1 PROBLEM DEFINITION – EXAMPLE 1

1.1 Geometry, basic assumptions and computational steps

The geometry of the problem follows from Figure 1. The domain analysed has been chosen as follows: width = 150 m, depth = 100 m. The mesh consists of approximately 1800 6-noded elements, which is refined in areas where high stress gradients can be expected. The mesh was deliberately chosen to be relatively fine in order to minimize the discretisation error (Figure 2). The finite element code Plaxis is used for all analyses presented in this paper.

The following assumptions have been postulated:
- plane strain
- influence of diaphragm wall construction is neglected, i.e. initial stresses without wall, then wall "wished-in-place" (weight of wall, $\gamma_b = 24$ kN/m$^3$)
- diaphragm wall modelling: beam elements ($E_b = 306$ kPa, $\nu_b = 0.15$, $d = 0.8$ m)
- interface elements between wall and soil
- horizontal hydraulic cut off at -30.00 m is not considered as structural support, the same mechanical properties as for the surrounding soil are assumed
- hydrostatic water pressures corresponding to water levels inside and outside excavation (ground-water lowering is performed in steps in advance to the respective excavation step)
- anchors are modelled as rods, the grouted body as membrane element (geotextile element in Plaxis terminology) which guarantee a continuous load transfer to the soil

Fig 1. Geometry and excavation stages

ABSTRACT: The influence of different modelling assumptions on the results of numerical analyses of a deep excavation problem is discussed. Based on a reference solution a comprehensive parametric study is performed, identifying modelling assumptions which may have a significant influence on the calculated displacement behaviour and the bending moments in the wall. The parameters investigated include wall friction, domain chosen for the analysis, constitutive models and modelling of the grout body. In a second example the influence of the design approaches defined in Eurocode7 for ULS-design are investigated in connection with numerical methods. It can be concluded from this study that care must be taken when setting up a numerical model because the sum of various assumptions, not considered being of large importance when looked at it individually, may significantly influence the outcome of the numerical calculation.
The following computational steps have been performed:

- **stage 0:** initial stress state (given by $\sigma_v = \gamma z$, $\sigma_h = K_0 \gamma z$, $K_0 = 0.43$)
- **stage 1:** activation of diaphragm wall and groundwater lowering to -4.90 m
- **stage 2:** excavation step 1 (to level -4.80 m)
- **stage 3:** activation of anchor 1 at level -4.30 m and prestressing
- **stage 4:** groundwater lowering to -9.40 m
- **stage 5:** excavation step 2 (to level -9.30 m)
- **stage 6:** activation of anchor 2 at level -8.80 m and prestressing
- **stage 7:** groundwater lowering to -14.50 m
- **stage 8:** excavation step 3 (to level -14.35 m)
- **stage 9:** activation of anchor 3 at level -13.85 m and prestressing
- **stage 10:** groundwater lowering to -17.90 m
- **stage 11:** excavation step 4 (to level -16.80 m)

Distance and prestressing loads for anchors follow from Figure 1.

### 1.2 Material Parameters for Hardening Soil Model

The so-called Plaxis Hardening Soil model (Brinkgreve, 2002) has been used as reference model. As the example is related to an actual project in Berlin, simplified however for the exercise discussed here, the basic set of material parameters used to obtain the reference solution is based on data available in the literature and also on published experimental data from triaxial and one-dimensional compression test for Berlin sand. Although the Hardening Soil model takes into account the stress dependency of stiffness for primary loading as well as unloading/reloading stress paths, three layers (see Table 1) are introduced in order to increase this effect and to take into account the high stiffness at low strains, which will be prevailing in most of the deeper layers of the domain analysed, at least in a very approximate way. $R_{\text{inter}}$ in Table 1 determines the reduction of strength parameters $\varphi$ and $c$ in the interface elements as compared to the surrounding soil ($\tan \varphi_{\text{inter}} = R_{\text{inter}} \tan \varphi$, $c_{\text{inter}} = R_{\text{inter}} c$). The stiffness of the interface is reduced as well. A value of 1 kPa is introduced for the cohesion which improves numerical stability, this is however not strictly required.

