

COMPUTATIONAL INFRASTRUCTURE FOR GEODYNAMICS (CIG)  
PRINCETON UNIVERSITY (USA)  
CNRS and UNIVERSITY OF MARSEILLE (FRANCE)

# SPECFEM 2D

User Manual  
Version 7.0



# **SPECFEM2D**

## **User Manual**

© Princeton University (USA) and CNRS / University of Marseille (France)  
Version 7.0

March 22, 2015

## Authors

The SPECFEM2D package was first developed by Dimitri Komatitsch and Jean-Pierre Vilotte at IPG in Paris (France) from 1995 to 1997 and then by Dimitri Komatitsch at Harvard University (USA), Caltech (USA) and then CNRS and University of Pau (France) from 1998 to 2005. The story started on April 4, 1995, when Prof. Yvon Maday from CNRS and University of Paris, France, gave a lecture to Dimitri Komatitsch and Jean-Pierre Vilotte at IPG about the nice properties of the Legendre spectral-element method with diagonal mass matrix that he had used for other equations. We are deeply indebted and thankful to him for that. That followed a visit by Dimitri Komatitsch to OGS (Istituto Nazionale di Oceanografia e di Geofisica Sperimentale) in Trieste, Italy, in February 1995 to meet with Géza Seriani and Enrico Priolo, who introduced him to their 2D Chebyshev version of the spectral-element method with a non-diagonal mass matrix. We are deeply indebted and thankful to them for that.

Since then it has been developed and maintained by a development team: in alphabetical order, Alexis Bottero, Paul Cristini, Dimitri Komatitsch, Jesús Labarta, Nicolas Le Goff, Pieyre Le Loher, Qinya Liu, Roland Martin, René Matzen, Christina Morency, Daniel Peter, Carl Tape, Jeroen Tromp, Jean-Pierre Vilotte, Zhinan Xie.

The code is released open-source under the CeCILL version 2 license, see the license at the end of this manual.

## Current and past main participants or main sponsors of the SPECFEM project (in no particular order)



Aix\*Marseille  
université

**ETH** Zürich



**Inria**  
INVENTEURS DU MONDE NUMÉRIQUE



Agence Nationale de la Recherche  
**ANR**



# Contents

<b>Contents</b>	<b>3</b>
<b>1 Introduction</b>	<b>4</b>
1.1 Citation . . . . .	5
1.2 Support . . . . .	6
<b>2 Getting Started</b>	<b>7</b>
2.1 Visualizing the subroutine calling tree of the source code . . . . .	8
2.2 Becoming a developer of the code, or making small modifications in the source code . . . . .	8
<b>3 Mesh Generation</b>	<b>9</b>
3.1 How to use SPECFEM2D . . . . .	9
3.2 How to use Gmsh to generate an external mesh . . . . .	11
3.3 Controlling the quality of an external mesh . . . . .	13
3.4 Controlling how the mesh samples the wave field . . . . .	13
<b>4 Running the Solver xspecefem2D</b>	<b>15</b>
4.1 How to run P-SV or SH (membrane) wave simulations . . . . .	17
4.2 How to use anisotropy . . . . .	18
4.3 How to use poroelasticity . . . . .	18
4.4 How to set plane waves as initial conditions . . . . .	20
4.5 How to choose the time step . . . . .	20
<b>5 Adjoint Simulations</b>	<b>22</b>
5.1 How to obtain Finite Sensitivity Kernels . . . . .	22
5.2 Remarks about adjoint runs and solving inverse problems . . . . .	23
5.3 Caution . . . . .	23
<b>6 Oil and gas industry simulations</b>	<b>24</b>
<b>A Troubleshooting</b>	<b>27</b>
<b>B License</b>	<b>28</b>

# Chapter 1

## Introduction

SPECFEM2D facilitates 2D simulations of acoustic, (an)elastic, and poroelastic seismic wave propagation. The 2D spectral-element solver accommodates regular and unstructured meshes, generated for example by Cubit (<http://cubit.sandia.gov>), Gmsh (<http://geuz.org/gmsh>) or GiD (<http://www.gid.cimne.upc.es>). Even mesh creation packages that generate triangles, for instance Delaunay-Voronoi triangulation codes, can be used because each triangle can then easily be decomposed into three quadrangles by linking the barycenter to the center of each edge; while this approach does not generate quadrangles of optimal quality, it can ease mesh creation in some situations and it has been shown that the spectral-element method can very accurately handle distorted mesh elements.

With version 7.0, the 2D spectral-element solver accommodates Convolution PML absorbing layers and well as higher-order time schemes (4th order Runge-Kutta and LDDRK4-6). Convolution or Auxiliary Differential Equation Perfectly Matched absorbing Layers (C-PML or ADE-PML) are described in ????.

The solver has adjoint capabilities and can calculate finite-frequency sensitivity kernels [??] for acoustic, (an)elastic, and poroelastic media. The package also considers 2D SH and P-SV wave propagation. Finally, the solver can run both in serial and in parallel. See SPECFEM2D (<http://www.geodynamics.org/cig/software/packages/seismo/specfem2d>) for the source code.

The SEM is a continuous Galerkin technique [??], which can easily be made discontinuous [?????????]; it is then close to a particular case of the discontinuous Galerkin technique [?????????????????????], with optimized efficiency because of its tensorized basis functions [??]. In particular, it can accurately handle very distorted mesh elements [?].

It has very good accuracy and convergence properties [?????????]. The spectral element approach admits spectral rates of convergence and allows exploiting  $hp$ -convergence schemes. It is also very well suited to parallel implementation on very large supercomputers [?????] as well as on clusters of GPU accelerating graphics cards [????]. Tensor products inside each element can be optimized to reach very high efficiency [?], and mesh point and element numbering can be optimized to reduce processor cache misses and improve cache reuse [?]. The SEM can also handle triangular (in 2D) or tetrahedral (in 3D) elements [?????] as well as mixed meshes, although with increased cost and reduced accuracy in these elements, as in the discontinuous Galerkin method.

Note that in many geological models in the context of seismic wave propagation studies (except for instance for fault dynamic rupture studies, in which very high frequencies or supershear rupture need to be modeled near the fault, see e.g. ???) a continuous formulation is sufficient because material property contrasts are not drastic and thus conforming mesh doubling bricks can efficiently handle mesh size variations [?????].

For a detailed introduction to the SEM as applied to regional seismic wave propagation, please consult ??? and in particular ?????. A detailed theoretical analysis of the dispersion and stability properties of the SEM is available in ?, ?, ?, ?, ? and ?.

The SEM was originally developed in computational fluid dynamics [??] and has been successfully adapted to address problems in seismic wave propagation. Early seismic wave propagation applications of the SEM, utilizing Legendre basis functions and a perfectly diagonal mass matrix, include ?, ?, ?, ?, ? and ?, whereas applications involving Chebyshev basis functions and a non-diagonal mass matrix include ?, ? and ?.

All SPECFEM2D software is written in Fortran2003 with full portability in mind, and conforms strictly to the Fortran2003 standard. It uses no obsolete or obsolescent features of Fortran. The package uses parallel programming based upon the Message Passing Interface (MPI) [??].

The next release of the code will include support for GPU graphics card acceleration [?????].

The code uses the plane strain convention when the standard P-SV equation case is used, i.e., the off-plane strain  $\epsilon_{zz}$  is zero by definition of the plane strain convention but the off-plane stress  $\sigma_{zz}$  is not equal to zero, one has  $\sigma_{zz} = \lambda(\epsilon_{xx} + \epsilon_{yy})$ . This implies, as in any plain strain software package, that the P-SV source is a line source along the direction perpendicular to the plane (see file discussion\_of\_2D\_sources\_and\_approximations\_from\_Pilant\_1979.pdf for more details).

## 1.1 Citation

If you use this code for your own research, please cite at least one article written by the developers of the package, for instance:



If you use the kernel capabilities of the code, please cite at least one article written by the developers of the package, for instance:

- ?,
  - ?,
  - ?,
  - ?.

If you use the SCOTCH / CUBIT non-structured capabilities, please also cite:

- ?.

The corresponding BibTeX entries may be found in file doc/USER MANUAL/bibliography.bib.

## 1.2 Support

This material is based upon work supported by the USA National Science Foundation under Grants No. EAR-0406751 and EAR-0711177, by the French CNRS, French Inria Sud-Ouest MAGIQUE-3D, French ANR NUMASIS under Grant No. ANR-05-CIGC-002, and European FP6 Marie Curie International Reintegration Grant No. MIRG-CT-2005-017461. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the authors and do not necessarily reflect the views of the USA National Science Foundation, CNRS, Inria, ANR or the European Marie Curie program.

# Chapter 2

## Getting Started

To download the SPECFEM2D software package, type this:

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

We recommend that you add `ulimit -S -s unlimited` to your `.bash_profile` file and/or limit `stacksize unlimited` to your `.cshrc` file to suppress any potential limit to the size of the Unix stack.

