MASTER'S DISSERTATION AT STRUCTURAL MECHANICS

DEPARTMENT OF CONSTRUCTION SCIENCES | FACULTY OF ENGINEERING LTH | LUND UNIVERSITY

ARGETA JUPOLLI
ar6074ju-s@student.lu.se

PRESENTATION
MAY 2025

REPORT
Will be published as
Report TVSM-5278

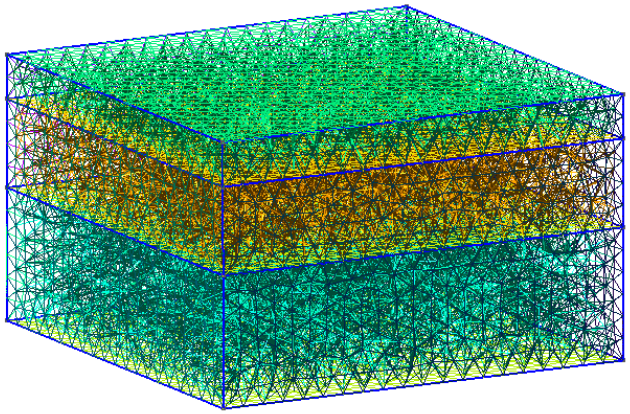
SUPERVISOR
LINUS ANDERSSON *Assistant Professor*
Div. of Structural Mechanics, LTH

ASSISTANT SUPERVISORS
Dr JONAS LINDEMANN
Div. of Structural Mechanics, LTH | Lunarc
KHUONG AN UNG *Doctoral Student*
Div. of Structural Mechanics, LTH

EXAMINER
PETER PERSSON *Associate Professor*
Div. of Structural Mechanics, LTH

THE WORK IS PERFORMED AT
DIVISION OF STRUCTURAL MECHANICS, LTH

MODELING TRAFFIC-INDUCED VIBRATIONS IN ABAQUS USING PYTHON SCRIPTING



BACKGROUND

When traffic-induced vibrations are analyzed using the FE software Abaqus with the purpose of determining its effects on a structure, a large soil domain needs to be modelled in order to properly capture the nature of the vibrations that reach the structure. This results in large FE models that are time consuming for the software to analyze. With the purpose of computational efficiency, models could be created with different sized soil domains and elements in relation to the frequency range that is to be studied, thus creating optimized models. As a finite soil domain is used, an important aspect of the modelling is non-reflective boundaries.

AIM

The aim is to develop a program using Python that will generate ready-to-use optimized Abaqus models for the analyzation of traffic-induced vibrations on a structure. The following objectives will be investigated:

- What vibrational phenomenon arise when a finite soil domain is modelled using a FE software?
- How can these phenomena be managed so that the vibrations behave as they would in an infinite soil domain, i.e. where the soil is represented by a semi-infinite sphere?
- Which conditions are relevant for the modelling, for example soil materials, frequency ranges?
- How much time can be saved in Abaqusanalysis through the use of optimized models?

METHOD

Python will be used to create input files for Abaqus as well as a user interface from which the user can specify the conditions of the analysis. The soil domain as well as the element sizes of the model will be determined by the frequency. Other areas of interests will be modeling of soil stratification and potentially adjusting mesh sizes by proximity to the structure and creating tie constraints to the structure.

DIVISION OF STRUCTURAL MECHANICS
Faculty of Engineering LTH, Lund University, Box 118, SE-221 00 Lund, Sweden
• Tel: + 46 (0)46-222 73 70 • Fax: + 46 (0)46-222 44 20 • www.byggmek.lth.se

xpTVSM-5278 (2025-03)