3D-interaction of Structure and Subsoil for a New Fly Ash Silo, Maasvlakte Rotterdam
Validation and Application of the Embedded Pile Row-Feature in PLAXIS 2D
3D Finite Element Analyses – Interaction between Abutment and Adjacent Earth Embankment of a Modern Railway Bridge
Table of contents

03 Editorial
04 New developments
05 PLAXIS Expert Services update
06 3D-interaction of Structure and Subsoil for a New Fly Ash Silo, Maasvlakte Rotterdam
10 Validation and Application of the Embedded Pile Row-Feature in PLAXIS 2D
14 3D Finite Element Analyses – Interaction between Abutment and Adjacent Earth Embankment of a Modern Railway Bridge
18 Recent activities
20 Upcoming events

Colophon

The Plaxis Bulletin is a publication of Plaxis bv and is distributed worldwide among Plaxis subscribers.

Editorial board:
Ronald Brinkgreve
Erwin Beemink
Arny Lengkeek

Design:
Jori van den Munckhof

Any correspondence regarding the Plaxis Bulletin can be sent by e-mail to:
bulletin@plaxis.nl
or by regular mail to:
Plaxis Bulletin
c/o Annelies Vogelezang
PO Box 572
2600 AN Delft
The Netherlands

For information about PLAXIS software contact your local agent or Plaxis main office:
Plaxis bv
PO. Box 572
2600 AN Delft
The Netherlands
info@plaxis.nl
www.plaxis.nl
Tel: +31 (0)15 251 7720
Fax: +31 (0)15 237 3107
We are very pleased to publish this 20 years edition of the Plaxis Bulletin. This year we are celebrating the 20th anniversary of Plaxis bv. Surrounding this celebration we have several activities, such as the Master Thesis Competition, celebrations at the 20th Edition of the European Plaxis Users Meeting, and the upcoming release of the new PLAXIS 2D version that will be restyled with a new user interface.

The new developments column focuses on the new PLAXIS 2D version that will be released. Since the release of the general PLAXIS 3D package in 2010, users have been asking for a similar command-driven 2D user interface with model explorer and integrated calculation mode.

The first user’s article involves a 3D-interaction of Structure and Subsoil for a New Fly Ash Silo as part of the new power plant at the E.ON site at the Maassvlakte Rotterdam, Netherlands. The advantages of this interaction analysis in comparison to the general accepted design approaches with a conventional elastic support model are discussed.

The second paper discusses the principle and validation of the embedded pile row feature in PLAXIS 2D, which has been performed in a MSc thesis study (TU Delft). The article takes a closer look at the recent application of the feature by Witteveen+Bos in the design of a quay wall for the General Cargo terminal of the New Baku International Sea Trade Port.

The third user’s contribution shows a 3D FE analyses of the interaction between abutment and adjacent earth embankment of a modern railway bridge. The high requirements for the settlement prognosis for the interaction system “abutment – adjacent embankment” necessitated a 3D finite element modeling of the abutment foundation, the embankment and the surrounding subsoil. The following FE analyses were performed: Class A prediction with 3DFoundation (PLAXIS 3D was not yet available) in the framework of the observation method, subsequent analysis with PLAXIS 3D 2012 to allow for a more realistic model geometry.

In the PLAXIS Expert Services update we take a look at an In-house training that Plaxis developed and provided. The course focused on the use of PLAXIS 2D and PLAXIS 3D for reservoir geomechanics with special emphasis on surface subsidence and wellbore stability assessment during reservoir depletion.

In the recent activities section you can read all about 20 years of Plaxis bv and the related Master Thesis Competition. The nominees of this competition are announced, and the final winners will be announced at the European Plaxis Users Meeting. Furthermore there is news about the latest product updates, and we discuss the recent activities in North America and Asia Pacific.

As always you can also find all our upcoming events on the back.

We trust to have again compiled an interesting Plaxis Bulletin for you, and look forward to celebrating this 20th year of Plaxis bv with you.

The Editors
New developments

Ronald Brinkgreve, Plaxis bv

Since the release of the general PLAXIS 3D package in 2010, users have been asking for a similar command-driven 2D user-interface with model explorer and integrated calculation mode. It is a pleasure to announce that this version is now about to be released.

With the new 2D user-interface, PLAXIS complies with the highest software standards, and it has become even more user-friendly than before. In addition to a redesign of all existing features as available in the 2012 version, it contains tools that are already ‘common’ in the 3D version.

Just to name a few:
- Using ‘boreholes’ (with CPT import) to define soil layers.
- Multi-select: adding/changing properties of multiple objects
- Advanced tunnel designer, including sub-sections and loads on curved lines
- New Phases window with an overview of all calculation settings
- Fast definition of construction stages (all part of Input) and fast stage regeneration after geometric changes
- Defining multiple independent dynamic loads

For advanced users, the new version includes a scripting interface, which enables labour-intensive tasks to be automated, to run PLAXIS remotely, or to interact with PLAXIS in external applications. In this way, PLAXIS can be used, for example, with probabilistic or inverse analysis tools. The new manual contains some elaborated examples of this new scripting interface feature.

The release of the new 2D package coincides more-or-less with the celebration of the 20-year anniversary of the Plaxis company. This memorable event is celebrated during the 20th edition of the European Plaxis Users Meeting in Karlsruhe, Germany. Participants will have the chance to ‘play’ with the new version and experience its new features. For more information see the Events agenda at the Plaxis website.

