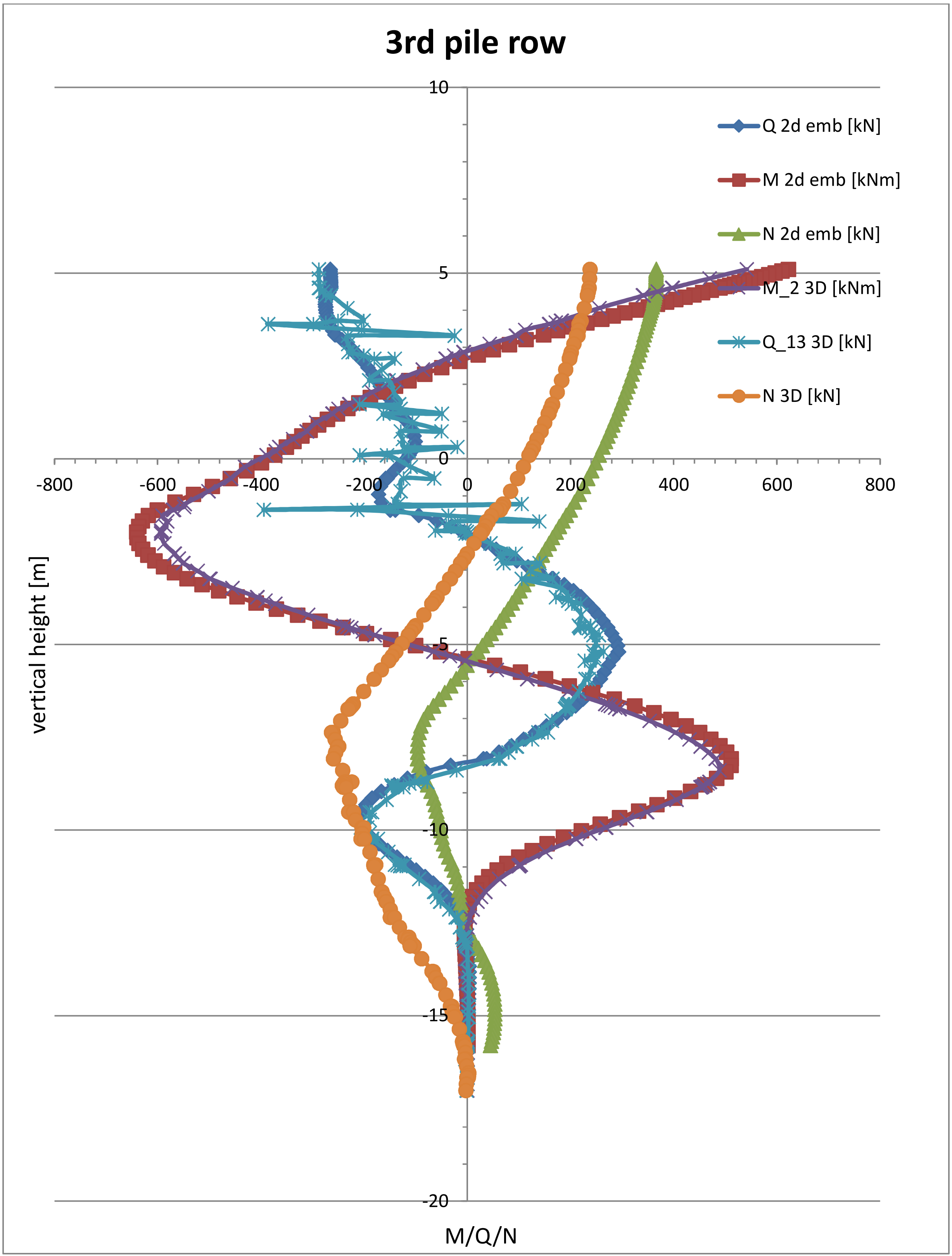


Figure A.9. Structural forces of 3rd pile row

Annex B: 2D model with plate elements

To be able to compare the 2D embedded pile row model results with a comparable 2D plate model, results of the 2D plate model have been presented in the annex.

