



3D-interaction of Structure and Subsoil for a New Fly Ash Silo, Maasvlakte Rotterdam

ir. L.F.J. Bekken & ing. W.A. Nohl, Fugro GeoServices B.V., Rotterdam, The Netherlands

A new fly ash silo is built with a construction of reinforced concrete (height = 55 m, diameter = 24 m) as part of the new power plant at the E.ON site at the Maasvlakte Rotterdam, Netherlands. The total weight of the silo is 400 MN when filled with fly ash. The silo is constructed on a 2.5 m thick concrete base slab and is founded on 59 large diameter bored piles (\varnothing 1.5 m) with a length of 32 m. The soil investigation revealed a mainly sand profile with a clay layer at 10 m below pile tip level, which varies in presence and thickness. The heterogeneity of the subsoil may result in differential settlements, which will lead to higher stresses in the construction depending on the stiffness of the construction. Detailed information on the heterogeneity of the subsoil, the stiffness of the structure and the pile behavior is necessary to analyze the settlement behavior and the forces in one interaction model, where all components with its properties are incorporated. A realistic interaction analysis is performed in 3D with the finite element program PLAXIS 3D. The advantages of this approach in comparison to the general accepted design approaches with a conventional elastic support model are defined.

➤ As part of the new power plant at the E.ON site at the Maasvlakte, a new fly ash silo is built with a construction of reinforced concrete with a total height of 55 m and a diameter of 24 m. The total weight of the silo is 140 MN, and 400 MN when filled with fly ash. The silo is constructed on a 2.5 m thick concrete base slab with a diameter of 32 m. Figure 1 shows a cross section of the silo.

For the foundation design a grid of Cone Penetration Tests (CPT's) with depths varying from 40 to 70 m were performed. In addition, a borehole was carried out with selected undisturbed samples taken for geotechnical laboratory tests. Since deformations of the clay layers were expected to have a relatively large influence on the structure, mainly oedometer tests had been performed on the undisturbed clay samples. From the CPT's and borehole logs the soil profile had been determined. The global soil profile is presented in table 1.

The total structure is founded on 59 large diameter bored piles (\varnothing 1.5 m) with pile tip level in the second sand layer (NAP -29 m). The calculated design pile bearing capacity is 11.5 MN.

The settlements of the pile group are mainly caused by compression of the 2nd clay layer below

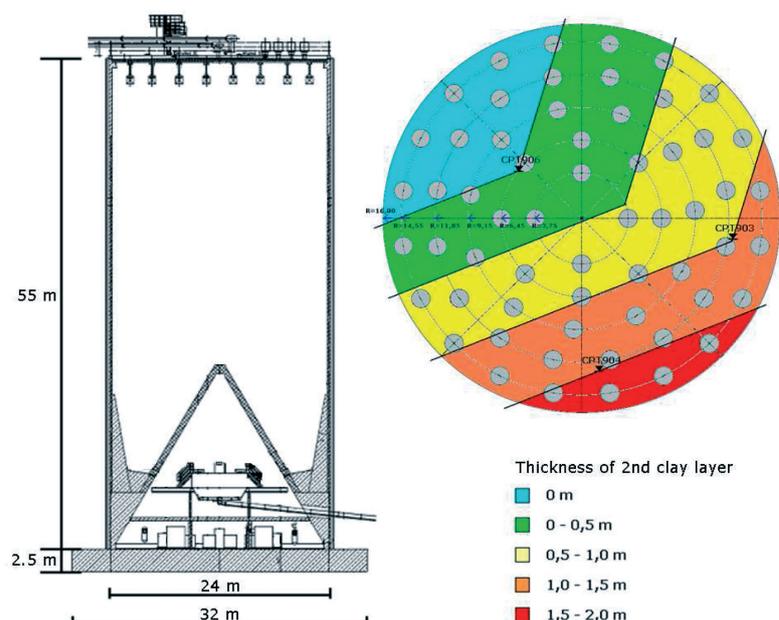


Figure 1: Cross section of the fly ash silo with top view of the pile plan and the thickness of the 2nd clay layer



Top of layer [NAP m]	Soil description	Soil layer
+5	SAND, medium dense to very dense	1st sand layer
-20	CLAY	1st clay layer
-21	SAND, very dense	2nd sand layer
-40	CLAY	2nd clay layer
-41	SAND, (medium) dense with laminations of clay	3rd sand layer
-65	Max. investigation depth	

Table 1: Global soil profile

pile tip level due to an increase of stresses. This 2nd clay layer varies in presence and thickness; at some locations the thickness is 1.5 m, but the layer is not encountered at some other locations, see figure 1. The differential settlements were restricted by the Clients specifications.

The heterogeneity of the subsoil may result in differential settlements, which will lead to higher stresses in the construction depending on the stiffness of the construction. The (differential) settlements were calculated both with the analytical method and with a 3D finite element model. With the analytical method only the behavior of the subsoil can be modeled and with a finite element model both the behavior of the subsoil and the interaction with the structure can be modeled. As a consequence of the interaction also stresses and bending moments in the structure can be determined.

The advantages of the 3D finite element analysis in comparison to the general accepted design approaches with a conventional elastic support model are defined in this article.

2. Settlement analyses with analytical model

2.1 General

The total settlement (s_d) of the pile group was

calculated according to the following relation [1]:

$$s_d = s_{t,d} + s_{2,d} \quad (1)$$

in which $s_{t,d}$ is the settlement of the pile head, consisting of the pile tip deformation ($s_{b,d}$) including the elastic pile deformation ($s_{el,d}$) and $s_{2,d}$ is the settlement caused by compression of (cohesive) layers below pile tip level.

2.2 Settlement s_t

The settlement s_t was calculated by dividing the pile force by the axial spring stiffness of a single pile. The axial spring stiffness was calculated according to the Dutch code [1]. It should be emphasized that the spring stiffness is not a constant value and depends on the actual forces.

The shaft friction only takes place effectively in the dense sand layer underneath the 1st clay layer. In reality the behavior will be stiffer on short term because the piles will also mobilize shaft friction in the top sand layers. But due to settlements in the 1st clay layer, the mobilized shaft friction in the top sand layers will be reduced considerably. Therefore the shaft friction of the top sand layers was ignored in the basic calculation of the pile bearing capacities.

2.3 Settlement s_2

The settlements of the whole pile group are mainly caused by compression of cohesive layers below the pile tip level. The settlements were calculated with the computer program DSettlement, which uses the Dutch method of Koppejan (more or less equivalent to the Bjerrum method). DSettlement calculates the settlements in the subsoil caused by an increase of stresses. Load spreading according to the Boussinesq method is hereby taken into account.

The load of the construction was modeled as a uniform distributed load at pile tip level. It is not possible to model the stiffness of the structure in detail. However it is possible to simulate a certain stress distribution depending on an estimate of stress concentration from the structure. The two extreme stress distributions are:

- Flexible plate ($\alpha = 1$): uniform distributed load with value p ;
- Infinite rigid plate ($\alpha = 0$): distributed load with a value $2p$ at the perimeter and a value of 0 in the centre of the plate.

The calculated settlements for the simulated flexible plate are presented in figure 2. The maximum settlement occurs in the centre of the plate. The maximum settlement is 18 cm with the 1.5 m thick 2nd clay layer taken into account and 5 cm without the presence of this clay layer.

The calculated settlements for the simulated infinite rigid plate are presented in figure 3. The maximum settlement occurs at the perimeter of the plate. The maximum settlement is 14 cm with the 1.5 m thick 2nd clay layer taken into account and 4 cm without the presence of this clay layer.

The settlements of the base slab according to the two stress distributions were not very realistic. Due to the relatively stiff base slab a uniform settlement of the base slab should be expected. Therefore the factor α was varied between 0 and 1. A uniform settlement of the concrete base slab was obtained with $\alpha = 0.6$, see figure 4.

