A Quick Guide for SPECFEM3D_GLOBAL

Chuangxin Lin, Wenjie Lei

August 6, 2015

1 CODE DOWNLOAD

Stable version:

```
git clone https://github.com/geodynamics/specfem3d_globe.git
```

Develop version (**recommend**):

```
git clone --recursive --branch devel
https://github.com/geodynamics/specfem3d_globe.git
```

2 COMPILE

2.1 CONFIGURE

Load the modules needed for compilation(copy modules.bash from github files to the code home dir):

```
source modules.bash
```

Run the *configure* shell script:

```
./configure FC=ifort MPIFC=mpif90
```

Specify the compiler and flags when necessary!

Tips: For 3D version, you could also compile the code under your preferred directory. Just call the *configure* shell script correctly.

This generates the Makefile, but you need to set up your parameters before you make the executables. This is done by editing `Par_file` under the `DATA` directory.

2.2 PAR_FILE

Copy the file(`DATA/Par_file`) from github to its corresponding place.

`SIMULATION_TYPE` = 1 (1 is for forward simulations, 2 for adjoint simulations for sources and 3 for kernel simulations.)

`SAVE_FORWARD` is set to `.true.` (for a forward simulation with the last frame of the simulation saved, as part of the finite-frequency kernel calculations later.)

`NCHUNKS` = 1 (must be set to 6 for global simulations).

Set the following parameters depending on the shortest period of the simulation wavefield.

$$\text{shortest period (s)} \simeq (256/\texttt{NEX\_XI}) \times 17 \tag{2.1}$$

 # number of elements at the surface along the two sides of the first chunk
*# (must be multiple of 16 and 8 * multiple of NPROC below)*
NEX_XI = 64
NEX_ETA = 64

 # number of MPI processors along the two sides of the first chunk
NPROC_XI = 2
NPROC_ETA = 2

Clean:

```
make clean
```

Then type:

```
make -j8
```

`-j n` just use *n* threads to accelerate the compilation.

Tips: Everytime you modify any parameters in `Par_file` which will effect the simulation resolution , you must recompile the code.

3 PREPARE SOURCE AND STATIONS

`CMTSOLUTION`: obtained from the Harvard CMT catalog.

`STATIONS`: stations listed here. Formatted as.

Station	network	Latitude (deg)	Longitude (deg)	Elevation (m)	Burial (m)

Copy the files(`DATA/CMTSOLUTION, DATA/STATIONS`) from github to their corresponding places.

4 WRITE YOUR PBS FILE

Tips: Require the computational resource you necessarily need, and have a rough estimate on the walltime
Example: go_mesher_solver_pbs.bash (See Appendix)

5 SUBMIT YOUR JOB

Copy go_mesher.pbs and go_solver.pbs from github files to code home dir.

qsub go_mesher.pbs *: submit the job to run the mesher*

qsub go_solver.pbs *: submit the job to run forward simulation*

6 MONITOR YOUR JOB

Some useful command:

showq -u yourid *: display all your submitted jobs*

checkjob jobid *: check the status of jobid*

showstart jobid *: estimate the start time of jobid*

canceljob jobid *: cancel jobid*

* `yourid`='hpstrnXX'. `jobid` *is the identify returned when you 'qsub' your* `pbs` *file*

7 APPENDEX

```
#!/bin/bash
## job name and output file
#PBS -N go_mesher_solver
#PBS -e OUTPUT_FILES/job_forward.e
#PBS -o OUTPUT_FILES/job_forward.o

#############################################################
# USER PARAMETERS

#PBS -l nodes=1:ppn=8,walltime=00:10:00

#############################################################

cd $PBS_O_WORKDIR

source modules.bash

BASEMPIDIR=`grep LOCAL_PATH DATA/Par_file | cut -d = -f 2 `

# script to run the mesher and the solver
# read DATA/Par_file to get information about the run
# compute total number of nodes needed
NPROC_XI=`grep NPROC_XI DATA/Par_file | cut -d = -f 2 `
NPROC_ETA=`grep NPROC_ETA DATA/Par_file | cut -d = -f 2`
NCHUNKS=`grep NCHUNKS DATA/Par_file | cut -d = -f 2 `

# total number of nodes is the product of the values read
numnodes=$(( $NCHUNKS * $NPROC_XI * $NPROC_ETA ))

mkdir -p OUTPUT_FILES
mkdir -p DATABASES_MPI

# backup files used for this simulation
cp DATA/Par_file OUTPUT_FILES/
cp DATA/STATIONS OUTPUT_FILES/
cp DATA/CMTSOLUTION OUTPUT_FILES/

# obtain job information
cat $PBS_NODEFILE > OUTPUT_FILES/compute_nodes
echo "$PBS_JOBID" > OUTPUT_FILES/jobid
##
```

```
## mesh generation
##
sleep 2

echo
echo `date`
echo "starting␣MPI␣mesher␣on␣$numnodes␣processors"
echo

mpiexec −np $numnodes $PWD/bin/xmeshfem3D

echo "␣␣mesher␣done:␣`date`"
echo

# backup important files addressing.txt and list∗.txt
#cp OUTPUT_FILES/∗.txt $BASEMPIDIR/


##
## forward simulation
##

# set up addressing
#cp $BASEMPIDIR/addr∗.txt OUTPUT_FILES/
#cp $BASEMPIDIR/list∗.txt OUTPUT_FILES/

sleep 2

echo
echo `date`
echo starting run in current directory $PWD
echo
mpiexec −np $numnodes $PWD/bin/xspecfem3D

echo "finished␣successfully"
echo `date`
```