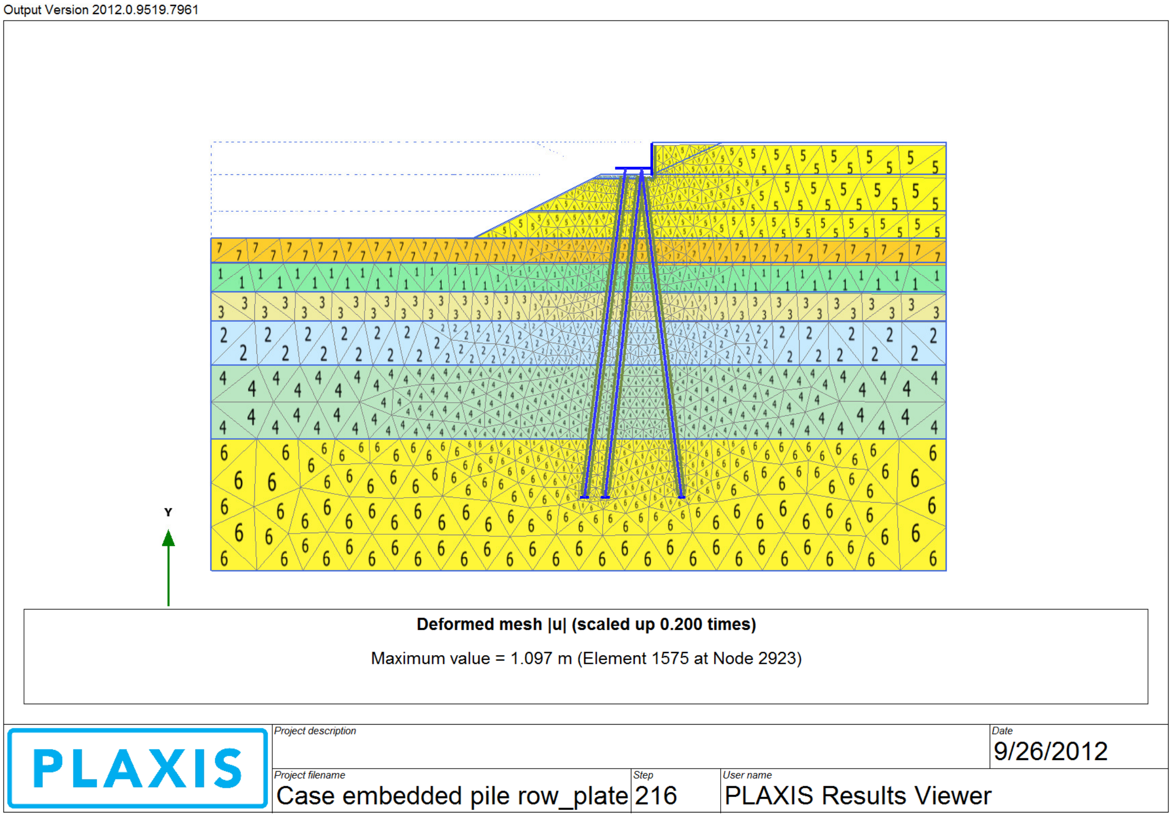
The model set up is the same as the 2D model, with the differences being that piles are modeled using plates and interfaces.

Properties per pile
EA pile = 4.58E6 kN
EI pile = 83.48E3 kNm2
Weight = 15 kN/m3 * 0.229 m2 = 3.44 kN/m

Plate properties
EA plate = 4.58E6 kN / 2.4 m = 1.91E6 kN/m
EI plate = 83.48E3 kNm2 / 2.4 m = 34.78E3 kNm2/m
Weight = 3.44 kN/m / 2.4 m = 1.43 kN/m/m

Note:

- Make a choice how to model the pile foot.
- Be sure to deactivate the interfaces along the piles/plates in Water Conditions mode to allow for consolidation flow.



