INDIANA UNIVERSITY, BLOOMINGTON, IN

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 --branch devel
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 ploting 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: inn 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).