Tutorial

Reinforcement Design of a Pile Cap
Outline

1 Introduction 3
2 Strut-and-Tie Approach for Pile Cap Design 4
3 Parametric Design using Python 8
4 Case 1: Pile Cap - 2 Piles 9
   4.1 User Input ................................................. 10
   4.2 Material Model ............................................ 12
   4.3 Finite Element Model ..................................... 15
      4.3.1 Composed Line Elements ............................. 16
      4.3.2 Linear Elastic Analysis ............................. 19
      4.3.3 Reinforcement Calculation ............................. 20
      4.3.4 Detailing Rules ........................................ 21
      4.3.5 Nonlinear Material Models ............................. 22
   4.4 Structural Nonlinear Analyses ............................. 25
   4.5 Results .................................................. 26
5 Case 2: Pile Cap - 4 Piles 31
   5.1 User Input ................................................. 32
   5.2 Finite Element Model ..................................... 34
      5.2.1 Composed Line Elements ............................. 34
      5.2.2 Linear Elastic Analysis ............................. 35
      5.2.3 Reinforcement Calculation ............................. 36
   5.3 Structural Nonlinear Analyses ............................. 37
   5.4 Results .................................................. 38
Appendix A Additional Information 44
1 Introduction

In this tutorial it is explained how to use Nonlinear Finite Element Analysis (NLFEA) to determine the reinforcement design for a pile cap which complies with the Eurocode 2 (EC2)\(^1\) and the fib Model Code 2010 (fib2010)\(^2\).

Eurocode 2 (NEN-EN 1992-1-1, 5.7) allows NLFEA:

"(1) Non-linear methods of analysis may be used for both ULS and SLS, provided that equilibrium and compatibility are satisfied and an adequate non-linear behavior for materials is assumed. The analysis may be first or second order."

However, there are few rules and guidelines presented for NLFEA in the EC2. Therefore, in this tutorial, whenever EC2 lacks rules or guidelines which are needed for a correct NLFEA, the fib Model Code 2010 is applied.

For repetitive structural calculations, such as a reinforcement design of a pile cap, it is beneficial to automate the process. Therefore, a parametric python script has been set up, which will be used to automatically determine the reinforcement design for a pile cap.

---


\(^2\) FIB, fib Model Code for Concrete Structures 2010, 2013
2 Strut-and-Tie Approach for Pile Cap Design

In the EC2 it is stated that for the design of discontinuity regions strut-and-tie models should be used (NEN-EN 1992-1-1, 6.1(1)). In this example the reinforcement of a pile cap is determined using strut-and-tie models (NEN-EN 1992-1-1, 6.5).

The strut-and-tie model for a pile cap supported by two piles is depicted in Figure 3. First the designer has to set up the strut-and-tie model in which either the internal arm \( z \) or the internal angle \( \theta \) has to be assumed. Subsequently the forces of the model can be calculated.

Some national annexes give guidelines or rules for the internal calculation of the internal arm \( z \) or the internal angle \( \theta \). For example the Dutch annex of the EC2 6.1(10) gives a rule to determine the internal arm \([\text{Eq. (1)}]\).

When the internal arm is known, the forces of the strut-and-tie model can be calculated.

\[
z = 0.2l + 0.4h \leq 0.6l
\]

(1)

The workflow for a strut-and-tie model is:
- Assume \( z \) or \( \theta \)
- Calculate forces (and check nodes)
- Determine reinforcement
- Calculate crack width
When the forces of the strut-and-tie model are known, a node check can be performed (NEN-EN 1992-1-1, 6.5.4). The EC2 presents rules to check whether the compression force can be resisted [Fig. 5]. The maximum resistance force depends on the type of node.

![Strut-and-tie model and node check](image)

![Nodes (NEN-EN 1992-1-1, 6.5.4)](image)
From the forces the reinforcement can be determined. The amount of longitudinal reinforcement can directly be calculated from the force in the bottom tie. The amount of stirrups and web reinforcement has to be derived from the diagonal strut. By using the rules from EC2 section 6.5.3 [Fig. 7] the tensile force $T$ (perpendicular to the strut) caused by the dispersion of the concentrated compression force can be calculated. From the tensile force $T$ and the angle $\theta$ the amount of reinforcement for the stirrups and web reinforcement can be determined.

