Interactive 3-D Analysis Method of Piled Raft Foundation for High-rise Buildings

Méthode d’analyse 3-D interactive de fondations mixte radier pieux pour immeubles de grande hauteur

Jeong S.J., Cho J.
Department of Civil and Environmental Engineering, Yonsei University, Seoul, Korea

ABSTRACT: The settlement behavior of piled raft foundations were investigated by a numerical analysis and field case studies. Special attention is given to the improved analytical method (YSPR) and interactive analysis proposed by considering raft flexibility and soil nonlinearity. The proposed method has been verified by comparing the results with other numerical methods and field case studies on piled raft. In the interactive analysis of super- and sub-structure, the loading conditions of super-structure, stiffness of sub-structure and soil conditions were considered. Through comparative studies, it is found that the proposed method in present study is in good agreement with general trend observed by field measurements and, thus, represents a significant improvement in the prediction of settlement behavior.

RÉSUMÉ : La stabilité des fondations mixtes de type radier pieux, d’un cas d’étude, est analysée numériquement. Une attention particulière est portée à l’amélioration de la méthode analytique (YSPR) et à l’analyse interactive proposée en considérant la flexibilité du radier et la non linéarité du sol. La méthode proposée a été vérifiée en comparant les résultats obtenus à d’autres récupérés sur des cas d’étude. Avec l’analyse interactive de la super structure et de la sous structure, les conditions de chargement, la rigidité et les états du sol ont été considérés. En comparant les études, nous montrons que la méthode utilisée est en bon accord avec les observations de terrain. Elle représente donc une amélioration significative par rapport au risque de tassements prévisionnels.

KEYWORDS: High-rise buildings, Piled raft foundation, Interactive analysis, computer based analytical method

1 INTRODUCTION

In South Korea, a number of huge construction projects involving high-rise buildings are being undertaken and the application of piled rafts for high-rise building is becoming an important issue in foundation design. The behavior of a structure is affected by the 3D interaction between the soil, super- and sub-structure(piled raft foundation). Therefore, a proper analytical model is needed to evaluate these interactions.

Numerical methods, which are approximate, have been developed widely in the last two decades because numerical methods are less costly and may be used to consider many kinds of different soil and foundation geometries compared to field and model tests. There are three broad classes of numerical analysis methods: (1) simplified calculation methods (Poulos, 1994; Randolph, 1983) (2) approximate computer-based methods (Clancy and Randolph, 1993; Russo, 1998) and (3) more rigorous computer-based methods (Katzenbach et al., 1998; Lee et al., 2010). Poulos(2001) noted that the most feasible method of analysis was the three-dimensional linear/nonlinear FE method.

The overall objective of this study is focused on the application of interactive analysis method for predicting behavior of super- and sub-structures. An analytical method is proposed for the interaction between the superstructure and the piled raft. In this study, a numerical method is used to combine the pile stiffness with the stiffness of the raft, in which the flexible raft is modeled as flat shell element and the piles as beam-column element, and the soil is treated as linear and nonlinear springs. Based on the proposed analysis method, approximate analysis computer program of YSPR for raft and piled raft foundations are developed respectively. In order to examine the validity of YSPR, the analysis results are compared with the available solutions from previous researches. In the field case study, comparative analyses between YSPR and a three-dimensional finite element analysis are carried out for the settlement behavior.

2 APPROXIMATE ANALYTICAL METHOD FOR PILED RAFT FOUNDATIONS

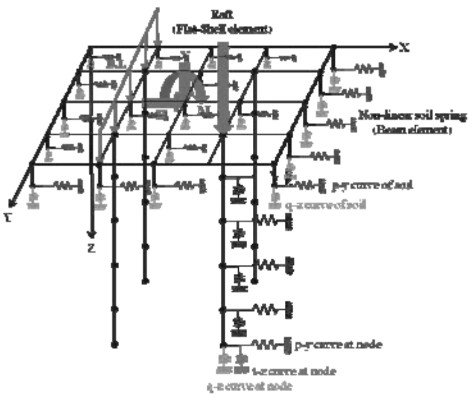
2.1 Modeling of piled raft foundation

A raft was treated as a 4 node flat shell element including torsional degrees of freedom. An improved four-node flat shell element, which combines a Mindlin’s plate element and a membrane element with torsional degrees of freedom as shown in figure 1, is adopted in this study. This element having six degrees of freedom per node permits an easy connection with other elements such as beams or folded elements.

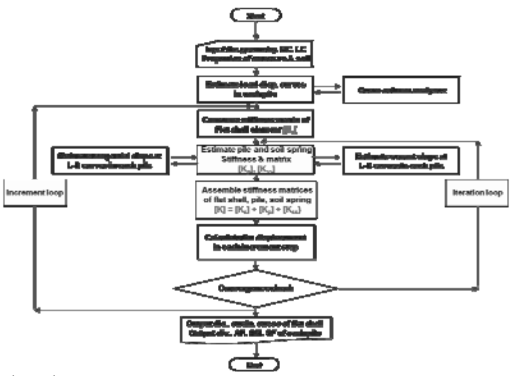
A pile was modeled as a beam-column element in this proposed method. The behavior of individual piles in a group was controlled by using stiffness matrices of pile heads, and the soil-structure interaction between soil and pile was modeled by using nonlinear load-transfer curves (t-z, q-z and p-y curves).

As a final outcome, a numerical method was developed to analyze the response of a raft and a piled raft considering pile-soil-pile and raft-soil interactions. In the present method, the raft was simulated with four-node flat shell elements, the piles with beam-column elements, and the soil with nonlinear load transfer curves (figure 1(a)).

A nonlinear analysis algorithm was proposed using a mixed incremental and iterative technique in this study. The stiffness matrix of piles, soil and that of a flat shell element are combined and a coupled analysis method of a piled raft is developed including a nonlinear analysis algorithm. Figure 1(b) shows the flow chart of the present method (YSPR).



(a) Modeling of a piled raft



(b) Flowchart

Figure 1. YSPR(Yonsei Piled Raft)

2.2 Comparison with case history

The validity of the proposed method was tested by comparing the results from the present approach with some of the case history.

As shown in Fig. 2, preliminary design case of piled raft (OO super tower) conducted at high-rise building construction sites in Korea were representatively selected for the design application. The construction site is comprised mainly of normally banded gneiss, brecciated gneiss and fault core zones. Based on the results of pressure meter, Goodman Jack and plate load tests carried out in the field, a non-linear elastic modulus design line is established to represent the stiffness of the ground.

A schematic diagram of a raft foundation with piles is shown in Fig. 2. This structure consists of a raft, and 112 of ground strengthen piles. The piles have an embedded length of 30 m, a diameter of 1.0 m. A large raft size 71.7x71.7m with a thickness of 6.0 m is resting on a banded gneiss. The raft and ground strengthen piles, with a Young's modulus of 30GPa and 28GPa respectively, is subjected to a vertical load (P_{total}=6,701MN). Table 1 summarizes the material properties used in the case studies.

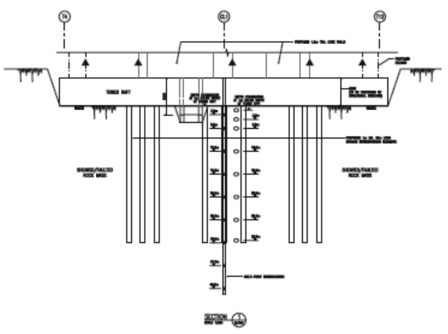
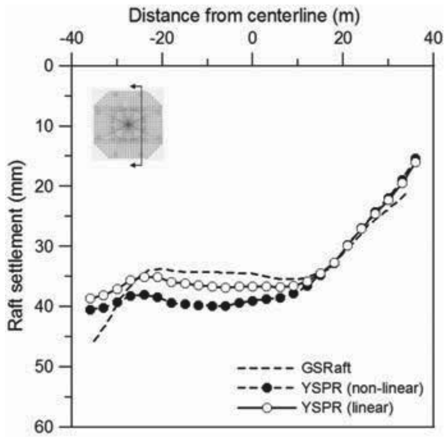


Figure 2. Preliminary design case

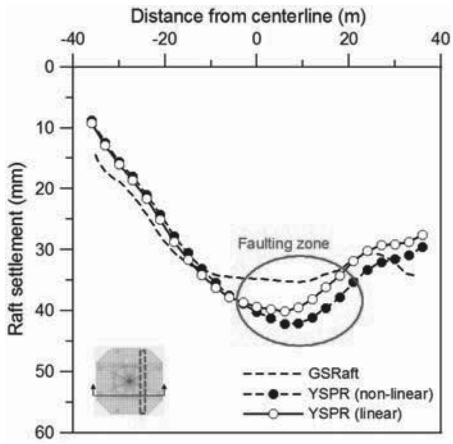
Table 1. Material propertiess

	Type	Depth(m)	E (MPa)	ν
Pile	Concrete	0~-30	28,000	0.2
Raft	Concrete	0~6.0	33,234	0.15
Soil	Gneiss	Soil spring stiffness (kPa/m)		
		0~204,250		

Fig. 3(a) and (b) shows the raft settlement at different section predicted by GSRaft and YSPR. Agreement between the GSRaft and YSPR of settlement is generally good; however there is a slight difference in prediction of settlement in the faulting zone which the sudden drop of the magnitudes were occurred. This can be attributed to the appropriate assumption of material properties due to no accurate ground investigation data on this section. As shown in this result, the prediction by the proposed method has a considerably larger settlement than the settlement calculated by the existing solution. This is because the existing method ignores lateral displacement due to membrane action of flexible raft and, thus, overestimates the lateral stiffness of raft and small displacement in raft lateral behavior. Although there are no measured profiles of raft settlement, the proposed analysis method showed reasonably good correspondence with well-known in-house program.



(a) Section1



(b) Section2

Figure 3. Raft settlement distribution

3 INTERACTIVE ANALYSIS OF SUPER- AND SUB-STRUCTURE

3.1 General

The unified analysis of piled foundation in long span bridges and buildings has become an important issue in structure design. There has been much work done on pile-soil interaction. A relatively less work has been done on the unified analysis that includes both piled foundation and superstructures. Because the mechanism of load transfer between piled foundation and superstructures involves a complex interaction between structures, raft, piles, and surrounding soil. The analysis process is affected by many factors such as: column and wall geometry, design load, raft and pile group geometry, soil properties, and interaction between different structure elements. Accordingly, there are currently no methods available to predict behavior of the entire structures due to the difficulty and uncertainty in quantifying these factors.

Therefore, the overall objective of this study focuses on application of interactive analysis method for predicting behavior of entire structures. A series of numerical analyses was performed to verify the interactive analysis routine in comparison to the unified analysis method. For this purpose, the three-dimensional (3D) Finite-Element (FE) analysis has been carried out. For the unified analysis of super and sub-structures, the numerical analyses were performed via the FE code PLAXIS 3D foundation, with column, floor and piled raft foundation systems placed on a typical weathered soil in Korea.

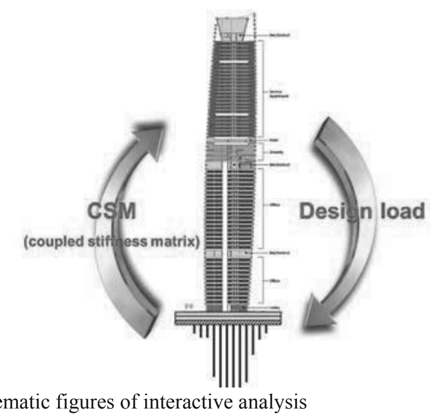
3.2 Interactive analysis

In most of the design field, the analysis of super-structure was conducted without modeling the foundation system and the foundation was designed without considering the rigidity of super-structure. It may result in overestimation of forces, the bending moment, settlement of super- and sub-structure.

