International Journal of GEOMATE, Jan., 2018, Vol.14, Issue 41, pp.13-19
Geotec., Const. Mat. & Env., DOI: https://doi.org/10.21660/2018.41.18791
ISSN: 2186-2982 (Print), 2186-2990 (Online), Japan

FUEL TANK FOUNDATION IMPROVEMENT SYSTEM ON SOFT SOIL LAYER BASED ON 3D NUMERICAL SIMULATION

Adhitya Yoga Purnama[1], *Ahmad Rifa'i[1] and Hary Christady Hardiyatmo[1]

[1]Faculty of Engineering, Universitas Gadjah Mada, Indonesia

*Corresponding Author, Received: 9 Aug. 2017, Revised: 16 Sept. 2017, Accepted: 20 Oct. 2017

ABSTRACT: Differential settlement on the fuel tank storage foundation in Banjarmasin fuel depot, Indonesia, cause the tilting of fuel tank storage. Due to this condition, the volume measurement of fuel in the tank becomes inaccurate and difficult to be predicted. This problem affects the fuel distribution in Banjarmasin area if not immediately solved. The structure of tank foundation was built on the soft soil without another support structure. This research carried out the study on improvement of foundation structure system with concrete slab supported by piles on the outside around and "Sistem Cakar Ayam" that spread evenly in the area of the foundation. The numerical study of the proposed improvement of structure model was conducted using SAP2000 and ABAQUS. The vertical and horizontal spring elements were chosen as the support system on the foundation model in SAP2000. In ABAQUS, the model defined as a three-dimensional model. The soils idealized by elasto-plastic Mohr-Coulomb model. The slab, piles, and "Sistem Cakar Ayam" defined as elastic material. Based on the numerical analysis results, the analysis using SAP2000 are used to determine the optimum configuration of the pile and there is no excessive differential settlement. A three-dimensional numerical analysis using ABAQUS can represent actual field conditions because the boundary conditions in two-dimensional analysis can be reduced. Reduction of deformations using the proposed system is 70.79%. The proposed foundation system using slab supported by piles and "Sistem Cakar Ayam" are quite effective in reducing the differential settlement due to fuel tank load.

Keywords: Numerical analysis, Fuel tank foundation, Improvement system, Differential settlement

1. INTRODUCTION

Terminal Bahan Bakar Minyak (TBBM) Banjarmasin is a fuel storage depot for distribution of fuel in Banjarmasin and its surrounding area. The fuel is stored temporarily in a tank with a certain capacity. However, there is a settlement that occurred in the fuel tank foundation and it will destabilize the fuel tank. The settlement in the fuel tank foundation is differential settlements that cause some of the tanks become tilt as shown in Fig. 1. The slope of the fuel tank foundation causes the volume measurement of fuel in the tank becomes inaccurate and difficult to be predicted. It is highly affects the distribution of fuel in Banjarmasin area.

Most tanks that occurred the differential settlement is a tank that supported by a shallow foundation using concrete slab foundation. The foundation was built on soft soil that is thick enough so that the possibility of soil settlement due to the consolidation process becomes very high. In order to support the fuel tank, the concrete slab should have a good performance to reduce the settlement. The proposed design to improve the performance of the fuel tank foundation are using concrete slab that supported by pile in outside around the slab and "Sistem Cakar Ayam" foundation that spread in the area of the foundation.

Fig. 1 Settlement of existing fuel tank foundation

The proposed improvement foundation design is selected based on the limited area of construction that not possible to move the tank or doing pile installation with heavy equipment in the middle of the fuel tank foundation. Therefore, the behavior of this structural system needs to be investigated to determine the performance of proposed foundation in order to solve the problem of the differential settlement at the fuel tank foundation in TBBM Banjarmasin.

Reference [1] conducted a research of raft foundation system model on the ABAQUS program. The model assumed relies on rigid layer, and the vertically boundary condition on the left and right that are assumed as rollers to allow vertical movement. Pile cap was connected rigidly to the concrete slab. In addition, as a reference for the behavior of pile-raft foundation, the behavior of raft foundation also analyzed. Soft clay, stiff clay and subgrade layer are modeled by Mohr-Coulomb model. Groundwater table is located on the surface of the clay in which assumed that there is a hydrostatic water pressure distribution. Thus, the consolidation effect is negligible. Plate foundations and pile are modeled as isotropic elastic element. Slip model used to describe the behavior of the contact (interface) between the pile and soil. Three-dimensional FEM model validation was examined by a comparison of the analyzed results with the field measurement for vertically loaded pile.