Output Version 2012.0.9519.7961

Deformed mesh |u| (scaled up 0.200 times)

Maximum value = 1.097 m (Element 1575 at Node 2923)

Project description

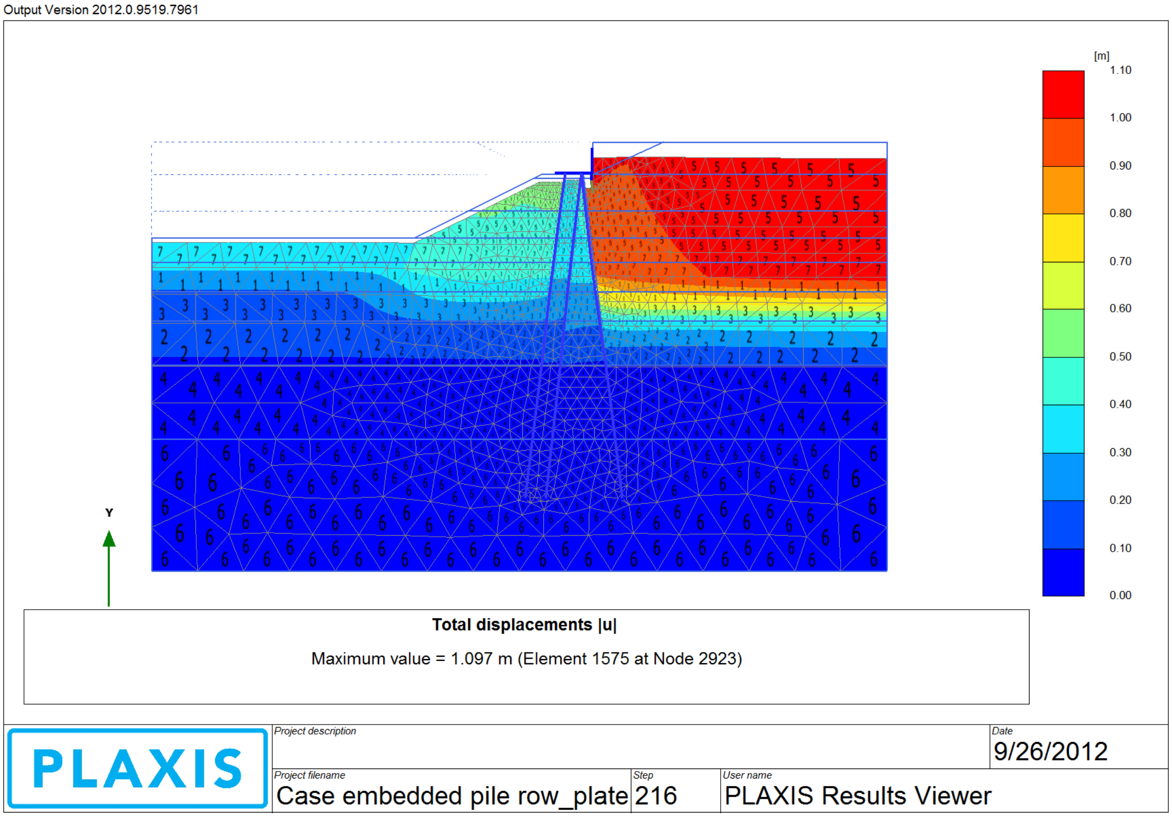
Date: 9/26/2012

Project filename: Case embedded pile row_plate

Step: 216

User name: PLAXIS Results Viewer

Figure B.1. Model overview last phase



Output Version 2012.0.9519.7961

Total displacements |u|

Maximum value = 1.097 m (Element 1575 at Node 2923)

PLAXIS

Project description:

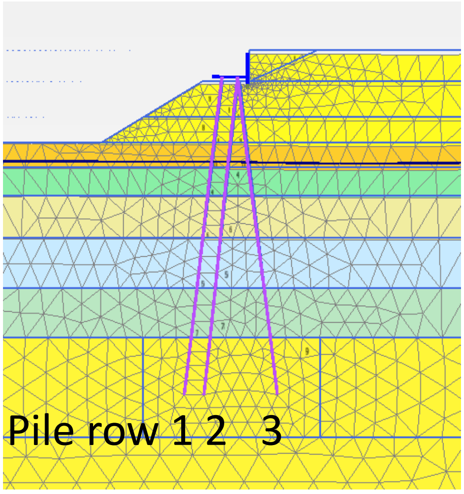
Date: 9/26/2012

Project filename: Case embedded pile row_plate

Step: 216

User name: PLAXIS Results Viewer

Figure B.2. Total deformations last phase



Note: the 2D structural forces have been multiplied with the ctc distance whereas the values of the “half piles” from the 3D program are multiplied with a factor of 2.

