10th International Congress
of Croatian Society of Mechanics
September, 28-30, 2022
Pula, Croatia

10thICCSM

Adaptation of Abaqus output data for application in seismic analysis of buildings

Romano Jevtić Rundek*, Ante Pilipović*, Mario Uroš*, Marija Demšić*

*Faculty of civil engineering in Zagreb; Fra Andrije Kačića-Miošića 26, 10 000 Zagreb
E-mails: {romano.jevtic.rundek; ante.pilipovic; mario.uros; marija.demsic} @grad.unizg.hr

Keywords: Abaqus, Phyton, Seismic analysis, Masonry

1. Introduction

Abaqus is a quality engineering simulation software that offers FEM, DEM, MEM and other simulation methods to the user. In seismic analysis of structures, it is a valuable tool for performing time-history analyses because it provides complex simulation of material behaviour under dynamic excitation, geometric nonlinearities, and has a very efficient solver. While Abaqus is powerful and can perform a wide range of general-purpose engineering simulations, it also comes at a cost of inefficient workflows for some specific tasks required in the analysis of buildings. In calculating the response of masonry walls and buildings to seismic action using Abaqus time-history analysis, the main problem is the interpretation of the results, as a large number of elements need to provide output data for each time step. That is a large number of walls, consisting of a large number of 3D elements. It is impractical to do this in the native Abaqus GUI.

To analyse the seismic response of a building, the analysis results for each wall need to be compared with the predefined limit states for each time step of the analysis. Two groups of analysis results are used: the internal forces acting on the top and bottom of the wall and the in-plane wall drift. Both datasets can easily be extracted directly from Abaqus through native GUI, however only for one wall and one frame at a time. To analyse the building properly, the forces and drifts of each wall need to be extracted for every step of time-history analysis, which is impractical. This is streamlined using Python scripts, which can go through all time steps, extract the necessary datasets, and even perform a check of wall stability. It needs to be said, the current scripts are made for a model consisting of 3D finite elements, and the purpose of the scripts is extracting the forces and displacements from the model in a timely and clear manner.

Abaqus includes a Python interface. It is easy to log or translate user activity in the GUI into Python commands, and it is also easy to run scripts through the GUI. This allows the user to quickly learn and use Abaqus specific commands.

2. Structure of the scripts and workflow

In the Python code itself, a combination of native Abaqus commands and standard Python commands is used to extract data from Abaqus. In the scripts provided, element cross sections in the model are identified, their time-history outputs requested [1] and extracted into Excel or another common data manipulation software (Mathematica, MatLab etc.). It is necessary to define which outputs are required, this is done through defining SETs. Elements contained in a SET define which points/planes on the model need to be tracked, while the name of the set defines the way the values will be processed, and which values will be requested in the history output (force or displacement or both). This workflow is shown in Fig 1. The provided scripts are also suitable for parametric studies and can be further expanded.

2.1. Benchmarks:

These scripts will be used to perform benchmarks on several walls in Abaqus, calibrate material properties and mesh size. The calibrated values are then used to calculate the response of a full-scale building consisting of these walls, which have now been calibrated to empirical data (Fig 2). The wall in Fig 2 is a 150/100/60 cm stone wall, described by Kržan et al. [2], the church tower in Fig 2 is a Stone tower described by Pojatina et al.[3]. Calibration is tested on the wall, and result export scripts are tested on the tower model.

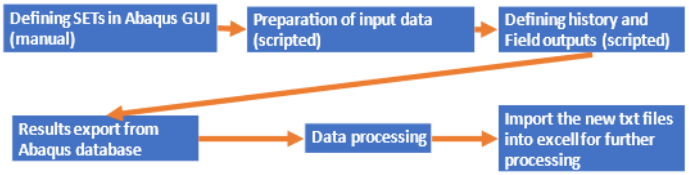

Figure 1: Workflow diagram

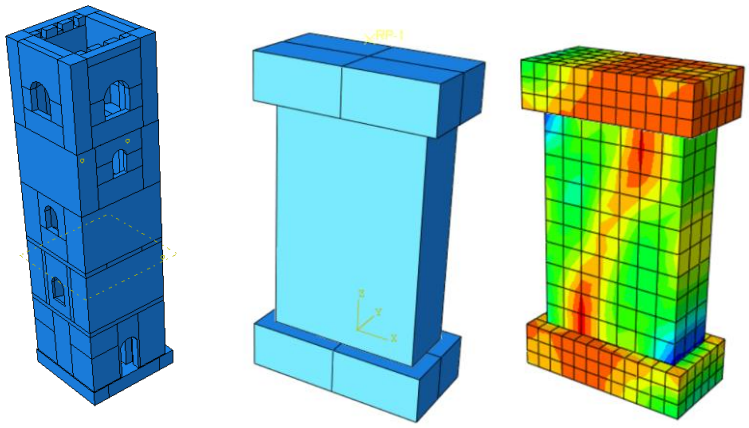

Figure 2: Benchmark of a wall, and a building model

3. Conclusion:

Abaqus provides nonlinear time-history analysis that is superior to most software packages available today, especially in civil engineering. Detailed input data is required to create a quality model, but much better results can be obtained. In this paper inefficiency in data interpretation was addressed, when analyzing a seismic time-history(TH) analysis of a building modeled in 3D finite elements. Obtaining the results and compiling them for interpretation is done through Python scripts. This improves the workflow of performing a time history seismic analysis by a few orders of magnitude compared to stock Abaqus GUI, depending on building size. In their current form, the scripts allow a quick export of data, in a form that allows easy interpretation. In their future form, they could allow somewhat automatic construction of fragility curves for buildings. When such an improvement on the scripts is performed, it will introduce new challenges of checking the result validity, for an example, checking automatically if there is some critical crack forming, or a local collapse mechanism etc… In conclusion, for the purpose of seismic analysis, Abaqus is capable of producing models of greater quality, but simultaneously requires more detailed input data and scripts for result extraction and interpretation.

Acknowledgments

This script is being developed as a part of the 2BESAFE project, we gratefully acknowledge financial support from the Croatian Science Foundation (grant number UIP-2020-02-1128)

Literature:

[1] Abaqus manual

[2] Kržan, M., Gostič, S., Cattari, S., & Bosiljkov, V. (2015). Acquiring reference parameters of masonry for the structural performance analysis of historical buildings. *Bulletin of earthquake engineering*, 13(1), 203-236.

[3] Pojatina, J., Horvat, T., Uroš, M., Baniček, M., Pristup konstrukcijskoj obnovi crkve sv. Mihaela u Gračanima. In: Lakušić, Stjepan, ed., *Zbornik sažetaka predavanja*, pages 43-43. Dani Hrvatske komore inženjera građevinsratva Opatija, 17. – 19. 6. 2021., Zagreb, 2021.