Then, to configure the software for your system, run the `configure` shell script. This script will attempt to guess the appropriate configuration values for your system. However, at a minimum, it is recommended that you explicitly specify the appropriate command names for your Fortran compiler (another option is to define FC, CC and MPIF90 in your `.bash_profile` or your `.cshrc` file):

```
./configure FC=ifort
```

If you want to run in parallel, i.e., using more than one processor core, then you would type

```
./configure FC=ifort MPIFC=mpif90 --with-mpi
```

Before running the `configure` script, you should probably edit file `flags.guess` to make sure that it contains the best compiler options for your system. Known issues or things to check are:

**Intel ifort compiler** See if you need to add `-assume byterecl` for your machine. **In the case of that compiler, we have noticed that versions dot zero sometimes have bugs or issues that can lead to wrong results when running the code, thus we strongly recommend using versions dot one or above (for instance version 13.1 instead of 13.0, version 14.1 instead of 14.0 and so on).**

**IBM compiler** See if you need to add `-qsave` or `-qnosave` for your machine.

**Mac OS** You will probably need to install XCODE.

**IBM Blue Gene machines** Please refer to the manual of SPECFEM3D\_Cartesian, which contains detailed instructions on how to run on Blue Gene.

The SPECFEM2D software package relies on the SCOTCH library to partition meshes. The SCOTCH library [?] provides efficient static mapping, graph and mesh partitioning routines. SCOTCH is a free software package developed by François Pellegrini et al. from LaBRI and Inria in Bordeaux, France, downloadable from the web page <https://gforge.inria.fr/projects/scotch/>. In case no SCOTCH libraries can be found on the system, the configuration will bundle the version provided with the source code for compilation. The path to an existing SCOTCH installation can be set explicitly with the option `--with-scotch-dir`. Just as an example:

```
./configure FC=ifort MPIFC=mpif90 --with-mpi --with-scotch-dir=/opt/scotch
```

If you use the Intel ifort compiler to compile the code, we recommend that you use the Intel icc C compiler to compile Scotch, i.e., use:

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

For further details about the installation of SCOTCH, go to subdirectory `scotch_5.1.11/` and read `INSTALL.txt`. You may want to download more recent versions of SCOTCH in the future from ([http://www.labri.fr/perso/pelegrin/scotch/scotch\\_en.html](http://www.labri.fr/perso/pelegrin/scotch/scotch_en.html)) . Support for the METIS graph partitioner has been discontinued because SCOTCH is more recent and performs better.

When compiling the SCOTCH source code, if you get a message such as: "ld: cannot find -lz", the Zlib compression development library is probably missing on your machine and you will need to install it or ask your system administrator to do so. On Linux machines the package is often called "zlib1g-dev" or similar. (thus "sudo apt-get install zlib1g-dev" would install it)

You may edit the `Makefile` for more specific modifications. Especially, there are several options available:

- `-DUSE_MPI` compiles with use of an MPI library.
- `-DUSE_SCOTCH` enables use of graph partitioner SCOTCH.

After these steps, go back to the main directory of SPECFEM2D/ and type

```
make
```

to create all executables which will be placed into the folder `./bin/`.

By default, the solver runs in single precision. This is fine for most application, but if for some reason you want to run the solver in double precision, run the `configure` script with option "`--enable-double-precision`". Keep in mind that this will of course double total memory size and will also make the solver around 20 to 30% slower on many processors.

If your compiler has problems with the `use mpi` statements that are used in the code, use the script called `replace_use_mpi_with_include_mpif_dot_h.pl` in the root directory to replace all of them with include '`mpif.h`' automatically.

## 2.1 Visualizing the subroutine calling tree of the source code

Packages such as `doxywizard` can be used to visualize the subroutine calling tree of the source code. `Doxywizard` is a GUI front-end for configuring and running `doxygen`.

## 2.2 Becoming a developer of the code, or making small modifications in the source code

If you want to develop new features in the code, and/or if you want to make small changes, improvements, or bug fixes, you are very welcome to contribute. To do so, i.e. to access the development branch of the source code with read/write access (in a safe way, no need to worry too much about breaking the package, there is a robot called BuildBot that is in charge of checking and validating all new contributions and changes), please visit this Web page: <https://github.com/geodynamics/specfem2d/wiki/Using-Hub>.

# Chapter 3

## Mesh Generation

### 3.1 How to use SPECFEM2D

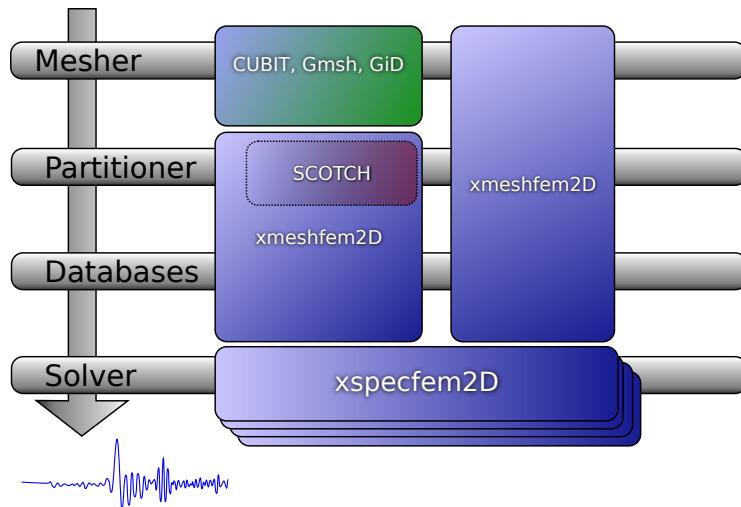


Figure 3.1: Schematic workflow for a SPECFEM2D simulation. The executable `xmshfem2D` creates the GLL mesh points and assigns specific model parameters. The executable `xspecfem2D` solves the seismic wave propagation.

To run the mesher, please follow these steps:

- edit the input file `DATA/Par_file`, which describes the simulation. The default `DATA/Par_file` provided with the code contains detailed comments and should be almost self-explanatory (note that some of the older `DATA/Par_file` files provided in the `EXAMPLES` directory work fine but some of the comments they contain may be obsolete or even wrong; thus refer to the default `DATA/Par_file` instead for reliable explanations). If you need more details we do not have a detailed description of all the parameters for the 2D version in this manual but you can find useful information in the manuals of the 3D versions, since many parameters and the general philosophy is similar. They are available at (<https://github.com/geodynamics/specfem3d/tree/master/doc/USER MANUAL>). To create acoustic (fluid) regions, just set the S wave speed to zero and the code will see that these elements are fluid and switch to the right equations there automatically, and automatically match them with the solid regions
- if you are using an external mesher (like GiD or CUBIT / Treliis), you should set `read_external_mesh` to `.true.`:

**mesh\_file** is the file describing the mesh : first line is the number of elements, then a list of 4 nodes (quadrilaterals only) forming each elements on each line.

**nodes\_coords\_file** is the file containing the coordinates ( $x$  and  $z$ ) of each node: number of nodes on the first line, then coordinates  $x$  and  $z$  on each line.

**materials\_file** is the number of the material for every element : an integer ranging from 1 to nbmodels on each line.

**free\_surface\_file** is the file describing the edges forming the acoustic free surface: number of edges on the first line, then on each line: number of the element, number of nodes forming the free surface (1 for a point, 2 for an edge), the nodes forming the free surface for this element. If you do not want any free surface, just put 0 on the first line.

**absorbing\_surface\_file** is the file describing the edges forming the absorbing boundaries: number of edges on the first line, then on each line: number of the element, number of nodes forming the absorbing edge (must always be equal to 2), the two nodes forming the absorbing edge for this element, and then the type of absorbing edge: 1 for BOTTOM, 2 for RIGHT, 3 for TOP and 4 for LEFT. Only two nodes per element can be listed, i.e., the second parameter of each line must always be equal to 2. If one of your elements has more than one edge along a given absorbing contour (e.g., if that contour has a corner) then list it twice, putting the first edge on the first line and the second edge on the second line. Do not list the same element with the same absorbing edge twice or more, otherwise absorption will not be correct because the edge integral will be improperly subtracted several times. If one of your elements has a single point along the absorbing contour rather than a full edge, do NOT list it (it would have no weight in the contour integral anyway because it would consist of a single point). If you use 9-node elements, list only the first and last points of the edge and not the intermediate point located around the middle of the edge; the right 9-node curvature will be restored automatically by the code.

**tangential\_detection\_curve\_file** contains points describing the envelope, that are used for the `source_normal_to_surface` and `rec_normal_to_surface`. Should be fine grained, and ordered clockwise. Number of points on the first line, then ( $x,z$ ) coordinates on each line.

- if you have compiled with MPI, you must specify the number of processes.

Then type

```
./bin/xmeshfem2D
```

to create the mesh (which will be stored in directory `OUTPUT_FILES/`). `xmeshfem2D` is serial; it will output several files called `Database??????`, one for each process.

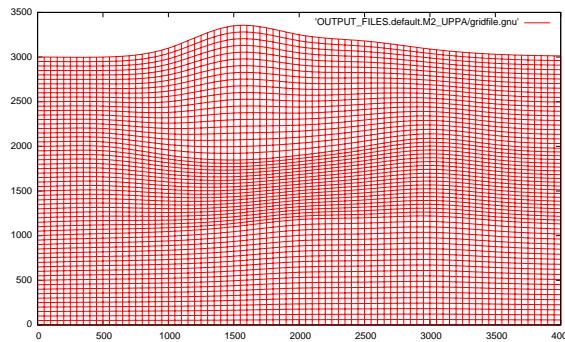


Figure 3.2: Example of a grid file generated by `xmeshfem2D` and visualized with `gnuplot` (within `gnuplot`, type ‘`plot "OUTPUT_FILES/gridfile.gnu" w l`’).

Regarding mesh point numbering in the files created by the mesher, we use the classical convention of 4-node and 9-node finite elements:

```

4 . . . . 7 . . . . 3
.
.
. eta .
.
. | .
8 9--xi 6
.
.
.
.
1 . . . . 5 . . . . 2

```

the local coordinate system being  $\xi$  and  $\eta$  (`xi` and `eta`). Note that this convention is used to describe the geometry only. In the solver the wave field is then described based on high-order Lagrange interpolants at Gauss-Lobatto-Legendre points, as is classical in spectral-element methods.

## 3.2 How to use Gmsh to generate an external mesh

Gmsh<sup>1</sup> is a 3D finite element grid generator which can be used for the generation of quadrangle and hexahedral meshes. It is therefore a good candidate for generating meshes which can be processed by SPECFEM2D. Only two modules of Gmsh are of interest for the SPECFEM2D users : the geometry and the mesh modules. An example is given in directory `EXAMPLES/Gmsh_example` which illustrates the generation of an external mesh using these two modules. The model that is considered consists of a homogeneous square containing two circles filled with a different material.

The geometry is generated by loading file `SqrCirc.geo` into Gmsh. The end of the `.geo` file contains several lines which are required in order to define the sides of the box and the media. This is done using the following conventions :

```

Physical Line("Top") = {1}; line corresponding to the top of the box
Physical Line("Left") = {2}; line corresponding to the left side of the box
Physical Line("Bottom") = {3}; line corresponding to the bottom of the box
Physical Line("Right") = {4}; line corresponding to the right side of the box
Physical Surface("M1") = {10}; surrounding medium
Physical Surface("M2") = {11,12}; interior of the two circles

```

For instance, if you want to fill the two circles with two different materials, you will have to write :

```

Physical Surface("M1") = {10}; surrounding medium
Physical Surface("M2") = {11}; interior of the big circle
Physical Surface("M3") = {12}; interior of the small circle

```

and, consequently, you will have to define a new medium numbered 3 in the `Par_file`.

Then, a 2D mesh can be created and saved after selecting the appropriate options in Gmsh : All quads in Subdivision algorithm and 1 or 2 in Element order whether you want a 4 or 9 node mesh. This operation will generate a `SqrCirc.msh` file which must be processed to get all the files required by SPECFEM2D when using an external mesh (see previous section). This is done by running a python script called `LibGmsh2Specfem.py`, located in directory `utils/Gmsh`:

```
python LibGmsh2Specfem.py SqrCirc -t A -b A -r A -l A
```

Where the options `-t`, `-b`, `-r` and `-l` represent the different sides of the model (top, bottom, right and left) and can take the values `A` or `F` if the corresponding side is respectively absorbing or free. All boundaries are absorbing by default. The connections of the generated filenames to the filenames indicated in the previous section are :

- Mesh\_`SqrCirc` is the **mesh\_file**

---

<sup>1</sup>freely available at the following address : <http://www.geuz.org/gmsh/>

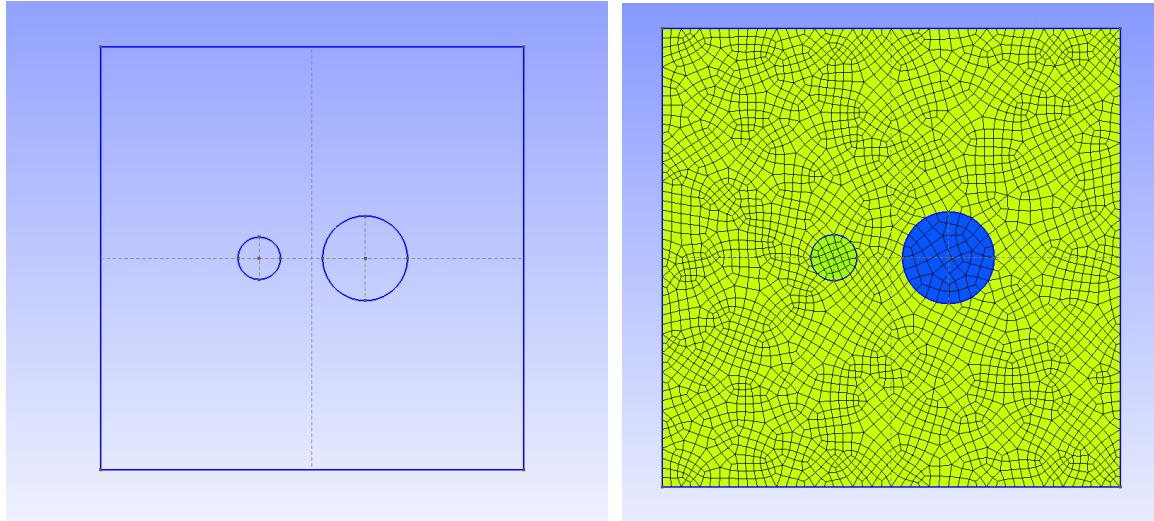


Figure 3.3: Geometry and mesh of the two circle model generated with Gmsh

- Material\_SqrCirc is the **material\_file**
- Nodes\_SqrCirc is the **nodes\_coords\_file**
- Surf\_abs\_SqrCirc is the **absorbing\_surface\_file**
- Surf\_free\_SqrCirc is the **free\_surface\_file**

In addition, four files like `free_surface_file` corresponding to the sides of the model are generated.

If you use CPML, an additional file listing the CPML elements is needed. Its first line is the total number of CPML elements, and then a list of all the CPML elements, one per line. The format of these lines is: in the first column the CPML element number, and in the second column a flag as follows:

Table 3.1: Definition of flags for CPML elements

Flag	Meaning
1	element belongs to a X CPML layer only (either in $X_{\min}$ or in $X_{\max}$ )
2	element belongs to a Y CPML layer only (either in $Y_{\min}$ or in $Y_{\max}$ )
3	element belongs to both a X and a Y CPML layer (i.e., to a CPML corner)

In order to see how to add PML layers to a mesh / model created with an external mesher such as ‘Gmsh’, see the examples in directory `EXAMPLES/CPML_absorbing_layers`.

If you use PML, the mesh elements that belong to the PML layers can be acoustic or elastic, but not viscoelastic nor poroelastic. Then, when defining your model, you should define these absorbing elements as either acoustic or elastic. If you forget to do that, the code will fix the problem by automatically converting the viscoelastic or poroelastic PML elements to elastic. This means that strictly speaking the PML layer will not be perfectly matched any more, since the physical model will change from viscoelastic or poroelastic to elastic at the entrance of the PML, but in practice this is sufficient and produces only tiny / negligible spurious reflections.

If you use PML and an external mesh (created using an external meshing tool such as Gmsh, CUBIT/TRELIS or similar), try to have elements inside the PML as regular as possible, i.e. ideally non-deformed rectangles obtained by ‘extrusion’ of the edge mesh elements meshing the outer edges of the computational domain without PML; by doing so, the PMLs obtained will be far more stable in time (PML being weakly unstable from a mathematical point of view, very deformed mesh elements inside the PMLs can trigger instabilities much more quickly).

When you have an existing CUBIT (or similar) mesh stored in SPECFEM2D format but do not know how to assign CPML flags to it, in the 3D version of the code (SPECFEM3D) we have created a small serial Fortran

program that will do that automatically for you, thus you can adapt that program to SPECFEM2D. That program is `utils/CPML/convert_external_layers_of_a_given_mesh_to_CPML_layers.f90` in the source code of SPECFEM3D, and a small Makefile is provided in that directory (`utils/CPML`). Note from Dimitri: we will soon provide a clean 2D version in `utils/CPML` of the SPECFEM2D source code; Andrea Colombi from CNRS Grenoble, France, is currently working on that.

If you use PML and an external velocity and density model (e.g., setting flag “`MODEL`” to `tomography`), you should be careful because mathematically a PML cannot handle heterogeneities along the normal to the PML edge inside the PML layer. This comes from the fact that the damping profile that is defined assumes a constant velocity and density model along the normal direction. Thus, you need to modify your velocity and density model in order for it to be 1D inside the PML, as shown in Figure 3.4. This applies to the bottom layer as well; there you should make sure that your model is 1D and thus constant along the vertical direction. To summarize, only use a 2D velocity and density model inside the physical region, and in all the PML layers extend it by continuity from its values along the inner PML edge.

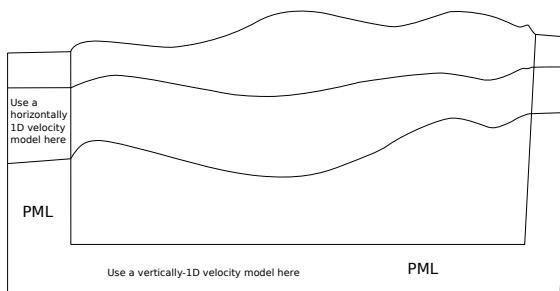


Figure 3.4: How to modify your external 2D velocity and density model in order to use PML. Such a modification is not needed when using Stacey absorbing boundary conditions (but such conditions are significantly less efficient).