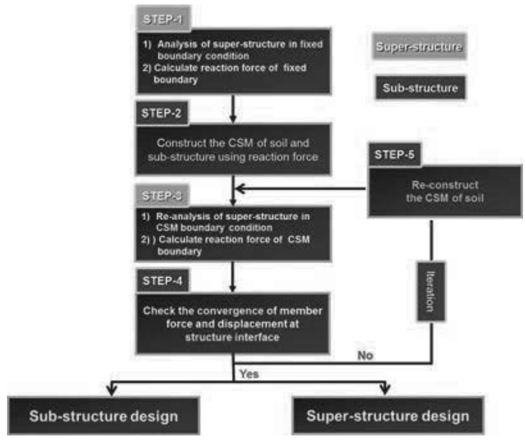
In this study, the interactive analysis procedure is proposed for predicting the behavior of entire structure by considering soil-foundation stiffness and rigidity of super-structure. The procedure of interactive analysis in this study include the following steps:

(a) Compute the reaction force of structure-foundation interface in fixed boundary condition.
(b) Construct the CSM(coupled stiffness matrix) of soil and foundation using reaction force.
(c) Re-compute the reaction force of interface in CSM boundary condition.
(d) Calculate the member force and displacement of foundation.
(e) Check the convergence of member force and displacement at interface node from step (c) and (d).

The procedure described above is iterated until the error between the super- and sub-structure displacements falls within a tolerance limit. Figure 4 shows a schematic diagram and flowchart of the interactive analysis.



(a) Schematic figures of interactive analysis



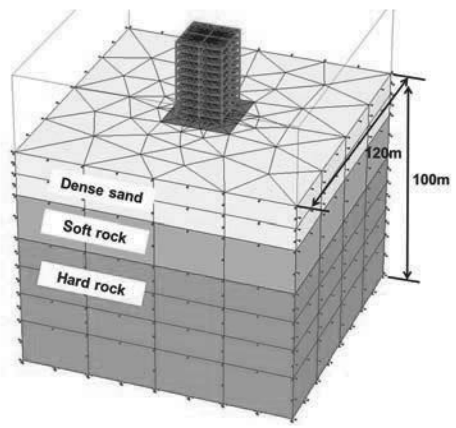
(b) Flowchart of interactive analysis

Figure 4. Interactive analysis between super- and sub-structure

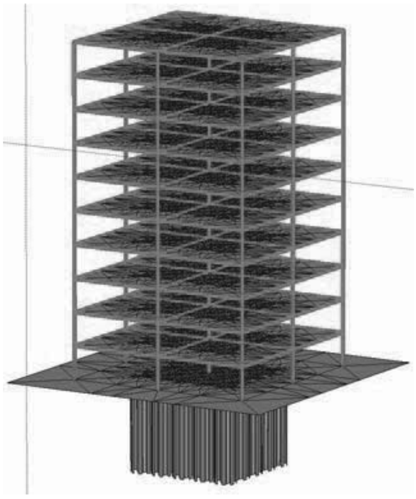
3.3 Verification of proposed method with finite-element analysis

In this section, the validation of the proposed method with numerical analysis is discussed. The three-dimensional finite-element mesh used for the structure-foundation-soil system is shown in Figure 5. The structure consists of columns, 10 floors, a raft, and 25 identical vertical piles, which are spaced by 2.5m (= 2.5D, where D is a pile diameter).

The piles are 1m in diameter and 10m in embedded length. A square raft has a width of 15m with fixed pile head conditions. The columns are 3.5m in length and 0.6m in width. At the left- and right-hand vertical boundaries, lateral displacements were restrained, whereas fixed supports were applied to the bottom boundaries. The specified initial stress distributions should match with a calculation based on the self-weight of the material. After the initial step, the applied loading was simulated by a self-weight of super-structure.



(a) Structure-foundation-soil system



(b) Modeling of structure-piled raft foundation
Figure 5. Typical 3D model for FE analysis

The pile is considered as linear-elastic material at all times, while for the surrounding soil layer the Mohr-Coulomb non-associated flow rule is adopted. The interface element modeled by the bilinear Mohr-Coulomb model is employed to simulate the pile–soil interface. The interface element is treated as a zone of virtual thickness. It behaves as an element with the same material properties as the adjacent soil elements before slip occur. A decreased value of shear modulus is assigned to the interface element when a slip mode occurs in the interface element. The decrease of strength for the interface element is represented by a strength reduction factor R_{inter} in PLAXIS.

Fig. 6 shows the computed settlement of a raft with different analysis methods. It is observed that the unified method using PLAXIS 3D Foundation predicts smaller settlement compared with interactive analysis in fixed and CSM boundary condition. Although, a reasonably good agreement between the unified analysis method and the proposed interactive analysis was obtained for the same loading step, the fixed boundary analyses have a larger displacement than that of the proposed interactive analysis and unified analysis.

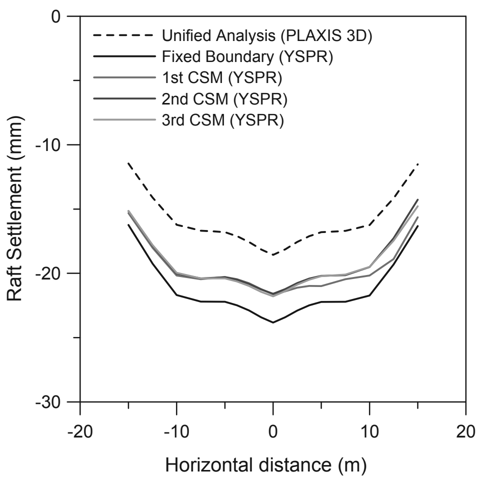


Figure 6. Raft settlement

Fig. 7 shows the computed bending moment of the raft. The figure also demonstrates a good agreement between the interactive analysis and the more rigorous finite element approach. These comparisons suggest that the proposed interactive analysis is fairly capable of predicting the deformation and load distribution of sub-structure.
Figure 7. Raft bending moment distribution

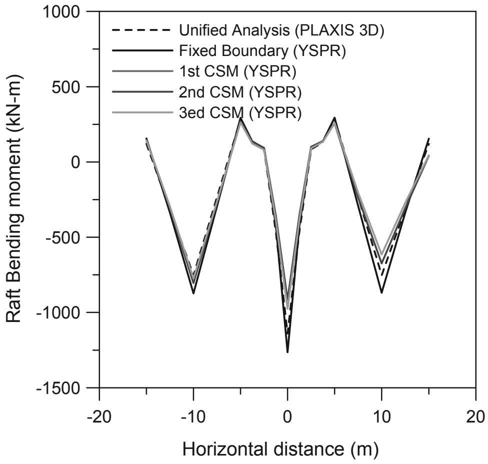


Figure 7. Raft bending moment distribution

4 CONCLUSIONS

The primary objective of this study was to propose improved analytical method and interactive analysis for super- and sub-structure. A series of analytical studies were conducted. Additionally, the analytical method is intermediate in theoretical accuracy between general three-dimensional FE analysis and the conventional numerical method. From the findings of this study, the following conclusions can be drawn:

Proposed analytical method produce a considerably larger settlement of piled raft than the results obtained by the conventional methods (GSRaft). When compared with the results of case histories, the proposed method is shown to be capable of predicting the behavior of a large piled raft. Nonlinear load-transfer curve and flat-shell element can overcome the limitations of existing numerical methods, to some extent, by considering the realistic nonlinear behavior of soil and membrane action of flexible raft. Therefore, the proposed method could be used in the design of large piled rafts for high-rise buildings.

Based on a numerical analysis for the structure-piled raft-soil system, it is found that the CSM boundary condition provide more realistic behavior of piled raft foundation than the result obtained by the fixed boundary condition.

5 REFERENCES

Clancy, P., and Randolph, M. F. (1993): An approximate analysis procedure for piled raft foundations, International Journal for Numerical and Analytical Methods in Geomechanics, 17(12), 849-869.

Katzenbach, R., Arslan, U., Moorman, C. and Reul, O. (1998): Piled raft foundation- Interaction between piles and raft, International Conference on Soil-Structure Interaction in Urban Civil Engineering, Darmstadt, 2(4), 279-296.

Lee, J. H., Kim, Y. H., and Jeong, S. S. (2010): Three-dimensional analysis of bearing behavior of piled raft on soft clay, Computers & Geotechnics, 37, 103-114.

Poulos, H. G. (1994): An approximate numerical analysis of pile-raft interaction, International Journal for Numerical and Analytical Methods in Geomechanics, 18 (2), 73–92.

Poulos, H, G. (2001): Piled raft foundations : design and applications, Geotechnique, 51(2), 95-113.

Randolph, M. F. (1983): Design of piled foundations, Research Report Soils TR143. Cambridge: Cambridge University Engineering Department.

Russo, G. (1998): Numerical analysis of piled rafts, International Journal for Numerical and Analytical Methods in Geomechanics, 22(6), 477-493.