VIP users will be able to update their 2D package towards the new version soon after the users meeting. If you are not yet a VIP user, please contact our sales department to become one. We are confident that the new 2D version will suit your professional geotechnical modelling requirements. Nevertheless, we are looking forward to receive your comments and suggestions on this new milestone in the succeeding PLAXIS story.
Plaxis trained several researchers and engineers on the use of PLAXIS 2D and 3D for the typical analyses in reservoir geomechanics such as reservoir depletion, fault stability during depletion and injection and wellbore stability. The intention was to offer to engineers and researchers a course with a balanced mix of lectures and practical hands-on exercises highlighting the main PLAXIS modeling capabilities in relation to their most relevant practical applications.

The detailed training objectives were:
- Get familiar with the both PLAXIS 2D and 3D software modeling workflow
- Understand the basis of the FE methods
- Understand the modeling basics of soil constitutive behavior and more particularly the use of the tension cut off and shear failure criteria (Mohr-Coulomb) for cracking and sand production assessment
- Be able to create a reservoir scale model
- Investigate geomechanical effects of hydrocarbon production such as: Stress changes, Strains, Deformations, Subsidence

Proposed course schedule
The first day of this course was organized with fundamental lectures on PLAXIS 2D modeling workflow and features and on the use of the software in the most relevant aspects involved in FEM analysis of reservoir depletion. The second day was more oriented towards relevant practical application with specific hands-on exercises along with an introduction on PLAXIS 3D.

First day:
- Introduction to FEM in Geo-engineering
- Understanding PLAXIS 2D modeling workflow:
  - Input program, Calculation facilities, Output post-processing
  - Overview of Soil Material Models in PLAXIS
  - Initial Stresses and Initial Pore Pressure Definition in PLAXIS
  - Introduction Hands-On Exercise

Second day:
- Practical application I: Reservoir Depletion Analysis in PLAXIS 2D
- Non-linear computations in PLAXIS:
  - Understanding Convergence Parameters
  - Introduction to PLAXIS 3D
- Practical application II: Inclined Wellbore stability Analysis in PLAXIS 3D with simplified consideration of filtercake permeability

Conclusions
Upon request Plaxis has provided high-level technical assistance in setting up a two-day fit-for-purpose training course which has been customized to Statoil’s specific requirements. PLAXIS Expert Services has boosted Statoil engineers and researchers analysis skills in the field of reservoir modeling.

“A useful introduction to the whole group so everyone can build various types of models quickly for a wide range of applications” - Ole Kristian Søreide, Statoil

The Company
Statoil is an international energy company with operations in 34 countries. Building on 40 years of experience from oil and gas production on the Norwegian continental shelf, Statoil is committed to accommodating the world’s energy needs in a responsible manner, applying technology and creating innovative business solutions. Statoil is headquartered in Stavanger, Norway with approx. 23,000 employees worldwide.

Pore pressure definition in reservoir and overburden

3d model of an inclined borehole
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 (Ø 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.
calculated according to the following relation (1):

\[ s_d = s_{1,d} + s_{2,d} \]  

in which \( s_{1,d} \) 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 \( s_1 \)

The settlement \( s_1 \) 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 of 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 \( \alpha \) was varied between 0 and 1. A uniform settlement of the concrete base slab was obtained with \( \alpha = 0.6 \), see figure 4.

### Table 1: Global soil profile

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

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 \( s_J \) of the pile group was calculated according to the following relation (1):

\[ s_J = s_{1,J} + s_{2,J} \]  

in which \( s_{1,J} \) is the settlement of the pile head, consisting of the pile tip deformation \( (s_{b,J}) \) including the elastic pile deformation \( (s_{el,J}) \) and \( s_{2,J} \) is the settlement caused by compression of (cohesive) layers below pile tip level.

### 2.2 Settlement \( s_1 \)

The settlement \( s_1 \) 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 of 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 \( \alpha \) 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 (HSmall). The soil stiffness parameters of the HSmall 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
The bending moments (M) in the concrete base slab are calculated from the settlement graphs according to the following relation:

\[ M = -EI \frac{d\phi}{dx} \]  

(2)

where EI is the bending stiffness of the concrete base slab and \( \frac{d\phi}{dx} \) 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 concrete base slab is 3,600 kPa. The corresponding bending moment is calculated according to the following relation:

\[ M = \sigma W \]  

(3)

where \( \sigma \) is the normal stress and \( W \) is the section modulus. The corresponding maximum bending moment is 3,750 kNm.

### 3.2 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 sand 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.

<table>
<thead>
<tr>
<th>Stiffness parameters</th>
<th>2nd sand layer (Normal model)</th>
<th>2nd clay layer (OCC model)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_{\text{ref}} )</td>
<td>70,000 kPa</td>
<td>70,000 kPa</td>
</tr>
<tr>
<td>( E_{\text{sat}} )</td>
<td>210,000 kPa</td>
<td>70,000 kPa</td>
</tr>
<tr>
<td>( m )</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>( \mu )</td>
<td>1.0</td>
<td>0.062</td>
</tr>
<tr>
<td>small strain stiffness ( \gamma_0 )</td>
<td>0.0024</td>
<td>0.0024</td>
</tr>
<tr>
<td>( G_{\text{sat}} )</td>
<td>265,000 kPa</td>
<td>265,000 kPa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Strength parameters</th>
<th>2nd sand layer</th>
<th>2nd clay layer</th>
</tr>
</thead>
<tbody>
<tr>
<td>( c' )</td>
<td>0.1 kPa</td>
<td>3.0 kPa</td>
</tr>
<tr>
<td>( c'' )</td>
<td>35.0 kPa</td>
<td>22.5 kPa</td>
</tr>
<tr>
<td>( \phi )</td>
<td>5.0 kPa</td>
<td>0.0 kPa</td>
</tr>
</tbody>
</table>

Table 2: Representative soil parameters
In the past, plates and node-to-node anchors have been used to model piles in PLAXIS 2D. Both methods have some advantages compared to each other, but both also have clear drawbacks:

- Plate elements have pile properties, converted to properties per unit width in out-of-plane direction. Interface elements are used to manipulate the pile-soil interaction. However, the use of interface elements separates the soil mesh, which results in a minor interaction between both sides of the pile. Using a plate element is therefore limited to low out-of-plane spacing \( L_{\text{spacing}} \) compared to the pile diameter \( D_{\text{eq}} \) (e.g. \( L_{\text{spacing}} / D_{\text{eq}} < 2 \) to 3).