### 3.3 Controlling the quality of an external mesh

To examine the quality of the elements in your externally build mesh, type

```
./bin/xcheck_quality_external_mesh
```

(and answer “3” to the first question asked). This code will tell you which element in the whole mesh has the worst quality (maximum skewness, i.e. maximum deformation of the element angles) and it should be enough to modify this element with the external software package used for the meshing, and to repeat the operation until the maximum skewness of the whole mesh is less or equal to about 0.75 (above is dangerous: from 0.75 to 0.80 could still work, but if there is a single element above 0.80 the mesh should be improved).

The code also shows a histogram of 20 classes of skewness which tells how many element are above the skewness = 0.75, and to which percentage of the total this amounts. To see this histogram, you could type:

```
gnuplot plot_mesh_quality_histogram.gnu
```

This tool is useful to estimate the mesh quality and to see it evolve along the successive corrections.

### 3.4 Controlling how the mesh samples the wave field

To examine (using Gnuplot) how the mesh samples the wave field, type

```
gnuplot plot_points_per_wavelength_histogram.gnu
```

and also check the following histogram printed on the screen or in the output file:

histogram of min number of points per S wavelength (P wavelength in acoustic regions)  
(too small: poor resolution of calculations – too big = wasting memory and CPU time)  
(threshold value is around 4.5 points per wavelength in elastic media and 5.5 in acoustic media)

If you see that you have a significant number of mesh elements below the threshold indicated, then your simulations will not be accurate and you should create a denser mesh. Conversely, if you have a significant number of mesh elements above the threshold indicated, the mesh you created is too dense, it will be extremely accurate but the simulations will be slow; using a coarser mesh would be sufficient and would lead to faster simulations.

## Chapter 4

# Running the Solver xspecfem2D

To run the solver, type:

```
./bin/xspecfem2D
```

to run the main solver (use `mpirun` or equivalent if you compiled with parallel support). This will output the seismograms and snapshots of the wave fronts at different time steps in directory `OUTPUT_FILES/`. To visualize them, type "`gs OUTPUT_FILES/vect*.ps`" to see the Postscript files (in which the wave field is represented with small arrows, fluid/solid matching interfaces with a thick pink line, and absorbing edges with a thick green line) and "`gimp OUTPUT_FILES/image*.gif`" to see the colour snapshot showing a pixelized image of one of the two components of the wave field (or pressure, depending on what you have selected for the output in `DATA/Par_file`).

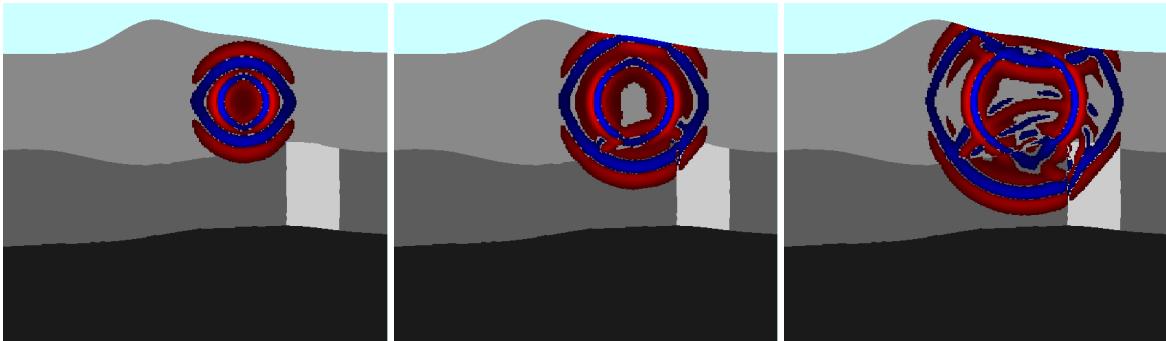


Figure 4.1: Wavefield snapshots of the default example generated by `xspecfem2D` when parameter `output_color_image` is set to `.true.`. To create smaller (subsampled) images you can change double precision parameter `factor_subsample_image = 1.0` to a higher value in file `DATA/Par_file`. This can be useful in the case of very large models. The number of pixels of the image in each direction must be smaller than parameter `NX_NZ_IMAGE_MAX` defined in file `SETUP/constants.h.in`, again to avoid creating huge images in the case of very large models.

Please consider these following points, when running the solver:

- the `DATA/Par_file` given with the code works fine, you can use it without any modification to test the code
- the seismograms `OUTPUT_FILES/* .sem*` are simple ASCII files with two columns: time in the first column and amplitude in the second, therefore they can be visualized with any tool you like, for instance “`gnuplot`”; if you prefer to output binary seismograms in Seismic Unix format (which is a simple binary array dump) you can use parameter `SU_FORMAT`, in which case all the seismograms will be written to a single file with the extension `*.bin`. Depending on your installation of the Seismic Unix package you can use one of these two commands:

```
surange < Uz_file_single.bin  
suoldtonew < Uz_file_single.bin | surange
```

to see the header info. Replace `surange` with `suxwigb` to see wiggle plots for the seismograms.

- if flag `MODEL` in `DATA/Par_file` is set to `default`, the velocity and density model is determined using the `nbmodels` and `nbregions` devices. Otherwise, `nbmodels` values are ignored and the velocity and density model is determined from a user supplied file or subroutine.
- when compiling with Intel ifort, use “`-assume byterecl`” option to create binary PNM images displaying the wave field
- there are a few useful scripts and Fortran routines in directory `utils/`.
- you can find a Fortran code to compute the analytical solution for simple media that we use as a reference in benchmarks in many of our articles at (<http://www.spice-rtn.org/library/software/EX2DDIR>). That code is described in: ?

The `SOURCE` file located in the `DATA/` directory should be edited in the following way:

**source\_surf** Set this flag to `.true.` to force the source to be located at the surface of the model, otherwise the sol be placed inside the medium

**xs** source location *x* in meters

**zs** source location *z* in meters

**source\_type** Set this value equal to 1 for elastic forces or acoustic pressure, set this to 2 for moment tensor sources. For a plane wave including converted and reflected waves at the free surface, P wave = 1, S wave = 2, Rayleigh wave = 3; for a plane wave without converted nor reflected waves at the free surface, i.e. the incident wave only, P wave = 4, S wave = 5. (incident plane waves are turned on by parameter `initialfield` in `DATA/Par_file`).

**time\_function\_type** Choose a source-time function: set this value to 1 to use a Ricker, 2 the first derivative, 3 a Gaussian, 4 a Dirac or 5 a Heaviside source-time function.

**f0** Set this to the dominant frequency of the source. For point-source simulations using a Heaviside source-time function (`time_function_type = 5`), we recommend setting the source frequency parameter `f0` equal to a high value, which corresponds to simulating a step source-time function, i.e., a moment-rate function that is a delta function.

The `half` duration of a source is obtained by  $1/f_0$ . If the code will use a Gaussian source-time function (`time_function_type = 3`) (i.e., a signal with a shape similar to a ‘smoothed triangle’, as explained in ? and shown in Fig 4.2), the source-time function uses a half-width of `half` duration. We prefer to run the solver with `half` duration set to zero and convolve the resulting synthetic seismograms in post-processing after the run, because this way it is easy to use a variety of source-time functions. ? determined that the noise generated in the simulation by using a step source time function may be safely filtered out afterward based upon a convolution with the desired source time function and/or low-pass filtering. Use the serial code `convolve_source_timefunction.f90` and the script `convolve_source_timefunction.sh` for this purpose, or alternatively use signal-processing software packages such as SAC ([www.llnl.gov/sac](http://www.llnl.gov/sac)). Type

```
make xconvolve_source_timefunction
```

to compile the code and then set the parameter `hdur` in `convolve_source_timefunction.sh` to the desired half-duration.

**t0** For single sources, we recommend to set the time shift parameter `t0` equal to 0.0. The time shift parameter would simply apply an overall time shift to the synthetics (according to the time shift of the first source), something that can be done in the post-processing. This time shift parameter can be non-zero when using multiple sources.

**anglesource** angle of the source (for a force only); for a plane wave, this is the incidence angle. For moment tensor sources this parameter is unused.

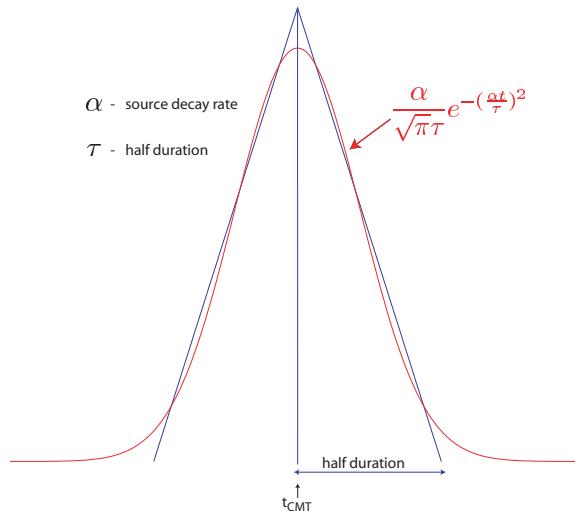


Figure 4.2: Comparison of the shape of a triangle and the Gaussian function actually used.