### Table 1. Material parameters for HS-Model - reference solution

<table>
<thead>
<tr>
<th>Depth of layer (m)</th>
<th>0 - 20</th>
<th>20 - 40</th>
<th>&gt; 40</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{50}^{\text{ref}}$ (kPa)</td>
<td>45.000</td>
<td>75.000</td>
<td>105.000</td>
</tr>
<tr>
<td>$E_{\text{ur}}^{\text{ref}}$ (kPa)</td>
<td>180.000</td>
<td>300.000</td>
<td>315.000</td>
</tr>
<tr>
<td>$E_{\text{om}}^{\text{ref}}$ (kPa)</td>
<td>45.000</td>
<td>75.000</td>
<td>105.000</td>
</tr>
<tr>
<td>$\varphi$</td>
<td>35</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>$\psi$</td>
<td>5</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>$c$ (kPa)</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>$v_{\text{ur}}$</td>
<td>0.2</td>
<td>0.2</td>
<td>0.2</td>
</tr>
<tr>
<td>$p_{\text{ref}}$ (kPa)</td>
<td>100</td>
<td>100</td>
<td>100</td>
</tr>
<tr>
<td>$m$</td>
<td>0.55</td>
<td>0.55</td>
<td>0.55</td>
</tr>
<tr>
<td>$R_f$</td>
<td>0.9</td>
<td>0.9</td>
<td>0.9</td>
</tr>
<tr>
<td>$R_{\text{inter}}$</td>
<td>0.8</td>
<td>0.8</td>
<td>-</td>
</tr>
</tbody>
</table>

### 1.3 Material Parameters for Structural Elements

- **Diaphragm wall**
  - $E_A = 2.4\times10^7$ kN/m
  - $E_I = 1.28\times10^6$ kNm²/m
  - $\nu = 0.15$
  - $w = 7.5$ kN/m/m
- **Anchor row 1**
  - $E_A = 2.87\times10^5$ kN
- **Anchor rows 2 and 3**
  - $E_A = 3.22\times10^5$ kN
- **Membrane elements for modelling grout body (anchor row 1)**
  - $E_A = 4.92\times10^5$ kN/m
- **Membrane elements for modelling grout body (anchor rows 2 and 3)**
  - $E_A = 8.38\times10^5$ kN/m
2 RESULTS FOR REFERENCE SOLUTION

In the following the most relevant results obtained for the reference solution are presented. Unlike otherwise stated the last construction stage is considered. In addition a few results for the first excavation step (no anchors installed) are shown. In Figure 3 the deformed mesh is shown and in Figure 6 the surface settlements are plotted for the first and final excavation stage. Settlements increase from approximately 5 mm for the first stage to over 15 mm for the final stage, which can be considered to be a very plausible result. Figure 4 depicts the lateral displacement of the wall together with the inclinometer measurements, again for the first and final excavation step. The measurements for the final stage have been corrected for lateral movement of the base of the wall which is not reflected in the inclinometer measurement but most likely to occur (based on measurements under similar conditions). Figure 5 shows calculated bending moments.

Fig 3. Deformed mesh (detail) – reference solution

Fig 4. Wall deflection – reference solution

Fig 5. Bending moments – reference solution

Fig 6. Surface settlements – reference solution
3 INFLUENCE OF VARIOUS MODELLING ASSUMPTIONS

In this section modelling assumptions such as the dimensions of the domain analysed, modelling of wall friction and grout body of anchors are investigated. The influence of the constitutive model is addressed in section 4.

3.1 Influence of wall friction

For the reference solution the interface elements implemented in the code Plaxis have been used in a standard way, i.e. the factor \( R_{\text{inter}} \) was used in order to reduce the strength properties of the interface elements with respect to the surrounding soil. The elastic stiffness of the interface elements is governed by a so-called "virtual thickness" which is based on the average element size of the mesh adjacent to the wall. For the reference solution \( R_{\text{inter}} = 0.8 \) has been assumed, a value which is based on experience. In order to study the effect of this parameter an analysis has been performed changing \( R_{\text{inter}} \) to 0.5. It follows from Figures 7 to 9 that this parameter has a significant influence on the displacements. The horizontal displacement of the top of the wall increases by approx. 25 mm and the settlement behind the wall by approx. 15 mm. Bending moments do not change significantly. In order to evaluate the influence of the elastic properties of the interface elements a calculation was performed with a reduced virtual thickness as compared to the default value set in Plaxis, thus the stiffness of the interface is increased. This results in a reduction of the maximum horizontal displacement of the wall in the order of 5 mm (Figure 8). It is obvious from these results that input parameters for modelling wall / soil interaction have to be chosen very carefully, which is however a difficult task because the elastic stiffness of an interface is not a well defined mechanical property. Although results presented here are related to the particular interface element formulation implemented in Plaxis it can be expected that other formulations will show a similar sensitivity to input parameters.

