

ABAQUS Modelling: A Weekly Report



Kyeong Sun Kim

Civil and Environmental Engineering

Seoul National University

Sept, 2021

Contents

1	First Meeting	1
1.1	진행상황	1
1.2	하고있는 일들	2

1

First Meeting

Contents

1.1	진행상황	1
1.1.1	Abaqus Modeling: 3D	1
1.1.2	문헌조사	2
1.2	하고있는 일들	2
1.2.1	Paramteric Study	2
1.2.2	Including images	2

1.1 진행상황

1.1.1 Abaqus Modeling: 3D

Problems

- Contact Interface

1. *First Meeting*

- Element distortion
- Partitioning and meshing matter
- Computational difficulty

1.1.2 문헌조사

Interface Elements

- thickness = 25mm

Beam Element for Structure

Pressure Loading

Different Soil Tank Size

1.2 하고 있는 일들

1.2.1 Paramteric Study

contact friction μ

ϕ

Elastic Modulus E

- $E_s = \frac{E}{50}$

Flow Angle

1.2.2 Including images

Code chunks are also used for including images, with `include_graphics` from the `knitr` package, as in [Figure 1.1](#)

1. First Meeting

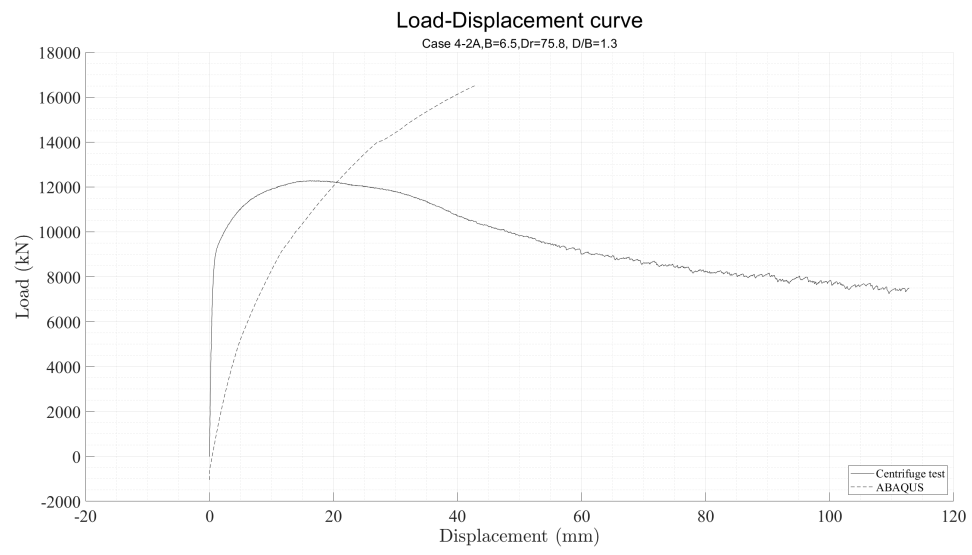


Figure 1.1: figure 4-2A