2CroCEE
22 - 24 March 2023

Proceedings of the 2nd Croatian Conference on Earthquake Engineering - 2CroCEE
Zagreb, Croatia - March 22 to 24, 2023

DOI: https://doi.org/10.5592/CO/2CroCEE.2023.140

EXTRACTION AND PROCESSING OF ABAQUS OUTPUT DATA FOR USE IN SEISMIC ANALYSIS

Romano Jevtić Rundek(1), Ante Pilipović (2), Marija Demšić (3) Mario Uroš (4)

(1) *Assistant, Faculty of civil engineering in Zagreb, romano.jevtic.rundek@grad.unizg.hr*
(2) *Assistant, Faculty of civil engineering in Zagreb, ante.pilipovic@grad.unizg.hr*
(3) *Associate professor, Faculty of civil engineering in Zagreb, marija.demsic@grad.unizg.hr*
(4) *Associate professor, Faculty of civil engineering in Zagreb, mario.uros@grad.unizg.hr*

Keywords: Abaqus, Python, Seismic analysis, Time history, FEM, post processing

1. Introduction

Abaqus [1] 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 as it provides complex simulation of material behaviour under dynamic excitation, geometric nonlinearities, and has a 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 seismic 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, so Python [2] scripts are developed to automate this workflow.

2. Structure of the scripts, workflow and output description

The scripts are in essence a sequence of commands that extracts processes and stores data. They can be run in different python environments. In this paper models consisting of 3D volumetric elements are considered. The main function of the scripts is identifying cross sections of PIER, SPANDREL or other elements in a model and outputting relevant forces and displacements. To obtain forces on cross section level, nodal forces are integrated across the cross sections. Cross section displacements are obtained by defining a regression plane for the field of deformed cross section nodes (example of such field output on Fig 1a)). Scripts will also monitor displacements of stories to provide story drift data.

Once analysis is complete, a script extracts field output data from the model (example used in this paper on Fig 1c)), process it into cross section based data and saves it in a database as forces, moments, displacements and deflection angles of cross sections. Then a postprocessing script is used to extract relevant data for PIER or SPANDREL elements, story drift data, etc. This processed data is exported in a text file that is suited for importing into excel, which allows easy further data visualisation. In terms of modifying the model for this scripted output and processing there are 2 options. One is defining each cross section as a set of surfaces before running the analysis. The other is identifying the finite element nodes of each cross section, if definition of sections prior to running the analysis wasn't possible. The second way requires some knowledge of coding in Python.

The scripts for data extraction are run inside Abaqus GUI, while further postprocessing is done using standard python code that can be run by a interface like IDLE. Output data can be customised when running the postprocessing script. It is possible to export specific components of forces of displacements, or select specific postprocessing like only PIER elements or only story drifts etc. Example of processed output data is shown on Fig 1b) and Fig 1d). Displayed are hysteresis of one PIER element in a building, analysed in the Dubrovnik B-17 study [3], for 3 different earthquake loads (analysis) on Fig 1b). On Fig 1d) are extremes of the drift, force and moment of the same PIER.

2CroCEE
22 - 24 March 2023

Proceedings of the 2nd Croatian Conference on Earthquake Engineering - 2CroCEE
Zagreb, Croatia - March 22 to 24, 2023

DOI: https://doi.org/10.5592/CO/2CroCEE.2023.140

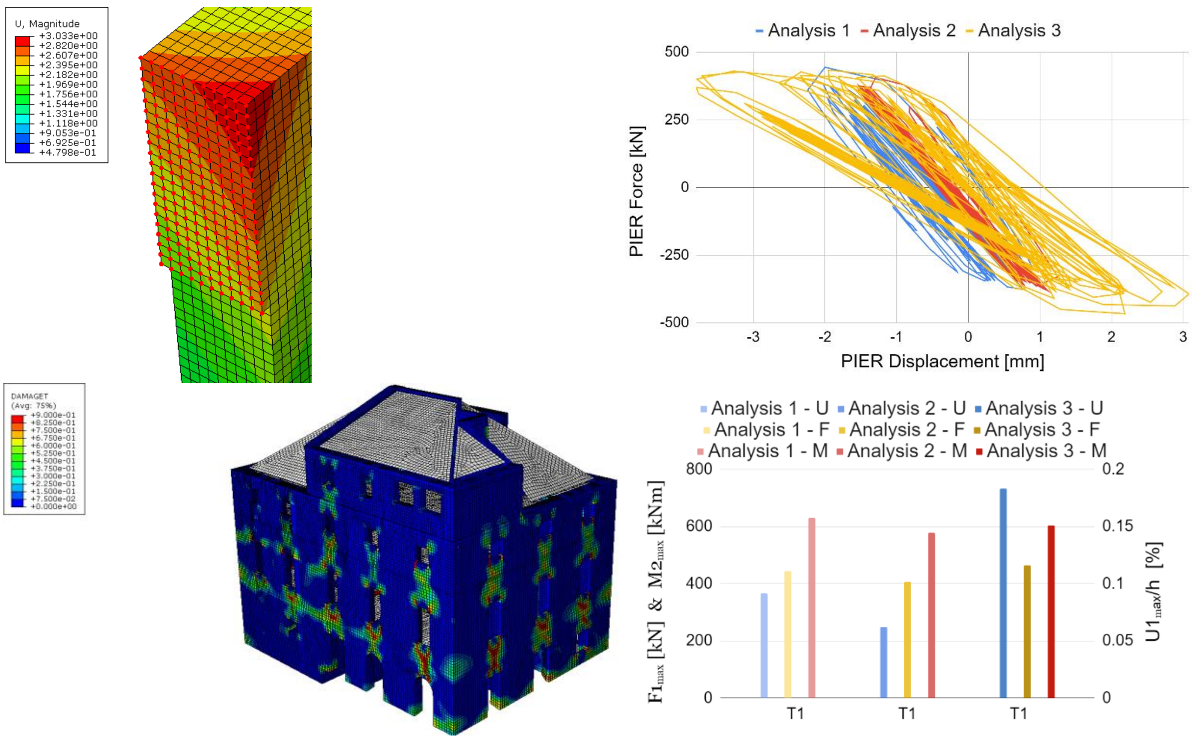

Figure 1. a)[top left] Displacement field on a cross section; b) [top right] Hysteresis of a PIER section in 3 different analysis; c)[bottom-left] FEM model [3]; d)[bottom-right] Maximum values of Force, Moment and drift of a PIER element for 3 different analysis

3. Conclusion

Abaqus data postprocessing presents a better insight into vast quantities of data and enables interpretation of more complex analysis or larger numbers of analysis. Good compiling of data for interpretation cannot be overstated as it is crucial for more extensive application of complex models to real world buildings. While the scripts in their current form offer good insight, there is still a significant issue with interpreting the results. Challenges presented here include estimating cross section based values from fields of values, shown on Fig 1a), identifying damage states of structural elements from time history data show on Fig 1b) and identifying structures health (for use as engineering demand parameter) from data shown in Fig 1d). Estimating cross section based values from a field output is currently done by a regression plane, with the assumption that cross sections are mostly flat enough after deformation. Identification of damage states of elements needs to be automated if multiple analysis on larger models are to be conducted. Once individual element damage states can be identified it is beneficial to able and interpret the whole structures health, since failure of 1 element doesn't always mean collapse of a structure. These challenges will be addressed in the future.

Acknowledgements

This work received financial support from the Croatian Science Foundation under project UIP – 2020-02-1128.

References

[1] Abaqus manual, software version: Abaqus 2021; http://130.149.89.49:2080/v6.14/

[2] Python manual, software version: Python 3.10.0; https://docs.python.org/3/

[3] Uroš, M., Demšić, M., Atalić, J., Gidak, P., Šavor Novak, M., Duvnjak, I., Baniček, M., Jevtić Rundek, R., Pilipović, A., Oreb, J.(2022): Procjena otpornosti bloka B-17 u gradskog jezgri u Dubrovniku i utjecaj ojačanja kamenih zidova na njegovo ponašanje pri djelovanju potresa. Faculty of civil engineering in Zagreb, Croatia