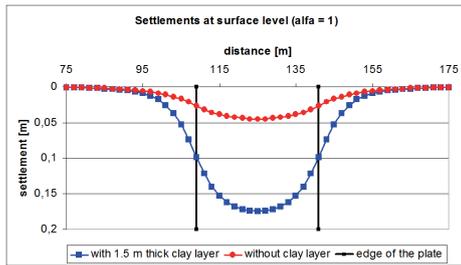


Figure 2: Settlements of the simulated flexible plate

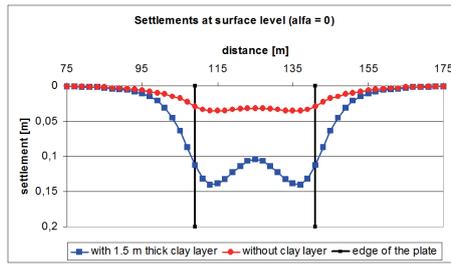


Figure 3: Settlements of the simulated infinite rigid plate

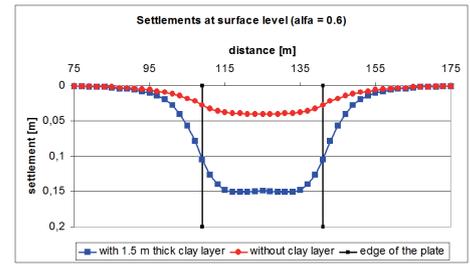


Figure 4: Settlements of the base slab

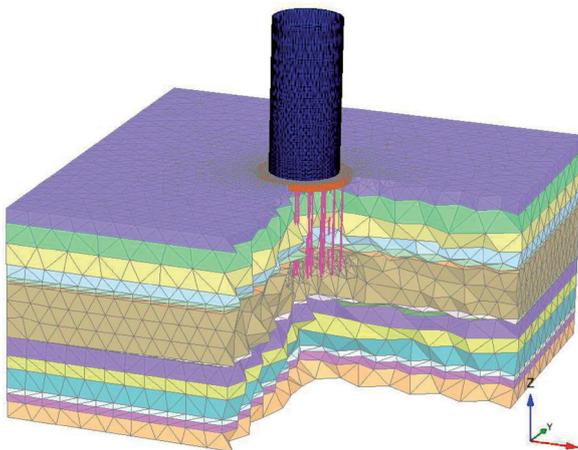


Figure 5: Input in PLAXIS 3D

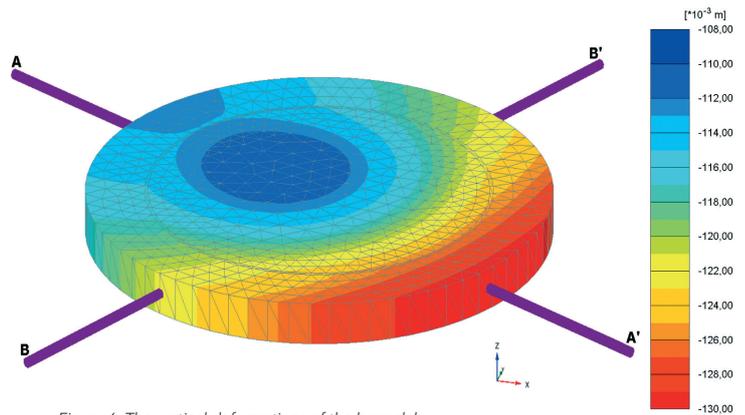


Figure 6: The vertical deformations of the base slab

From figure 4 it was concluded that the uniform settlement of the base slab is 15 cm with the 1.5 m thick 2nd clay layer taken into account and 4 cm without the presence of this clay layer. The differential settlement is therefore 11 cm. From these calculations it could not be concluded that the (differential) settlements meet the Clients requirements.

2.4 Limitations of the analytical model

A limitation of the analytical method is that no structure stiffness can be implemented. It is only possible to model a certain stress distribution. However this stress distribution again depends on the structure stiffness.

In addition, the load redistribution in the structure as a result of the difference in soil stiffness response is also not implemented in the analytical model. The soil without the presence of the clay layer behaves stiffer than the soil with a 1.5 m thick clay layer.

The actual stress distribution is determined by the interaction of the structure and soil stiffness. A realistic interaction analysis, in which both the heterogeneity of the subsoil and the stiffness of the structure are incorporated, was performed in 3D with the finite element program PLAXIS 3D.

3. Finite element model

3.1 General

The main advantages of the finite element analyses are:

- Stress redistribution as a result of the structure stiffness;
- Stress redistribution as a result of the soil stiffness;

- Modeling of the pile-soil interaction;
- More advanced soil models are implemented, in which it is possible to model a strain-level dependent soil stiffness.

The main choices for a 3D model were:

- The geometry of the structure is 3D (circular plate with piles);
- Variation of the thickness and depth of the clay layer in three directions;
- 3D stress distribution in the subsoil.