Figure 6: Strut-and-tie model and reinforcement determination

Figure 7: Concentrated force (NEN-EN 1992-1-1, 6.5.3)
After the determination of the reinforcement, two nonlinear analyses have to be performed: Service Limit State (SLS) and Ultimate Limit State (ULS).

In the SLS calculation, mean material parameters with SLS load are used ($f_{ctm}$ is used for the tensile strength). The crack width is checked in this limit state.

In the ULS calculation, design values for material and loading are used. This limit state checks the capacity of the structure. Note that the node check only evaluates the compressive failure. Therefore, it is possible to perform this check before determination of the reinforcement.

Figure 8: Reinforcement design
3 Parametric Design using Python

We will perform a parametric design using strut-and-tie models. For that we present the following:

- **Python script:**
  A parametric python script for DIANA has been set up to automatically determine the reinforcement design for a pile cap using NLFEA that complies with the building codes (EC2 and fib2010). The python script can be found in your DIANA folder:

  \share\Tutorials\PileCap

- **User input:**
  Between “START USER INPUT” and “END USER INPUT” the user has to specify the geometry, loading, material input etc.

- **Workflow:**
  In the next slides the workflow of the script will be explained in detail through two cases:
  - a pile cap supported by 2 piles [Section 4]
  - a pile cap supported by 4 piles [Section 5]
4 Case 1: Pile Cap - 2 Piles

The first case study of a pile cap supported by two piles is presented in Figure 10. The loading is applied on the top column and is listed in Table 1. The concrete class is C25/30.

![Figure 10: Pile cap with 2 piles (dimensions in mm)](image)

<table>
<thead>
<tr>
<th>Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G_k = 1000$ kN with $\gamma_k = 1.2$</td>
</tr>
<tr>
<td>$Q_k = 1000$ kN with $\gamma_q = 1.5$</td>
</tr>
<tr>
<td>$F_{rep} = 2000$ kN (SLS)</td>
</tr>
<tr>
<td>$F_d = 2700$ kN (ULS)</td>
</tr>
</tbody>
</table>

Table 1: Loading
4.1 User Input

Figure 11 presents the following user input information in the python script:

- **Model info**
  The element size can be set for the FE model (25-50 mm is recommended) and the user can indicate whether symmetry should be applied.

- **Geometry input data**
  The dimensions of the pile cap have to be specified in this section.
Figure 12 presents the following user input information in the Python script:

- **Load input data**
  Specify the SLS load and ULS load.

- **Material input data**
  Provide all the material data regarding the piles (stiffness and bearing capacity), the concrete class and the reinforcement. Choose the material model for concrete: EC2, fib MC2010 or TSCM - RWS (more information about the material models on the next slides).

- **Limit state**
  Choose SLS or ULS (the user can also change the limit state after running the script).

- **Reinforcement library**
  Input both the allowable diameters of the reinforcement bars, as well as the, allowable spacing for the reinforcement bars.
4.2 Material Model

In the python script you can choose three concrete material models: EC2, fib MC2010 and TSCM RWS.

Figure 13 presents the model from EC2. The plasticity and cracking behavior of the Eurocode 2 EN 1992-1-1 is described by a Total Strain Rotating Crack model with the brittle predefined tension softening function and the EC2 predefined compression curve with maximum strain $\varepsilon_{cu}$. Damage based Poisson’s ratio reduction will be used for the EC2.

![Figure 13: Stress-strain diagram according to Eurocode 2 EN 1992-1-1](image-url)
Figure 14 presents the model from fib MC2010. The plasticity and cracking behavior of the fib Model Code for Concrete Structures 2010 is described by a Total Strain Rotating Crack model, with the fib MC2010 predefined tension softening function and the fib MC2010 predefined compression curve with maximum strain $\varepsilon_{c,\text{lim}}$. The stress confined model is used (Selby and Vecchio)\textsuperscript{3} to take into account multiaxial behavior of concrete.