Table 1 Comparison of the results [1]

Result		Settlement (mm)
Field measurement		124
stiff clay	slip analysis	114
	no-slip analysis	92
soft clay	slip analysis	743*
	no-slip analysis	640*
*maximum value		

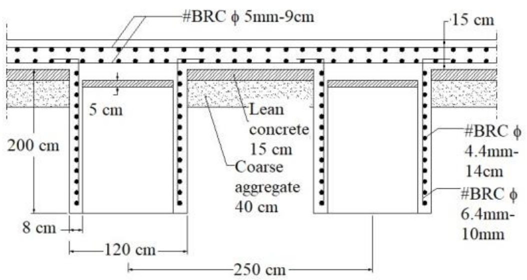


Fig. 2 “Sistem Cakar Ayam” foundation

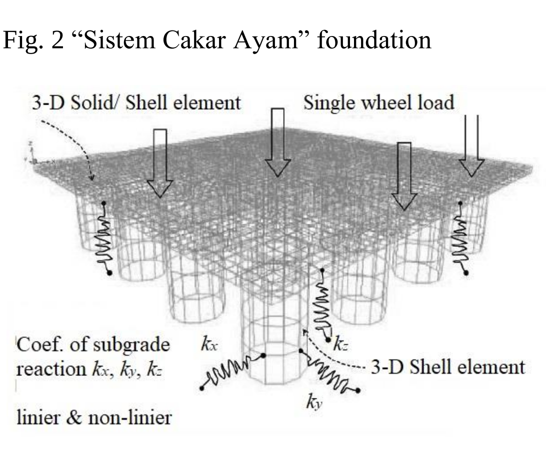


Fig. 3 Three dimensional finite element model [3]

Based on Table 1, the result of three dimensional Finite Element Method (3D FEM) analysis are smaller than the field measurement results. However, there was reasonably good agreement between the result of 3D FEM model with an interface and that from the measured. For a soft clay on the piled raft, the settlement of the raft was larger than that of a Frankfurt clay (stiff clay), and the non-slip analysis was also higher than the value from the slip analysis.

Based on physical laboratory test [2], the behavior of the “Sistem Cakar Ayam” foundation and the results have been outlined in the analytical formula that practically can make a real contribution in the preliminary design of “Sistem Cakar Ayam” foundation. The geometry of “Sistem Cakar Ayam” foundation that shown in Fig. 2. Numerically, after various trials, modification, and verification, adequate finite element modeling of pavement systems using “Sistem Cakar Ayam” foundation has been found [3]. Finite element model for ¼ “Cakar Ayam” section with 16 pipe that shown in Fig. 3. The analysis is based on the Beam on Elastic Foundation theory or BoEF.

2. METHODS

According to [4], in the three-dimensional finite element analysis, solid element shape and structure can be modeled without geometric simplification. Finite element model of this type has the advantage in terms of performance. Geometry assumption, constitutive model and load are necessary to reduce the dimension problem, such as planar or axisymmetric behavior. The boundary conditions on the force and displacement can be more realistically modeled. Another interesting feature is that the finite element meshing looks like a physical system. SAP2000 is used to analyze the foundation improvement system in three dimensional approach. The improvement structure modeled with the fuel loading of 2500 kL. Some idealization and assumptions in numerical analysis, such as, the whole pile and “Sistem Cakar Ayam” divided into small mesh about half a meter in thickness (0.5 m). Joint between each mesh is a joint point for input the coefficient of subgrade soil both horizontally and vertically. The value of spring stiffness horizontal direction (*Sh*) is multiplied between the coefficients of subgrade soil horizontal direction (*kh*) to the width of the mesh that are contacted directly by the soil in the horizontal direction [5]. It is also applied to the value of the soil spring stiffness in vertical direction. Based on appropriate investigation results, it can be obtained coefficient subgrade horizontal and vertical directions coefficient subgrade.

