High-performance Abaqus simulations in soil mechanics

H.M. Hügel, S. Henke, S. Kinzler

Institute for Geotechnics and Construction Management, Hamburg University of Technology, Germany

Abstract: *Abaqus is often applied to solve geomechanical boundary value problems. Several Abaqus built-in features enable a wide range of simulating such problems. For complex problems Abaqus can be extended via user subroutines. Several extensions for soil mechanics purposes are discussed and corresponding case studies are presented.*

Keywords: *Abaqus, soil mechanics, soil plasticity, hypoplasticity, pile penetration analysis, soil compaction analysis, quay wall deformation analysis, quay wall optimization analysis, Abaqus and MATLAB*

1. Introduction

Boundary value problems in soil mechanics are non-linear due to material non-linearity (always), geometrical non-linearity (sometimes) and non-linear boundary conditions for problems with soil-structure-interaction (often). They can imply complex construction processes like penetration of structures into subsoil causing propagation of waves in subsoil and actions on existing structures. Soils are modelled as single-, two- or three-phase materials depending on the existence and volume fractions of soil constituents like soil particles, pore water and pore air. Corresponding mechanical models range from classical continuum mechanics up to theories of mixtures like Theory of Porous Media. Moreover boundary value problems are commonly characterized by a disadvantageous ratio of characteristic lengths of the subsoil section to smallest dimension of structures, which makes three-dimensional models rather complex. Sophisticated computer simulations in soil mechanics need FE-formulations for finite deformations, contact algorithms for finite relative motions and parallelization algorithms. Abaqus offers a wide range of built-in features for soil mechanics purposes and can be extended using several user subroutines.

2. Abaqus built-in features for soil mechanics purposes

A listing of Abaqus built-in features to solve boundary value problems in soil mechanics reads without demand on completeness:

- Several **finite elements** with displacement, pore water pressure, temperature and concentration degrees of freedom can be used to discretize the soil body for one-, two- and three-dimensional stress-deformation, seepage, heat transfer and diffusion problems.