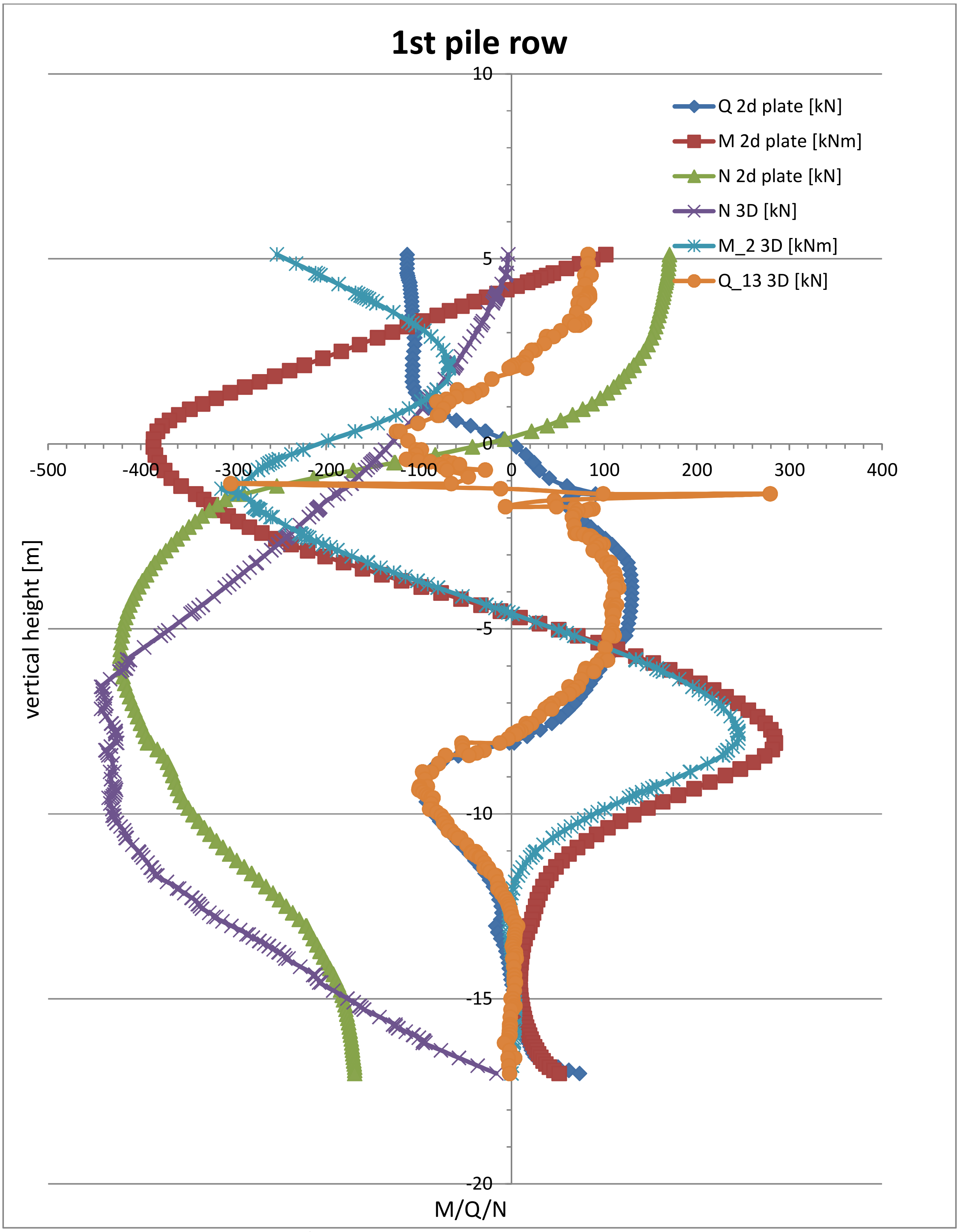


Figure B.3. Structural forces 1st pile row

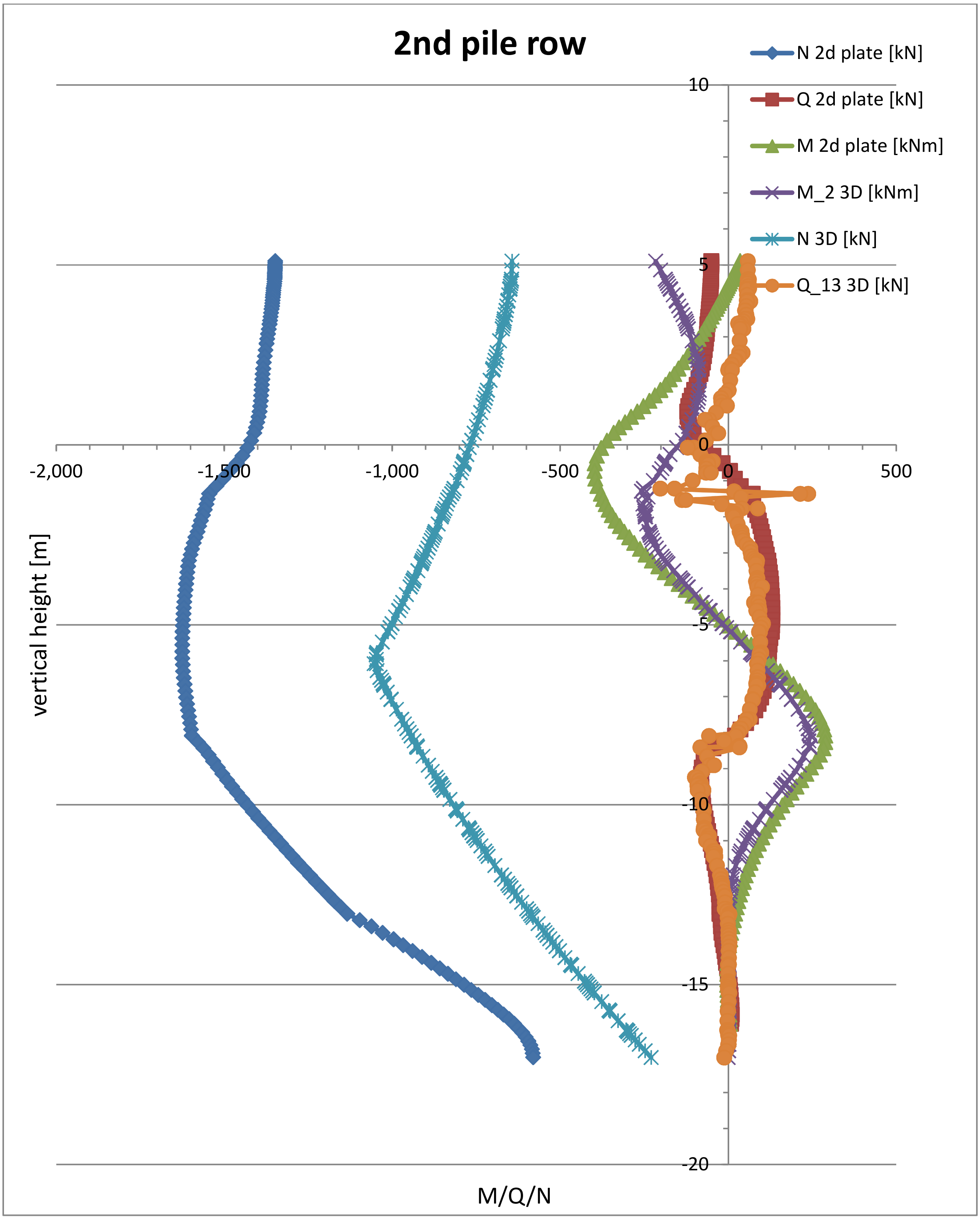


Figure B.4. Structural forces 2nd pile row

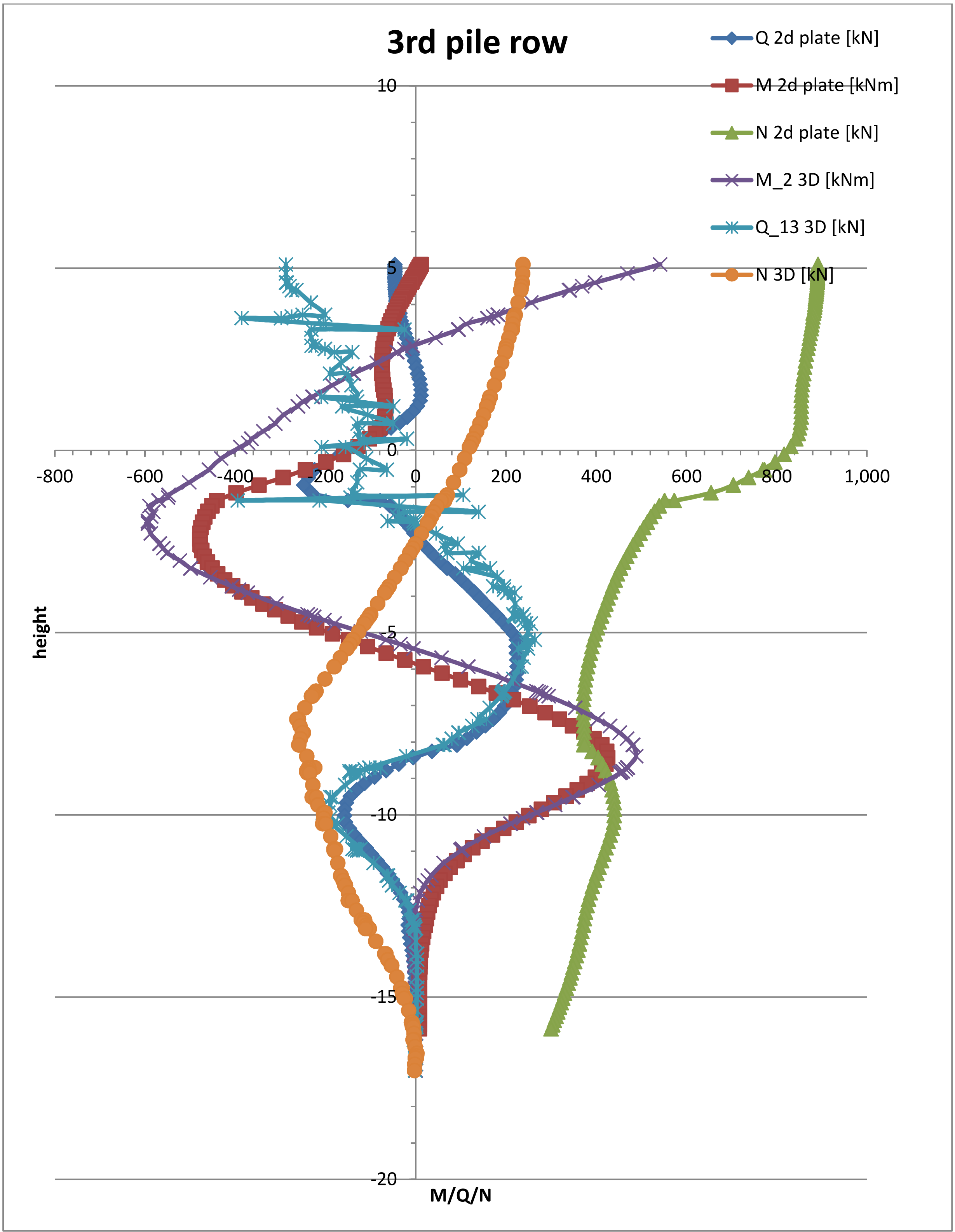


Figure B.5. Structural forces 3rd pile row

Resume
There is a large difference in normal forces in the 2nd and 3rd pile row. This is probably a result of the stiffer behaviour of the “pile rows” in this model due to the larger skin/shaft area.

For the top of the pile rows the shear force and bending moments are less realistic (qualitatively and quantitatively).

Further adjustment of the R_inter values may help to improve results somewhat.