Reference [6] shows that data limitation available problem and the uncertainty of soil condition was solved by the proposed Vesic's empirical equations that shown in Eq. (1).

$$k_v = k_s = \frac{0.65E_s}{B(1-\mu^2)} \sqrt[12]{\frac{E_s B^4}{E_p I_p}} \quad (1)$$

Based on Eq. (1), the modulus of sub grade reaction (k_v) depends not only on the width of the foundation, B, but also on the elastic parameters of soils, *Es* and μ, Elasticity of pile, E_p and shape factor of the foundation, *Ip*.

Alternative foundation system analysis also can be used by ABAQUS. This program consists of Standard and Explicit analysis. Standard analysis is used to resolve the case of the general finite element analysis, whereas Explicit analysis is used for dynamic analysis that more complex [7]. The three-dimensional analysis model was done with the fuel tank capacity of 2500 kL. Soil modeled as elasto-plastic material based on Mohr-Coulomb model, while the concrete structure is modeled as an isotropic elastic material. Fuel tank load that applied on the foundation is assumed to be uniformly distributed load working on the concrete slab. Structural modeling foundation with ABAQUS program is divided into two phases which geostatic and loading phase. In phase geostatic analysis to determine the initial stress that occurs in the soil. After the initial stress is obtained, followed by analysis using a loading phase by adding a load of tanks and carried through the pore water pressure reaches a minimum.

The boundary conditions on the vertical side using a roller nodes fixity so the vertical displacement allowed to occur, while the horizontal displacement is considered there is no movement. At the lower limit, the entire nodal cannot move in vertical and horizontal directions. As for the upper limit, the nodes are not restrained so completely can move vertically and horizontally. Clay soil in this modeling idealized as the of Mohr-Coulomb criteria which have permeability coefficient (k) of 2.079x10^{-4} m/day. Concrete material in the foundation system is modeled according to the finite element meshing procedure.

3. RESULTS AND DISCUSSIONS

3.1 Soil Profile

Field drilling test and SPT test results was discovered in the area of TBBM Banjarmasin. Field test consisted of 5 drill point and SPT test, the BH-1 to the BH-5 is located inside the area TBBM Banjarmasin. From the results of drilling test, the soils are dominated gray clay soil to a depth of 32 m and there is a layer of sand was from the depths of 32 to a depth of 50 m. From the drill test results showed that the soil layer is relatively flat and uniform at every point of testing. The upper layer is generally identified as a very soft clay soil that has very low strength. The stratification of soil layers and SPT based on drill testing are shown in Fig. 4.

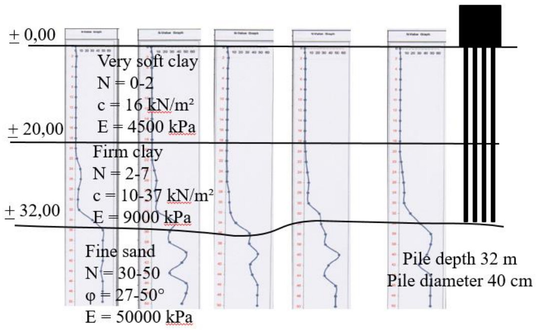


Fig. 4 Soil stratification layer in the fuel tank area

3.2 Data Input

The parameter input is conducted based on the field test and laboratory test. The input parameters used in the modeling are shown in Table 2.

Table 2 Data input

Material type	Elastic		Plastic	
	E (kN/m^2)	v	c (kN/m^2)	ϕ (°)
Very soft clay	4500	0,35	16,00	1,00
Firm clay	9000	0,35	27,92	1,63
Sand	50000	0,30	8,50	44,56
Concrete (K-300)	23452	0,20	-	-
Concrete (K-500)	30277	0,20	-	-

Loading input is calculated based on the maximum capacity of 2500 kL tank that applied to the foundation structure. This tank is one of the tank that suffered severe enough settlement. Loading input values consider the applied loading for maximum capacity of fuel, while the weight of structure is calculated by the software. Loading input values shown in Table 3.

Table 3 Loading input parameter

Capacity	2500	m^3
Unit weight of fuel	0,80	ton/m^3
Force	19620	kN
Diameter of slab	18,00	m
Area	254,47	m^2
Load pressure	77,10	kN/m^2

3.3 Numerical Analysis Results of Existing Foundation

The results in the analysis using the existing foundation of 3D ABAQUS is shown in Fig. 5. In the results of ABAQUS model of the existing fuel tank foundation, the maximum settlement of 2500 kL tank capacity at the beginning of loading is 334.60 mm and increased during the 2495-day period of consolidation become 462.50 mm. Based on this results, it is not advisable to use the existing foundation system to hold the fuel load.

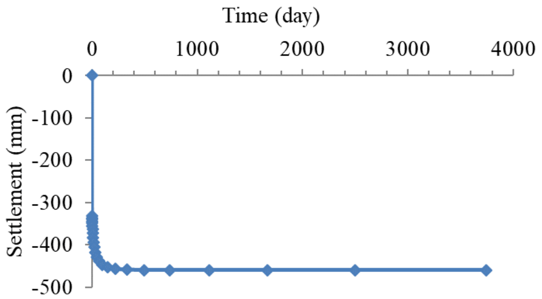


Fig. 5 Analysis results of existing foundation maximum settlement

Based on the topography survey results on the settlement of the foundation slab is 14 cm on the edge of foundation slab. In ABAQUS modeling the foundation settlement on the edge of the slab is about 15-25 cm. The results of the existing foundation structure modeling as valid by the field conditions so that the modeling parameters can be used for further modeling.

3.4 Numerical Analysis Results of Proposed Improvement Foundation

The structure of the fuel tank that indicated the settlement is the fuel tank with a capacity of 2500 kL with foundation slab diameter is 17 m. To accommodate the retrofitting system, an addition piles outside the tank needed, it would require the addition of a diameter of 1 m so that the final diameter used in planning is 18 m. Design improvements to the structure of the oil tank capacity of 2500 kL is the structure of the circular slab with diameter of 18 m which is supported by the arrangement of the 16 piles on the outside which are all driven as deep as 32 m from the original ground level. It is also supported by the structure of the “Sistem Cakar Ayam” as many as 19 units and depth of 2 m. Piles used is spun pile type A1 with a diameter of 45 cm. Diameter of “Sistem Cakar Ayam” is 1.2 m from concrete material. Foundation system configuration is shown in Fig. 6.