**Mxx, Mzz, Mxz** Moment tensor components (valid only for moment tensor sources, `source_type = 2`). Note that the units for the components of a moment tensor source are different in SPECFEM2D and in SPECFEM3D:

**SPECFEM3D:** in SPECFEM3D the moment tensor components are in dyne\*cm

**SPECFEM2D:** in SPECFEM2D the moment tensor components are in N\*m

To go from strike / dip / slip to CMTSOLUTION moment-tensor format using the classical formulas (of e.g. ? you can use these two small C programs from SPECFEM3D\_GLOBE:

```
./utils/strike_dip_rake_to_CMTSOLUTION.c
./utils/CMTSOLUTION_to_AkiRichards.c
```

but then it is another story to make a good 2D approximation of that, because in plain-strain P-SV what you get is the equivalent of a line source in the third direction (orthogonal to the plane) rather than a 3D point source For more details on this see e.g. Section 7.3 "Two-dimensional point sources" of the book of ?. That book being hard to find, we scanned the related pages in file discussion\_of\_2D\_sources\_and\_approximations\_from\_Pilant\_1979.pdf in the same directory as this users manual. Another very useful reference addressing that is ? and its recent extension [?].

**factor** amplification factor

Note, the zero time of the simulation corresponds to the center of the triangle/Gaussian, or the centroid time of the earthquake. The start time of the simulation is  $t = -1.2 * \text{half duration} + t0$  (the factor 1.2 is to make sure the moment rate function is very close to zero when starting the simulation; Heaviside functions use a factor 2.0), the half duration is obtained by  $1/f0$ . If you prefer, you can fix this start time by setting the parameter `USER_T0` in the `constants.h` file to a positive, non-zero value. The simulation in that case would start at a starting time equal to  $-USER_T0$ .

## Coupled Simulations

The code supports acoustic/elastic, acoustic/poroelastic, elastic/poroelastic, and acoustic, elastic/poroelastic simulations. Elastic/poroelastic coupling supports anisotropy, but not attenuation for the elastic material.

### 4.1 How to run P-SV or SH (membrane) wave simulations

For elastic materials, you have these additional options:

**P-SV:** To run a P-SV waves calculation propagating in the  $x$ - $z$  plane, set `p_sv = .true.` in the `Par_file`.

**SH:** To run a SH (membrane) waves calculation travelling in the  $x$ - $z$  plane with a  $y$ -component of motion, set `p_sv = .false.`

This feature is only implemented for elastic materials and sensitivity kernels can be calculated (see `?` for details on membrane surface waves).

A useful Python script called `SEM_save_dir.py`, written by Paul Cristini from Laboratoire de Mecanique et d'Acoustique, CNRS, Marseille, France, is provided. It allows one to automatically save all the parameters and results of a given simulation.

## 4.2 How to use anisotropy

Following `?`, we use the classical reduced Voigt notation to represent symmetric tensors [??]:

The constitutive relation of a heterogeneous anisotropic and elastic solid is expressed by the generalized Hooke's law, which can be written as

$$\sigma_{ij} = c_{ijkl}\varepsilon_{kl}, \quad i, j, k = 1, \dots, 3,$$

where  $t$  is the time,  $\mathbf{x}$  is the position vector,  $\sigma_{ij}(\mathbf{x}, t)$  and  $\varepsilon_{ij}(\mathbf{x}, t)$  are the Cartesian components of the stress and strain tensors respectively, and  $c_{ijkl}(\mathbf{x})$  are the components of a fourth-order tensor called the elasticites of the medium. The Einstein convention for repeated indices is used.

To express the stress-strain relation for a transversely isotropic medium we introduce a shortened matrix notation commonly used in the literature. With this convention, pairs of subscripts concerning the elasticities are replaced by a single number according to the following correspondence:

$$\begin{array}{lll} (11) \rightarrow 1, & (22) \rightarrow 2, & (33) \rightarrow 3, \\ (23) = (32) \rightarrow 4, & (31) = (13) \rightarrow 5, & (12) = (21) \rightarrow 6. \end{array}$$

Thus in the most general 2D case we have the following convention for the stress-strain relationship:

```
! implement anisotropy in 2D
sigma_xx = c11*dux_dx + c13*duz_dz + c15*(duz_dx + dux_dz)
sigma_zz = c13*dux_dx + c33*duz_dz + c35*(duz_dx + dux_dz)
sigma_xz = c15*dux_dx + c35*duz_dz + c55*(duz_dx + dux_dz)

! sigma_yy is not equal to zero in the plane strain formulation
! but is used only in post-processing if needed,
! to compute pressure for display or seismogram recording purposes
sigma_yy = c12*dux_dx + c23*duz_dz + c25*(duz_dx + dux_dz)
```

where the notations are for instance  $\text{duz}_\text{dx} = \frac{d(U_z)}{dx}$ .

## 4.3 How to use poroelasticity

Check the following new inputs in `Par_file`:

In section "# geometry of model and mesh description":

`TURN_VISCATTENUATION_ON`, `Q0`, and `FREQ0` deal with viscous damping in a poroelastic medium. `Q0` is the quality factor set at the central frequency `FREQ0`. For more details see `?`.

In section "# time step parameters":

`SIMULATION_TYPE` defines the type of simulation

- (1) forward simulation

- (2) UNUSED (purposely, for compatibility with the numbering convention used in our 3D codes)
- (3) adjoint method and kernels calculation

In section "# source parameters":

The code now support multiple sources. NSOURCE is the number of sources. Parameters of the sources are displayed in the file SOURCE, which must be in the directory DATA/. The components of a moment tensor source must be given in N.m, not in dyne.cm as in the DATA/CMTSOLUTION source file of the 3D version of the code.

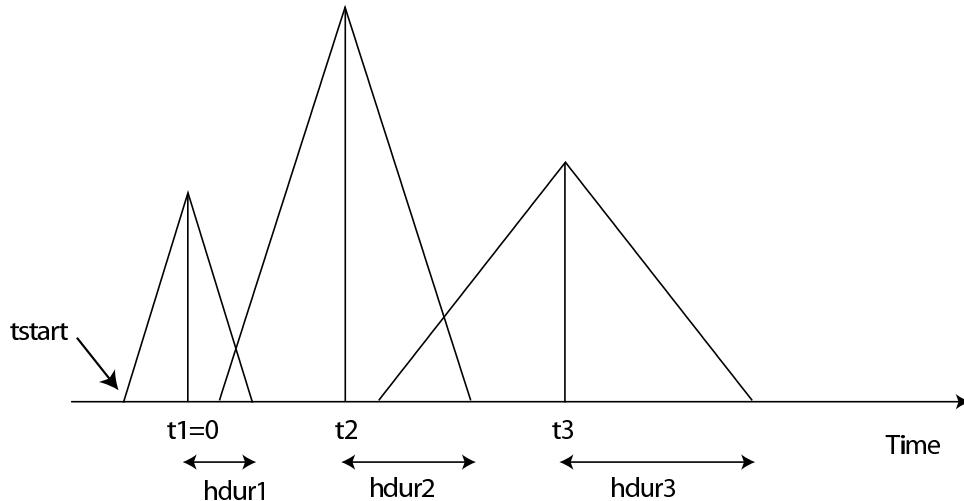


Figure 4.3: Example of timing for three sources. The center of the first source triangle is defined to be time zero. Note that this is NOT in general the hypocentral time, or the start time of the source (marked as `tstart`). The time shift parameter  $t_0$  in the SOURCE file would be  $t_1(= 0)$ ,  $t_2$ ,  $t_3$  in this case, and the half-duration parameter,  $f_0$ , would be  $hdur_1 = 1/f_{01}$ ,  $hdur_2 = 1/f_{02}$ ,  $hdur_3 = 1/f_{03}$  for the sources 1, 2, 3 respectively.

In section "# receiver line parameters for seismograms":

`SAVE_FORWARD` determines if the last frame of a forward simulation is saved (`.true.`) or not (`.false.`)

In section "# define models....":

There are three possible types of models:

```
I: (model_number 1 rho Vp Vs 0 0 QKappa Qmu 0 0 0 0 0 0) or
II: (model_number 2 rho c11 c13 c15 c33 c35 c55 c12 c23 c25 0 0 0) or
III: (model_number 3 rhos rhof phi c kxx kxz kzz Ks Kf Kfr etaf mufr Qmu).
```

For isotropic elastic/acoustic material use I and set Vs to zero to make a given model acoustic, for anisotropic elastic use II, and for isotropic poroelastic material use III. The mesh can contain acoustic, elastic, and poroelastic models simultaneously.

For anisotropic elastic media the last three parameters,  $c_{12}$   $c_{23}$   $c_{25}$ , are used only when the user asks the code to compute pressure for display or seismogram recording purposes. Thus, if you do not know these parameters for your anisotropic material and/or if you do not plan to display or record pressure you can ignore them and set them to zero. When pressure is used these three parameters are needed because the code needs to compute  $\sigma_{yy}$ , which is not equal to zero in the plane strain formulation.

**rho\_s** = solid density

**rho\_f** = fluid density

**phi** = porosity

```

tort = tortuosity
permxx = xx component of permeability tensor
permxz = xz,zx components of permeability tensor
permzz = zz component of permeability tensor
kappa_s = solid bulk modulus
kappa_f = fluid bulk modulus
kappa_fr = frame bulk modulus
eta_f = fluid viscosity
mu_fr = frame shear modulus
Qmu = shear quality factor

```

Note: for the poroelastic case, **mu\_s** is irrelevant. For details on the poroelastic theory see ?.