In this tutorial the fib MC2010 material will be used instead of the EC2 material. The main reason for this choice is the fact that EC2 does not have a tension softening curve, which is needed to accurately describe the crack development and the order of magnitude of the crack width.

\textsuperscript{3}Selby and Vecchio, *Three-dimensional Constitutive Relations for Reinforced Concrete*, 1993
The Ministry of Infrastructure and the Environment (RWS) has set up a technical document named “Guidelines for Nonlinear Finite Element Analysis of Concrete Structures” (RTD 1016-1:2017) to overcome the lack of rules and guidelines in the Eurocode 2 regarding NLFEA.

The plasticity and cracking behavior is described by a Total Strain Rotating Crack model with the Hordyk tension softening function and the parabolic compression curve [Fig. 15]. The lateral influence of cracking on the compressive behavior and damage based Poisson’s ratio reduction are applied. The stress confined model is used (Selby and Vecchio) to take into account the multiaxial behavior of concrete.

![Figure 15: Predefined compression behavior and tensile behavior for the TSCM-RWS model](image)

---

*Hendriks et al., Guidelines for Nonlinear Finite Element Analysis of Concrete Structures, 2017*
4.3 Finite Element Model

The first step after the user input is to set up the Finite Element (FE) model for a linear elastic calculation. This model is used to calculate the strut-and-tie model and determine the reinforcement.

The model is 2D and consists of plane stress elements (for pile cap and pile), line interface elements (for pile behavior) and composed line elements (to determine the forces for the strut-and-tie model). Symmetry is used to reduce the model size (symmetry supports are on the right side of the model).

Composed line elements (dashed lines in Figure 17 and presented in Figure 18) are used to extract the forces from the linear elastic calculation, to generate the strut-and-tie model. In this model we assume that the diagonal strut has a direct load path from the load to the pile. Also the splitting force (tensile force $T$) of the compression strut can be directly read out from the composed element. It is not necessary to indirectly calculate this force from the compression strut.
4.3.1 Composed Line Elements

Composed line elements are used to determine forces from a stress field. By integrating the internal stresses or internal forces in the finite elements over the cross-section plane of the composed line, the forces can be determined. Therefore, it is important that the correct thickness is specified (this determines the area of the cross-section plane) in order to get the a strut-and-tie model that is in equilibrium [Fig. 19].

The composed line elements for the bottom tie calculates the tensile force in the bottom part of the pile cap. In case there is perfect bending over the cross section, then 0.5 $h$ is the tensile zone (non-deep beams). In case of a deep beam the height of the tensile zone can be calculated with $x \tan(\theta)$ [Fig. 20].

$$t_1 = 2 \times \min \left\{\frac{0.5h}{x \tan(\theta)}\right\} \quad \text{[Fig. 20]}$$

![Figure 19: Thickness of composed element 1](image1)

![Figure 20: Height of the tensile zone](image2)
As a result of dispersion of the concentrated force in the compression strut, different thicknesses have to be applied for the composed line elements. Therefore, the composed line has been divided in three parts [Fig. 21]. The thicknesses have been derived according to the EC2 (NEN-EN 1992-1-1 - 6.5.3) [Fig. 22].

\[ t_2 = a \] (depends on load surface)  
\[ t_3 = b_F \] (effective width at the middle of the strut)  
\[ t_4 = a \] (depends on pile surface)

Figure 21: Thickness of composed element 2, 3 and 4  
Figure 22: Concentrated load (NEN-EN 1992-1-1 - 6.5.3)
As a result of dispersion of the concentrated force in the compression strut, perpendicular to this strut splitting tensile stresses are occurring. These splitting stresses are largest in the middle of the compression strut. Therefore, the thickness of composed element 5 is equal to the length of the middle part of the compression strut [Fig. 23].

\[ t_5 = \text{length of middle strut} \]

Figure 23: Thickness of composed element 5
4.3.2 Linear Elastic Analysis

We first perform a linear static analysis. The results of the linear elastic analysis are depicted below. The output of the stress tensors are presented in Figure 24. The figure confirms the assumption about the direct loading path from the column to the pile. The internal arm that follows from the model is larger than the internal arm calculated from the code (NEN-EN 1992-1-1 - 6.1(10)).