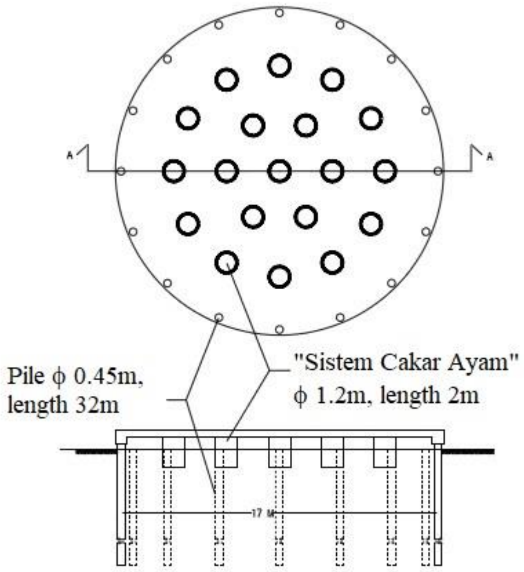


Fig. 6 Proposed foundation system configuration

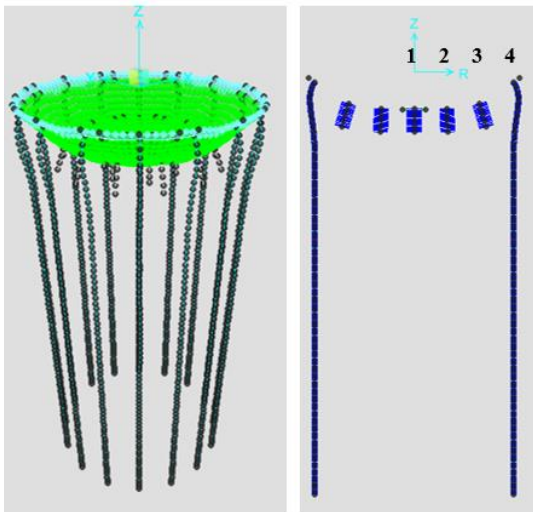


Fig. 7 Results of analysis using SAP2000 for a tank capacity of 2500 kL

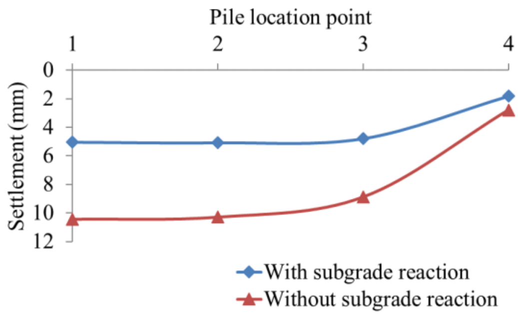


Fig. 8 Settlement of the improved foundation system in radial direction

Fig. 7 shown the results of the structural system improvement analysis with capacity of 2500 kL using SAP2000 are settlement in the foundation due to load on top of the tank foundation. The performance of the structural foundation is reviewed by the largest deformation of the structure taken in the direction of radial direction. Based on the results the indication of the placement of “Sistem Cakar Ayam” that scattered in the area of the foundation provides behavioral uniform settlement in the middle area of the foundation. A uniform settlement in the middle area of the foundation can reduce the possibility of cracking in the slab foundation.

Fig. 8 shows the settlement in the fuel tank slab foundation for each point placement of concrete foundation and piles radially. Analysis of improvement foundation using SAP2000. Based on Fig. 8, indicating that the placement of concrete foundation is scattered in the central part of the foundation provides uniform displacement in the middle area of the foundation. This uniform displacement can reduce the possibility of cracking in the slab foundation.

Numerical analysis of improvement foundation system using two conditions, that are when there is an interaction between slab and soil (k_v = 10000 kN/m^2) and the conditions there is no soil support on the plate (k_v = 0 kN/m^2). The conditions when there is an interaction between the slab and soil is relatively in short-term at the beginning of loading occurs. In this condition the soil have not experienced plastic conditions so that soil still has a strength capacity that is modeled as k_v value 10000 kN/m^2. The other conditions were when there is no soil support on the slab or in long-term condition that the ground has become plastic and it has no interaction with the slab. Soil and slab that do not have interaction is assumed to have a value of k_v = 0 kN/m^2.

The three dimensional analysis using ABAQUS also conducted to analyze the behavior of soil more clearly. The model assumed relies on rigid layer, and the vertical boundary condition on the left and right are assumed as rollers to allow movement down the soil. The size of the full-scale model dimensions created with the dimensions of the ground part of 30 m x 30 m with a depth of 50 m. The mesh element of three dimensional analysis using ABAQUS was shown in Fig. 9. The results of numerical analysis using ABAQUS in the initial phase of loading obtained the maximum settlement of 106.70 mm in the central part of the foundation. In the end of consolidation period, the maximum settlement is 135.10 mm as shown in Fig. 10. During the loading, soil pore water pressure will change. The pore water pressure changes due to load in the initial phase of loading tanks can be seen in Fig. 11a. Changes in pore water pressure due to the load tank and can also be seen in Fig. 11b. The time needed until the consolidation process ends is 513 days.

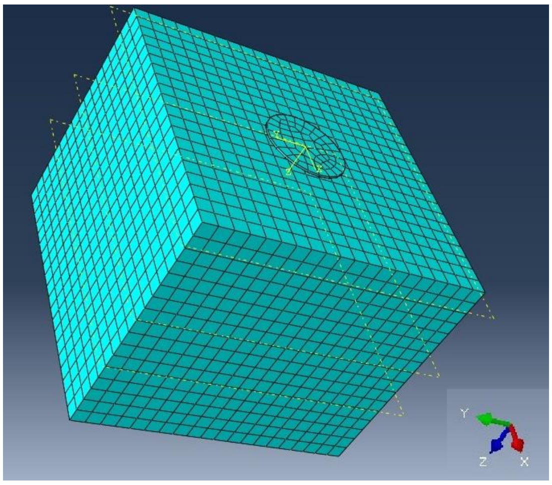


Fig. 9 Mesh module in improvement foundation system analysis

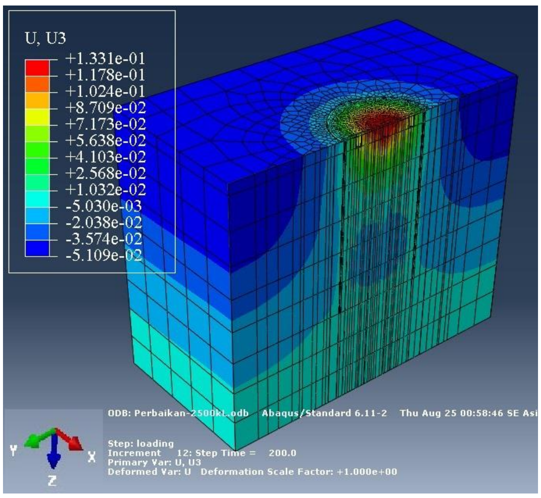
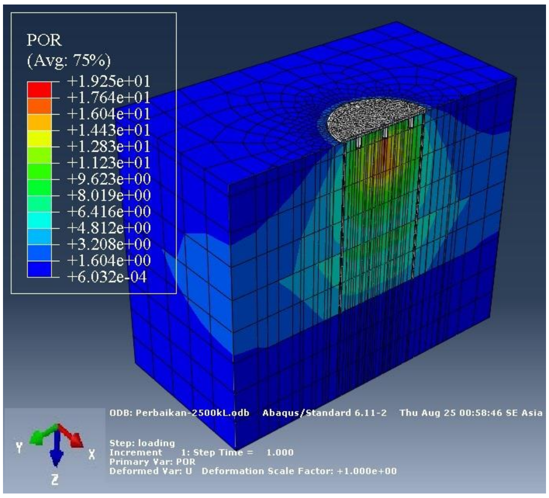
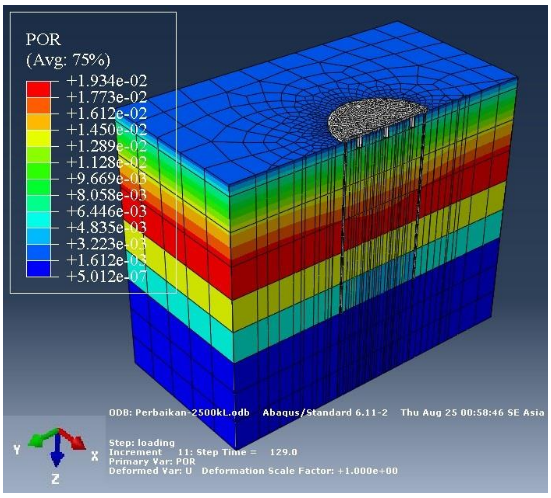


Fig. 10 Displacement of improvement foundation system analysis results in ABAQUS



(a)



(b)

Fig. 11 Excess pore pressure changes during loading on improvement system; a) initial phase of loading b) final phase of consolidation

3.5 Comparison of Numerical Analysis Results

The results of numerical analysis using SAP2000 gives good behavior for piling structure and “Cakar Ayam” foundation system, but the settlement value is very small. This is because of the settlement is the immediate settlement after the load is applied so it is not representing the behavior of soils in field conditions. Modeling using SAP2000 is very effective to do a trial and error of pile configuration because it does not require a complex model so that the analysis to determine the optimum configuration of foundation will be faster.

Three-dimensional numerical analysis using ABAQUS can represent the actual field conditions due to the constraints in a two-dimensional analysis that can be reduced. In addition to the modeling of ABAQUS, soil behavior and the foundation structure can be modeled more realistic. The results of numerical analysis using ABAQUS 3D delivers maximum settlement of 106 mm in the beginning of loading and 135 mm after the consolidation period ends. The settlement in the foundation slab occurs uniformly at the center of the foundation because of the “Sistem Cakar Ayam” foundation behaves as the slab stiffeners. The change of settlement during consolidation process is also quite small at 29 mm for 513 days. Behavior of the proposed improvement foundation system is appropriate as expected, but the value of the maximum settlement is quite large and exceeds the maximum limit of the settlement in a foundation.

Based on the results of numerical analysis, improvement foundation system using slab supported by piles and “Cakar Ayam” is quite effective in reducing the differential settlement due to fuel load. The slope average on each pile and the “Cakar Ayam” are 0.09% (SAP2000) and 0.16% (ABAQUS 3D). The differentials in settlement of the fuel tank foundation are become insignificant. Besides the influence of consolidation settlement is also very small and consolidation also happened in a long time about 1-2 years for 3D modeling using ABAQUS. The settlement of the foundation system after it is repaired using the proposed system can be reduced by 70.79%.

4. CONCLUSIONS

The results of numerical analysis using SAP2000 gives good behavior for the structure of piles and “Sistem Cakar Ayam”, but the settlement results are the immediate settlement after the applied load, so it cannot represent the behavior of soils in actual field conditions. Modeling of pile foundation structure using SAP2000 can be used as a preliminary analysis to determine the configuration of piles and “Sistem Cakar Ayam” that give a good foundation behavior.

Numerical analysis using ABAQUS can represent actual field conditions because it can model the behavior of soil and foundation structure in three dimensional models, so that the restrictions contained in the two-dimensional analysis can be reduced. Numerical analysis using ABAQUS is not suitable to simulate a model that has a lot of elements, which are required the high computing power, for calculations process. The results of numerical analysis using ABAQUS 3D delivers maximum settlement value of 106 mm in the beginning of loading and 135 mm after 513 days during the consolidation period ends. The maximum settlement after the foundation of the system is repaired using the proposed system can be reduced by 70.79% compared with the existing foundation system.

The consolidation settlement is very high and it can lead to structural collapse, so it should be analyzed by relating to the maximum load which can be retained by the foundation system configuration. Proposed foundation system can be applied to support the weight of the fuel tank but the subgrade soil properties should be improved to increase the strength of the soil so the foundation system can withstand the load optimally. There is a need of convergence in meshing element dimension of finite element modeling as to obtain the value of an appropriate stress. The further research using similar modeling methods can be done to simulate the different foundation configuration systems.

5. ACKNOWLEDGEMENTS

The authors express their gratitude to Marketing Operation Region VI – Fuel Marketing & Distribution, Marketing & Trading Directorate, PT. Pertamina Persero for the facilities during this research period. The authors also would like to thanks to colleagues of Department of Civil and Environmental Engineering, UGM for their helpful and fruitful discussion and spirit in conducting field and laboratory tests.

6. REFERENCES

[1] Jeong S, Cho J,” Three Dimensional Analysis of Piled Raft Foundation in Clay Soils”, International Journal of Geo-Engineering, 2012, pp. 11-22.

[2] Hardiyatmo HC, Suhendro B, Hutagamissufardal, Susanto HA, ”Perilaku Fondasi Cakar Ayam pada Model di Laboratorium-Kontribusi untuk Perancangan”, Proceding Seminar Nasional Geoteknik. Yogyakarta, Indonesia, 1999.

[3] Suhendro B,” Pemodelan Elemen Hingga dan Studi Eksperimental Perilaku Struktural Sistem Perkerasan Cakar Ayam di Bandara Soekarno-Hatta”, Proceding Seminar Nasional Metode Elemen Hingga, Bandung, Indonesia, 1999.

[4] Cook RD, Malkus DS and Plesha ME,” Concepts and Application of Finite Element Methods”, 3rd ed., New York, Wiley,1989.

[5] Rahman T,” Evaluasi Kapasitas Dukung dan Nilai PCN Runway Utara Sistem Cakar Ayam Bandar Udara Soekarno-Hatta dengan Pemodelan Elemen Hingga”, Thesis, Yogyakarta, Indonesia, 2015.

[6] Shukla SJ, Desai AK and Solanki, CH, “Behavioral Study of Dynamic Soil Structure Interaction for Piled Raft Foundation with Variable Sub Soils by Time History FEM Model”, Vol. 8, Int. J. of GEOMATE, 2015, pp. 1288-1292.

[7] Khennane A,” Finite Element Analysis Using MATLAB and Abaqus”, CRC Press, 2013.