

# COUPLING AXISEM AND SPECFEM3D

**Authors :** V. Monteiller, C. Durochat

NOTE : We have to finally choose which AxiSEM version we add and definitely use for coupling with SPECFEM3D.

The paths present here are form specfem3d home directory.

Here, 'run' is an alias for Curie defined as : `run='ccc_msub -q standard'`.

## 1 STEP 1 : meshfem3D

- Go in 'EXAMPLES/coupling\_with\_EXTERNAL\_CODES/with\_AxiSEM/example\_1st\_for\_validation', (OR, create another subdirectory in '[...]/with\_AxiSEM', copy files from '[...]/example\_1st\_for\_validation' and modify them if you need)
- **CAUTION!!** : Check the file 'paths\_for\_coupling\_SPECFEM3D\_AxiSEM.sh' and modify the variable 'HOME\_SPECFEM3D'  $\implies$  this is the only path you have to modify for specfem3d
- Type :

```
./clean_this_example_dir.sh
```

```
run ./batch_coupling_step1_create_dir_and_paths+_meshfem3d_for_AxiSEM.sh
```

It generate mesh files (on 1 proc on Curie), and copy files 'list\_gll\_boundary\_spherical.txt' and 'list\_gll\_boundary\_cartesian.txt' in AxiSEM solver dir.

Also, check Meshfem3D with parameters  $\implies$  In the directory MESH/ :

- 'ParFileMeshChunk' : parameters for the chunk meshing
- 'iasp91\_dsm' or 'prem\_dsm' : background model used in both AxiSEM ans Specfem3D

In the Directory DATA/ :

- 'CMTSOLUTION' : with 0 in order to not use source inside
- 'coeff\_poly\_deg12' : to generate smooth intitial solution
- 'STATIONS' : stations files
- Set the file 'Par\_file' with these parmeters :
  - SIMULATION\_TYPE=1
  - SAVE\_FORWARD = .false.
  - COUPLE\_WITH\_EXTERNAL\_CODE = .true
  - EXTERNAL\_CODE\_TYPE = 2

## 2 STEP 2 : AxiSEM mesher

Go in : 'utils/EXTERNAL\_CODES\_coupled\_with\_SPECFEM3D/AxiSEM\_for\_SPECFEM3D/AxiSEM\_modif\_for\_coupling\_with\_specfem/MESHER'.

Check the file 'inparam\_mesh'

Type : ./submit.csh

It run the mesher in serial : check file OUTPUT (wait for "....DONE WITH MESHER" to appear in OUTPUT)

Type : ./movemesh.csh NAME\_OF\_DESTINATION\_MESH\_DIR

It moves mesh files to ../SOLVER/MESHES/NAME\_OF\_DESTINATION\_MESH\_DIR

For more details, see Axisem manual.

## 3 STEP 3 : AxiSEM solver

2 files produced by meshfem3D (input\_box.txt and input\_box\_sem\_cart.txt) were copied (and the first line indicating the number of line to be read were add at the beginning of the files too) during the STEP 1 in the AxiSEM SOLVER dir.

Go in 'utils/EXTERNAL\_CODES\_coupled\_with\_SPECFEM3D/AxiSEM\_for\_SPECFEM3D/AxiSEM\_modif\_for\_coupling\_with\_specfem/SOLVER'

Check 'inparam\_basic' (set the value for MESHNAME to the meshname from above, among others), and also (differences with \*.TEMPLATES provided by AxiSEM) :

- ATTENUATION false
- SAVE\_SNAPSHOTS true

Check 'inparam\_advanced' :

- KERNEL\_WAVEFIELDS true
- KERNEL\_IBEG 0
- KERNEL\_IEND 4

Type : ./sub\_launch\_script\_for\_run\_AxiSEM\_Curie\_CD.csh NAME\_OF\_RESULTS\_DIR

!! CAUTION : don't forget "NAME\_OF\_RESULTS\_DIR" (it will be your results folder, and a sub-folder of SOLVER/)

It run, on N procs in batch on Curie, the compilation and the execution of AxiSEM solver. Among others, this script calls :

./add\_line\_number\_in\_points\_lists.sh (rename files 'list\_gll\_boundary\_spherical.txt' & 'list\_gll\_boundary\_cartesian.txt', and add number of lines)

run ./sub\_called\_batch\_for\_AxiSEM\_Curie\_CD.sh (submit the job on Curie)

## 4 STEP 4 : Specfem Partitionning & STEP 5 : Specfem Generate database

Go back in 'EXAMPLES/coupling\_with\_EXTERNAL\_CODES/with\_AxiSEM/example\_1st\_for\_validation'

Type : `run ./batch_coupling_step4_and_step5_scoth_part_and_generdata_for_AxiSEM.sh`

!! CAUTION : the number of procs in DATA/Par\_file, have to be the same as in the 'inparam\_mesh' (cf STEP 2) of AxiSEM!!

