

Assignment 2 – How does a planet deform? - Finite element model for viscous deformation

Responsible instructor

Dr.ir. Wouter van der Wal

Email: w.vanderwal@tudelft.nl

Group Size: max. 2 Students

Due: 13:30h, Thursday 8 June, 2023

Estimated time: 16 hours

Report page limit: 7 pages

General introduction

Viscous deformation of planets occurs after surface loading by for example topography or glaciers. The same equations hold for loading by a potential (tidal potential) or internal loading (post-seismic deformation). In this assignment you will use a finite-element model which can be used for a surface load on a realistic planet, and you will investigate some properties of the model such as the effect of an elastic layer.

Assignment description

Objective: Assess the differences between a simplified model for surface relaxation from a textbook and a model based on finite element software.

Conceptual model

The conceptual model is equation 6-103 of Turcotte and Schubert, see the pdf on Brightspace. As parameters you can use the planet model that you developed in assignment 1 or select another one of your choice.

Finite element model

Input file for ABAQUS finite element software is prepared and can be run on a server of the planetary exploration group. Please email w.vanderwal@tudelft.nl with the names of you and your collaborator and you will receive a login.

Use secure shell to login to the server:

Host: hipparchos.lr.tudelft.nl

User: will be provided

Password: will be provided

All files you need are in the directory /home/ae4893. You can copy them to your own directory: cp /home/ae4893/* /home/studentxx/. The easiest is to use secure ftp to transfer files from the server to your own laptop. If you use windows there is easy free software to do this with drag and drop, which is called mobaxterm.

load: contains 3 columns: element number, direction of the load (P2 is a downward load), magnitude. You can create your own load with the small MATLAB program **makeload.m**. It uses the file elements.txt which has the x and y coordinate of each element in the 2nd and 3rd column.

surface.el contains a list of elements in the top layer. These files do not need to be modified.

Input.inp contains the input for Abaqus. It reads **load** and **surface.el**. You will modify it for your purpose by changing:

Thickness of the layers

Material parameters (viscosity, density and Young's modulus).

Number and length of time steps (hints: based your time steps on the time constant that you derive from the conceptual model. First test the model with a single short time step, before letting it run longer)

See the pdf file **input.inp.pdf** on Brightspace which has comments. Abaqus keywords are preceded by a *. If you want to know what their input is you can google 'Abaqus keyword manual', see for example this link: <https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/key/default.htm?startat=ch15abk01.html>

To run the finite element model, type the following command:

abaqus job=job input=input cpus=1 mp_mode=mpi interactive &

A run takes about 10 minutes if you take one short timestep and one timestep of 1000 years. Please do not increase the number of cpus because we will run into the limit of the licenses for the software.

The output is in the job.dat file. Look for the values below the string "N O D E O U T P U T". If you want to see the increments that ABAQUS uses within one time step for the numerical integration, you can look at the file **job.sta**. Other Abaqus output files are not needed, but you can look at messages in **job.msg** if you are interested.

Deliverable

Your report should contain (answers to) the following. No reporting overhead such as cover page, table of contents is required. However, explain all parameter choices that you made, and all changes made to the input files provided.

1. Show the implementation of the buoyancy force for one of the layer interfaces. (10%)
2. Plot of deformation over time for uniform mantle viscosity for the simplified and ABAQUS model for the same settings, and explanation of the differences. You can use

the body for which you obtained parameters in assignment 1 and you can consider deformation of a single point. Hint: to make the models similar, adjust the initial displacement in the conceptual model to the deformation from the FE model that you get after letting the load rest on the surface for a very long time. (25%)

3. Change the horizontal size of the load in both the simplified model and the ABAQUS model and plot the deformation over time and analyze and explain the differences. (20%)
4. Show if and how the computation time changes as a function of the viscosity and provide an explanation. (20%)
5. Measuring deformation can constrain interior properties, but this is difficult if there is no unique set of interior properties that fits the observations. Explain if you think there are different combinations of load and layering that can give the same deformation. Support your argument with simulations results. (25%)

Relevant items from the rubrics are: physical model, numerical model, model setup, model runs and analysis.

Prerequisites for the report to be assigned a grade

The report should contain the following:

- Your name(s)
- Your study number(s)
- An estimate of how many hours you spent on the assignment per person.
- Literature referencing when necessary

In addition, the report has to fit within the page limit

Finite element runs should be placed in your directory on the server in separate subdirectories: /home/studentxx/questiona, /home/studentxx/questionb etc., which contain all the input and output necessary.