- Using a node-to-node anchor, the soil mesh is continuous and there is no interaction with the soil. Soil can ‘flow’ independent from the pile, however, in reality there is always some sort of interaction. Moreover, a node-to-node anchor has no properties in lateral direction, which limits the use of this method to fully axial loaded piles only. Some sort of pile foot modelling is needed in order to sustain axial forces in the pile, since a single node at the foot is generally insufficient and leads to mesh dependent results.

The embedded pile row combines the advantages of the plate and node-to-node anchor. It has pile properties similar to the plate element and a continuous mesh similar to a node-to-node anchor. This is done by separating the pile and the soil. The pile, represented by a Mindlin beam element, is not ‘in’ the 2D mesh, but superimposed ‘on’ the mesh. A special out-of-plane interface is developed to connect the beam with the soil nodes and represents the pile-soil interaction.

The modelling of piles in a 2D finite element model brings limitations because pile-soil interaction is a strongly 3D phenomenon. Pile-soil interaction is difficult to model and traditional methods in which pile rows are modelled either as plates or as node-to-node anchors have clear drawbacks. The embedded pile row has been developed to model a row of piles in the out-of-plane direction, which is available in PLAXIS 2D 2012. It is supposed to result in a more realistic pile-soil interaction behaviour compared to other methods. This article discusses the principle and validation of the feature, which has been performed in a MSc thesis study (TU Delft). Recently, the embedded pile row feature has been applied by Witteveen+Bos in the design of a quay wall for the General Cargo terminal of the New Baku International Sea Trade Port.

**Validation and Application of the Embedded Pile Row-Feature in PLAXIS 2D**

Main author: Jasper Sluis MSc, Witteveen+Bos, The Netherlands
Co-author: Floris Besseling MSc, Paul Stuurwold MSc & Arny Lengkeek MSc, Witteveen+Bos, The Netherlands

The modelling of piles in a 2D finite element model brings limitations because pile-soil interaction is a strongly 3D phenomenon. Pile-soil interaction is difficult to model and traditional methods in which pile rows are modelled either as plates or as node-to-node anchors have clear drawbacks. The embedded pile row has been developed to model a row of piles in the out-of-plane direction, which is available in PLAXIS 2D 2012. It is supposed to result in a more realistic pile-soil interaction behaviour compared to other methods. This article discusses the principle and validation of the feature, which has been performed in a MSc thesis study (TU Delft). Recently, the embedded pile row feature has been applied by Witteveen+Bos in the design of a quay wall for the General Cargo terminal of the New Baku International Sea Trade Port.

**Fig. 1: Principle of the embedded pile row (Sluis, 2012)**
Along the pile there is a line-to-line interface represented by springs with numerical stiffnesses in axial and lateral direction ($R_S$ and $R_N$). The springs in axial direction are limited by a plastic slider, representing the shaft capacity of the pile. There is no limitation of the spring forces in lateral direction. At the base there is a point-to-point interface, represented by a spring and plastic slider, which take care of the end bearing of the pile. The interface is visualised in Figure 1 (sliders are not shown in this figure).

The pile capacity is an input parameter, similar to the embedded pile in PLAXIS 3D, for which a shaft capacity $T_{S,\text{max}}$ and base capacity $F_{B,\text{max}}$ should be defined. The pile capacity is defined in the material set, together with the stiffness, weight and dimensions of the pile. These pile properties are entered per pile, which are converted to properties per unit width in out-of-plane direction during the calculation. This is done by using the out-of-plane centre-to-centre distance of the pile row $L_{\text{spacing}}$, which is also an input parameter.

The deformation behaviour between the pile and the soil is an interaction between pile stiffness, soil stiffness and interface stiffness, as shown in Figure 1. The interface stiffnesses $R_S$, $R_N$ and $K_F$ are related to the shear modulus of the soil $G_{\text{soil}}$ and the out-of-plane centre-to-centre distance $L_{\text{spacing}}$ and radius of the pile $r$:

$$R_S = ISF_{RS} \frac{G_{\text{soil}}}{L_{\text{spacing}}}$$

$$R_N = ISF_{RN} \frac{G_{\text{soil}}}{L_{\text{spacing}}}$$

$$K_F = ISF_{KF} \frac{G_{\text{soil}}}{r}$$

$ISF_{RS}$, $ISF_{RN}$ and $ISF_{KF}$ are the interface stiffness factors, a dimensionless factor to manipulate the deformation behaviour. Default values have been derived by Plaxis as part of the validation (Sluis, 2012) and are related to the out-of-plane spacing and pile diameter:

$$ISF_{RS} = 2.5 \left( \frac{L_{\text{spacing}}}{D_{\text{pile}}} \right)^{0.75}$$

$$ISF_{RN} = ISF_{RS} = 2.5 \left( \frac{L_{\text{spacing}}}{D_{\text{pile}}} \right)^{0.75}$$

$$ISF_{KF} = 25 \left( \frac{L_{\text{spacing}}}{D_{\text{pile}}} \right)^{0.75}$$

The interface stiffness factors can be overruled by the user, because the formulas are derived for only a limited number of cases. Moreover, by overruling the default values the user is able to fit the load-displacement curve of the embedded pile row with for example measurement data from a pile load test.