Figure 25 shows the normal forces of the composed elements. These represent the forces in the strut-and-tie model. The normal force in the bottom composed elements will be used to determine the longitudinal reinforcement. The normal force in the composed elements perpendicular to the compression strut (splitting force of the strut) will be used to determine the stirrup and web reinforcement.

Figure 24: Stress tensor

Figure 25: Normal force in composed elements
4.3.3 Reinforcement Calculation

We calculate the reinforcement areas [Fig. 27] based on the forces from the composed elements [Fig. 26].

The bottom reinforcement: \( A_{s,req} = \frac{N_x}{f_{yd}} \rightarrow A_{s,req} = 8.86 \times 10^5 / 435 = 2037 \text{ mm}^2 \)

\[ \rightarrow \text{Longitudinal reinforcement: } A_{s,prov} = 7620 = 2199 \text{ mm}^2 \]

The splitting reinforcement: \( A_{s,req} = \frac{N_x}{(f_{yd} \times \ell)} \rightarrow A_{s,req} = 2.96 \times 10^5 / (435 \times 0.340) = 2001 \text{ mm}^2/\text{m}^1 \)

The calculated reinforcement is perpendicular to the compression strut (with angle \( \theta \)) and therefore needs to be decomposed in vertical and horizontal reinforcement.

\[ \rightarrow \text{Stirrup: } A_{s,str} = \cos(\theta) \times A_{s,req} = \cos(55) \times 2001 = 1148 \text{ mm}^2/\text{m}^1 \rightarrow A_{s,prov} = 2 \times \phi_{12} - 150 = 1508 \text{ mm}^2/\text{m}^1 \]

\[ \rightarrow \text{Web reinforcement: } A_{s,fl} = \sin(\theta) \times A_{s,req} = \cos(55) \times 2001 = 1639 \text{ mm}^2/\text{m}^1 \rightarrow A_{s,prov} = 2 \times \phi_{12} - 125 = 1810 \text{ mm}^2/\text{m}^1 \]
4.3.4 Detailing Rules

After calculating the amount of reinforcement due to the acting forces, the detailing rules are applied and the reinforcement is updated accordingly. In this case only the top reinforcement has to be updated.

Applied detailing rules:

- EC2 section 8.2 – Spacing of bars: check minimum and maximum spacing
- EC2 section 9.7 – Deep beams (minimum for orthogonal reinforcement mesh of each face)
- EC2 section 9.2.1.2(1) - Other detailing arrangement (top reinforcement)

Note that “EC2 section 8.4 – Anchor length” is not applied. Due to modeling of the actual bond-slip behavior, the anchor length is implicitly taken into account in the model.

Figure 28: Geometry with reinforcement

Figure 29: Geometry with updated reinforcement
4.3.5 Nonlinear Material Models

After determination of the reinforcement, the SLS (for crack width check) and ULS (for node and reinforcement check) nonlinear calculations can be performed. The fib model code material model is used to model the concrete behavior. Bond-slip reinforcement (type truss bond-slip bar) with Von Mises plasticity is used to model the reinforcement behavior. For the bond-slip behavior a multilinear bond-slip curve is defined according to fib model code 2010 (6.1.1 Local bond-slip relationship).

Figure 30: Geometry with reinforcement

Figure 31: Mesh with reinforcement
We define the nonlinear material models. Figure 32 and Figure 33 show the properties of the fib MC2010 concrete material for SLS and ULS, respectively.

