9th International Congress on Civil Engineering, May 8-10, 2012
Isfahan University of Technology (IUT), Isfahan, Iran

Piled Raft Foundation (PRF) Systems Behavior under Earthquake Loading

Mohammad M. Eslami[1], Mohammad M. Ahmadi[2]

1- M. Sc., Structural Engineering, Dept. of Civil Eng., Sharif University of Technology, International Campus, Kish Island, Iran

2- Associate Professor, Dept. of Civil Eng., Sharif University of Technology, Tehran, Iran

<Mandro.es@gmail.com>

Abstract

Behavior of piled rafts foundations as an efficient alternative system for pile groups subjected to earthquake loading has been studied and realized. The raft interaction with subsoil and also superstructure can affect the seismic response and load sharing of involved elements. ABAQUS software was employed based on the Finite Element method and constitutive soil models to investigate soil pile structure interaction (SPSI). For calibration and verification of the procedure of modeling with two other experimental studies, results compared by the analysis of models under single frequency sinusoidal dynamic loads. Afterwards, analyses of the same models were applied under the acceleration time history of the El-Centro earthquake. Results indicate that the participation of the raft interaction with soil under earthquake loading, well improves the seismic response and behavior of a piled foundation system. Reductions of maximum acceleration response, horizontal displacement and bending moment in the piled raft system, are some important achievements in this study. Moreover, more uniform stress distribution in soil deposit and prevention of stress concentration leads to the reduction of the PRF system settlement during earthquakes.

Keywords: Piled Raft Foundations (PRF), FEM, ABAQUS, Earthquake Loading, SPSI

1. INTRODUCTION

For a structure, if a shallow foundation is not adequate, it is common in foundation engineering to design a fully piled foundation in which the entire loads are transferred to the subsoil by piles [1]. In traditional methods of piled foundation design, because of the occurrence of large settlements under the pile cap resulted in the separation of the raft and soil, therefore in the calculations of bearing capacity of foundations only the piles were considered and no emphasis was made on the raft as a load sharing element.

To date, according to the advanced numerical analysis, one can consider the interaction between a raft and the soil in foundation design. Among such design improvements, it is common for a raft to be part of the foundation system. Recently, there has been an increasing recognition that the use of piles to reduce rafts total and differential settlements which can be lea to efficient design. Such a foundation makes use of both the raft and the piles, and is referred to as a piled raft foundation (PRF) [2].

In the design of piled rafts the load sharing between the piles and raft is taken into account. The use of PRF is an effective way of minimizing both total and differential settlements, improving the bearing capacity of a shallow foundation, and effectively reducing the internal stress levels and bending moments within a pile [3].

The vertical load bearing mechanism has been extensively investigated by a number of researchers by applying the elasticity theory [4, 5] and the finite element method [6, 7]. On these results, piled raft foundations are becoming popular in practical use and with this type of foundation design becoming a common procedure, the need of study on its seismic behavior seems necessary. The study on the load bearing mechanism under horizontal loading or during earthquakes, however, is very limited in the literature. Since the behavior of a piled raft foundation during earthquakes is considered fairly complex due to dynamic interaction among a raft, piles and a soil, the design procedure should include the effect of this mechanism in an appropriate manner. The objective of this study is to generalize this behavior under earthquake loading and by three dimensional Finite element modeling in custom engineering scales.

2. NUMERICAL MODELING PROCESS

The main objective of Finite Element modeling via ABAQUS is the investigation of pile-soil interactions under seismic or earthquake loading. In this study effective parameters in the modeling are checked in order to reach a better procedure of analyzing the soil-pile interactions. The main points of this model are as follows [1]: