**Running OQ**E. R. Abbott, J Moratalla

Last modified 7 August 2020

If you are comfortable with the information in your input files, including checking the location to which your output is being sent (ie. the export\_dir variable toward the bottom of your input file), you are ready to run OpenQuake.

While this command can be run from anywhere, it is easiest to do it from your input directory (ie. where your job.ini file, source models, and logic trees are located).

1. Navigate to your input directory
2. Run the following command:

oq engine --run job.ini --exports csv

NOTE: those dashes are TWO dashes.  
NOTE: job.ini is your input file, which can have any name as long as it ends in .ini  
**NOTE:** HDF5 files generated for your hazard run will not be stored in your export directory, rather you will find them in /home/openquake/oqdata.   
The calculation number of the file you want corresponds to the calculation run number follows the date in the bracketed statement at the start of each line of the engine’s calculation terminal output.

e.g. [2020-08-05 00:38:14 #28 INFO]  
 The calculation number above is 28.

You should see a series of statements appearing in your terminal screen throughout the process, including % of the calculation complete.

If you encounter any WARNING statements or other unusual remarks, READ THEM.   
Warnings usually exist to explain something about the program and why it’s running a certain way (e.g. too much memory is being used, so the program provides a warning and processing slows down).

When the calculation has finished you should see something along the lines of:

[2020-08-05 00:38:14 #28 INFO] Calculation 28 finished correctly in 5 seconds

id | name

6 | Full Report

7 | Hazard Curves

8 | Hazard Maps

9 | Input Files

10 | Realizations

The values below ‘id | name’ are dependent on what you selected in your input file.

The output from the hazard run should have appeared in whatever directory you specified in the export\_dir variable in your input file.