

# A Quick Guide for SPECFEM2D

---

Wenjie Lei, Chuangxin Lin

August 6, 2015

\*This tutorial is prepared for the IRIS-Earthscope USArray Data Processing and Analysis Short Course.

## 1 CODE DOWNLOADING

Download code from Github. Type the following command in the terminal:

```
git clone --recursive https://github.com/geodynamics/specfem2d.git
```

## 2 CHANGE INPUT FILES

Change directory to specfem2d(code home dir). Input files are sitting in the *DATA* directory. Two files you may want to have a look:

### 1. DATA/Par\_file

- SIMULATION\_TYPE=1 (for forward simulation, default value)
- MODEL=default (keep the model as default here)
- nt=1600 (number of forward step, keep as default value)
- deltat=1.1d-3 (each time step length, keep as default)
- modelvect=.true. (for plotting purpose, plot the velocity model)
- GPU\_MODE=.false. (turn GPU\_MODE off, default)

Also check other variables if you have time.

## 2. DATA/SOURCE

We will just leave the SOURCE file as the default status. But you should know the source can be changed to the type you want.

## 3 CODE COMPILATION

**1. Configuration:** change directory to the code home(specfem2d home). Use configure file to generate the Makefile, type in command in the code home directory:

```
./configure FC=gfortran
```

"FC=gfortran" specifies the fortran compiler we used here is gnu(gfortran). If you want to use intel fortran compiler, you can use "FC=ifort". See detailed instructions about configuration in the user manual(/doc/USER\_MANUAL/manual\_SPECFEM2D.pdf)

**2. Compile the code:** in the code home dir, type in:

```
make clean; make
```

## 4 RUN THE CODE

**1. run the Mesher:** in the code home dir, type in:

```
./bin/xmeshfem2d
```

Check out what files has been generated by mesher in the directory *OUTPUT\_FILES*.

**2. run the Solver:** in the code home dir, type in:

```
./bin/xspecfem2d
```

Again, check out what files has been generated by mesher in the directory *OUTPUT\_FILES*.

## 5 EXTRA WORK

If you have time left, check the "DATA/Par\_file" file to see how the mesh of the default model is generated using the internal mesher. Clues in Par\_file:

- 1. nbmodels
- 2. interfacesfile
- 3. nbregions

Compare these values with the picture of the mesh and find the answer out.

\*Notes: this is just a quick and dirty guide. For more detailed information, please refer to the official user manual (/doc/USER\_MANUAL/manual\_SPECFEM2D.pdf).