**Validation**

The embedded pile row has been tested and validated as part of a MSc thesis study (TU Delft). The behaviour of the embedded pile row, soil displacement and pile displacement, was evaluated for four loading directions:

- Axial compression loading
- Axial tension loading
- Lateral loading by external force
For lateral loading, interface stiffness factor ISF$_{\text{IN}}$ was derived by comparing the 2D embedded pile row with the 3D embedded pile, using a similar model as shown in Figure 2. When using ISF$_{\text{IN}} = \text{ISF}_{\text{IN}}$, a satisfying fit is obtained for relatively small pile spacing ($L_{\text{cons}}/D_{\text{st}} < 4$). For larger pile spacing, the results deviate from 3D calculations, which is probably caused by the absence of a plastic slider in lateral direction.

For example, in 3D a large horizontal force on top of the pile results in soil failure near the pile top, which results in the pile being pulled through the soil, giving large deformations. In 2D, the force should be divided by the pile spacing. For large pile spacing this results in a relatively low line load and no soil failure. To fit the displacement at the pile head, a relatively low ISF should be applied. However, this also results in larger displacements at the pile base. This is illustrated in Figure 5.

In this figure, the 3D embedded pile is also compared with a 3D volume pile, part of a recent study by Witteveen+Bos for the design of a quay wall, to better understand the behaviour of the embedded pile row in lateral loading conditions. In this study the embedded pile row was validated by applying the project-specific pile properties and soil types. A better fit of the pile displacement and bending moment as shown in figure 5 was not possible by varying the interface stiffness.

Because installation effects are not taken into account in PLAXIS, the embedded pile row is most suitable for bored piles. Curve 3 of Figure 4 was used for fitting. For lateral loading, interface stiffness factor ISF$_{\text{IN}}$ was derived by comparing the 2D embedded pile row with the 3D embedded pile, using a similar model as shown in Figure 2. When using ISF$_{\text{IN}} = \text{ISF}_{\text{IN}}$, a satisfying fit is obtained for relatively small pile spacing ($L_{\text{cons}}/D_{\text{st}} < 4$). For larger pile spacing, the results deviate from 3D calculations, which is probably caused by the absence of a plastic slider in lateral direction.

For example, in 3D a large horizontal force on top of the pile results in soil failure near the pile top, which results in the pile being pulled through the soil, giving large deformations. In 2D, the force should be divided by the pile spacing. For large pile spacing this results in a relatively low line load and no soil failure. To fit the displacement at the pile head, a relatively low ISF should be applied. However, this also results in larger displacements at the pile base. This is illustrated in Figure 5.

In this figure, the 3D embedded pile is also compared with a 3D volume pile, part of a recent study by Witteveen+Bos for the design of a quay wall, to better understand the behaviour of the embedded pile row in lateral loading conditions. In this study the embedded pile row was validated by applying the project-specific pile properties and soil types. A better fit of the pile displacement and bending moment as shown in figure 5 was not possible by varying the interface stiffness.

For the design of the quay wall, the default values were applied and a workaround has been found and applied to overcome the current limitation of lateral loading for relatively large pile spacing ($L_{\text{cons}}/D_{\text{st}} > 4$).

Application

The ministry of Transport of Azerbaijan intends to make a new port 65 km outside of Baku. A map of the area including the location of the new port are presented in figure 6.

Witteveen+Bos is assisting a local contractor Evrascon to make an alternative design for the quay wall. The construction consists of an anchored diaphragm wall connected to a deck on bored concrete piles. The anchor structure consists of a bored pile wall.

A schematic cross section is presented in figure 7.

The subsoil consists of 2.5m general fill, 2m earth fill, 4-8m very silty sand layer, 40m stiff to hard clay. The first three row of piles are Ø1.5m bored piles with a spacing of 6m and the last row Ø1.2m bored piles with a spacing of 3m.

In PLAXIS 2D it is difficult to calculate accurate forces in the piles, without using the embedded pile feature. This is related to the pile spacing of 4D which exceeds the range where modelling the pile row as a plate is realistic. On the other hand modelling of the piles as node-to-node anchors will also give unrealistic interaction behaviour and result in too high forces in the frontwall. The importance of soil-structure interaction and the effect of interaction stiffness on the distribution of structural forces has been the reason why it was decided to use the embedded pile row feature.

The bored piles below the deck are modelled with the embedded pile feature of PLAXIS 2D. The properties for ultimate shaft friction are determined based on ‘floating’ pile calculation according to Pouls (Pouls, 2007). Base resistance of the piles is calculated according to the API/ISO (ISO 19902). The bored pile anchor wall is modelled in PLAXIS 2D as a plate because the pile spacing is approximately equal to the pile diameter.

Outputs of the model are the distribution of bending moments, axial forces, shear forces and displacements along the pile. In figure 8, an overview of the model is presented. In figure 9, output figures are given for bending moments and axial forces.

In the embedded pile feature, a lateral slider is currently not included. Therefore, the PLAXIS 2D embedded pile row cannot be used as a tool for lateral capacity design of the piles. Within this project the capacity of the soil in lateral direction is checked separately. Ultimate lateral capacities are calculated based on p-y expressions provided by API / ISO 19902 standards. In this calculation soil layering and pile group effects are included as well. First the lateral mobilisation of the soil by the piles is calculated from the embedded pile row shear force distribution, thereafter it is assessed whether the lateral capacity of the soil is exceeded. This assessment is done over the total length of the pile.

The poorest interaction performance of the
extended the range of practical applications of the embedded pile row feature in PLAXIS 2D.

Conclusions and Recommendations

The embedded pile row feature in PLAXIS 2D brings new possibilities for the modeling of line elements with soil-structure interaction, compared to ‘old’ methods with plates and node-to-node anchors. It combines the advantages of these methods, having pile properties and a continuous mesh. The special developed interface between the pile and the soil represents the soil-structure interaction.

As part of a MSc thesis study the embedded pile row has been tested and validated in various situations and loading conditions. The soil displacement is found to be independent of the interface stiffness and is an average of the out-of-plane soil displacement. Formulas have been derived for the interface stiffness to fit the pile displacement. For axial loading the pile displacement was fitted with the load-displacement curves from the Dutch annex of Eurocode 7. For lateral loading a comparison was made with PLAXIS 3D embedded pile. The formulas provide default values for the interface stiffness factor (ISF) as a function of the out-of-plane pile spacing. The values can be overruled by PLAXIS users to match the pile displacement for their specific case.

Recently, the embedded pile row feature has been applied by Witteveen+Bos in the design of a quay wall for the General Cargo terminal of the New Baku International Sea Trade Port. Limitations were found with respect to lateral loading of the pile, due to the absence of a plastic slider in lateral direction, which limits the soil mobilisation. A workaround was found and successfully applied to overcome this limitation. The implementation of a plastic slider in lateral direction is however strongly recommended for future updates of PLAXIS 2D, as it will significantly extend the range of practical applications of the embedded pile row feature in PLAXIS 2D.

References

- Poulos, H.G., A practical design approach for piles with negative skin friction, 2007
- ISO 19902:2006 Petroleum and Natural Gas Industries - Fixed Steel Offshore Structures

Fig. 6: Location of the new Baku port development

Fig. 7: Sketch cross section quay wall structure

Fig. 8: Overview model

Fig. 9: Output pile bending moments and axial forces

Fig. 10: Soil mobilisation (note the too high mobilisation of shallow layers)
The foundation pile design of the eastern abutment was influenced by a change of the construction sequence; namely the abutment had to be built prior to the embankment. The correct prediction of the development of consequential settlements due to the construction of the adjacent embankment was crucial for the design. Figure 1 depicts the construction design of the abutment including the abutment (wing) walls and the adjacent earth embankment, which is supported by hydraulically (cement) stabilised fill. The subsoil consists of the following layers:

Slope loam (approx. 2 m), underlain by completely to highly weathered (decomposed) rock (approx. 3 m) underlain by decreasingly weathered rock with depth.

In the area of the abutment the slope loam was exchanged for hydraulically stabilised material. The abutment and the adjacent embankment are designed based on the principles of the observation method with the following main foci:

• Design of a deformation-resistant bored pile foundation, consisting of 14 piles 10 m in length and 1.5 m in diameter
• Deformation prognosis with respect to the settlements due to the construction of the embankment, for the interaction system “abutment – adjacent embankment”.
• Settlement monitoring of the abutment and the embankment during and after construction of the embankment

The allowable settlement for the abutment was set to 5 cm. Settlements up to this value can be compensated later by the abutment structure within the meaning of the observation method.

Monitoring
Figure 2 shows the layout of measuring points at the abutment, below the footprint of the embankment and on top of the embankment. The settlements at the foundation of the abutment and along the track axis on top of the embankment were measured by means of geodetic leveling. Additionally, one settlement marker was installed at each of the abutment walls. The settlement measurements below the footprint of the embankment were performed by means of two multi-extensometers (E1 and E2, cf. Figure 2) along the track axis, reaching depths down to 20 m. This instrumentation rendered possible the settlement measurement of three different depth intervals (0 – 5 m, 0 – 10 m and 0 – 20 m).

The relevant settlement measurements together with the calculated values are depicted in Figure 7 and 8 and will be discussed below. The two diagrams also show the relevant construction stages, for each of which measurements were performed. These construction stages were also modeled in the 3D finite element analyses.

3D Finite Element Analyses
The high requirements for the settlement prognosis for the interaction system “abutment – adjacent embankment” necessitated a 3D finite element modeling of the abutment foundation, the embankment and the surrounding subsoil.

The following FE analyses were performed:
Class A prediction with 3DFoundation (PLAXIS 3D was not yet available) in the framework of the observation method

Subsequent analysis with PLAXIS 3D 2012 to allow for a more realistic model geometry

The symmetry of the abutment geometry is incorporated in both FE models in terms of a vertical symmetry plain along the track axis (cf. Figure 3 and 4).

Class A prediction with 3DFoundation

Due to the limitations of 3DFoundation with respect to the generation of complex 3D geometries, the following simplifications were made:

- horizontal surface and layering of the subsoil
- modeling of the foundation slab with 2D plate elements
- modeling of the abutment by means of vertical surface loads
- modeling of the superstructure of the bridge by means of vertical point loads

Using the symmetry of the system a FE mesh consisting of 6690 15-node 3D wedge continuum elements was generated.

Subsequent analyses with PLAXIS 3D 2012

The subsequent analyses with PLAXIS 3D 2012 aimed at

- modeling of the complex geometry of the abutment including the abutment walls and especially of the adjacent embankment supported by a hydraulically stabilised wedge as well as the inclined ground surface
- and comparing the results with those obtained from the simplified 3DFoundation model.