Fig 7. Surface settlements – influence of wall friction

Fig 8. Wall deflection – influence of wall friction

Fig 9. Bending moments – influence of wall friction
3.2 Influence of domain analysed

For the reference solution the domain analysed was chosen as 150 x 100 m for width (W) and depth (D) of the mesh respectively. In order to study the influence of the discretized domain chosen the following analyses have been performed: W x D = 150 x 70, W x D = 100 x 100, W x D = 100 x 70 and W x D = 200 x 150 m. It follows from Figures 10 to 12 that bending moments are hardly influenced but displacements (horizontal displacements of wall and surface settlements) differ in the order of approx. 6 mm, the deeper meshes resulting in smaller surface settlements. This is a result of the vertical upwards displacements caused by the elastic unloading due to excavation which increases with deeper meshes. This effect would be even more pronounced with a Mohr-Coulomb model but would practically vanish when applying a constitutive model taking into account small strain stiffness effects. Similar observations are usually made when analysing excavation of tunnels where surface settlements may depend significantly on the depth of the mesh below the tunnel when linear elastic - perfectly plastic constitutive models are used (Schweiger et al., 1999).

3.3 Influence of modelling ground anchors

When using the code Plaxis the load transfer from the free length of the ground anchors into the ground can be conveniently modelled with membrane elements. These elements, which have no bending stiffness but axial stiffness only, allow a continuous load transfer from the membrane element to the ground along its entire length and avoid a concentrated point load at the end of the free anchor length. Of course this modelling technique is only applicable for working load conditions because the limiting pull
out force cannot be taken into account correctly with this simple model.

To emphasize the importance of a continuous load transfer along the grout body two analyses without membrane elements have been performed. In the first one the free anchor length has been kept the same as in the reference solution and in the second one the free anchor length has been increased by half of the length of the grout body in order to compensate for not modelling the load transfer in more detail. Figures 13 to 15 clearly show that care must be taken when choosing the model representing the ground anchor and grout body.

Fig 13. Wall deflection – influence of modelling grout body

Fig 14. Bending moments – influence of modelling grout body

Fig 15. Surface settlements – influence of modelling grout body

4 INFLUENCE OF CONSTITUTIVE MODEL

Because linear elastic - perfectly plastic constitutive models are still widely used in practice some analyses are performed with a Mohr-Coulomb model and compared to the reference solution. The Young's moduli are back-calculated for each layer corresponding to the initial stiffness introduced in the Hardening Soil model of the reference solution in the middle of each layer (Table 2). MC_3 uses the loading stiffness (E_{st}^{ref}) and MC_4 the unloading stiffness (E_{rel}^{ref}). It follows from the results shown in Figures 16 to 18 that high differences are obtained for displacements, with the reference solution of the HS-Model being in between the two MC-solutions. A similar picture is obtained for bending moments. The deficiencies of elastic-perfectly plastic models become apparent when looking at the displacements behind the wall in Figure 18 where all MC-models show unrealistic heave of the surface.

Table 2. Material parameters MC-model 3 and 4

<table>
<thead>
<tr>
<th>depth of layer</th>
<th>E (MC_3)</th>
<th>E (MC_4)</th>
<th>φ</th>
<th>ψ</th>
<th>c</th>
<th>ν</th>
<th>R_{inter}</th>
</tr>
</thead>
<tbody>
<tr>
<td>m kPa</td>
<td>kPa</td>
<td>kPa</td>
<td>kPa</td>
<td>kPa</td>
<td>kPa</td>
<td>kPa</td>
<td>kPa</td>
</tr>
<tr>
<td>0 - 20</td>
<td>32 000</td>
<td>128 000</td>
<td>35</td>
<td>5</td>
<td>1.0</td>
<td>0.3</td>
<td>0.8</td>
</tr>
<tr>
<td>20 - 40</td>
<td>90 000</td>
<td>360 000</td>
<td>38</td>
<td>6</td>
<td>1.0</td>
<td>0.3</td>
<td>0.8</td>
</tr>
<tr>
<td>&gt; 40</td>
<td>196 000</td>
<td>588 000</td>
<td>38</td>
<td>6</td>
<td>1.0</td>
<td>0.3</td>
<td>-</td>
</tr>
</tbody>
</table>
4.1 Parameter variation with MC-model 1

