

The authors of this paper present a spherical self-gravitating finite element model generated using the commercial code ABAQUS. The main purpose of the paper is to provide a numerical platform (ABAQUS input files) and associated benchmark useful for future studies of far-field postseismic (viscoelastic) deformation triggered by large earthquakes. A Spherical domain is required to overcome the limits imposed by flat surface analytical or semi-analytical solutions, and self-gravitation is important especially when we want to combine the deformation field from sources other than earthquakes (e.g. post-glacial rebound) or for multi-cycle simulations.

The ABAQUS input file presented in the paper is structured in two parts. The first part (*Static) controls the purely elastic coseismic phase and the implementation of fault slip using kinematic constraint equations. The second part (*Visco) controls the long term postseismic phase (viscoelastic relaxation) using Maxwell or Burgers rheology. The structure of the ABAQUS file is user friendly because provides a clear separation between the different phases of the simulation and is generally easy to modify from other people.

Large part of the paper is correctly focused on the benchmark of the coseismic and postseismic simulation against known analytical solutions. More specifically, for the coseismic phase, the authors compare the ABAQUS predicted displacement field with the well known Okada (1985) solutions for a rectangular source implemented in a homogeneous isotropic half-space. They show a general good agreement between the numerical and the analytical approaches. For the postseismic viscoelastic deformation they compare the ABAQUS predictions with the spherical harmonics code VISCO1D, showing again good agreement. The correct reproduction of analytical solutions using ABAQUS is very useful and proves the correct implementation of fault slip and viscoelastic relaxation.

In my opinion, this paper (technical activity) requires one or two additions before publication. I explain my point below.

I commend the authors for working on a such important and complex topic. However, I believe there is at least one main issue to be discussed before publication.

1. I think it will be helpful for future users of your ABAQUS input files if you could provide at least one forward model that implements a different rheologic structure. For example PREM or any recent model with lateral variations in material properties (e.g. viscosity). This way any future user will have an example to learn and to modify for its own project. The same thing is noted from Reviewer 1 although you have not updated your work to accommodate this specific request. You have already successfully completed the benchmark. For that reason, you don't need to benchmark again against a code that implements PREM (although it will be useful). I feel that this work will be significantly improved if you present what described above.

2. I'm not sure If I completely understand the problem regarding the implementation of multiple faults. If I understand correctly, implementing multiple faults will require a verification of the mesh quality. Verifying the quality of the mesh is part of the FE modeling work anyway. ABAQUS will give you a warning if the elements are distorted, although it will most probably finish the calculation. You can easily implement multiple sources (or variable fault geometry) in ABAQUS, you will just have to take the time to verify your mesh using the appropriate metrics. Another solution will be to use an external meshing software like Trelis/Cubit, that will give you better control on the generation of the mesh, and subsequently use this mesh to generate the ABAQUS input file. I will agree however, that for the purpose of investigating far-field deformation, details regarding the fault geometry might be of secondary order.