In analogy to the 3DFoundation model, the superstructure of the bridge was modeled by means of vertical point loads. The FE mesh of the

<table>
<thead>
<tr>
<th>HS Small Model</th>
<th>Slope loam</th>
<th>Decomposed rock</th>
<th>Rock</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculation Type</td>
<td>Drained</td>
<td>Drained</td>
<td>Drained</td>
</tr>
<tr>
<td>$\tau_{\text{me}}$ [kN/m²]</td>
<td>21,0</td>
<td>21,0</td>
<td>22,0</td>
</tr>
<tr>
<td>$\tau_{\text{mr}}$ [kN/m²]</td>
<td>21,0</td>
<td>21,0</td>
<td>22,0</td>
</tr>
<tr>
<td>$k$ [m/day]</td>
<td>8,64E-3</td>
<td>8,64E-3</td>
<td>8,64E-5</td>
</tr>
<tr>
<td>$E_{\text{me}}$ [kN/m⁴]</td>
<td>7,000,0</td>
<td>12,000</td>
<td>55,500</td>
</tr>
<tr>
<td>$E_{\text{mr}}$ [kN/m⁴]</td>
<td>7,000,0</td>
<td>12,000</td>
<td>54,000</td>
</tr>
<tr>
<td>power (m) [-]</td>
<td>0,90</td>
<td>0,60</td>
<td>0,40</td>
</tr>
<tr>
<td>$c_{\text{me}}$ [kN/m²]</td>
<td>12,5</td>
<td>20,0</td>
<td>50,0</td>
</tr>
<tr>
<td>$\varphi$ [°]</td>
<td>22,5</td>
<td>22,5</td>
<td>45,0</td>
</tr>
<tr>
<td>$\psi$ [°]</td>
<td>0,0</td>
<td>0,0</td>
<td>10,0</td>
</tr>
<tr>
<td>$E_{\text{me}}$ [kN/m⁴]</td>
<td>21,000</td>
<td>36,000</td>
<td>166,500</td>
</tr>
<tr>
<td>$G_{\text{mr}}$ [kN/m²]</td>
<td>40,000</td>
<td>70,000</td>
<td>2,080,000</td>
</tr>
<tr>
<td>$\tau_{\text{f}}$ [-]</td>
<td>2,0E-4</td>
<td>2,0E-4</td>
<td>4,0E-5</td>
</tr>
<tr>
<td>$v$ [-]</td>
<td>0,20</td>
<td>0,20</td>
<td>0,20</td>
</tr>
<tr>
<td>$\rho_{\text{mr}}^*$ [kN/m³]</td>
<td>100,0</td>
<td>100,0</td>
<td>100,0</td>
</tr>
<tr>
<td>Tensile Strength [kN/m²]</td>
<td>0,0</td>
<td>0,0</td>
<td>0,0</td>
</tr>
<tr>
<td>$R_{\text{ur}}$ [-]</td>
<td>1,0</td>
<td>1,0</td>
<td>1,0</td>
</tr>
</tbody>
</table>

Table 1: Material parameters of the HSsmall constitutive model of the slope loam
3D Finite Element Analyses – Interaction between Abutment and Adjacent Earth Embankment of a Modern Railway Bridge

Fig. 3: FE model – 3DFoundation

Fig. 4: FE model – PLAXIS 3D

Fig. 5: 3DFoundation analysis – Settlements of embankment base and abutment

Fig. 6: PLAXIS 3D analysis – Settlements of embankment base and abutment
model depicted in figure 4 consists of 108784 10-node 3D tetrahedral continuum elements, with a total of 150790 nodes. Constitutive models of soils and structural members:

In both FE models the following constitutive equations were used:

- linear elasticity for the abutment, modeled with continuum elements in PLAXIS 3D and with plate elements (foundation slab) in 3DFoundation, respectively and for the bored piles
- linear-elastic perfectly plastic model with Mohr-Coulomb failure criterion for the hydraulically stabilised material
- Hardening Soil Small Strain Model (HSsmall model) for the soils/rock

Stiffness values according to the concrete strength class C35/45 were assigned to the structural members. A cohesion of 15 kPa with tension cut off was assigned to the hydraulically stabilised material. The applied material parameters for the structural members and the hydraulically stabilised layer, were determined based on relevant codes and experience, respectively.

The HSsmall model was not only used for modeling the behaviour of the slope loam and weathered rock, but also for the only slightly weathered to fresh (solid) rock. Especially with respect to the two parameters governing the small strain behaviour, namely $G_s$ and $Y_s$, there is only limited experience for rocks. Table 1 summarises the sets of material parameters of the HSsmall constitutive model of the slope loam, weathered rock and rock.

The less pronounced pressure dependence of the stiffness of the rock is reflected in the low value of the exponent ($m = 0.4$) of the stiffness-pressure relation according to OHDE or JANBU, respectively. The stiffness for small strains of the solid rock, which lies in the range of the values of concretes, was determined based on data sets given by BENZ (2007) for solid rock and the relation between the “static” and “dynamic” stiffness according to ALPAN (1970) (cf. PLAXIS material models manual, Figure 6.7). The very high small strain stiffness of the rock means only the stiffness of the compact rock is relevant. Hence, that the influence of joints is irrelevant.

Calculation Phases

In both FE analyses the construction stages "installation of the pile foundation" and "construction of the abutment" were modeled (calculation type: plastic) after the initial phase (calculation type: gravity loading).

Performing the Class A prediction with 3DFoundation, the construction of the embankment was modeled in one step only, since a work schedule was not available. The calculation phase “embankment construction” was carried out as a consolidation analysis to obtain the development of settlements with time (Figures 7 and 8).

In the subsequent analysis with PLAXIS 3D the “construction of the embankment” is divided into three phases, according to the by then known work schedule. These calculation phases were carried out as plastic analyses.

Results

Figure 7 depicts the measured settlements due to the construction of the embankment together with the results of the Class A prediction and the subsequent analysis. The agreement between the measurements and the numerical results is good, with the results of the class A prediction representing a lower bound and the ones of the subsequent analysis an upper bound of the measurement results.

Figure 8 shows the development of the settlements measured in the observed soil layers by means of the multi-extensiometers together with the results of the two FE analyses. For the extensiometer E1 (Figure 8a and b), the corresponding numerical results exceed the measured values. Whereas the calculated values for the extensiometer E2 closer to the abutment lie in the same range as the measurements.

In the face of the principal lack of information with respect to the subsoil, the agreement between the measured and calculated development of settlements is satisfactory. The measured developments of settlements differ more significantly for the different depth intervals ($0 \sim 5$ m, $0 \sim 10$ m and $0 \sim 20$ m) than the calculated values, which yield more or less the same value for all intervals. The main reason for this is the rock layer was assumed to be homogeneous. With the rock layer divided into sublayers with increasing stiffness and strength characteristics with depth and related to the degree of weathering (decomposition) one can expect more realistic results. However, especially in case of the rock a reliable base of investigation data is missing.

Conclusions

The two different FE analyses yielded the following conclusions:

1. The HSsmall model is applicable not only for soils but also for (weathered) rock. The results show only little lateral spreading of the settlement troughs as well as a small range of influence of settlements with respect to depth. The settlements mainly occur in the upper part of the weathered/decomposed rock layer.

2. Complex model geometries can be generated and imported from CAD data with PLAXIS 3D. However, the expenditure of human labour should not be underestimated.

3. It is not always necessary to model geometries as realistic as possible. It is of much more importance, that the modeling engineer makes adequate simplifications where applicable.

Fig. 7: Measured and calculated development of settlements of the abutment, a) 3DFoundation, b) PLAXIS 3D

Fig. 8: Measured and calculated development of settlements (extensiometers E1 (a+b), extensiometers E2 (c+d)), a)+c) 3DFoundation, b)+d) PLAXIS 3D
Recent activities

Plaxis 20 years
September 2013 marked the 20th anniversary of Plaxis bv. This year will also be the 20th edition of the European Plaxis Users Meeting to be held 6 – 8 November in Karlsruhe. In order to celebrate this we planned some extra activities during the user meeting. Alongside the regular schedule, there will be time to look back at 20 years of Plaxis bv and an even longer history of PLAXIS software, winners from the 20 years Master Thesis Competition will be presenting, and there will be some extra time to socialise on Thursday evening.

Master Thesis Competition
Without students and universities our brand would not exist. The PLAXIS code itself is a spin off from a project at the TU Delft. For this reason we wanted to get the students actively involved, and developed the idea of a Master Thesis Poster Competition. Winners will get the opportunity to show off their work at the European Plaxis Users Meeting this year.

We invited young enthusiast engineers and researchers from all around the world to submit a poster of their recent graduation work involving numerical analysis of engineering problems with the PLAXIS finite element software or research which contributed on the development of PLAXIS tools and methods.

Over the last two months we have received many great and interesting posters for the Master Thesis Competition. The committee has now reviewed all the posters and the final nominees are: J. Sluis, Z. Orazalin, and C.M. Elescano.

These three nominees will be at the European Plaxis Users Meeting to present their work, where the final standing of the competition with the winner of each prize will be announced.

Product Updates
After the successful release of PLAXIS 3D 2013 including the new 3D PlaxFlow module earlier this year, the new PLAXIS 2D is planned to be released later in the year. The user interface of this new version of our classic software has been restyled to follow the flexible and easy to use workflow of the PLAXIS 3D program.

In this new graphical user interface the geometric modelling and staged construction will be integrated allowing for quick and easy switching between input (geometry) and calculation phases. The new version will also include other new and improved features such as a command line and the command runner, a borehole wizard, CPT import, remote scripting API, and more.

The release of 3D Dynamics in 2011 and the recent release of 3D PlaxFlow, brings the PLAXIS 2D and PLAXIS 3D suites closer together. Furthermore

Screenshots from the new 3D PlaxFlow
this upcoming release of the new and improved PLAXIS 2D program is another step towards aligning the two versions of our software. Keep an eye out on our website for more information about this new release in the coming months.

North America Update
May 2013 saw yet another advanced course organized by Plaxis, this time in Berkeley CA. We plan to continue to organize these advanced courses on a frequent basis (in addition to the standard course). We do our best to have our courses in different parts of the US and Canada, so keep an eye on our newsletters or events page for a course near you (or enjoy visiting a different part of the country!). Upcoming standard course is in Toronto, 5 - 8 November.

We have also been active in reaching out to and being in touch with the North American geotechnical community at conferences organized by key organizations serving the community. Several specialty events provided us the opportunity to highlight some new developments in our software, e.g. showing new 2D and 3D tunneling tools to RETC attendees, and seepage and rapid drawdown analysis in the new 3D PlaxFlow module to Dam Safety conference attendees. We hope to see you at an upcoming event.

Plaxis AsiaPac
Many courses, workshops, and seminars were organized around the region throughout the year.

In association with our Vietnam agent, CIC, organised the first official three day Introductory Standard Course on Computational Geotechnics from the 10th to 12th of July 2013 in Da Nang, Vietnam. The course was a great success. It was attended by 40 engineers and academicians coming from various parts of Viet Nam. We are looking forward to be back in Vietnam in 2014.