In this section a limited parametric study, similar to the one presented in section 3, is performed with the parameter set of MC_1 (Table 3) as basic analysis. It is interesting to see that with certain, however not very realistic, assumptions the Mohr-Coulomb model calculates a similar lateral deflection of the wall as the Hardening Soil model (Figure 19). A match of surface settlements however cannot be achieved (Figure 21). Bending moments are not influenced so much (Figure 20). Again the strong influence on the results of the assumptions made for wall friction is obvious.
5 PROBLEM DEFINITION – EXAMPLE 2

The second, simpler example briefly addresses the influence of the design approach according to Eurocode7 when performing ULS-design with finite elements. Eurocode7 allows for three different design approaches DA1 to DA3 which differ in the application of the partial factors of safety on actions, soil properties and resistances.

EC7 states: "It is to be verified that a limit state of rupture or excessive deformation will not occur with the sets of partial factors" as given in Tables 4 and 5 for all three approaches. It is noted that 2 separate analyses are required for design approach 1. The problem which arises for numerical analyses is also immediately apparent because DA1/1 and DA2 require permanent unfavourable actions to be factored by a partial factor of safety, e.g. the earth pressure acting on retaining structures. This is however not readily taken into account in numerical analyses because the earth pressure is not an input but a result of the analysis.

Table 4. Partial factors for actions according to EC7

<table>
<thead>
<tr>
<th>Design approach</th>
<th>Permanent unfavourable (γ)</th>
<th>Variable (γ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DA1/1</td>
<td>1.35</td>
<td>1.50</td>
</tr>
<tr>
<td>DA1/2</td>
<td>1.00</td>
<td>1.30</td>
</tr>
<tr>
<td>DA2</td>
<td>1.35</td>
<td>1.50</td>
</tr>
<tr>
<td>DA3</td>
<td>Geot. 1.00</td>
<td>1.30</td>
</tr>
<tr>
<td></td>
<td>Struct. 1.35</td>
<td>1.50</td>
</tr>
</tbody>
</table>

Table 5. Partial factors for soil properties and resistances according to EC7

<table>
<thead>
<tr>
<th>Design approach</th>
<th>tanα</th>
<th>c'</th>
<th>c</th>
<th>Unit weight</th>
<th>Passive</th>
<th>Anchor</th>
</tr>
</thead>
<tbody>
<tr>
<td>DA1/1</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.10</td>
</tr>
<tr>
<td>DA1/2</td>
<td>1.25</td>
<td>1.25</td>
<td>1.25</td>
<td>1.00</td>
<td>1.00</td>
<td>1.10</td>
</tr>
<tr>
<td>DA2</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.10</td>
</tr>
<tr>
<td>DA3</td>
<td>1.25</td>
<td>1.25</td>
<td>1.25</td>
<td>1.00</td>
<td>1.00</td>
<td>1.10</td>
</tr>
</tbody>
</table>

5.1 Geometry, parameters and computational steps

The geometry of the problem follows from Figure 22. The following construction steps have been considered in the analysis:
- initial phase (K0 = 0.5)
- activation of diaphragm wall (wished-in-place)
- activation of surcharge loads
- excavation step 1 to level -2.0 m
- activation of strut at level -1.50 m, excavation step 2 to level -4.0 m,
- groundwater lowering inside excavation to level -6.0 m
- excavation step 3 to level -6.0 m
The surcharge of 10 kPa is a permanent load, the surcharge of 50 kPa is a variable load. Bedrock was assumed at a depth of 20 m below ground surface. The following parameters have been used.

**Soil properties:**
- \( E = 30 \, \text{MPa} \)
- \( \nu = 0.3 \)
- \( \phi = 27.5^\circ \)
- \( c = 10 \, \text{kPa} \)
- \( \gamma_{\text{saturated}} = 20 \, \text{kN/m}^3 \)
- \( \gamma_{\text{above water table}} = 19 \, \text{kN/m}^3 \)

**Diaphragm wall:**
- \( E = 3.0E7 \, \text{kN/m}^2 \)
- \( \nu = 0.18 \)
- \( \gamma = 24 \, \text{kN/m}^3 \)
- \( d = 0.8 \text{m} \)

**Strut:**
- \( EA = 1.5E6 \, \text{kN/m} \)

### 6 RESULTS – EXAMPLE 2

The following results were evaluated applying DA2 and DA3 as described above: required embedment depth, design strut force, design bending moment. It is acknowledged that in general the required embedment would not be determined by means of finite element analyses but by employing a more conventional approach. However, it is done here for highlighting differences in design approaches.

In order to determine the embedment depth analyses were performed with different wall lengths. Starting with an embedment depth of 5 m the wall was shortened in 0.5 m intervals and for each wall length a new analysis was performed. Horizontal displacements of the base of the wall, bending moments, strut forces and the factor of safety, obtained by means of a strength reduction technique, were evaluated. Figure 23 shows the increase of horizontal deformation of the base of the wall when decreasing the length of the wall for DA2, DA3 and for an analysis with characteristic parameters. The characteristic analysis and DA2 is almost the same because the only difference is the factor of 1.11 for the variable load in DA2. Numerical convergence could not be achieved for an embedment depth of 1.5 m for DA2 and characteristic parameters and a depth of 3.0 m for DA3 respectively.

![Figure 23. Horizontal wall displacement vs. embedment depth](image)

Figure 24 plots the safety factor obtained from a strength reduction technique for different embedment depths. Again values for DA2 and characteristic parameters do not vary much and values for DA3 are much smaller because the soil strength is already factored at the beginning of the analysis. Figure 24 features an additional line, namely the value obtained for DA2 divided by the partial factor which has been put on the strength parameters in DA3. They compare well for embedment depths between 3.5 and 4.5 m but for factors around or below 1 they differ. To some extent this can be attributed to details of the iteration procedure and convergence settings which become more sensitive for states at or very near to failure. No attempt has been made to achieve a closer matching by tightening tolerance factors because this was not the main goal of this investigation.
Figure 25 plots bending moments versus embedment depths. It is interesting to see that the design bending moment of DA2 (which is the result of the DA2 analysis multiplied by the partial factor of 1.35) coincides with DA3 for an embedment depth of 4 m but is different for 3.5 and 4.5 m. The same holds for the strut forces (Figure 26), with DA3 resulting in higher strut forces than DA2 where the resulting force is factored. When applying DA2 an additional check has to be made with respect to the passive resistance for which a partial factor of 1.4 (see Table 5) has to be applied. One way of checking this is to compare the passive earth pressure obtained from the finite element analysis (multiplied by the partial factor of 1.35) to the maximum theoretical passive pressure divided by 1.4. This corresponds roughly to consider a mobilisation of 50% of the passive resistance which is sometimes assumed in practical design. Of course different theoretical solutions can be used to obtain the maximum passive resistance but here Coulomb's solution is employed for simplicity. By doing so the required embedment depth is around 3.0 m for DA2, which is less than for DA3 where a minimum embedment depth of approximately 3.5 m is required for a factor of safety > 1.0 (Figure 24). However this result is considered to be quite acceptable given the various modelling assumptions involved in the two approaches.

Discussion on the merits of numerical analyses in combination with Eurocode7 design approaches can also be found in Schweiger (2005), Bauduin, De Vos & Frank (2003) and Simpson (2000).

7 SUMMARY AND CONCLUSION

A reference solution for a deep excavation problem utilizing the finite element method and an elastic-plastic constitutive model as been presented. The problem has been specified by the German Working Group 1.6 of the DGGT and has been used, in addition to the work presented here, as benchmark problem (Schweiger 2002). Based on the reference solution a comprehensive study was performed in order to evaluate quantitatively the influence of various modelling assumptions on calculated displacements and bending moments. Some analyses assuming elastic-perfectly plastic material behaviour confirmed the well known fact that these very simple constitutive models are not well suited for predicting realistic deformations for these types of problems.

Secondly, the influence on the design approaches as defined in Eurocode7 in ULS-design employing the finite element method has been discussed. It is shown that different design approaches can be readily accommodated for in finite element analyses but design strut forces and design bending moments depend on the design approach used. Based on the results shown here and from other investigations it can be concluded that no general trend can be observed which design approach (DA1, DA2 or DA3) leads to higher design forces. This depends to a large extent on the problem under consideration and in particular
on the relative stiffness between support system (wall and struts/anchors) and soil.

Finally it is emphasized that numerical methods provide a valuable tool in assessing serviceability limit states (SLS) because it is possible to model soil structure interaction adequately. However, care must be taken in setting up the model because a number of assumptions which have to be made by the modeller will not be found in any code of practice or geotechnical report. As far as Ultimate Limit State (ULS) design is concerned, numerical methods can be used but more experience is still required with respect to model soil structure interaction in the ultimate limit state in order to guarantee a consistent level of safety factor in structural elements and soil.

REFERENCES