`get_poroelastic_velocities.f90` allows to compute **cpI**, **cpII**, and **cs** function of the source dominant frequency. Notice that for this calculation we use **permxx** and the dominant frequency of the first source, **f0(1)**. Caution if you use several sources with different frequencies and if you consider anisotropic permeability.

## 4.4 How to set plane waves as initial conditions

To simulate propagation of incoming plane waves in the simulation domain, initial conditions based on analytical formulae of plane waves in homogeneous model need to be set. No additional body or boundary forces are required. To set up this scenario:

`Par_file`:

- switch on **initialfield** = .true.
- at this point setting **add\_bielak\_condition** does not seem to help with absorbing boundaries, therefore, it should be turned off.

`SOURCE`:

- **zs** has to be the same as the height of the simulation domain defined in **interfacesfile**.
- **xs** is the *x*-coordinate of the intersection of the initial plane wave front with the free surface.
- **source\_type** = 1 for a plane P wave, 2 for a plane SV wave, 3 for a Rayleigh wave.
- **angleforce** can be negative to indicate a plane wave incident from the right (instead of the left)

## 4.5 How to choose the time step

Three different explicit conditionally-stable time schemes can be used for elastic, acoustic (fluid) or coupled elastic/acoustic media: the Newmark method, the low-dissipation and low-dispersion fourth-order six-stage Runge-Kutta method (LDDRK4-6) presented in ?, and the classical fourth-order four-stage Runge-Kutta (RK4) method. Currently the last two methods are not implemented for poroelastic media. According to ? and ?, with different degrees  $N = NGLLX - 1$  of the GLL basis functions the CFL bounds are given in the following tables. Note that by default the SPECFEM solver uses  $NGLLX = 5$  and thus a degree  $N = 4$ , which is thus the value you should use in most cases in the following tables. You can directly compare these values with the value given in sentence ‘Max stability for P wave velocity’ in file `output_solver.txt` to see whether you set the correct  $\Delta t$  in `Par_file` or not. For elastic simulation, the CFL value given in `output_solver.txt` does not consider the  $V_p/V_s$  ratio, but the CFL limit slightly decreases when  $V_p/V_s$  increases. In viscoelastic simulations the CFL limit does not change compared to the elastic case because we use a rational approximation of a constant quality factor  $Q$ , which has no attenuation effect on zero-frequency waves. Additionally, if you use C-PML absorbing layers in your simulations, which are implemented for the Newmark and LDDRK4-6 techniques but not for the classical RK4), the CFL upper limit decreases to approximately 95% of the limit without absorbing layers in the case of Newmark and to 85% in the case of LDDRK4-6.

Table 4.1: CFL upper bound for an acoustic (fluid) simulation.

Degree $N$	Newmark	LDDRK4-6	RK4
1	0.709	1.349	1.003
2	0.577	1.098	0.816
3	0.593	1.129	0.839
<b>4</b>	<b>0.604</b>	<b>1.150</b>	<b>0.854</b>
5	0.608	1.157	0.860
6	0.608	1.157	0.860
7	0.608	1.157	0.860
8	0.607	1.155	0.858
9	0.607	1.155	0.858
10	0.607	1.155	0.858

Table 4.2: CFL upper bound for an elastic simulation with  $V_p/V_s = \sqrt{2}$ .

Degree $N$	Newmark	LDDRK4-6	RK4
1	0.816	1.553	1.154
2	0.666	1.268	0.942
3	0.684	1.302	0.967
<b>4</b>	<b>0.697</b>	<b>1.327</b>	<b>0.986</b>
5	0.700	1.332	0.990
6	0.700	1.332	0.990
7	0.700	1.332	0.990
8	0.699	1.330	0.989
9	0.698	1.328	0.987
10	0.698	1.328	0.987

# Chapter 5

## Adjoint Simulations

### 5.1 How to obtain Finite Sensitivity Kernels

1. Run a forward simulation:

- SIMULATION\_TYPE = 1
- SAVE\_FORWARD = .true.
- seismotype = 1 (we need to save the displacement fields to later on derive the adjoint source. Note: if the user forgets it, the program corrects it when reading the proper SIMULATION\_TYPE and SAVE\_FORWARD combination and a warning message appears in the output file)

Important output files (for example, for the elastic case, P-SV waves):

- absorb\_elastic\_bottom\*\*\*\*.bin
- absorb\_elastic\_left\*\*\*\*.bin
- absorb\_elastic\_right\*\*\*\*.bin
- absorb\_elastic\_top\*\*\*\*.bin
- lastframe\_elastic\*\*\*\*.bin
- S\*\*\*\*.AA.BXX.semd
- S\*\*\*\*.AA.BXZ.semd

2. Define the adjoint source:

- Use adj\_seismogram.f90
- Edit to update NSTEP, nrec, t0, deltat, and the position of the cut to pick any given phase if needed (tstart,tend), add the right number of stations, and put one component of the source to zero if needed.
- The output files of adj\_seismogram.f90 are S\*\*\*\*.AA.BXX.adj and S\*\*\*\*.AA.BXZ.adj, for P-SV waves (and S\*\*\*\*.AA.BXY.adj, for SH (membrane) waves). Note that you will need these three files (S\*\*\*\*.AA.BXX.adj, S\*\*\*\*.AA.BXY.adj and S\*\*\*\*.AA.BXZ.adj) to be present in the SEM/ directory together with the absorb\_elastic\_\*\*\*\*.bin and lastframe\_elastic.bin files to be read when running the adjoint simulation.

3. Run the adjoint simulation:

- Make sure that the adjoint source files absorbing boundaries and last frame files are in the OUTPUT\_FILES/ directory.
- SIMULATION\_TYPE = 3
- SAVE\_FORWARD = .false.

Output files (for example for the elastic case):

- `snapshot_rho_kappa_mu*****`
- `snapshot_rhop_alpha_beta*****`

which are the primary moduli kernels and the phase velocities kernels respectively, in ascii format and at the local level, that is as “`kernels(i, j, ispec)`”.

## 5.2 Remarks about adjoint runs and solving inverse problems

SPECFEM2D can produce the gradient of the misfit function for a tomographic inversion, but options for using the gradient within an iterative inversion are left to the user (e.g., conjugate-gradient, steepest descent). The plan is to include some examples in the future.

The algorithm is simple:

1. calculate the forward wave field  $\mathbf{s}(x, t)$
2. calculate the adjoint wave field  $\mathbf{s}^\dagger(x, t)$
3. calculate their interaction  $\mathbf{s}(x, t) \cdot \mathbf{s}^\dagger(x, T - t)$  (these symbolic, temporal and spatial derivatives should be included)
4. integrate the interactions, which is summation in the code.

That is all. Step 3 has some tricks in implementation, but which can be skipped by regular users.

If you look into SPECFEM2D, besides “`rhop_ac_k1`” and “`rho_ac_k1`”, there are more variables such as “`kappa_ac_k1`” and “`rho_el_k1`” etc. “`rho`” denotes density  $\rho$  (“`kappa`” for bulk modulus  $\kappa$  etc.), “`ac`” denotes acoustic (“`el`” for elastic), “`k1`” means kernel (and you may find “`k`” as well, which is the interaction at each time step, i.e., before doing time integration).

## 5.3 Caution

Please note that:

- at the moment, adjoint simulations do not support anisotropy, attenuation, and viscous damping.
- you will need `S*****.AA.BXX.adj`, `S*****.AA.BXY.adj` and `S*****.AA.BXZ.adj` to be present in directory SEM/ even if you are just running an acoustic or poroelastic adjoint simulation.
  - `S*****.AA.BXX.adj` is the only relevant component for an acoustic case.
  - `S*****.AA.BXX.adj` and `S*****.AA.BXZ.adj` are the only relevant components for a poroelastic case.

## Chapter 6

# Oil and gas industry simulations

The SPECFEM2D package provides compatibility with industrial (oil and gas industry) types of simulations. These features include importing Seismic Unix (SU) format wavespeed models into SPECFEM2D, output of seismograms in SU format with a few key parameters defined in the trace headers and reading adjoint sources in SU format etc. There is one example given in EXAMPLES/INDUSTRIAL\_FORMAT, which you can follow.

We also changed the relationship between adjoint potential and adjoint displacement in fluid region (the relationship between forward potential and forward displacement remains the same as previously defined). The new definition is critical when there are adjoint sources (in other words, receivers) in the acoustic domain, and is the direct consequence of the optimization problem.

$$\mathbf{s} \equiv \frac{1}{\rho} \nabla \phi$$
$$p \equiv -\kappa (\nabla \cdot \mathbf{s}) = -\partial_t^2 \phi$$

$$\partial_t^2 \mathbf{s}^\dagger \equiv -\frac{1}{\rho} \nabla \phi^\dagger$$
$$p^\dagger \equiv -\kappa (\nabla \cdot \mathbf{s}^\dagger) = \phi^\dagger$$

# Acknowledgments

The Gauss-Lobatto-Legendre subroutines in `gll_library.f90` are based in part on software libraries from the Massachusetts Institute of Technology, Department of Mechanical Engineering (Cambridge, Massachusetts, USA). The non-structured global numbering software was provided by Paul F. Fischer (Brown University, Providence, Rhode Island, USA, now at Argonne National Laboratory, USA).