In Japan a Seminar & Workshop was held from the 17th to 18th July 2013, Tokyo. A seminar was delivered by one of key staff member. During the event new features and developments were presented. With the devastation of the Tohuku earthquake in 2011, practitioners and academicians are relying much more on the use of PLAXIS 2D and PLAXIS 3D for the analysis of soil-structure interaction problems due to earthquake loads. The seminar was well received and attended by expert users. This event was organised in association with our local counterpart Jip-Techno Sciences Tokyo office.

In September 2013 Plaxis returned to India to deliver the second Advanced Course on Computational Geotechnics, which was held from the 18th to the 20th of September 2013 in Chennai. The course was co-organised by RamCadds Pvt Ltd, IIT- Madras and Plaxis AsiaPac at ICSR Building Indian Institute of Technology Madras. The course was well received as it was attended by 40 participants. The next course will be held in New Delhi in 2014. Plaxis will also be back in India next year for the third Plaxis Users Meeting and this will be held in Kolkata.

We had the first inaugural training on the use of PLAXIS software in Sri Lanka on 24th & 25th September 2013. The event was organised by Sri Lankan Geotechnical Society in association with Plaxis AsiaPac. The training was officiated by Prof H. S. Thilakasiri from the University of Moratuwa and currently the President of the Sri Lankan Geotechnical Society. There are altogether 25 engineers attended the event that was held at Sri Lanka Institute for Information Technology, Malabe.
### Upcoming Events 2013

<table>
<thead>
<tr>
<th>Date</th>
<th>Event</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>October 29, 2013</td>
<td>Seminar on Numerical Methods in Geotechnical Engineering</td>
<td>Shanghai, China</td>
</tr>
<tr>
<td>October 31, 2013</td>
<td>Chinese Plaxis Users Meeting / 全国用户应用技术交流大会暨</td>
<td>Shanghai, China</td>
</tr>
<tr>
<td>November 1, 2013</td>
<td>Türkiye Plaxis Kullanıcıları Toplantısı</td>
<td>İstanbul, Turkey</td>
</tr>
<tr>
<td>November 5 - 8, 2013</td>
<td>Short Course on Computational Geotechnics &amp; 3D Modelling</td>
<td>Toronto, Canada</td>
</tr>
<tr>
<td>November 5, 2013</td>
<td>Workshop on the use of PLAXIS 2D for Modelling Groundwater and Slope Stability</td>
<td>Oslo, Norway</td>
</tr>
<tr>
<td>November 5, 2013</td>
<td>Geotechniekdag 2013</td>
<td>Breda, The Netherlands</td>
</tr>
<tr>
<td>November 6 - 8, 2013</td>
<td>European Plaxis Users Meeting</td>
<td>Karlsruhe, Germany</td>
</tr>
<tr>
<td>November 6, 2013</td>
<td>Workshop on Advanced Modelling in PLAXIS</td>
<td>Delft, The Netherlands</td>
</tr>
<tr>
<td>November 19 - 22, 2013</td>
<td>Pratique Eclairée des Éléments Finis en Geotechnique</td>
<td>Paris, France</td>
</tr>
<tr>
<td>November 19 - 22, 2013</td>
<td>Curso de Geotécnia Computacional</td>
<td>Bogotá, Colombia</td>
</tr>
<tr>
<td>November 26, 2013</td>
<td>Waterbouwdag 2013</td>
<td>Rotterdam, The Netherlands</td>
</tr>
<tr>
<td>November 27 - 28, 2013</td>
<td>Training on the use of PLAXIS 3D</td>
<td>Singapore</td>
</tr>
<tr>
<td>December 2 - 5, 2013</td>
<td>Southeastern Transportation Geotechnical Engineering Conference</td>
<td>Baton Rouge LA, U.S.A.</td>
</tr>
<tr>
<td>December 4, 2013</td>
<td>Workshop on Modelling Dynamics in PLAXIS</td>
<td>Delft, The Netherlands</td>
</tr>
<tr>
<td>December 4 - 5, 2013</td>
<td>Training on the use of PLAXIS 2D</td>
<td>Kuala Lumpur, Malaysia</td>
</tr>
</tbody>
</table>

### Upcoming Events 2014

<table>
<thead>
<tr>
<th>Date</th>
<th>Event</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>January 8, 2014</td>
<td>Workshop on Modelling Dynamics in PLAXIS</td>
<td>Singapore</td>
</tr>
<tr>
<td>January 13 - 16, 2014</td>
<td>Standard Course on Computational Geotechnics</td>
<td>Schiphol, The Netherlands</td>
</tr>
<tr>
<td>February 10 - 13, 2014</td>
<td>Finite Elemente in der Geotechnik &amp; 3D Analysen - Theorie und Praxis</td>
<td>Ostfildern, Germany</td>
</tr>
<tr>
<td>March 10 - 13, 2014</td>
<td>Advanced Course on Computational Geotechnics</td>
<td>Schiphol, The Netherlands</td>
</tr>
<tr>
<td>March 31 - April 2, 2014</td>
<td>Special Course on Computational Geotechnics</td>
<td>Istanbul, Turkey</td>
</tr>
<tr>
<td>May 5 - 8, 2014</td>
<td>Offshore Technology Conference 2014</td>
<td>Houston TX, U.S.A.</td>
</tr>
<tr>
<td>June 18 - 20, 2014</td>
<td>Numerical Methods in Geotechnical Engineering</td>
<td>Delft, The Netherlands</td>
</tr>
</tbody>
</table>