The concrete properties are determined according to fib MC2010. The sections of the fib MC2010, which have been used, are presented below:

For SLS mean values:
- 5.1.4 Compressive strength $f_{cm}$
- 5.1.5 Tensile strength and fracture properties $f_{ctm}$ and $G_F$
- 5.1.6 Strength under multiaxial states of stress
- 5.1.7 Modulus of elasticity and Poisson’s ratio $E_c$ and $\nu_c$
- 5.1.8 Stress-strain relations for short term loading $\varepsilon_{c1}$ and $\varepsilon_{cu1}$

For ULS design values:
- 7.2.3 Dimensioning values $\gamma_c$

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear material properties</td>
<td></td>
</tr>
<tr>
<td>Elasticity</td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>32000.3 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
</tr>
<tr>
<td>Mass</td>
<td></td>
</tr>
<tr>
<td>Mass density</td>
<td>2.40-09 T/mm²</td>
</tr>
<tr>
<td>Total strain based crack model</td>
<td></td>
</tr>
<tr>
<td>Crack orientation</td>
<td>Rotating</td>
</tr>
<tr>
<td>Tensile behavior</td>
<td></td>
</tr>
<tr>
<td>Tensile curve</td>
<td>fib Model Code for Concrete Structure…</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>2.56496 N/mm²</td>
</tr>
<tr>
<td>Mode-I tensile fracture energy</td>
<td>0.156975 N/mm²</td>
</tr>
<tr>
<td>Crack bandwidth specification</td>
<td>Govindjee</td>
</tr>
<tr>
<td>Poisson’s ratio reduction</td>
<td>Damage based</td>
</tr>
<tr>
<td>Compressive behavior</td>
<td></td>
</tr>
<tr>
<td>Compressive curve</td>
<td>fib Model Code for Concrete Structure…</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>33 N/mm²</td>
</tr>
<tr>
<td>Strain at maximum stress</td>
<td>0.902</td>
</tr>
<tr>
<td>Strain at ultimate stress</td>
<td>0.9035</td>
</tr>
<tr>
<td>Stress confinement</td>
<td>Seilby and Vecchio</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear material properties</td>
<td></td>
</tr>
<tr>
<td>Elasticity</td>
<td></td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>20674.4 N/mm²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.2</td>
</tr>
<tr>
<td>Mass</td>
<td></td>
</tr>
<tr>
<td>Mass density</td>
<td>2.40-09 T/mm²</td>
</tr>
<tr>
<td>Total strain based crack model</td>
<td></td>
</tr>
<tr>
<td>Crack orientation</td>
<td>Rotating</td>
</tr>
<tr>
<td>Tensile behavior</td>
<td></td>
</tr>
<tr>
<td>Tensile curve</td>
<td>fib Model Code for Concrete Structure…</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>1.19698 N/mm²</td>
</tr>
<tr>
<td>Mode-I tensile fracture energy</td>
<td>0.059267 N/mm²</td>
</tr>
<tr>
<td>Crack bandwidth specification</td>
<td>Govindjee</td>
</tr>
<tr>
<td>Poisson’s ratio reduction</td>
<td>Damage based</td>
</tr>
<tr>
<td>Compressive behavior</td>
<td></td>
</tr>
<tr>
<td>Compressive curve</td>
<td>fib Model Code for Concrete Structure…</td>
</tr>
<tr>
<td>Compressive strength</td>
<td>16.6667 N/mm²</td>
</tr>
<tr>
<td>Strain at maximum stress</td>
<td>0.002</td>
</tr>
<tr>
<td>Strain at ultimate stress</td>
<td>0.0035</td>
</tr>
<tr>
<td>Stress confinement</td>
<td>Seilby and Vecchio</td>
</tr>
</tbody>
</table>

Figure 32: SLS material properties
Figure 33: ULS material properties
We define the material model for the reinforcement.