Please e-mail your feedback, bug reports, questions, comments, and suggestions to the CIG Computational Seismology Mailing List ([cig-seismo@geodynamics.org](mailto:cig-seismo@geodynamics.org)).

# **Copyright**

Main historical authors: Dimitri Komatitsch and Jeroen Tromp

Princeton University, USA, and CNRS / University of Marseille, France

© Princeton University and CNRS / University of Marseille, July 2012

# Appendix A

## Troubleshooting

### FAQ

#### Regarding the structure of some of the database files

**Question:** Can anyone tell me what the columns of the SPECFEM2D boundary condition files in SPECFEM2D/DATA/Mesh\_canyon are?

SPECFEM2D/DATA/Mesh\_canyon/canyon\_absorbing\_surface\_file  
SPECFEM2D/DATA/Mesh\_canyon/canyon\_free\_surface\_file

**Answer:** `canyon_absorbing_surface_file` refers to parameters related to the absorbing conditions: The first number (180) is the number of absorbing elements (nelemabs in the code). Then the columns are:

column 1: the element number

column 2: the number of nodes of this element that form the absorbing surface

column 3: the first node

column 4: the second node

`canyon_free_surface_file` refers to the elements of the free surface (relevant for enforcing free surface condition for acoustic media): The first number (160) is the number of elements of the free surface. Then the columns are (similar to the absorbing case):

column 1: the element number

column 2: the number of nodes of this element that form the absorbing surface

column 3: the first node

column 4: the second node

Concerning the free surface description file, nodes/edges pertaining to elastic elements are discarded when the file is read (if for whatever reason it was simpler to include all the nodes/edges on one side of a studied area and that there are among them some elements that are elastic elements, only the nodes/edges of acoustic elements are kept).

These files are opened and read in `meshfem2D.F90` using subroutines `read_abs_surface()` and `read_acoustic_surface()`, which are in `part_unstruct.F90`

## **Appendix B**

## **License**

### **CeCILL FREE SOFTWARE LICENSE AGREEMENT**

Available online at [http://www.cecill.info/licences/Licence\\_CeCILL\\_V2-en.html](http://www.cecill.info/licences/Licence_CeCILL_V2-en.html)

#### **Notice**

This Agreement is a Free Software license agreement that is the result of discussions between its authors in order to ensure compliance with the two main principles guiding its drafting:

- firstly, compliance with the principles governing the distribution of Free Software: access to source code, broad rights granted to users,

- secondly, the election of a governing law, French law, with which it is conformant, both as regards the law of torts and intellectual property law, and the protection that it offers to both authors and holders of the economic rights over software.

The authors of the CeCILL license (CeCILL stands for Ce(a) C(nrs) I(nria) L(ogiciel) L(ibre)) are:

Commissariat à l'Energie Atomique - CEA, a public scientific, technical and industrial research establishment, having its principal place of business at 25 rue Leblanc, immeuble Le Ponant D, 75015 Paris, France.

Centre National de la Recherche Scientifique - CNRS, a public scientific and technological establishment, having its principal place of business at 3 rue Michel-Ange, 75794 Paris cedex 16, France.

Institut National de Recherche en Informatique et en Automatique - Inria, a public scientific and technological establishment, having its principal place of business at Domaine de Voluceau, Rocquencourt, BP 105, 78153 Le Chesnay cedex, France.

#### **Preamble**

The purpose of this Free Software license agreement is to grant users the right to modify and redistribute the software governed by this license within the framework of an open source distribution model.

The exercising of these rights is conditional upon certain obligations for users so as to preserve this status for all subsequent redistributions.

In consideration of access to the source code and the rights to copy, modify and redistribute granted by the license, users are provided only with a limited warranty and the software's author, the holder of the economic rights, and the successive licensors only have limited liability.

In this respect, the risks associated with loading, using, modifying and/or developing or reproducing the software by the user are brought to the user's attention, given its Free Software status, which may make it complicated to use, with the result that its use is reserved for developers and experienced professionals having in-depth computer knowledge. Users are therefore encouraged to load and test the suitability of the software as regards their requirements in conditions enabling the security of their systems and/or data to be ensured and, more generally, to use and operate it in the same conditions of security. This Agreement may be freely reproduced and published, provided it is not altered, and that no provisions are either added or removed herefrom.

This Agreement may apply to any or all software for which the holder of the economic rights decides to submit the use thereof to its provisions.

#### **Article 1 - DEFINITIONS**

For the purpose of this Agreement, when the following expressions commence with a capital letter, they shall have the following meaning:

Agreement: means this license agreement, and its possible subsequent versions and annexes.

**Software:** means the software in its Object Code and/or Source Code form and, where applicable, its documentation, "as is" when the Licensee accepts the Agreement.

**Initial Software:** means the Software in its Source Code and possibly its Object Code form and, where applicable, its documentation, "as is" when it is first distributed under the terms and conditions of the Agreement.

**Modified Software:** means the Software modified by at least one Contribution.

**Source Code:** means all the Software's instructions and program lines to which access is required so as to modify the Software.

**Object Code:** means the binary files originating from the compilation of the Source Code.

**Holder:** means the holder(s) of the economic rights over the Initial Software.

**Licensee:** means the Software user(s) having accepted the Agreement.

**Contributor:** means a Licensee having made at least one Contribution.

**Licensor:** means the Holder, or any other individual or legal entity, who distributes the Software under the Agreement.

**Contribution:** means any or all modifications, corrections, translations, adaptations and/or new functions integrated into the Software by any or all Contributors, as well as any or all Internal Modules.

**Module:** means a set of sources files including their documentation that enables supplementary functions or services in addition to those offered by the Software.

**External Module:** means any or all Modules, not derived from the Software, so that this Module and the Software run in separate address spaces, with one calling the other when they are run.

**Internal Module:** means any or all Module, connected to the Software so that they both execute in the same address space.

**GNU GPL:** means the GNU General Public License version 2 or any subsequent version, as published by the Free Software Foundation Inc.

**Parties:** mean both the Licensee and the Licensor.

These expressions may be used both in singular and plural form.

## Article 2 - PURPOSE

The purpose of the Agreement is the grant by the Licensor to the Licensee of a non-exclusive, transferable and worldwide license for the Software as set forth in Article 5 hereinafter for the whole term of the protection granted by the rights over said Software.

## Article 3 - ACCEPTANCE

3.1 The Licensee shall be deemed as having accepted the terms and conditions of this Agreement upon the occurrence of the first of the following events:

(i) loading the Software by any or all means, notably, by downloading from a remote server, or by loading from a physical medium;

(ii) the first time the Licensee exercises any of the rights granted hereunder. 3.2 One copy of the Agreement, containing a notice relating to the characteristics of the Software, to the limited warranty, and to the fact that its use is restricted to experienced users has been provided to the Licensee prior to its acceptance as set forth in Article 3.1 hereinabove, and the Licensee hereby acknowledges that it has read and understood it.

## Article 4 - EFFECTIVE DATE AND TERM

### 4.1 EFFECTIVE DATE

The Agreement shall become effective on the date when it is accepted by the Licensee as set forth in Article 3.1.

### 4.2 TERM

The Agreement shall remain in force for the entire legal term of protection of the economic rights over the Software.

## Article 5 - SCOPE OF RIGHTS GRANTED

The Licensor hereby grants to the Licensee, who accepts, the following rights over the Software for any or all use, and for the term of the Agreement, on the basis of the terms and conditions set forth hereinafter.

Besides, if the Licensor owns or comes to own one or more patents protecting all or part of the functions of the Software or of its components, the Licensor undertakes not to enforce the rights granted by these patents against successive Licensees using, exploiting or modifying the Software. If these patents are transferred, the Licensor undertakes to have the transferees subscribe to the obligations set forth in this paragraph.

### 5.1 RIGHT OF USE

The Licensee is authorized to use the Software, without any limitation as to its fields of application, with it being hereinafter specified that this comprises:

permanent or temporary reproduction of all or part of the Software by any or all means and in any or all form.

loading, displaying, running, or storing the Software on any or all medium.

entitlement to observe, study or test its operation so as to determine the ideas and principles behind any or all constituent elements of said Software. This shall apply when the Licensee carries out any or all loading, displaying, running, transmission or storage operation as regards the Software, that it is entitled to carry out hereunder.

### 5.2 ENTITLEMENT TO MAKE CONTRIBUTIONS

The right to make Contributions includes the right to translate, adapt, arrange, or make any or all modifications to the Software, and the right to reproduce the resulting software.

The Licensee is authorized to make any or all Contributions to the Software provided that it includes an explicit notice that it is the author of said Contribution and indicates the date of the creation thereof.

### 5.3 RIGHT OF DISTRIBUTION

In particular, the right of distribution includes the right to publish, transmit and communicate the Software to the general public on any or all medium, and by any or all means, and the right to market, either in consideration of a fee, or free of charge, one or more copies of the Software by any means.

The Licensee is further authorized to distribute copies of the modified or unmodified Software to third parties according to the terms and conditions set forth hereinafter.

#### 5.3.1 DISTRIBUTION OF SOFTWARE WITHOUT MODIFICATION

The Licensee is authorized to distribute true copies of the Software in Source Code or Object Code form, provided that said distribution complies with all the provisions of the Agreement and is accompanied by:

a copy of the Agreement,

a notice relating to the limitation of both the Licensor's warranty and liability as set forth in Articles 8 and 9,

and that, in the event that only the Object Code of the Software is redistributed, the Licensee allows future Licensees unhindered access to the full Source Code of the Software by indicating how to access it, it being understood that the additional cost of acquiring the Source Code shall not exceed the cost of transferring the data.

#### 5.3.2 DISTRIBUTION OF MODIFIED SOFTWARE

When the Licensee makes a Contribution to the Software, the terms and conditions for the distribution of the resulting Modified Software become subject to all the provisions of this Agreement.

The Licensee is authorized to distribute the Modified Software, in source code or object code form, provided that said distribution complies with all the provisions of the Agreement and is accompanied by:

a copy of the Agreement,

a notice relating to the limitation of both the Licensor's warranty and liability as set forth in Articles 8 and 9,

and that, in the event that only the object code of the Modified Software is redistributed, the Licensee allows future Licensees unhindered access to the full source code of the Modified Software by indicating how to access it, it being understood that the additional cost of acquiring the source code shall not exceed the cost of transferring the data.

#### 5.3.3 DISTRIBUTION OF EXTERNAL MODULES

When the Licensee has developed an External Module, the terms and conditions of this Agreement do not apply to said External Module, that may be distributed under a separate license agreement.

#### 5.3.4 COMPATIBILITY WITH THE GNU GPL

The Licensee can include a code that is subject to the provisions of one of the versions of the GNU GPL in the Modified or unmodified Software, and distribute that entire code under the terms of the same version of the GNU GPL.

The Licensee can include the Modified or unmodified Software in a code that is subject to the provisions of one of the versions of the GNU GPL, and distribute that entire code under the terms of the same version of the GNU GPL.

### Article 6 - INTELLECTUAL PROPERTY

#### 6.1 OVER THE INITIAL SOFTWARE

The Holder owns the economic rights over the Initial Software. Any or all use of the Initial Software is subject to compliance with the terms and conditions under which the Holder has elected to distribute its work and no one shall be entitled to modify the terms and conditions for the distribution of said Initial Software.

The Holder undertakes that the Initial Software will remain ruled at least by this Agreement, for the duration set forth in Article 4.2.

#### 6.2 OVER THE CONTRIBUTIONS

The Licensee who develops a Contribution is the owner of the intellectual property rights over this Contribution as defined by applicable law.

#### 6.3 OVER THE EXTERNAL MODULES

The Licensee who develops an External Module is the owner of the intellectual property rights over this External Module as defined by applicable law and is free to choose the type of agreement that shall govern its distribution.

#### 6.4 JOINT PROVISIONS

The Licensee expressly undertakes:

not to remove, or modify, in any manner, the intellectual property notices attached to the Software;  
to reproduce said notices, in an identical manner, in the copies of the Software modified or not.

The Licensee undertakes not to directly or indirectly infringe the intellectual property rights of the Holder and/or Contributors on the Software and to take, where applicable, vis-a-vis its staff, any and all measures required to ensure respect of said intellectual property rights of the Holder and/or Contributors.

#### Article 7 - RELATED SERVICES

7.1 Under no circumstances shall the Agreement oblige the Licenser to provide technical assistance or maintenance services for the Software.

However, the Licensor is entitled to offer this type of services. The terms and conditions of such technical assistance, and/or such maintenance, shall be set forth in a separate instrument. Only the Licensor offering said maintenance and/or technical assistance services shall incur liability therefor.

7.2 Similarly, any Licensor is entitled to offer to its licensees, under its sole responsibility, a warranty, that shall only be binding upon itself, for the redistribution of the Software and/or the Modified Software, under terms and conditions that it is free to decide. Said warranty, and the financial terms and conditions of its application, shall be subject of a separate instrument executed between the Licensor and the Licensee.

#### Article 8 - LIABILITY

8.1 Subject to the provisions of Article 8.2, the Licensee shall be entitled to claim compensation for any direct loss it may have suffered from the Software as a result of a fault on the part of the relevant Licensor, subject to providing evidence thereof.

8.2 The Licensor's liability is limited to the commitments made under this Agreement and shall not be incurred as a result of in particular: (i) loss due the Licensee's total or partial failure to fulfill its obligations, (ii) direct or consequential loss that is suffered by the Licensee due to the use or performance of the Software, and (iii) more generally, any consequential loss. In particular the Parties expressly agree that any or all pecuniary or business loss (i.e. loss of data, loss of profits, operating loss, loss of customers or orders, opportunity cost, any disturbance to business activities) or any or all legal proceedings instituted against the Licensee by a third party, shall constitute consequential loss and shall not provide entitlement to any or all compensation from the Licensor.

#### Article 9 - WARRANTY

9.1 The Licensee acknowledges that the scientific and technical state-of-the-art when the Software was distributed did not enable all possible uses to be tested and verified, nor for the presence of possible defects to be detected. In this respect, the Licensee's attention has been drawn to the risks associated with loading, using, modifying and/or developing and reproducing the Software which are reserved for experienced users.

The Licensee shall be responsible for verifying, by any or all means, the suitability of the product for its requirements, its good working order, and for ensuring that it shall not cause damage to either persons or properties.

9.2 The Licensor hereby represents, in good faith, that it is entitled to grant all the rights over the Software (including in particular the rights set forth in Article 5).

9.3 The Licensee acknowledges that the Software is supplied "as is" by the Licensor without any other express or tacit warranty, other than that provided for in Article 9.2 and, in particular, without any warranty as to its commercial value, its secured, safe, innovative or relevant nature.

Specifically, the Licensor does not warrant that the Software is free from any error, that it will operate without interruption, that it will be compatible with the Licensee's own equipment and software configuration, nor that it will meet the Licensee's requirements.

9.4 The Licensor does not either expressly or tacitly warrant that the Software does not infringe any third party intellectual property right relating to a patent, software or any other property right. Therefore, the Licensor disclaims any and all liability towards the Licensee arising out of any or all proceedings for infringement that may be instituted in respect of the use, modification and redistribution of the Software. Nevertheless, should such proceedings be instituted against the Licensee, the Licensor shall provide it with technical and legal assistance for its defense. Such technical and legal assistance shall be decided on a case-by-case basis between the relevant Licensor and the Licensee pursuant to a memorandum of understanding. The Licensor disclaims any and all liability as regards the Licensee's use of the name of the Software. No warranty is given as regards the existence of prior rights over the name of the Software or as regards the existence of a trademark.

#### Article 10 - TERMINATION

10.1 In the event of a breach by the Licensee of its obligations hereunder, the Licensor may automatically terminate this Agreement thirty (30) days after notice has been sent to the Licensee and has remained ineffective.

10.2 A Licensee whose Agreement is terminated shall no longer be authorized to use, modify or distribute the Software. However, any licenses that it may have granted prior to termination of the Agreement shall remain valid subject to their having been granted in compliance with the terms and conditions hereof.

#### Article 11 - MISCELLANEOUS

##### 11.1 EXCUSABLE EVENTS

Neither Party shall be liable for any or all delay, or failure to perform the Agreement, that may be attributable to an event of force majeure, an act of God or an outside cause, such as defective functioning or interruptions of the electricity or telecommunications networks, network paralysis following a virus attack, intervention by government authorities, natural disasters, water damage, earthquakes, fire, explosions, strikes and labor unrest, war, etc.

11.2 Any failure by either Party, on one or more occasions, to invoke one or more of the provisions hereof, shall under no circumstances be interpreted as being a waiver by the interested Party of its right to invoke said provision(s) subsequently.

11.3 The Agreement cancels and replaces any or all previous agreements, whether written or oral, between the Parties and having the same purpose, and constitutes the entirety of the agreement between said Parties concerning said purpose. No supplement or modification to the terms and conditions hereof shall be effective as between the Parties unless it is made in writing and signed by their duly authorized representatives.

11.4 In the event that one or more of the provisions hereof were to conflict with a current or future applicable act or legislative text, said act or legislative text shall prevail, and the Parties shall make the necessary amendments so as to comply with said act or legislative text. All other provisions shall remain effective. Similarly, invalidity of a provision of the Agreement, for any reason whatsoever, shall not cause the Agreement as a whole to be invalid.

##### 11.5 LANGUAGE

The Agreement is drafted in both French and English and both versions are deemed authentic.

#### Article 12 - NEW VERSIONS OF THE AGREEMENT

12.1 Any person is authorized to duplicate and distribute copies of this Agreement.

12.2 So as to ensure coherence, the wording of this Agreement is protected and may only be modified by the authors of the License, who reserve the right to periodically publish updates or new versions of the Agreement, each with a separate number. These subsequent versions may address new issues encountered by Free Software.

12.3 Any Software distributed under a given version of the Agreement may only be subsequently distributed under the same version of the Agreement or a subsequent version, subject to the provisions of Article 5.3.4.

#### Article 13 - GOVERNING LAW AND JURISDICTION

13.1 The Agreement is governed by French law. The Parties agree to endeavor to seek an amicable solution to any disagreements or disputes that may arise during the performance of the Agreement.

13.2 Failing an amicable solution within two (2) months as from their occurrence, and unless emergency proceedings are necessary, the disagreements or disputes shall be referred to the Paris Courts having jurisdiction, by the more diligent Party.

Version 2.0 dated 2006-09-05.