The finite element analysis was performed to determine the total and differential settlements of the concrete base slab and the silo wall due to the heterogeneity of the 2nd clay layer. Furthermore the analysis should prove whether excessive load concentrations in the structure could be expected and in what degree tilting might occur. The 3D finite element model is shown in figure 5.

3.2 Input

The stiffness of the superstructure was modeled by the dimensions and properties of the foundation slab and the silo wall. The base slab was modeled as a volume element with a linear elastic material behavior. The silo was modeled with plate elements with linear elastic material behavior. Furthermore structural properties like the connection of the silo wall and the foundation slab were included in the finite element model.

The foundation piles were modeled by "embedded piles" which distribute the structure load to the bearing sand layer underneath the 1st clay layer. The pile-soil interaction was modeled with a representative skin resistance of 0 kN/m until the 1st clay layer and 500 kN/m below this layer and with a representative base resistance of 13 MN according to the pile bearing design. The spring characteristic of a single large diameter

bored pile was checked with a calibration calculation for one foundation pile (see chapter 3.4).

3.3 Soil model and parameters

The sand layers were modeled with the Hardening Soil model with small strain stiffness (HSsmall). The soil stiffness parameters of the HSsmall model were based on the CPT results.

The clay layers were modeled with the Soft Soil Creep (SSC) model. The soil parameters were based on CPT and laboratory results. The SSC model is suitable when considering creep, i.e. secondary compression. The creep was taken into account during a period of 30 years.

The representative soil parameters of the 2nd sand layer and the 2nd clay layer are given in table 2.

3.4 Verification

Before modeling the total structure, the pile-soil interaction was verified for a single foundation pile in the 3D finite element model. One large diameter bored pile (Ø 1.5 m) with a length of 34 m was modeled in the subsoil.

From the finite element calculations it was concluded that the foundation pile would have a vertical displacement of 95 mm due to a representative vertical load of 9 MN. The corresponding axial pile spring stiffness was 95 MN/m. The axial spring stiffness of the pile corresponded with the calculated spring stiffness for a single pile according to the Dutch codes. Therefore, the pile-soil behavior of a single pile in the 3D finite element model was verified.

4. Results

4.1 Settlements

The vertical (differential) settlements of the

concrete base slab are shown in figure 6. Figure 7 shows a graph with the settlements of the base slab for cross sections A-A' and B-B'.

The figures 6 and 7 show an average settlement of 0.12 m for the concrete base slab. The figures also show a maximum difference in settlements of 20 mm between the centre and the perimeter of the base slab. The foundation tilts towards the positive x-axis. The settlement of the concrete base slab is 0.130 m with the 1.5 m thick 2nd clay layer taken into account and 0.113 m without the presence of this clay layer. Thus, the maximum differential settlement between the two edges of the base slab is 17 mm at a distance of 32 m, which results in a rotation of approximately 1 : 2,000.

Due to stress redistributions in the finite element program the rotation is smaller compared to the analytical model. From these calculations it could be concluded that the (differential) settlements meet the Clients requirements.

4.2 Bending moments

The bending moments (M) in the concrete base slab are calculated from the settlement graphs according to the following relation:

$$M = -EI \frac{d^2\phi}{dx^2} \quad (2)$$

in which EI is the bending stiffness of the concrete base slab and $d^2\phi/dx^2$ is the curvature. The corresponding graph is presented in figure 8. The bending moment in the centre of the base slab is 3,750 kNm.

The bending moment in the centre of the base slab can also be determined from the normal stresses, which follow directly from the finite element calculations. Figure 9 shows the normal stresses in the concrete base slab along cross section A-A'. The maximum normal stress in the centre of the base slab is 3,600 kPa. The corresponding bending moment is calculated according to the following relation:

$$M = \sigma \cdot W \quad (3)$$

In which σ is the normal stress and W is the section modulus. The corresponding maximum bending moment is 3,750 kNm.

4.3 Pile head forces

The representative pile head forces, as calculated in the 3D finite element analysis, are shown in figure 10. From the analysis it was concluded that about 80% of the structure loads were transferred to the piles. The remaining 20% of the structure loads were directly transferred from the plate to the subsoil. If the load is evenly distributed over the 59 piles, the representative pile head load will be 5,450 kN/pile. However due to stress redistribution the piles in the area without the 2nd clay layer carry more load than the piles with the presence of the 1.5 m thick 2nd clay layer. The stress redistribution in the 3D finite element analyses results in a reduction of differential settlements.

Figure 10 also shows that the pile head forces at the outer ring are larger than the pile head forces at the inner rings.

Conclusions

The following conclusions can be drawn:

- The differential settlements obtained with the analytical method are not realistic because

stress redistribution due to structure stiffness and differences in soil stiffness are not implemented.

- Due to the limitations of the analytical method, finite element analyses are performed in which the heterogeneity of the subsoil and the stiffness of the structure are incorporated in a realistic interaction calculation model. A 3D finite element model is selected since the geometry is 3D (circular base plate with foundation piles) and the heterogeneity of the subsoil varies in three directions.
- The calculated differential settlements of the base slab obtained with the 3D finite element model are considerably smaller than the differential settlements obtained with the analytical method in accordance with the usual design practice. The results obtained with the 3D finite element analysis are more realistic than the results obtained with the analytical method because the stress redistributions due to interaction is incorporated.
- The finite element analysis also provides additional information, like bending moments in the superstructure and pile forces.

References

- [1] NEN-EN 9997-1:2011 (NL), Geotechnical design of structures, Part 1: General rules (Eurocode 7), Nederlands Normalisatie-instituut, Delft, 2011.
- [2] PLAXIS manual, PLAXIS 3D 2010, Plaxis bv, Delft, 2010.

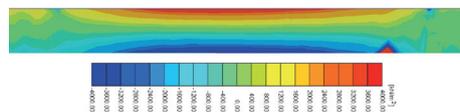


Figure 9: The normal stresses (σ_{yy}) in the base slab

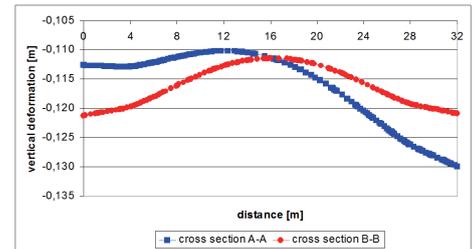


Figure 7: The vertical deformations of the base slab

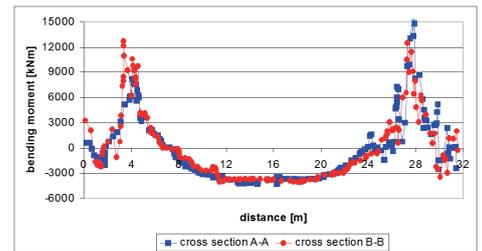


Figure 8: The bending moments in the base slab

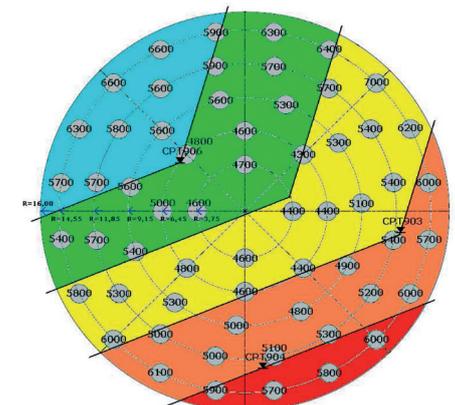


Figure 10: Representative pile head forces

2nd sand layer (HSsmall model)	2nd clay layer (SSC model)
$\gamma/\gamma_{sat} = 18 / 20 \text{ kN/m}^3$	$\gamma/\gamma_{sat} = 17 / 17 \text{ kN/m}^3$
Stiffness parameters	
$E_{50}^{ref} = 70,000 \text{ kPa}$	$\lambda^* = 0,062$
$E_{oed}^{ref} = 70,000 \text{ kPa}$	$\kappa^* = 0,012$
$E_{ur}^{ref} = 210,000 \text{ kPa}$	$\mu^* = 0,0024$
$m = 0,5$	
$p^{ref} = 100 \text{ kPa}$	
small strain stiffness	
$\gamma_0 = 0,0001$	
$G_0^{ref} = 265,000 \text{ kPa}$	
Strength parameters	
$c' = 0,1 \text{ kPa}$	$c' = 3,0 \text{ kPa}$
$\phi' = 35,0 \text{ kPa}$	$\phi' = 22,5 \text{ kPa}$
$\psi' = 5,0 \text{ kPa}$	$\psi' = 0,0 \text{ kPa}$

Table 2: Representative soil parameters