## 5 Interface $\implies$ STEP 6 : expand 2D to 3D & STEP 7 : reformat

Go in 'utils/EXTERNAL\_CODES\_coupled\_with\_SPECFEM3D/AxiSEM\_for\_SPECFEM3D/AxiSEM\_modif\_for\_coupling\_with\_specfem/SOLVER'

- Check the file 'expand\_2D\_3D.par', cf THE PATHS IN THE 3 LAST LINES (normally, these are the only paths you to modify in the AxiSEM dir)!!  $\implies$  Caution : for the tractions path (Tractions/1/), the directory need to be created!!  $\implies$

it is created in the script 'sub\_launch\_script\_for\_make+\_run\_expand2D3D\_and\_reformat\_Curie\_CD.sh', but if you don't use this script, you have to create the directory yourself

Example :

```
input_box.txt
input_box_sem_cart.txt
32                # number of AxiSEM mpi processes used in solver
90. 0.            # source position (lat lon)
0. 60.           # chunk center (lat lon)
1                # number of axisem simus depends on moment tensor used
32                # number of Specfem3D MPI processes
/[...]/example_1st_for_validation/MESH                # Specfem MESH dir
/[...]/example_1st_for_validation/OUTPUT_FILES/DATABASES_MPI # Specfem databases dir
/[...]/example_1st_for_validation/DATA/AxiSEM_tractions/1    # AxiSEM tractions dir
```

- Check the file 'reformat.par'

Example :

```
10.              # = 1/DT, i.e. output sampling in Hz (time step that will use in Specfem3D simu)
710. 1800.       # begin time and end time (s.)
```

You can launch the compilation (of expand\_2D\_3D and reformat) AND the execution of this two programs with the single script :

`./sub_launch_script_for_make+_run_expand2D3D_and_reformat_Curie_CD.sh NAME_OF_RESULTS_DIR`

!! CAUTION : don't forget "NAME\_OF\_RESULTS\_DIR"!!

It will do the compilation, launch `./sub_called_batch_for_expand2D3D_Curie_CD.sh`, and at the end, launch `./sub_called_batch_for_reformat_Curie_CD.sh`  $\Leftarrow$  IMPORTANT : for the expand\_2D\_3D part, the number processes is arbitrary, BUT for the reformat part you have to configure the batch with the **\*\*SAME\*\*** number of processes that Specfem3D will use. Onefile is created by one process for one Specfem3D partition of domain.

All the results will be in 'NAME\_OF\_RESULTS\_DIR'.

If you prefer you do this step by step :

- Go in 'utils/EXTERNAL\_CODES\_coupled\_with\_SPECFEM3D/AxiSEM\_for\_SPECFEM3D/UTILS\_COUPLING\_SpecFEM' to make the compilation.
- Go in NAME\_OF\_RESULTS\_DIR and type : `run ../sub_called_batch_for_expand2D3D_Curie_CD.sh` to submit expand\_2D\_3D
- Go in NAME\_OF\_RESULTS\_DIR and type : `run ../sub_called_batch_for_reformat_Curie_CD.sh` to submit reformat

## 6 STEP 8 : Running specfem3D

Go in 'EXAMPLES/coupling\_with\_EXTERNAL\_CODES/with\_AxiSEM/example\_1st\_for\_validation'

Check in 'DATA/Par\_file', the lines after the comment "# time step parameters", these parameters must coincide with the parameters in 'reformat.par' (cf STEP 7) :

- NSTEP = 10900 #  $\Rightarrow$  cf the two values in the second line of 'reformat.par'. This value equals to :

$$\frac{\text{second value} - \text{the first}}{DT} \left( = \frac{\text{duration of physical time}}{DT} \right)$$

So here, it is :  $\frac{1800 - 710}{0.1} = 10900$

- DT = 0.1 #  $\Rightarrow$  the inverse of the first line in 'reformat.par' (which is 1/DT)

Also check in 'DATA/Par\_file', the lines after the comment "# to couple with an external code (such as DSM, AxiSEM, or FK)" :

- COUPLE\_WITH\_EXTERNAL\_CODE = .true. #  $\Rightarrow$  always set it to '.true.' in this case
- EXTERNAL\_CODE\_TYPE = 2 # 1 = DSM, 2 = AxiSEM, 3 = FK  $\Rightarrow$  always set it to '2' for AxiSEM
- TRACTION\_PATH = ./DATA/AxiSEM\_tractions/1/ #  $\Rightarrow$  This is where there are the tractions from AxiSEM (from steps 6 and 7)!! CAUTION : verify that they are in this dir, in the form of files 'proc0000\*\_sol\_axisem', etc.!!
- MESH\_A\_CHUNK\_OF\_THE\_EARTH = .true. #  $\Rightarrow$  always set it to '.true.' in this case

And finally, type : `run ./batch_coupling_step8_xspecfem3d.sh`

## 7 STEP ? : Set up scripts