On the finite element bearing capacity analysis of a rib system to be used as shallow foundation construction

Soelarso¹, Antaluca E¹, Batoz, J.L¹, Lamarque F¹.

¹Equipe Avenues (EA 7284), Sorbonne Universities, Université de Technologie de Compiègne, France

E-mail: soelarso.soelarso@utc.fr

Abstract. The work is related to the Ph.D. subject of the first author on the analysis and design of shallow foundation systems in the context of earthquake resistant constructions in Indonesia. We focus here on a particular system called SNSF (Spider Net System Footing) that has been proposed and successfully used for around 10 years. This paper deals with the analysis of a unit cell of SNSF consisting of an assemblage of concrete vertical ribs and horizontal plate subjected to a central loading. We propose a geometrical 3D model and analyses based on 3D solid and shell elements for a detailed static analysis of the structure under central load or under self-weight. We also report some results considering the soil structure interaction, assuming linear elastic properties of the soil.

1. Introduction

The selection and design of foundation is one of the important stages in planning for an earthquake resistant building since the foundation is part of an engineering system that support the upper structure, transfer the permanent and service loads to the ground. If there is a construction disorder of the foundation, then damages to the building can be observed like cracking of walls, bumpy floors and differential settlements. The Spider Net System Footing (SNSF) is a shallow foundation proposed by Riyantori and Soetjipto Soedjono in 1976 [1] consisting in an assemblage of concrete plates and ribs with compacted soil inside the cells (see figure 1, [2]). The combination of compacted soil and reinforced concrete structural elements leads to a foundation system that has a much higher stiffness compared to other shallow foundation system [3]. Several buildings have already been erected with SNSF in Indonesia, and no severe damages due to recent earthquakes have been observed.

Figure 1. Spider Net System Footing (SNSF)[2] Figure 2. The unit cell (from reference [4])

2. Description of a particular SNSF unit cell

A unit SNSF cell has been studied by Darjanto in 2015 [3] and results published in [4]. The purpose of the work was to evaluate the bearing capacity of a unit cell, both experimentally and numerically. That
work is very interesting since we can find in [3] information on the geometry of the reinforced concrete structure, material properties of the soil and some experimental and numerical results. The dimensions of the cell are 2.6 m x 2.6 m (the plate) and a depth of 0.9 m (height of vertical external ribs) (figure 2).

The soil tests show that there are 2 different types of soil properties. The first soil layer for a depth of 2 meters is a soft soil. The deeper layer of 8 meters is considered as a medium soil. The loading system is described in figure 3. Static loads applied at the center are: 8 T, 15 T, 30 T and 60 T. It has been reported in [4] and [3] that up to 8 T the behaviour is linear.

A CAD 3D geometrical model has been built using the Revit software of Autodesk (figure 4).

3. FEM analysis and results of the concrete structure
The model of the unit cell was exported to Altair Hypermesh [5]. We performed finite element analysis assuming an elastic behaviour of both the concrete structure and the soil inside and outside the structure. We first study in details the structure alone, simply supported on a rigid ground. The relevant material properties for the linear analysis are given in table 1.
Table 1. Material properties

| Material   | Elasticity Modulus E (MPa) | Density $\rho$ (T/m$^3$) | Shear Modulus (G) (MPa) |
|------------|----------------------------|---------------------------|-------------------------|
| Concrete   | 21409,52                   | 2,4                       | 8243,43                 |
| Soft Soil  | 22,5                       | 1,7                       | 8,65                    |
| Medium Soil| 75                         | 1,74                      | 28,85                   |

The structure (plate and ribs) can be meshed either by 3D solid elements or by 3D shell elements. The solid element we use is a classical 4 nodes tetrahedron with three displacements $u$, $v$, and $w$ as degrees of freedom per node, (Batoz et Dhatt, 1990). For the modelling with shell elements we use the 3 and 4 nodes elements available in the Optistruct software (equivalent to NASTRAN software). Those two elements have 6 degrees of freedom per node, three displacements $(u,v,w)$ and three rotations around the three axis $x$, $y$ and $z$, (Batoz et Dhatt, 1992). The loadings are either the self-weight of the structure or a load of 8 T applied on a patch at the centre for the solid 3D elements and a concentrated load when using 3D shell elements (see figure 3).

On figure 5 we show two meshes of the structure. The first is a mesh with 3D solid tetrahedron elements and the second is a mesh based on 3 nodes shell elements.

![Meshes and central loading on the structure](image)

**Figure 5.** Meshes and central loading on the structure

The vertical displacement at the center is reported in Table 2 for different meshes. On figure 6 we present the isovalues of displacements for the case of 8T central loading using shell elements.

![Isovalues of displacements using Shell elements for 8 T central loading](image)

**Figure 6.** Isovalues of displacements using Shell elements for 8 T central loading
Table 2. Displacements using 3D Solid and Shell elements

| Loading (Ton) | Mesh size (mm) | 3D Solid Elements | 3D Shell Elements |
|--------------|----------------|-------------------|-------------------|
|              |                | Total of DOFs     | Central vertical Displacement (mm) | Total of DOFs | Central vertical Displacement (mm) |
| 8 Ton        | 400            | 1275              | 0.064             | 925           | 0.108 |
| Concentrated load | 100    | 12997             | 0.118             | 13705         | 0.145 |
| Self weight  | 400            | 1275              | 0.009             | 925           | 0.020 |
|              | 25             | 387646            | 0.156             | 220903        | 0.166 |

The central displacement as a function of the number of degrees of freedom is shown on figure 7, for the two load cases (8T and self weight) and using solid and shell elements (SW for self weight). These curves show the uniform convergence towards the unknown exact solutions. Good results can be obtained with shell elements for less than 2 times degrees of freedom compared to solid elements, for equivalent precision in displacements and stresses.

Different values of maximum stresses can be observed depending on the model used (3D solid or 3D shell), but the maximum values are always local values and despite that the order of magnitude is quite the same. The values are also compatible with admissible stresses for reinforced concrete (< 1.04 MPa in tension). The local maximum stresses occur in the center at the lowest levels of the ribs for the 8T load and also at the lower level of the limit ribs in contact with the soil. The isovalues of maximum von Mises stresses are presented in figure 8 and 9.
4. Soil structure interaction
The soil inside and below the cell is meshed with 4 nodes solid elements. The meshes obtained by Hypermesh for the soil and for the structure are fully compatible (continuity of displacements). A first simulation of soil structure interaction (based on 3D solid tetrahedron meshes) has been performed considering the structure, the inner soil and the outside soil. All materials are considered as linear elastic (table 1). The outside soil has dimensions (12 m x 12 m x 10 m) and at the limits (vertical and bottom planes we assume zero displacements), see figure 10. Our results for the vertical displacement considering a mesh size indicator of 100 mm (see table 2) can be compared with some numerical and experimental results reported in references [3] and [4] and obtained by Darjanto M (2015) using the SAP 3D software and the experimental setup (Table 3 and figure 11).

Figure 9. von Mises stresses using 3D shell elements

Figure 10. Soil structure interaction model

| Loading (Ton) | Size Mesh | Optistruct Displacement (mm) | SAP 3D | Experiment |
|--------------|-----------|-------------------------------|--------|------------|
|              | 200       | 100                           | 0      | -          |
| 0            | 200       | 100                           | 0      | -          |
| 8            | 200       | 100                           | 0,486  | 0,506      | 0,75 | 0,63 |
|              | 200       | 100                           | 0,912  | 0,949      | 1,19 | 1,2   |
| 30           | 200       | 100                           | 1,824  | 1,899      | 2,14 | 3   |
Displacements using Hypermesh and SAP 3D are linear but this is not case after 8 T for the experiments due to the nonlinear behavior of the soil. Our results are in fair agreement with the experimental result for 8T loading and with SAP3D regarding the slope but with a different value for 0 T. The main reason can be due to the influence of the self-weight of the structure. Since the extension of the SAP3D displacement for 0 T is not zero, we evaluate the influence of the self-weight of the cell (using SAP3D) as 0.243 mm (figure 11). The displacement values for different loadings and modelling conditions can be seen in Table 4; comparing lines 1 and 2 the displacement due to self weight (0 T) is 0.298 mm which is close to 0.243 mm based on SAP3D results.

Table 4. Displacement for several load conditions

| Load Conditions                                      | Size Mesh | Displacement (mm) |  |  |
|------------------------------------------------------|-----------|-------------------|---|---|
| Load 8 T with soil inside and outside                | 100       | 0.506             |  |  |
| Load 8 T + SW Concrete (4,799 T) with soil inside and outside | 100       | 0.804             | 0.75 | 0.63 |
| Load 8 T without soil inside but with soil outside   | 100       | 0.631             |  |  |

Comparing lines 3 and 1 of Table 4 we can also see that taking into account the compacted soil inside the cell will increase the stiffness of the cell by 20%. Figure 12 shows a 2D section of displacements. It is seen that the influence of boundary conditions on the outside soil planes has no effect on the results due to the dimensions of the volume of soil we consider for modeling the soil-structure interaction.
5. Concluding remarks
In this work we propose a 3D geometrical model of a unit cell representative of the SNSF system and we analyze in details the behaviour of the structure under a central loading and self weight. We also perform some computations to estimate the displacements and stresses considering the interaction between the structure and the soil inside and outside the structure. All results are satisfactory with respect to available experimental and numerical results.

6. References
[1] https://id.wikipedia.org/wiki/soetjipto_soedjono.
[2] http://www.katama.co.id
[3] Darjanto H, “Mekanisme Transfer Beban Pada Cell Fondasi Konstruksi Sarang Laba-Laba Melalui Beban Statis Vertikal Siklis Skala Penuh dan Analisis Numerik 3D”. Report Doctoral Dissertation, Civil Engineering, Diponegoro University, 2015.
[4] Darjanto H, Irsyam M, Retno S.R, “Full Scale Load Test on The Spider Net System”, 77:11 (2015) 73-82. Jurnal Teknologi (Sciences & Engineering). Universiti Teknologi Malaysia.
[5] https://www.altairhyperworks.com/
[6] Suhendro, B. Progress report III : Optimization Study on The Equation of Chicken Claws System Footing. Civil Engineering Department, Gadjah Mada University, 1992.
[7] Batoz J.L, Dhatt G. Modélisation des structures par éléments finis. Vol. 2 Poutres et Plaques, 1990 and Vol 3. Coques, 1992, Hermés Science Publications. France.
[8] https://www.autodesk.com/