

3 Dimensions Finite Element Analysis of Tunnel-Pile Interaction Scenarios Using Abaqus Software

Authors : Haitham J. M. Odeh

Abstract : This paper introduced an analysis of the effect of tunneling near pile foundations. Accomplished by three-dimensional finite element modeling. The numerical simulation is conducted using Abaqus finite element software. By examining different Tunnel-pile scenarios. The paper presents the tunnel induced pile responses, Such as pile settlement, pile internal forces, and the comments made on changing the vertical and transversal location of the tunnel related to the piles, the study contains two pile-supported structure cases, single and a group of piles. A comprehensive comparison between real case study results and numerical simulation is presented. The results of the analysis reveal the critical and safe location of tunnel construction and the positive effect of a group of piles existing instead of single piles. Also, demonstrates the changes in pile responses by changing the tunnel location.

Keywords : pile responses, single pile, group of piles, pile-tunnel interaction

Conference Title : ICSGM 2022 : International Conference on Structural and Geotechnical Modelling

Conference Location : Oslo, Norway

Conference Dates : June 23-24, 2022