The reinforcement properties are determined according to fib MC2010. The sections of the fib MC2010, which have been used, are presented below:

For reinforcement steel:

- 5.2.5 Mechanical properties $f_y$, $f_t$, and $\varepsilon_u$
- 5.2.8 Assumptions used for design

For bondslip of reinforcement:

- 6.1.1 Local bond-slip relationship

Figure 34: Reinforcement material properties
4.4 Structural Nonlinear Analyses

You can choose the type of nonlinear analysis, either “PILECAP2D” or “PILECAP2D – arclen” for SLS and ULS, respectively [Fig. 35]. The first analysis uses (regular or modified) Newton-Raphson with explicitly specified load step sizes. The second analysis uses arc length control to indirectly control the displacement.

For SLS analysis it is recommended to use “PILECAP2D”, because only minor cracking is expected (therefore minor nonlinear behavior). For ULS extended cracking and crushing of the concrete is expected (important nonlinear behavior), therefore arc length control is recommended.

Figure 35: Analyses
4.5 Results

We first present the results related to the SLS: displacement [Fig. 36] and crack width [Fig. 37].

For SLS we have to check:

\[ \text{Crack width} \ll 0.35 \text{ mm} - \text{satisfied} \]
We now show the results related to the ULS.

For ULS the compressive failure of the nodes is checked. At the location of the piles and column the pile cap is only partially loaded over the thickness. To take this into account in a two-dimensional analysis a thickness function is applied. In this way the calculation does not lead to an overshoot of the compressive failure of the pile cap. The thickness distribution in the model is represented in Figure 38.

Figure 38: Thickness of the cross section
We show the results of the crack width [Fig. 39] and stress tensor [Fig. 40].

For ULS the crack width does not have to be checked, but it is shown here to provide insight in the behavior. The stress tensor shows that the assumption of the compression strut was correct. Due to the large shear crack, there is a direct loading path from the load to the pile.
We show the stresses in the reinforcement [Fig. 41] and the bond-slip stresses along the reinforcement bars [Fig. 42]. In this case the reinforcement stresses are still below the yield value and therefore it satisfies with the Eurocode 2. Also Figure 41 and Figure 42 show that the anchorage of the reinforcement bar is sufficient. Too short anchorage length would lead to large slip of the reinforcement bar causing the structure to fail by pull out of the reinforcement.

For ULS we have to check:

→ Reinforcement stress < 435 N/mm² - satisfied
We present the second principal stress and strain distributions in the structure [Fig. 43] [Fig. 44].

Due to the biaxial stress state at the loading surface, we can observe an increase of compressive strength in that area. Both below the loading surface as above the pile the principal strain does not exceed 3.5‰ (check of nodes).

For ULS we have to check:

→ Check nodes: Principal strain $E_2 < 3.5‰$ - satisfied
5 Case 2: Pile Cap - 4 Piles

Figure 45 represents the second case study with four piles. The loading is applied on the top column and is listed in Table 2. The maximum bearing capacity of a pile is set to 3000 kN and the concrete class is C35/45.

The same methodology used in Case 1 will be followed in the present case.
### 5.1 User Input

```python
### MODEL INFO
A = 900  # maximum size [mm] for finite elements
S = 1   # apply symmetry (0=False, 1=True)

### GEOMETRY
b = 1500  # pile cap base
h = 1200  # pile cap height
t = 1000  # pile cap thickness

### Piles/support [mm]:
np = 5  # number of piles
bp = 600  # pile section dimension
tp = 500  # pile thickness
ip = 1000  # axial distance between piles (c.t.c. piles)

### Column/load [mm]:
boc = 600  # column section base
toc = 500  # column thickness
cor = 15  # coverage and grain size [mm]

c = 32  # max grain size of concrete
```

Figure 46: User input part 1
Figure 47 presents the second part of the user input.
5.2 Finite Element Model

5.2.1 Composed Line Elements

As explained in Section 4.3.1, it is important that the correct thickness/area is specified of the composed elements in order to get the a strut-and-tie model that is in equilibrium. In this case if $b_{ht}$ (EC2 section 6.5.3) would be used for the thickness of the composed elements (compression struts), the thickness/area of the struts will overlap each other, which will lead to too high compressive forces. Therefore, the pile cap is divided by the green dashed line into two mesh sets in which the composed line elements are only allowed to extract the force of the corresponding element set [Fig. 48].

Figure 48: Mesh
5.2.2 Linear Elastic Analysis

We perform a linear elastic analysis to determine the forces in the composed elements. The output of the stress tensors are presented in Figure 49. The figure shows the loading path to the piles. Figure 50 and Figure 51 show the normal forces of the composed elements for the two mesh sets of the pile cap. From these figures it can be clearly seen that most of the force is carried by the two inner piles.
5.2.3 Reinforcement Calculation

Similarly to Section 4.3.3, we calculate the reinforcement areas [Fig. 52] [Fig. 53] based on the forces determined from the composed elements.

Figure 52: Geometry with reinforcement

Figure 53: Mesh with reinforcement
5.3 Structural Nonlinear Analyses

Similarly to Section 4.4 we perform two nonlinear analyses: one for SLS and one for ULS.
5.4 Results

We first present the results related to the SLS: displacement [Fig. 54] and crack width [Fig. 55].

For SLS we have to check:

→ Crack width < 0.35 mm - satisfied
We now show the results related to the ULS.

For ULS the compressive failure of the nodes is checked. At the location of the piles and column the pile cap is only partially loaded over the thickness. To take this into account in a two-dimensional analysis a thickness function is applied. In this way the calculation does not lead to an overshoot of compressive failure of the pile cap. The thickness distribution in the model is represented in Figure 56.

Figure 56: Thickness of the cross section
We show the results of the crack width [Fig. 57] and stress tensor [Fig. 58] at 80% of the total load.

The crack width and the stress tensors are shown here to provide insight in the behavior (results at 0.80 of the total load). At this load the middle piles reach the pile capacity of 3.0 MN. Up till this phase most of the forces go to the middle piles, which can be clearly seen in Figure 58.

Figure 57: Crack width at 80% of total load

Figure 58: Stress tensor at 80% of total load
We show the results of the crack width [Fig. 59] and stress tensor [Fig. 60] at 100% of the total load.

The crack width and the stress tensors are shown here at the full load to provide insight in the behavior (results at 1.0 of the total load). After a load of 0.80 the middle piles start to deform plastically leading to large settlements. From this moment the outer piles start to take up the rest of the force. This effect can be seen in Figure 60.
We show the stresses and strains in the reinforcement [Fig. 61] [Fig. 62].

In this case the reinforcement is yielding ($S_{xx} \geq 435$ N/mm$^2$). Therefore, the reinforcement strain is checked. The strain in the reinforcement is lower than 25‰ and therefore it satisfies with the Eurocode 2.

For ULS we have to check:

→ Reinforcement stress $\leq 435$N/mm$^2$ → Reinforcement strain $< 25\%$ - satisfied
We present the second principal stress and strain distribution in the structure [Fig. 63] [Fig. 64]. Due to the biaxial stress state at the loading surface, we can observe an increase of compressive strength in that area. Both below the loading surface as above the pile the principal strain does not exceed 3.5‰ (check of nodes).

For ULS we have to check:

→ Check nodes: Principal strain $E_2 < 3.5\,\%$ - satisfied
Appendix A  Additional Information

Folder: Tutorials/PileCapReinfoDesign

Number of elements $\approx 1300$ and $10000$

Keywords:
- ANALYS: geomet nonlin physic.
- CLASS: large.
- CONSTR: suppor.
- ELEMEN: bar bondsl cable cl12i cl6tr cq16m cq22if ct12m interf pstres reinfo struct truss.
- LOAD: edge elemen force functi.
- MATERI: cebfip crack elasti harden isotro multil nonlin plasti rotati slip soften strain totstr vonmis.
- OPTION: arclen direct lagran linese newton normal regula select stop total units update.
- POST: binary ndiana.
- PRE: dianai.
- RESULT: cauchy crack crkwdt displa extern force green princ reacti strain stress total tracti.

References: