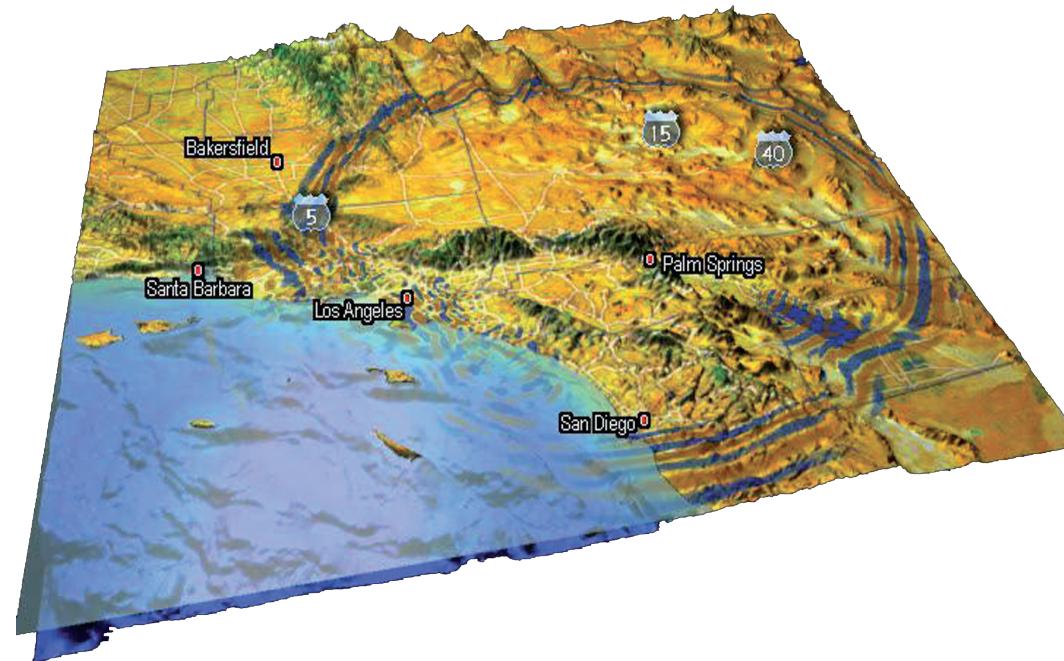


COMPUTATIONAL INFRASTRUCTURE FOR GEODYNAMICS (CIG)  
PRINCETON UNIVERSITY (USA)  
CNRS and UNIVERSITY OF MARSEILLE (FRANCE)  
ETH ZÜRICH (SWITZERLAND)

# SPECFEM 3D

## Cartesian

User Manual  
Version 3.0



Aix\*Marseille  
université **ETH** zürich

# **SPECFEM3D Cartesian**

## **User Manual**

©Princeton University (USA), CNRS and University of Marseille (France),  
and ETH Zürich (Switzerland)  
Version 3.0

December 23, 2016

## Authors

The SPECFEM3D package was first developed by Dimitri Komatitsch and Jean-Pierre Vilotte at Institut de Physique du Globe (IPGP) in Paris, France from 1995 to 1997 and then by Dimitri Komatitsch and Jeroen Tromp at Harvard University and Caltech, USA, starting in 1998. 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, Michael Afanasiev, Jean-Paul (Pablo) Ampuero, Kangchen Bai, Piero Basini, Céline Blitz, Alexis Bottero, Ebru Bozdağ, Emanuele Casarotti, Joseph Charles, Min Chen, Paul Cristini, Clément Durochat, Percy Galvez, Dominik Göddeke, Vala Hjörleifsdóttir, Sue Kientz, Dimitri Komatitsch, Jesús Labarta, Nicolas Le Goff, Pieyre Le Loher, Matthieu Lefebvre, Qinya Liu, Youshan Liu, David Luet, Yang Luo, Alessia Maggi, Federica Magnoni, Roland Martin, René Matzen, Dennis McRitchie, Matthias Meschede, Peter Messmer, David Michéa, Vadim Monteiller, Surendra Nadh Somala, Tarje Nissen-Meyer, Daniel Peter, Kevin Pouget, Max Rietmann, Elliott Sales de Andrade, Brian Savage, Bernhard Schuberth, Anne Sieminski, James Smith, Leif Strand, Carl Tape, Jeroen Tromp, Brice Videau, Jean-Pierre Vilotte, Zhinan Xie, Chang-Hua Zhang, Hejun Zhu.

The cover graphic of the manual was created by Santiago Lombeyda from Caltech's Center for Advanced Computing Research (CACR), USA, with free satellite clipart pictures from <http://www.4vector.com> and <http://www.clker.com> added to it.

The code is released open-source under the GNU 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>6</b>
1.1 Citation . . . . .	7
1.2 Support . . . . .	9
<b>2 Getting Started</b>	<b>10</b>
2.1 Adding OpenMP support in addition to MPI . . . . .	12
2.2 Compiling on an IBM BlueGene . . . . .	12
2.3 Visualizing the subroutine calling tree of the source code . . . . .	13
2.4 Using the ADIOS library for I/O . . . . .	15
2.5 Becoming a developer of the code, or making small modifications in the source code . . . . .	15
<b>3 Mesh Generation</b>	<b>16</b>
3.1 Meshing with CUBIT . . . . .	16
3.1.1 Creating the Mesh with CUBIT . . . . .	17
3.1.2 Exporting the Mesh with <code>run_cubit2specfem3d.py</code> . . . . .	18
3.1.3 Partitioning the Mesh with <code>xdecompose_mesh</code> . . . . .	22
3.2 Meshing with <code>xmeshfem3D</code> . . . . .	23
<b>4 Creating the Distributed Databases</b>	<b>28</b>
4.1 Main parameter file <code>Par_file</code> . . . . .	28
4.2 Choosing the time step <code>DT</code> . . . . .	34
<b>5 Running the Solver <code>xspecfem3D</code></b>	<b>36</b>
5.1 Note on the viscoelastic model used . . . . .	42
<b>6 Kinematic and dynamic fault sources</b>	<b>43</b>
6.1 Mesh Generation with Split Nodes . . . . .	43
6.2 CUBIT-Python Scripts for Faults . . . . .	45
6.3 Examples . . . . .	46
6.4 Sign Convention for Fault Quantities . . . . .	46
6.5 Input Files . . . . .	47
6.6 Setting the Kelvin-Voigt Damping Parameter . . . . .	49
6.7 Output Files . . . . .	50
6.8 Post-processing and Visualization . . . . .	51
<b>7 Adjoint Simulations</b>	<b>52</b>
7.1 Adjoint Simulations for Sources Only (not for the Model) . . . . .	52
7.2 Adjoint Simulations for Finite-Frequency Kernels (Kernel Simulation) . . . . .	53

<b>8 Doing tomography based on the sensitivity kernels obtained</b>	<b>55</b>
8.1 Tomographic full waveform inversion (FWI) / imaging using the sensitivity kernels obtained . . . . .	55
8.1.1 Principle . . . . .	55
8.1.2 Computation of the gradient based on the adjoint method . . . . .	56
8.2 Tomographic tools . . . . .	58
8.2.1 Summing kernels . . . . .	59
8.2.2 Smoothing and post-processing . . . . .	61
8.2.3 Model updating . . . . .	62
8.3 OLD VERSION OF THE SECTION, WILL SOON BE IMPROVED AND ENRICHED . . . . .	63
<b>9 Noise Cross-correlation Simulations</b>	<b>64</b>
9.1 Input Parameter Files . . . . .	64
9.2 Noise Simulations: Step by Step . . . . .	65
9.2.1 Pre-simulation . . . . .	65
9.2.2 Simulations . . . . .	66
9.2.3 Post-simulation . . . . .	67
9.3 Example . . . . .	67
<b>10 Gravity integral calculations for the gravity field of an Earth model</b>	<b>68</b>
<b>11 Graphics</b>	<b>69</b>
11.1 Meshes . . . . .	69
11.2 Movies . . . . .	69
11.2.1 Movie Surface and Shakemaps . . . . .	70
11.2.2 Movie Volume . . . . .	70
11.3 Finite-Frequency Kernels . . . . .	72
<b>12 Running through a Scheduler</b>	<b>75</b>
<b>13 Changing the Model</b>	<b>76</b>
13.1 Using external tomographic Earth models . . . . .	76
13.2 External (an)elastic Models . . . . .	78
13.3 Anisotropic Models . . . . .	79
13.4 Using external SEP models . . . . .	79
<b>14 Post-Processing Scripts</b>	<b>80</b>
14.1 Process Data and Synthetics . . . . .	80
14.1.1 Data processing script <code>process_data.pl</code> . . . . .	80
14.1.2 Synthetics processing script <code>process_syn.pl</code> . . . . .	81
14.1.3 Script <code>rotate.pl</code> . . . . .	81
14.2 Collect Synthetic Seismograms . . . . .	81
14.3 Clean Local Database . . . . .	82
14.4 Plot Movie Snapshots and Synthetic Shakemaps . . . . .	82
14.4.1 Script <code>movie2gif.gmt.pl</code> . . . . .	82
14.4.2 Script <code>plot_shakemap.gmt.pl</code> . . . . .	82
14.5 Map Local Database . . . . .	82
<b>15 Information for developers of the code, and for people who want to learn how the technique works</b>	<b>83</b>
<b>Notes &amp; Acknowledgments</b>	<b>85</b>
<b>Copyright</b>	<b>86</b>
<b>Bibliography</b>	<b>87</b>
<b>A Reference Frame Convention</b>	<b>96</b>

<b>CONTENTS</b>	<b>5</b>
-----------------	----------

<b>B Channel Codes of Seismograms</b>	<b>98</b>
<b>C Troubleshooting</b>	<b>100</b>
<b>D License</b>	<b>103</b>

# Chapter 1

## Introduction

The software package SPECFEM3D Cartesian simulates seismic wave propagation at the local or regional scale and performs full waveform imaging (FWI) or adjoint tomography based upon the spectral-element method (SEM). The SEM is a continuous Galerkin technique [Tromp et al., 2008, Peter et al., 2011], which can easily be made discontinuous [Bernardi et al., 1994, Chaljub, 2000, Kopriva et al., 2002, Chaljub et al., 2003, Legay et al., 2005, Kopriva, 2006, Wilcox et al., 2010, Acosta Minolia and Kopriva, 2011]; it is then close to a particular case of the discontinuous Galerkin technique [Reed and Hill, 1973, Lesaint and Raviart, 1974, Arnold, 1982, Johnson and Pitkäranta, 1986, Bourdel et al., 1991, Falk and Richter, 1999, Hu et al., 1999, Cockburn et al., 2000, Giraldo et al., 2002, Rivière and Wheeler, 2003, Monk and Richter, 2005, Grote et al., 2006, Ainsworth et al., 2006, Bernacki et al., 2006, Dumbser and Käser, 2006, De Basabe et al., 2008, de la Puente et al., 2009, Wilcox et al., 2010, De Basabe and Sen, 2010, Étienne et al., 2010], with optimized efficiency because of its tensorized basis functions [Wilcox et al., 2010, Acosta Minolia and Kopriva, 2011]. In particular, it can accurately handle very distorted mesh elements [Oliveira and Seriani, 2011].

**In fluids, when gravity is turned off, SPECFEM3D uses the classical linearized Euler equation; thus if you have sharp local variations of density in the fluid (highly heterogeneous fluids in terms of density) or if density becomes extremely small in some regions of your model (e.g. for upper-atmosphere studies), before using the code please make sure the linearized Euler equation is a valid approximation in the case you want to study, and/or see if you should turn gravity on. For more details on that see e.g. Jensen et al. [2011].**

It has very good accuracy and convergence properties [Maday and Patera, 1989, Seriani and Priolo, 1994, Deville et al., 2002, Cohen, 2002, De Basabe and Sen, 2007, Seriani and Oliveira, 2008, Ainsworth and Wajid, 2009, 2010, Melvin et al., 2012]. 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 [Komatitsch et al., 2003, Tsuboi et al., 2003, Komatitsch et al., 2008, Carrington et al., 2008, Komatitsch et al., 2010b] as well as on clusters of GPU accelerating graphics cards [Komatitsch, 2011, Michéa and Komatitsch, 2010, Komatitsch et al., 2009, 2010a]. Tensor products inside each element can be optimized to reach very high efficiency [Deville et al., 2002], and mesh point and element numbering can be optimized to reduce processor cache misses and improve cache reuse [Komatitsch et al., 2008]. The SEM can also handle triangular (in 2D) or tetrahedral (in 3D) elements [Wingate and Boyd, 1996, Taylor and Wingate, 2000, Komatitsch et al., 2001, Cohen, 2002, Mercerat et al., 2006] 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. Benjemaa et al. [2007, 2009], de la Puente et al. [2009], Tago et al. [2010]) a continuous formulation is sufficient because material property contrasts are not drastic and thus conforming mesh doubling bricks can efficiently handle mesh size variations [Komatitsch and Tromp, 2002a, Komatitsch et al., 2004, Lee et al., 2008, 2009a,b].

For a detailed introduction to the SEM as applied to regional seismic wave propagation, please consult Peter et al. [2011], Tromp et al. [2008], Komatitsch and Vilotte [1998], Komatitsch and Tromp [1999], Chaljub et al. [2007] and in particular Lee et al. [2009b,a, 2008], Godinho et al. [2009], van Wijk et al. [2004], Komatitsch et al. [2004]. A

detailed theoretical analysis of the dispersion and stability properties of the SEM is available in Cohen [2002], De Basabe and Sen [2007], Seriani and Oliveira [2007], Seriani and Oliveira [2008] and Melvin et al. [2012].

Effects due to lateral variations in compressional-wave speed, shear-wave speed, density, a 3D crustal model, topography and bathymetry are included. The package can accommodate full 21-parameter anisotropy (see Chen and Tromp [2007]) as well as lateral variations in attenuation [Savage et al., 2010]. Adjoint capabilities and finite-frequency kernel simulations are included [Tromp et al., 2008, Peter et al., 2011, Liu and Tromp, 2006, Fichtner et al., 2009a, Virieux and Operto, 2009].

The SEM was originally developed in computational fluid dynamics [Patera, 1984, Maday and Patera, 1989] 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 Cohen et al. [1993], Komatitsch [1997], Faccioli et al. [1997], Casadei and Gabellini [1997], Komatitsch and Vilotte [1998] and Komatitsch and Tromp [1999], whereas applications involving Chebyshev basis functions and a non-diagonal mass matrix include Seriani and Priolo [1994], Priolo et al. [1994] and Seriani et al. [1995]. In the Legendre version that we use in SPECFEM the mass matrix is purposely slightly inexact but diagonal (but can be made exact if needed, see Teukolsky [2015]), while in the Chebyshev version it is exact but non diagonal.

All SPECFEM3D 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) [Gropp et al., 1994, Pacheco, 1997].

SPECFEM3D won the Gordon Bell award for best performance at the SuperComputing 2003 conference in Phoenix, Arizona (USA) (see Komatitsch et al. [2003] and [www.sc-conference.org/sc2003/nrfinalaward.html](http://www.sc-conference.org/sc2003/nrfinalaward.html)). It was a finalist again in 2008 for a run at 0.16 petaflops (sustained) on 149,784 processors of the ‘Jaguar’ Cray XT5 system at Oak Ridge National Laboratories (USA) [Carrington et al., 2008]. It also won the BULL Joseph Fourier supercomputing award in 2010.

It reached the sustained one petaflop performance level for the first time in February 2013 on the Blue Waters Cray supercomputer at the National Center for Supercomputing Applications (NCSA), located at the University of Illinois at Urbana-Champaign (USA).

This new release of the code includes Convolution or Auxiliary Differential Equation Perfectly Matched absorbing Layers (C-PML or ADE-PML) [Martin et al., 2008b,c, Martin and Komatitsch, 2009, Martin et al., 2010, Komatitsch and Martin, 2007]. It also includes support for GPU graphics card acceleration [Komatitsch, 2011, Michéa and Komatitsch, 2010, Komatitsch et al., 2009, 2010a].

The next release of the code will use the PT-SCOTCH parallel and threaded version of SCOTCH for mesh partitioning instead of the serial version.

SPECFEM3D Cartesian includes coupled fluid-solid domains and adjoint capabilities, which enables one to address seismological inverse problems, but for linear rheologies only so far. To accommodate visco-plastic or non-linear rheologies, the reader can refer to the GeoELSE software package [Casadei and Gabellini, 1997, Stupazzini et al., 2009].

## 1.1 Citation

You can find all the references below in BIBTeXformat in file doc/USER\_MANUAL/bibliography.bib.

If you use SPECFEM3D Cartesian for your own research, please cite at least one of the following articles:

### Numerical simulations in general

Forward and adjoint simulations are described in detail in Tromp et al. [2008], Peter et al. [2011], Vai et al. [1999], Komatitsch et al. [2009, 2010a], Chaljub et al. [2007], Madec et al. [2009], Komatitsch et al. [2010b],

Carrington et al. [2008], Tromp et al. [2010a], Komatitsch et al. [2002], Komatitsch and Tromp [2002a,b, 1999] or Komatitsch and Vilotte [1998]. Additional aspects of adjoint simulations are described in Tromp et al. [2005], Liu and Tromp [2006], Tromp et al. [2008], Liu and Tromp [2008], Tromp et al. [2010a], Peter et al. [2011]. Domain decomposition is explained in detail in Martin et al. [2008a], and excellent scaling up to 150,000 processor cores is shown for instance in Carrington et al. [2008], Komatitsch et al. [2008], Martin et al. [2008a], Komatitsch et al. [2010a], Komatitsch [2011],

### **GPU computing**

Computing on GPU graphics cards for acoustic or seismic wave propagation applications is described in detail in Komatitsch [2011], Michéa and Komatitsch [2010], Komatitsch et al. [2009, 2010a].

If you use this new version, which has non blocking MPI for much better performance for medium or large runs, please cite at least one of these six articles, in which results of non blocking MPI runs are presented: Peter et al. [2011], Komatitsch et al. [2010a,b], Komatitsch [2011], Carrington et al. [2008], Martin et al. [2008a].

If you use the C-PML absorbing layer capabilities of the code, please cite at least one article written by the developers of the package, for instance:

- Xie et al. [2014],
- Xie et al. [2016].

If you use the UNDO\_ATTENUATION option of the code in order to produce full anelastic/viscoelastic sensitivity kernels, please cite at least one article written by the developers of the package, for instance (and in particular):

- Komatitsch et al. [2016].

More generally, if you use the attenuation (anelastic/viscoelastic) capabilities of the code, please cite at least one article written by the developers of the package, for instance:

- Komatitsch et al. [2016],
- Blanc et al. [2016].

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

- Tromp et al. [2008],
- Peter et al. [2011],
- Liu and Tromp [2006],
- Morency et al. [2009].

If you work on geophysical applications, you may be interested in citing some of these application articles as well, among others:

### **Southern California simulations**

Komatitsch et al. [2004], Krishnan et al. [2006a,b].

If you use the 3D southern California model, please cite Süss and Shaw [2003] (Los Angeles model), Lovely et al. [2006] (Salton Trough), and Hauksson [2000] (southern California). The Moho map was determined by Zhu and Kanamori [2000]. The 1D SoCal model was developed by Dreger and Helmberger [1990].

### **Anisotropy**

Chen and Tromp [2007], Ji et al. [2005], Chevrot et al. [2004], Favier et al. [2004], Ritsema et al. [2002], Tromp and Komatitsch [2000].

### **Attenuation**

Savage et al. [2010], Komatitsch and Tromp [2002a, 1999].

### **Topography**

Lee et al. [2009b,a, 2008], Godinho et al. [2009], van Wijk et al. [2004].

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 SPECFEM3D\_Cartesian software package, type this:

```
git clone --recursive --branch devel https://github.com/geodynamics/specfem3d.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 MPIFC90 in your `.bash_profile` or your `.cshrc` file):

```
./configure FC=gfortran CC=gcc
```

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

```
./configure FC=gfortran CC=gcc MPIFC=mpif90 --with-mpi
```

You can replace the GNU compilers above (gfortran and gcc) with other compilers if you want to; for instance for Intel ifort and icc use FC=ifort CC=icc instead.

Note that MPI must be installed with MPI-IO enabled because parts of SPECFEM3D perform I/Os through MPI-IO.

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.

When compiling on an IBM machine with the `xlf` and `xlc` compilers, we suggest running the `configure` script with the following options:

```
./configure FC=xlf90_r MPIFC=mpif90 CC=xlc_r CFLAGS="-O3 -q64" FCFLAGS="-O3 -q64" -with-scotch-dir=...
```

On SGI systems, `flags.guess` automatically informs `configure` to insert ‘‘TRAP\_FPE=OFF’’ into the generated Makefile in order to turn underflow trapping off.

You can add `-enable-vectorization` to the configuration options to speed up the code in the fluid (acoustic) and elastic parts. This works fine if (and only if) your computer always allocates a contiguous memory block

for each allocatable array; this is the case for most machines and most compilers, but not all. To disable this feature, use option `-disable-vectorization`. For more details see [github.com/geodynamics/specfem3d/issues/81](https://github.com/geodynamics/specfem3d/issues/81). To check if that option works fine on your machine, run the code with and without it for an acoustic/elastic model and make sure the seismograms are identical.

Note that we use CUBIT (now called Treliis) to create meshes of hexahedra, but other packages can be used as well, for instance GiD from <http://gid.cimne.upc.es> or Gmsh from <http://geuz.org/gmsh> [Geuzaine and Remacle, 2009]. Even mesh creation packages that generate tetrahedra, for instance TetGen from <http://tetgen.berlios.de>, can be used because each tetrahedron can then easily be decomposed into four hexahedra as shown in the picture of the TetGen logo at <http://tetgen.berlios.de/figs/Delaunay-Voronoi-3D.gif>; while this approach does not generate hexahedra 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 [Oliveira and Seriani, 2011].

The SPECFEM3D Cartesian software package relies on the SCOTCH library to partition meshes created with CUBIT. METIS [Karypis and Kumar, 1998a,c,b] can also be used instead of SCOTCH if you prefer, by editing file `src/decompose_mesh/decompose_mesh.F90` and uncommenting the flag `USE_METIS_INSTEAD_OF_SCOTCH`. You will also then need to install and compile Metis version 4.0 (do \*NOT\* install Metis version 5.0, which has incompatible function calls) and edit `src/decompose_mesh/Makefile.in` and uncomment the METIS link flag in that file before running `configure`.

The SCOTCH library [Pellegrini and Roman, 1996] 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 to be set explicitly with the option `--with-scotch-dir`. Just as an example:

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

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)

To compile a serial version of the code for small meshes that fits on one compute node and can therefore be run serially, run `configure` with the `--without-mpi` option to suppress all calls to MPI.

A summary of the most important configuration variables follows.

**F90** Path to the Fortran compiler.

**MP IF90** Path to MPI Fortran.

**MPI\_FLAGS** Some systems require this flag to link to MPI libraries.

**FLAGS\_CHECK** Compiler flags.

The configuration script automatically creates for each executable a corresponding `Makefile` in the `src/` subdirectory. The `Makefile` contains a number of suggested entries for various compilers, e.g., Portland, Intel, Absoft, NAG, and Lahey. The software has run on a wide variety of compute platforms, e.g., various PC clusters and machines from Sun, SGI, IBM, Compaq, and NEC. Select the compiler you wish to use on your system and choose the related optimization flags. Note that the default flags in the `Makefile` are undoubtedly not optimal for your system, so we encourage you to experiment with these flags and to solicit advice from your systems administrator. Selecting the right compiler and optimization flags can make a tremendous difference in terms of performance. We welcome feedback on your experience with various compilers and flags.

Now that you have set the compiler information, you need to select a number of flags in the `constants.h` file depending on your system:

**LOCAL\_PATH\_IS\_ALSO\_GLOBAL** Set to `.false.` on most cluster applications. For reasons of speed, the (parallel) distributed database generator typically writes a (parallel) database for the solver on the local disks of the compute nodes. Some systems have no local disks, e.g., BlueGene or the Earth Simulator, and other systems have a fast parallel file system, in which case this flag should be set to `.true..` Note that this flag is not used by the database generator or the solver; it is only used for some of the post-processing.

The package can run either in single or in double precision mode. The default is single precision because for almost all calculations performed using the spectral-element method using single precision is sufficient and gives the same results (i.e. the same seismograms); and the single precision code is faster and requires exactly half as much memory. Select your preference by selecting the appropriate setting in the `constants.h` file:

**CUSTOM\_REAL** Set to `SIZE_REAL` for single precision and `SIZE_DOUBLE` for double precision.

In the `precision.h` file:

**CUSTOM\_MPI\_TYPE** Set to `MPI_REAL` for single precision and `MPI_DOUBLE_PRECISION` for double precision.

On many current processors (e.g., Intel, AMD, IBM Power), single precision calculations are significantly faster; the difference can typically be 10% to 25%. It is therefore better to use single precision. What you can do once for the physical problem you want to study is run the same calculation in single precision and in double precision on your system and compare the seismograms. If they are identical (and in most cases they will), you can select single precision for your future runs.

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 Adding OpenMP support in addition to MPI

NOTE FROM JULY 2013: OpenMP support is maybe / probably not maintained any more. Thus the section below is maybe obsolete.

OpenMP support can be enabled in addition to MPI. However, in many cases performance will not improve because our pure MPI implementation is already heavily optimized and thus the resulting code will in fact be slightly slower. A possible exception could be IBM BlueGene-type architectures.

To enable OpenMP, uncomment the OpenMP compiler option in two lines in file `src/specfem3D/Makefile.in` (before running `configure`) and also uncomment the `#define USE_OPENMP` statement in file `src/specfem3D/specfem3D.F90`.

The DO-loop using OpenMP threads has a `SCHEDULE` property. The `OMP_SCHEDULE` environment variable can set the scheduling policy of that DO-loop. Tests performed by Marcin Zielinski at SARA (The Netherlands) showed that often the best scheduling policy is `DYNAMIC` with the size of the chunk equal to the number of OpenMP threads, but most preferably being twice as the number of OpenMP threads (thus chunk size = 8 for 4 OpenMP threads etc). If `OMP_SCHEDULE` is not set or is empty, the DO-loop will assume generic scheduling policy, which will slow down the job quite a bit.

## 2.2 Compiling on an IBM BlueGene

More recent installation instruction for IBM BlueGene, from April 2013:

Edit file `flags.guess` and put this for `FLAGS_CHECK`:

```
-g -qfullpath -O2 -qsave -qstrict -qtune=qp -qarch=qp -qcache=auto -qhalt=w
-qfree=f90 -qsuffix=f=f90 -qlanglvl=95pure -Q -Q+rank,swap_all -Wl,-relax
```

The most relevant are the `-qarch` and `-qtune` flags, otherwise if these flags are set to “auto” then they are wrongly assigned to the architecture of the front-end node, which is different from that on the compute nodes. You will need to set these flags to the right architecture for your BlueGene compute nodes, which is not necessarily “qp”; ask your system administrator. On some machines it is necessary to use `-O2` in these flags instead of `-O3` due to a compiler bug of the XLF version installed. We thus suggest to first try `-O3`, and then if the code does not compile or does not run fine then switch back to `-O2`. The debug flags (`-g`, `-ffullpath`) do not influence performance but are useful to get at least some insights in case of problems.

Before running `configure`, select the XL Fortran compiler by typing `module load bgq-xl/1.0` or `module load bgq-xl` (another, less efficient option is to load the GNU compilers using `module load bgq-gnu/4.4.6` or similar).

Then, to configure the code, type this:

```
./configure FC=bgxlf90_r MPIFC=mpixlf90_r CC=bgxlc_r LOCAL_PATH_IS ALSO_GLOBAL=true
```

In order for the SCOTCH domain decomposer to compile, on some (but not all) Blue Gene systems you may need to run `configure` with `CC=gcc` instead of `CC=bgxlc_r`.

Older installation instruction for IBM BlueGene, from 2011:

To compile the code on an IBM BlueGene, Laurent Léger from IDRIS, France, suggests the following: compile the code with

```
FLAGS_CHECK="-O3 -qsave -qstrict -qtune=auto -qarch=450d -qcache=auto \
-qfree=f90 -qsuffix=f=f90 -g -qlanglvl=95pure -qhalt=w -Q \
-Q+rank,swap_all -Wl,-relax"
```

Option `-Wl,-relax` must be added on many (but not all) BlueGene systems to be able to link the binaries `xmeshfem3D` and `xspecfem3D` because the final link step is done by the GNU `ld` linker even if one uses `FC=bgxlf90_r`, `MPIFC=mpixlf90_r` and `CC=bgxlc_r` to create all the object files. On the contrary, on some BlueGene systems that use the native AIX linker option `-Wl,-relax` can lead to problems and must be suppressed from `flags.guess`. Also, `AR=ar`, `ARFLAGS=cru` and `RANLIB=ranlib` are hardwired in all `Makefile.in` files by default, but to cross-compile on BlueGene/P one needs to change these values to `AR=bgar`, `ARFLAGS=cru` and `RANLIB=bgranlib`. Thus the easiest thing to do is to modify all `Makefile.in` files and the `configure` script to set them automatically by `configure`. One then just needs to pass the right commands to the `configure` script:

```
./configure --prefix=/path/to/SPECFEM3DG_SP --host=Babel --build=BGP \
FC=bgxlf90_r MPIFC=mpixlf90_r CC=bgxlc_r AR=bgar ARFLAGS=cru \
RANLIB=bgranlib LOCAL_PATH_IS ALSO_GLOBAL=false
```

This trick can be useful for all hosts on which one needs to cross-compile.

On BlueGene, one also needs to run the `xcreate_header_file` binary file manually rather than in the `Makefile`:

```
bgrun -np 1 -mode VN -exe ./bin/xcreate_header_file
```

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

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

To do your own call graphs, you can follow these simple steps below.

0. Install `Doxygen` and `graphviz` (the two are usually in the package manager of classic Linux distribution).
1. Run in the terminal : `doxygen -g`, which creates a `Doxyfile` that tells `doxygen` what you want it to do.
2. Edit the `Doxyfile`. Two `Doxyfile`-type files have been already committed in the directory `specfem3d/doc/Call_trees`:

- `Doxyfile_truncated_call_tree` will generate call graphs with maximum 3 or 4 levels of tree structure,
- `Doxyfile_complete_call_tree` will generate call graphs with complete tree structure.

The important entries in the Doxyfile are:

**PROJECT\_NAME**

**OPTIMIZE\_FOR\_FORTRAN** Set to YES

**EXTRACT\_ALL** Set to YES

**EXTRACT\_PRIVATE** Set to YES

**EXTRACT\_STATIC** Set to YES

**INPUT** From the directory `specfem3d/doc/Call_trees`, it is ".../src/"

**FILE\_PATTERNS** In SPECFEM case, it is `*.f90* *.F90* *.c* *.cu* *.h*`

**HAVE\_DOT** Set to YES

**CALL\_GRAPH** Set to YES

**CALLER\_GRAPH** Set to YES

**DOT\_PATH** The path where is located the dot program graphviz (if it is not in your \$PATH)

**RECURSIVE** This tag can be used to turn specify whether or not subdirectories should be searched for input files as well. In the case of SPECFEM, set to YES.

**EXCLUDE** Here, you can exclude:

```
.../src/specfem3D/older_not_maintained_partial_OpenMP_port
.../src/decompose_mesh/scotch
.../src/decompose_mesh/scotch_5.1.12b
```

**DOT\_GRAPH\_MAX\_NODES** to set the maximum number of nodes that will be shown in the graph. If the number of nodes in a graph becomes larger than this value, doxygen will truncate the graph, which is visualized by representing a node as a red box. Minimum value: 0, maximum value: 10000, default value: 50.

**MAX\_DOT\_GRAPH\_DEPTH** to set the maximum depth of the graphs generated by dot. A depth value of 3 means that only nodes reachable from the root by following a path via at most 3 edges will be shown. Using a depth of 0 means no depth restriction. Minimum value: 0, maximum value: 1000, default value: 0.

3. Run : doxygen Doxyfile, HTML and LaTeX files created by default in `html` and `latex` subdirectories.
4. To see the call trees, you have to open the file `html/index.html` in your browser. You will have many informations about each subroutines of SPECFEM (not only call graphs), you can click on every boxes / subroutines. It show you the call, and, the caller graph of each subroutine : the subroutines called by the concerned subroutine, and the previous subroutines who call this subroutine (the previous path), respectively. In the case of a truncated calling tree, the boxes with a red border indicates a node that has more arrows than are shown (in other words: the graph is truncated with respect to this node).

Finally, some useful links:

- a good and short summary for the basic utilisation of Doxygen:

<http://www.softeng.rl.ac.uk/blog/2010/jan/30/callgraph-fortran-doxygen/>,

- to configure the diagrams :

<http://www.stack.nl/~dimitri/doxygen/manual/diagrams.html>,

- the complete alphabetical index of the tags in Doxyfile:  
<http://www.stack.nl/~dimitri/doxygen/manual/config.html>,
- more generally, the Doxygen manual:  
<http://www.stack.nl/~dimitri/doxygen/manual/index.html>.

## 2.4 Using the ADIOS library for I/O

Regular POSIX I/O can be problematic when dealing with large simulations one large clusters (typically more than 10,000 processes). SPECFEM3D use the ADIOS library Liu et al. [2013] to deal transparently take advantage of advanced parallel file system features. To enable ADIOS, the following steps should be done:

1. Install ADIOS (available from <https://www.olcf.ornl.gov/center-projects/adios/>). Make sure that your environment variables reference it.
2. You may want to change ADIOS related values in the `constants.h.in` file. The default values probably suit most cases.
3. Configure using the `-with-adios` flag.

ADIOS is currently only usable for meshfem3D generated mesh (i.e. not for meshes generated with CUBIT). Additional control parameters are discussed in section 4.1.

## 2.5 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/specfem3d/wiki/Using-Hub>.

To visualize the call tree (calling tree) of the source code, you can see the Doxygen tool available in directory `doc/call_trees_of_the_source_code`.

# Chapter 3

## Mesh Generation

The first step in running a spectral-element simulation consists of constructing a high-quality mesh for the region under consideration. We provide two possibilities to do so: (1) relying on the external, hexahedral mesher CUBIT, or (2) using the provided, internal mesher `xmeshfem3D`. In the following, we explain these two approaches.

### 3.1 Meshing with CUBIT

CUBIT is a meshing tool suite for the creation of finite-element meshes for arbitrarily shaped models. It has been developed and maintained at Sandia National Laboratories and can be purchased for a small academic institutional fee at <http://cubit.sandia.gov>. Our experience showed that using CUBIT greatly facilitates and speeds up the generation and preparation of hexahedral, conforming meshes for a variety of geophysical models with increasing complexity.

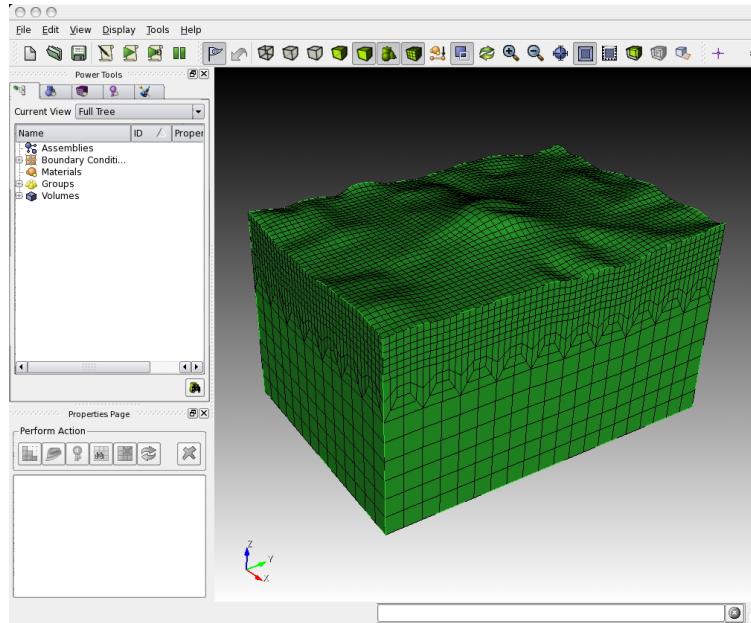


Figure 3.1: Example of the graphical user interface of CUBIT. The hexahedral mesh shown in the main display consists of a hexahedral discretization of a single volume with topography.

The basic steps in creating a load-balanced, partitioned mesh with CUBIT are:

1. setting up a hexahedral mesh with CUBIT,

2. exporting the CUBIT mesh into a SPECFEM3D Cartesian file format and
3. partitioning the SPECFEM3D Cartesian mesh files for a chosen number of cores.

Examples are provided in the SPECFEM3D Cartesian package in the subdirectory EXAMPLES/. We strongly encourage you to contribute your own example to this package by contacting the CIG Computational Seismology Mailing List ([cig-seismo@geodynamics.org](mailto:cig-seismo@geodynamics.org)).

### 3.1.1 Creating the Mesh with CUBIT

For the installation and handling of the CUBIT meshing tool suite, please refer to the CUBIT user manual and documentation. In order to give you a basic understanding of how to use CUBIT for our purposes, examples are provided in the SPECFEM3D Cartesian package in the subdirectory EXAMPLES/:

**homogeneous\_halfspace** Creates a single block model and assigns elastic material parameters.

**layered\_halfspace** Combines two different, elastic material volumes and creates a refinement layer between the two. This example can be compared for validation against the solutions provided in subdirectory VALIDATION\_3D\_SEM\_SIMPLER\_LAYER\_SOURCE\_DEPTH/.

**waterlayered\_halfspace** Combines an acoustic and elastic material volume as in a schematic marine survey example.

**tomographic\_model** Creates a single block model whose material properties will have to be read in from a tomographic model file during the databases creation by xgenerate\_databases.

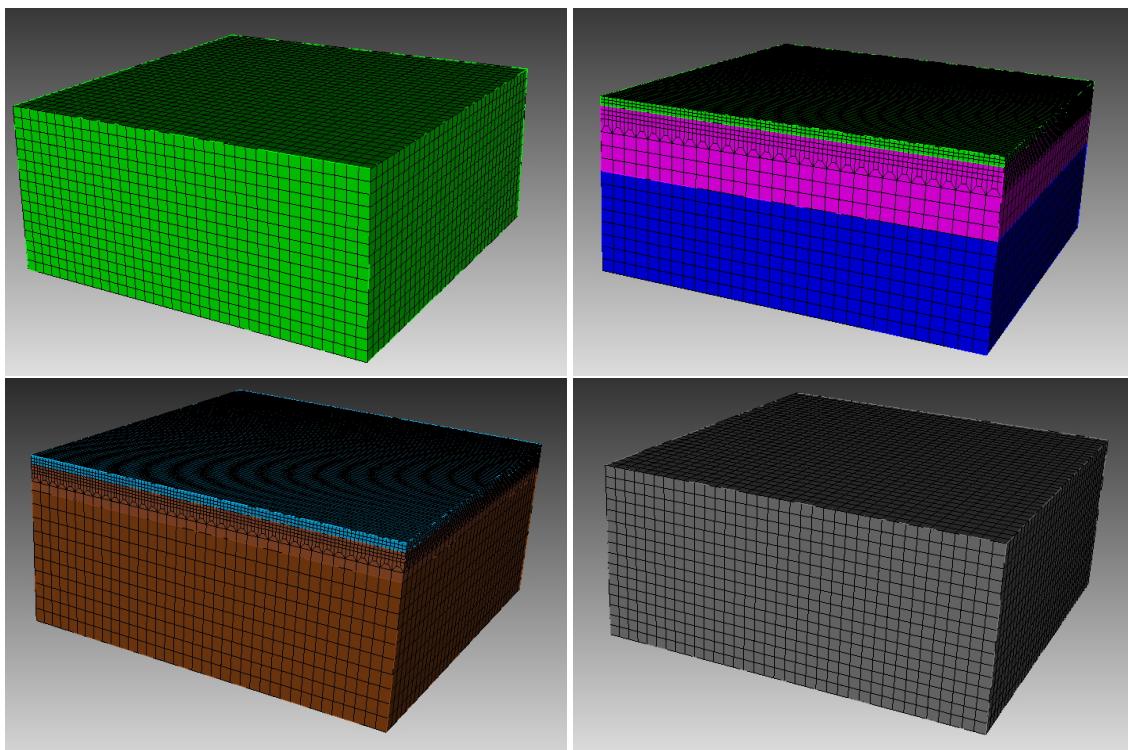


Figure 3.2: Screenshots of the CUBIT examples provided in subdirectory EXAMPLES/: homogeneous halfspace (top-left), layered halfspace (top-right), water layered halfspace (bottom-left) and tomographic model (bottom-right).

In each example subdirectory you will find a README file, which explains in a step-by-step tutorial the workflow for the example. Please feel free to contribute your own example to this package by contacting the CIG Computational

Seismology Mailing List ([cig-seismo@geodynamics.org](mailto:cig-seismo@geodynamics.org)).

In some cases, to re-create the meshes for the examples given, just type

```
claro ./create_mesh.py
```

or similar from the command line (claro is the command to run CUBIT from the command line).

**IMPORTANT:** In order to correctly set up GEOCUBIT and run the examples, please read the file called EXAMPLES/README; in particular, please make sure you correctly set up the Python paths as indicated in that file.

Concerning the script `create_mesh.py`, you may find the option `use_explicit` which chooses how to assign the material properties for the volume (or domain):

- (1) one way is to explicitly assign block attributes to the block of the corresponding volume, by commands like:

```
cubit.cmd('block '+str(id_block)+' attribute index 2 2800') # vp
```

This is done when using `use_explicit = 1` in the script. The final command:

```
cubit2specfem3d.export2SPECFEM3D('MESH/')
```

will then create the corresponding `MESH/nummaterial_velocity_file` for all such defined block volumes/domains.

- (2) the other option with `use_explicit = 0` is to let GEOCUBIT deal with a dummy entry and then overwrite the `nummaterial_velocity_file` at the end with a corresponding command section:

```
f = open(nummaterial_velocity_file,'w')
```

This second way is chosen by default because GEOCUBIT can handle partitions for several processes and glues everything together automatically. Thus, it adds some more sophistication when the model gets more complicated than just this single volume example.

You will find out by experimenting what is easier for your case.

### 3.1.2 Exporting the Mesh with `run_cubit2specfem3d.py`

Once the geometric model volumes in CUBIT are meshed, you prepare the model for exportation with the definition of material blocks and boundary surfaces. Thus, prior to exporting the mesh, you need to define blocks specifying the materials and absorbing boundaries in CUBIT. This process could be done automatically using the script `run_cubit2specfem3d.py` if the mesh meets some conditions or manually, following the block convention:

**material\_name** Each material should have a specific block defined by a unique name. The name convention of the material is to start with either '**elastic**' or '**acoustic**'. It must be then followed by a unique identifier, e.g. '**elastic 1**', '**elastic 2**', etc. The additional attributes to the block define the material description.

For an elastic material:

**material\_id** An integer value which is unique for this material.

**Vp** P-wave speed of the material (given in m/s).

**Vs** S-wave speed of the material (given in m/s).

**rho** density of the material (given in kg/m<sup>3</sup>).

**Q** quality factor to use in case of a simulation with attenuation turned on. It should be between 1 and 9000.

In case no attenuation information is available, it can be set to zero. You can either specify a single Q value, in which case it will be assumed to be pure shear attenuation  $Q_\mu$ , or two separate values for bulk and shear attenuation,  $Q_\kappa$  and  $Q_\mu$  respectively. Note that  $Q_\mu$  is always equal to  $Q_s$ , but  $Q_\kappa$  is in general not equal to  $Q_p$ . To convert one to the other see doc/note\_on\_Qkappa\_versus\_Qp.pdf and utils/attenuation/conversion\_from\_Qkappa\_Qmu\_to\_Qp\_Qs\_from\_Dahlen\_Tromp\_959\_960.f90.

Please note that your Vp- and Vs-speeds are given for a reference frequency. To change this reference frequency, you change the value of ATTENUATION\_f0\_REFERENCE in the main constants file `constants.h` found in subdirectory `src/shared/`. The code uses a constant  $Q$  quality factor, `write(IMAIN,*)` "but approximated based on a series of Zener standard linear solids (SLS). The approximation is thus performed in a given frequency band determined based on that ATTENUATION\_f0\_REFERENCE reference frequency.

**anisotropic\_flag** Flag describing the anisotropic model to use in case an anisotropic simulation should be conducted. See the file `model_aniso.f90` in subdirectory `src/generate_databases/` for an implementation of the anisotropic models. In case no anisotropy is available, it can be set to zero.

Note that this material block has to be defined using all the volumes which belong to this elastic material. For volumes belonging to another, different material, you will need to define a new material block.

For an acoustic material:

**material\_id** An integer value which is unique for this material.

**Vp** P-wave speed of the material (given in m/s).

**0** S-wave speed of the material is ignored.

**rho** density of the material (given in kg/m<sup>3</sup>).

**face\_topo** Block definition for the surface which defines the free surface (which can have topography). The name of this block must be 'face\_topo', the block has to be defined using all the surfaces which constitute the complete free surface of the model.

**face\_abs\_xmin** Block definition for the faces on the absorbing boundaries, one block for each surface with x=Xmin.

**face\_abs\_xmax** Block definition for the faces on the absorbing boundaries, one block for each surface with x=Xmax.

**face\_abs\_ymin** Block definition for the faces on the absorbing boundaries, one block for each surface with y=Ymin.

**face\_abs\_ymax** Block definition for the faces on the absorbing boundaries, one block for each surface with y=Ymax.

**face\_abs\_bottom** Block definition for the faces on the absorbing boundaries, one block for each surface with z=bottom.

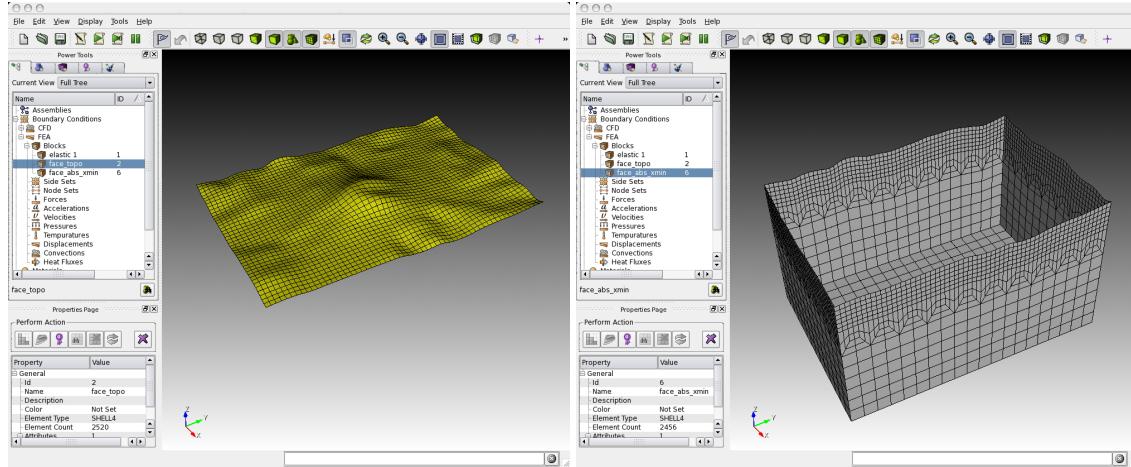


Figure 3.3: Example of the block definitions for the free surface 'face\_topo' (left) and the absorbing boundaries, defined in a single block 'face\_abs\_xmin' (right) in CUBIT.

Optionally, instead of specifying for each surface at the boundaries a single block like mentioned above, you can also specify a single block for all boundary surfaces and name it as one of the absorbing blocks above, e.g. 'face\_abs\_xmin'.

After the block definitions are done, you export the mesh using the script `cubit2specfem3d.py` provided in each of the example directories (linked to the common script `CUBIT_GEOCUBIT/cubit2specfem3d.py`). If the export was successful, you should find the following files in a subdirectory `MESH/`:

**absorbing\_cpml\_file (only needed in case of C-PML absorbing conditions)** Contains on the first line the total number of C-PML spectral elements in the mesh, and then on the following line the list of all these C-PML elements with two numbers per line: first the spectral element number, and then a C-PML flag indicating to which C-PML layer(s) that element belongs, according to the following convention:

- Flag = 1 : element belongs to a X CPML layer only (either in  $X_{\min}$  or in  $X_{\max}$ ),
- Flag = 2 : element belongs to a Y CPML layer only (either in  $Y_{\min}$  or in  $Y_{\max}$ ),
- Flag = 3 : element belongs to a Z CPML layer only (either in  $Z_{\min}$  or in  $Z_{\max}$ ),
- Flag = 4 : element belongs to a X CPML layer and also to a Y CPML layer,
- Flag = 5 : element belongs to a X CPML layer and also to a Z CPML layer,
- Flag = 6 : element belongs to a Y CPML layer and also to a Z CPML layer,
- Flag = 7 : element belongs to a X, to a Y and to a Z CPML layer, i.e., it belongs to a CPML corner.

Note that it does not matter whether an element belongs to a  $X_{\min}$  or to a  $X_{\max}$  CPML, the flag is the same in both cases; the same is true for  $Y_{\min}$  and  $Y_{\max}$ , and also  $Z_{\min}$  and  $Z_{\max}$ .

When you have an existing CUBIT (or similar) mesh stored in SPECFEM3D format, i.e., if you have existing `nodes_coords_file` and `mesh_file` files but do not know how to assign CPML flags to them, we have created a small serial Fortran program that will do that automatically for you, i.e., which will create the `absorbing_cpml_file` for you. That program is:

`utils/CPML/convert_external_layers_of_a_given_mesh_to_CPML_layers.f90`,  
and a small Makefile is provided in that directory (`utils/CPML`).

**IMPORTANT:** it is your responsibility to make sure that in the input CUBIT (or similar) mesh that this code will read in SPECFEM3D format from files `nodes_coords_file` and `mesh_file` you have created layers of elements that constitute a layer of constant thickness aligned with the coordinate grid axes (X, Y and/or Z), so that this code can assign CPML flags to them. This code does NOT check that (because it cannot, in any easy way). The mesh inside these CPML layers does not need to be structured nor regular, any non-structured mesh is fine as long as it has flat PML inner and outer faces, parallel to the axes, and thus of a constant thickness. The thickness can be different for the X, Y and Z sides. But for X it must not vary, for Y it must not vary, and for Z it must not vary. If you do not know the exact thickness, you can use a slightly LARGER value in this code (say 2% to 5% more) and this code will fix that and will adjust it; never use a SMALLER value otherwise this code will miss some CPML elements.

Note: in a future release we will remove the constraint of having CPML layers aligned with the coordinate axes; we will allow for meshes that are tilted by any constant angle in the horizontal plane. However this is not implemented yet.

Note: in the case of fluid-solid layers in contact, the fluid-solid interface currently needs to be flat and horizontal inside the CPML layer (i.e., bathymetry should become flat and horizontal when entering the CPML); this (small) constraint will probably remain in the code for a while because it makes fluid-solid matching inside the CPML much easier.

**materials\_file** Contains the material associations for each element. The format is:

```
element_ID material_ID
```

where `element_ID` is the element identifier and `material_ID` is a unique identifier, positive (for materials taken from this list of materials, i.e. for which each spectral element has constant material properties taken from this list) or negative (for tomographic models, i.e. for spectral element whose real velocities and density will be assigned later by calling an external function to define model variations, for instance in the case of tomographic models; in such a case, material properties can vary inside each spectral element, i.e. be different at each of its Gauss-Lobatto-Legendre grid points).

**nummaterial\_velocity\_file** Defines the material properties.

- For classical materials (i.e., spectral elements for which the velocity and density model will not be assigned by calling an external function to define for instance a tomographic model), the format is:

```
domain_ID material_ID rho vp vs Qkappa Qmu anisotropy_flag
```

where **domain\_ID** is 1 for acoustic and 2 for elastic or viscoelastic materials, **material\_ID** a unique identifier, **rho** the density in  $kg\ m^{-3}$ , **vp** the P-wave speed in  $m\ s^{-1}$ , **vs** the S-wave speed in  $m\ s^{-1}$ , **Q** the quality factor and **anisotropy\_flag** an identifier for anisotropic models. Note that both **Qkappa** and **Qmu** are ignored by the code unless ATTENUATION is set. If you want a model with no **Qmu** attenuation, both set ATTENUATION to .false. in the Par\_file and set **Qmu** to 9999 here. If you want a model with no **Qkappa** attenuation, set **Qkappa** to 9999 here. Note that **Qmu** is always equal to **Qs**, but **Qkappa** is in general not equal to **Qp**. To convert one to the other see doc/note\_on\_Qkappa\_versus\_Qp.pdf and utils/attenuation/conversion\_from\_Qkappa\_Qmu\_to\_Qp\_Qs\_from\_Dahlen\_Tromp.

- For tomographic velocity models, please read Chapter 13 and Section 13.1 ‘Using external tomographic Earth models’ for further details.

**nodes\_coords\_file** Contains the point locations in Cartesian coordinates of the mesh element corners.

**mesh\_file** Contains the mesh element connectivity. The hexahedral elements can have 8 or 27 nodes.

See picture doc/mesh\_numbering\_convention/numbering\_convention\_27\_nodes.jpg to see in which (standard) order the points must be cited. In the case of 8 nodes, just include the first 8 points.

**free\_or\_absorbing\_surface\_file\_zmax** Contains the free surface connectivity or the surface connectivity of the absorbing boundary surface at the top (Zmax), depending on whether the top surface is defined as free or absorbing (STACEY\_INSTEAD\_OF\_FREE\_SURFACE in DATA/Par\_file).

You should put both the surface of acoustic regions and of elastic regions in that file; that is, list all the element faces that constitute the surface of the model in that file.

**absorbing\_surface\_file\_xmax** Contains the surface connectivity of the absorbing boundary surface at Xmax (also needed in the case of C-PML absorbing conditions, in order for the code to be able to impose Dirichlet conditions on their outer edge).

**absorbing\_surface\_file\_xmin** Contains the surface connectivity of the absorbing boundary surface at Xmin (also needed in the case of C-PML absorbing conditions, in order for the code to be able to impose Dirichlet conditions on their outer edge).

**absorbing\_surface\_file\_ymax** Contains the surface connectivity of the absorbing boundary surface at Ymax (also needed in the case of C-PML absorbing conditions, in order for the code to be able to impose Dirichlet conditions on their outer edge).

**absorbing\_surface\_file\_ymin** Contains the surface connectivity of the absorbing boundary surface at Ymin (also needed in the case of C-PML absorbing conditions, in order for the code to be able to impose Dirichlet conditions on their outer edge).

**absorbing\_surface\_file\_bottom** Contains the surface connectivity of the absorbing boundary surface at the bottom (Zmin) (also needed in the case of C-PML absorbing conditions, in order for the code to be able to impose Dirichlet conditions on their outer edge).

These mesh files are needed as input files for the partitioner xdecompose\_mesh to load-balance the mesh. Please see the next section for further details.

In directory "CUBIT\_GEOCUBIT/" we provide a script that can help doing the above tasks of exporting a CUBIT mesh to SPECFEM3D format automatically for you: "run\_cubit2specfem3d.py". Just edit them to indicate the path to your local installation of CUBIT and also the name of the \*.cub existing CUBIT mesh file that you want to export to SPECFEM3D format. These scripts will do the conversion for you automatically except assigning material properties

to the different mesh layers. To do so, you will then need to edit the file called "nummaterial\_velocity\_file" that will have just been created and change it from the prototype created:

```
0 1 vol1 --> syntax: #material_domain_id #material_id #rho #vp #vs #Q_kappa #Q_mu #anisotropy
0 2 vol2 --> syntax: #material_domain_id #material_id #rho #vp #vs #Q_kappa #Q_mu #anisotropy
```

(where "vol1" and "vol2" here represent the volume labels that you have set while creating the mesh in CUBIT) to for instance

```
2 1 1500 2300 1800 9999.0 9999.0 0
2 2 1600 2500 20000 9999.0 9999.0 0
```

### Checking the mesh quality

The quality of the mesh may be inspected more precisely based upon the serial code in the file `check_mesh_quality_CUBIT_Abaqus.f90` located in the directory `src/check_mesh_quality_CUBIT_Abaqus/`. Running this code is optional because no information needed by the solver is generated.

Prior to running and compiling this code, you have to export your mesh in CUBIT to an ABAQUS (.inp) format. For example, export mesh block IDs belonging to volumes in order to check the quality of the hexahedral elements. You also have to determine a number of parameters of your mesh, such as the number of nodes and number of elements and modify the header of the `check_mesh_quality_CUBIT_Abaqus.f90` source file in directory `src/check_mesh_quality_CUBIT_Abaqus/`. Then, in the main directory, type

```
make xcheck_mesh_quality
```

and use

```
./bin/xcheck_mesh_quality
```

to generate an OpenDX output file (`DX_mesh_quality.dx`) that can be used to investigate mesh quality, e.g. skewness of elements, and a Gnuplot histogram (`mesh_quality_histogram.txt`) that can be plotted with gnuplot (type '`gnuplot plot_mesh_quality_histogram.gnu`'). The histogram is also printed to the screen. Analyze that skewness histogram of mesh elements to make sure no element has a skewness above approximately 0.75, otherwise the mesh is of poor quality (and if even a single element has a skewness value above 0.80, then you must definitely improve the mesh). If you want to start designing your own meshes, this tool is useful for viewing your creations. You are striving for meshes with elements with 'cube-like' dimensions, e.g., the mesh should contain no very elongated or skewed elements.

### 3.1.3 Partitioning the Mesh with `xdecompose_mesh`

The SPECFEM3D Cartesian software package performs large scale simulations in a parallel 'Single Process Multiple Data' way. The spectral-element mesh created with CUBIT needs to be distributed on the processors. This partitioning is executed once and for all prior to the execution of the solver so it is referred to as a static mapping.

An efficient partitioning is important because it leverages the overall running time of the application. It amounts to balance the number of elements in each slice while minimizing the communication costs resulting from the placement of adjacent elements on different processors. `decompose_mesh` depends on the SCOTCH library [Pellegrini and Roman, 1996], which 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 most cases, the configuration with `./configure FC=ifort` should be sufficient. During the configuration process, the script tries to find existing SCOTCH installations. In case your system has no pre-existing SCOTCH installation, we provide the source code of SCOTCH, which is released open source under the French CeCILL-C version 1 license, in directory `src/decompose_mesh/scotch_5.1.12b`. This version gets bundled with the compilation of the SPECFEM3D Cartesian package if no libraries could have been found. If this automatic compilation of the SCOTCH libraries fails, please refer to file `INSTALL.txt` in that directory to see further details how to compile it on your system. In case you want to use a pre-existing installation, make sure you have correctly specified the path of the SCOTCH library when using the option `--with-scotch-dir` with the `./configure` script. In the future you should be able to find more recent versions at <http://www.labri.fr/perso/pelegrin/scotch/>

`scotch_en.html`.

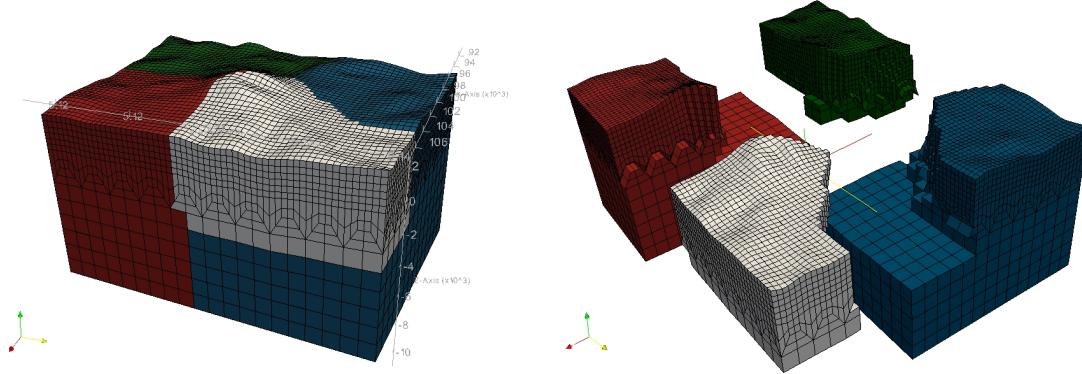


Figure 3.4: Example of a mesh partitioning onto four cores. Each single core partition is colored differently. The executable `xdecompose_mesh` can equally distribute the mesh on any arbitrary number of cores. Domain decomposition is explained in detail in Martin et al. [2008a], and excellent scaling up to 150,000 processor cores is shown for instance in Carrington et al. [2008], Komatitsch et al. [2008], Martin et al. [2008a], Komatitsch et al. [2010a], Komatitsch [2011].

When you are ready to compile, in the main directory type

```
make xdecompose_mesh
```

If all paths and flags have been set correctly, the executable `bin/xdecompose_mesh` should be produced.

The partitioning is done in serial for now (in the next release we will provide a parallel version of that code). It needs to be run in the main directory because it expects the `./DATA/Par_file`. The synopsis is:

```
./bin/xdecompose_mesh nparts input_directory output_directory
```

where

- `nparts` is the number of partitions, i.e., the number of cores for the parallel simulations,
- `input_directory` is the directory which holds all the files generated by the Python script `cubit2specfem3d.py` explained in the previous Section 3.1.2, e.g. `./MESH/`, and
- `output_directory` is the directory for the output of this partitioner which stores ASCII-format files named like `proc*****_Database` for each partition. These files will be needed for creating the distributed databases, and have to reside in the directory `LOCAL_PATH` specified in the main `Par_file`, e.g. in directory `./OUTPUT_FILES/DATABASES_MPI`. Please see Chapter 4 for further details.

Note that all the files generated by the Python script `cubit2specfem3d.py` must be placed in the `input_directory` folder before running the program.

## 3.2 Meshing with `xmeshfem3D`

In case you successfully ran the configuration script, you are also ready to compile the internal mesher. This is an alternative to CUBIT for the mesh generation of relatively simple models.

In the main directory, type

```
make xmeshfem3D
```

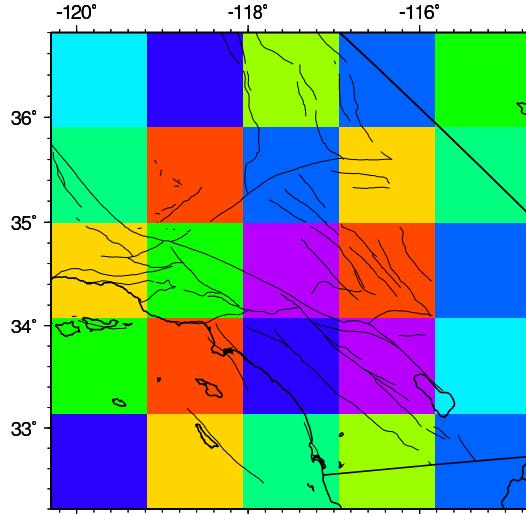


Figure 3.5: For parallel computing purposes, the model block is subdivided in `NPROC_XI`  $\times$  `NPROC_ETA` slices of elements. In this example we use  $5^2 = 25$  processors.

If all paths and flags have been set correctly, the mesher should now compile and produce the executable `bin/xmeshfem3D`. Please note that `xmeshfem3D` must be called directly from the main directory, as most of the binaries of the package.

Input for the mesh generation program is provided through the parameter file `Mesh_Par_file`, which resides in the subdirectory `DATA/meshfem3D_files/`. (to see how to use it, see the EXAMPLES specific to the internal mesher in directory `EXAMPLES/meshfem3D_examples/`). Before running the mesher, a number of parameters need to be set in the `Mesh_Par_file`. This requires a basic understanding of how the SEM is implemented, and we encourage you to read Komatitsch and Vilotte [1998], Komatitsch and Tromp [1999] and Komatitsch et al. [2004].

The mesher and the solver use UTM coordinates internally, therefore you need to define the zone number for the UTM projection (e.g., zone 11 for Los Angeles). Use decimal values for latitude and longitude (no minutes/seconds). These values are approximate; the mesher will round them off to define a square mesh in UTM coordinates. When running benchmarks on rectangular models, turn the UTM projection off by using the flag `SUPPRESS_UTM_PROJECTION`, in which case all ‘longitude’ parameters simply refer to the *x* axis, and all ‘latitude’ parameters simply refer to the *y* axis. To run the mesher for a global simulation, the following parameters need to be set in the `Mesh_Par_file`:

**LATITUDE\_MIN** Minimum latitude in the block (negative for South).

**LATITUDE\_MAX** Maximum latitude in the block.

**LONGITUDE\_MIN** Minimum longitude in the block (negative for West).

**LONGITUDE\_MAX** Maximum longitude in the block.

**DEPTH\_BLOCK\_KM** Depth of bottom of mesh in kilometers.

**UTM\_PROJECTION\_ZONE** UTM projection zone in which your model resides, only valid when `SUPPRESS_UTM_PROJECTION` is `.false.`. Use a negative zone number for the Southern hemisphere: the Northern hemisphere corresponds to zones +1 to +60, the Southern hemisphere to zones -1 to -60.

We use the WGS84 (World Geodetic System 1984) reference ellipsoid for the UTM projection. If you prefer to use the Clarke 1866 ellipsoid, edit file `src/shared/utm_geo.f90`, uncomment that ellipsoid and recompile the code.

From [http://en.wikipedia.org/wiki/Universal\\_Transverse\\_Mercator\\_coordinate\\_system](http://en.wikipedia.org/wiki/Universal_Transverse_Mercator_coordinate_system):

The Universal Transverse Mercator coordinate system was developed by the United States Army Corps of Engineers in the 1940s. The system was based on an ellipsoidal model of Earth. For areas within the contiguous United States the Clarke Ellipsoid of 1866 was used. For the remaining areas of Earth, including Hawaii, the International Ellipsoid was used. The WGS84 ellipsoid is now generally used to model the Earth in the UTM coordinate system, which means that current UTM northing at a given point can be 200+ meters different from the old one. For different geographic regions, other datum systems (e.g.: ED50, NAD83) can be used.

**SUPPRESS\_UTM\_PROJECTION** set to be `.false.` when your model range is specified in geographical coordinates, and needs to be `.true.` when your model is specified in Cartesian coordinates. UTM PROJECTION ZONE IN WHICH YOUR SIMULATION REGION RESIDES.

**INTERFACES\_FILE** File which contains the description of the topography and of the interfaces between the different layers of the model, if any. The number of spectral elements in the vertical direction within each layer is also defined in this file.

**NEX\_XI** The number of spectral elements along one side of the block. This number *must* be  $8 \times$  a multiple of `NPROC_XI` defined below. Based upon benchmarks against semi-analytical discrete wavenumber synthetic seismograms [Komatitsch et al., 2004], determined that a `NEX_XI = 288` run is accurate to a shortest period of roughly 2 s. Therefore, since accuracy is determined by the number of grid points per shortest wavelength, for any particular value of `NEX_XI` the simulation will be accurate to a shortest period determined by

$$\text{shortest period (s)} = (288/\text{NEX\_XI}) \times 2. \quad (3.1)$$

The number of grid points in each orthogonal direction of the reference element, i.e., the number of Gauss-Lobatto-Legendre points, is determined by `NGLLX` in the `constants.h` file. We generally use `NGLLX = 5`, for a total of  $5^3 = 125$  points per elements. We suggest not to change this value.

**NEX\_ETA** The number of spectral elements along the other side of the block. This number *must* be  $8 \times$  a multiple of `NPROC_ETA` defined below.

**NPROC\_XI** The number of processors or slices along one side of the block (see Figure 3.5); we must have `NEX_XI = 8 \times c \times NPROC_XI`, where  $c \geq 1$  is a positive integer.

**NPROC\_ETA** The number of processors or slices along the other side of the block; we must have `NEX_ETA = 8 \times c \times NPROC_ETA`, where  $c \geq 1$  is a positive integer.

**USE\_REGULAR\_MESH** set to be `.true.` if you want a perfectly regular mesh or `.false.` if you want to add doubling horizontal layers to coarsen the mesh. In this case, you also need to provide additional information by setting up the next three parameters.

**NDOUBLINGS** The number of horizontal doubling layers. Must be set at least to 1 if `USE_REGULAR_MESH` is set to `.true..` Multiple mesh doublings can be chosen, for which each an `NZ_DOUBLING_**` entry must be given. By default, we only provide two possible entries in the `Mesh_Par_file`. For higher numbers of doubling layers, additional entries must be added.

**NZ\_DOUBLING\_1** The position of the first doubling layer (only interpreted if `USE_REGULAR_MESH` is set to `.true..`).

**NZ\_DOUBLING\_2** The position of the second doubling layer (only interpreted if `USE_REGULAR_MESH` is set to `.true..` and if `NDOUBLINGS` is set to 2). Doubling layers must be at least 2 layers apart. The layer count starts from the bottom layer. More entries must be listed by the user if `NDOUBLINGS` is larger than 2.

**CREATE\_ABAQUS\_FILES** Set this flag to `.true.` to save Abaqus FEA ([www.simulia.com](http://www.simulia.com)) mesh files for subsequent viewing. Turning the flag on generates files in the `LOCAL_PATH` directory. See Section 11.1 for a discussion of mesh viewing features.

**CREATE\_DX\_FILES** Set this flag to `.true.` to save OpenDX ([www.opendx.org](http://www.opendx.org)) mesh files for subsequent viewing.

**LOCAL\_PATH** Directory in which the partitions generated by the mesher will be written. Generally one uses a directory on the local disk of the compute nodes, although on some machines these partitions are written on a parallel (global) file system (see also the earlier discussion of the LOCAL\_PATH\_IS\_ALSO\_GLOBAL flag in Chapter 2).

The mesher generates the necessary partitions in parallel, one set for each of the NPROC\_XI × NPROC\_ETA slices that constitutes the mesh (see Figure 3.5). After the mesher finishes, you can log in to one of the compute nodes and view the contents of the LOCAL\_PATH directory to see the files generated by the mesher. These files will be needed for creating the distributed databases, and have to reside in the directory LOCAL\_PATH specified in the main Par\_file, e.g. in directory OUTPUT\_FILES/DATABASES\_MPI. Please see Chapter 4 for further details.

**NMATERIALS** The number of different materials in your model. In the following lines, each material needs to be defined as:

```
material_ID rho vp vs Q anisotropy_flag domain_ID
```

where

- $Q$  : quality factor (0=no attenuation) for shear attenuation  $Q_\mu$
- anisotropy\_flag : 0=no anisotropy / 1,2,.. check with implementation in aniso\_model.f90
- domain\_id : 1=acoustic / 2=elastic

**NREGIONS** The number of regions in the mesh. In the following lines, because the mesh is regular or 'almost regular', each region is defined as:

```
NEX_XI_BEGIN NEX_XI_END NEX_ETA_BEGIN NEX_ETA_END NZ_BEGIN NZ_END material_ID
```

The INTERFACES\_FILE parameter of Mesh\_Par\_File defines the file which contains the settings of the topography grid and of the interfaces grids. Topography is defined as a set of elevation values on a regular 2D grid. It is also possible to define interfaces between the layers of the model in the same way. The file needs to define several parameters:

- The number of interfaces, including the topography. This needs to be set at the first line. Then, from the bottom to the top of the model, you need to define the grids with:
- SUPPRESS\_UTM\_PROJECTION flag as described previously,
- number of points along  $x$  and  $y$  direction (NXI and NETA),
- minimal  $x$  and  $y$  coordinates (LONG\_MIN and LAT\_MIN),
- spacing between points along  $x$  and  $y$  (SPACING\_XI and SPACING\_ETA) and
- the name of the file which contains the elevation values (in  $y.x$  increasing order).

At the end of this file, you simply need to set the number of spectral elements in the vertical direction for each layer. We provide a few models in the EXAMPLES/ directory.

Finally, depending on your system, you might need to provide a file that tells MPI what compute nodes to use for the simulations. The file must have a number of entries (one entry per line) at least equal to the number of processors needed for the run. A sample file is provided in the file mymachines. This file is not used by the mesher or solver, but is required by the go\_mesher and go\_solver default job submission scripts. See Chapter 12 for information about running the code on a system with a scheduler, e.g., LSF.

Now that you have set the appropriate parameters in the Mesh\_Par\_file and have compiled the mesher, you are ready to launch it! This is most easily accomplished based upon the go\_mesher script. When you run on a PC cluster, the script assumes that the nodes are named n001, n002, etc. If this is not the case, change the tr -d

'n' line in the script. You may also need to edit the last command at the end of the script that invokes the `mpirun` command. See Chapter 12 for information about running the code on a system with a scheduler, e.g., LSF.

Mesher output is provided in the `OUTPUT_FILES` directory in `output_mesher.txt`; this file provides lots of details about the mesh that was generated. Please note that the mesher suggests a time step `DT` to run the solver with. The mesher output file also contains a table about the quality of the mesh to indicate possible problems with the distortions of elements. Alternatively, output can be directed to the screen instead by uncommenting a line in `constants.h`:

```
! uncomment this to write messages to the screen
! integer, parameter :: IMAIN = ISTANDARD_OUTPUT
```

To control the quality of the mesh, check the standard output (either on the screen or in the `OUTPUT_FILES` directory in `output_mesher.txt`) and analyze the skewness histogram of mesh elements to make sure no element has a skewness above approximately 0.75, otherwise the mesh is of poor quality (and if even a single element has a skewness value above 0.80, then you must definitely improve the mesh). To draw the skewness histogram on the screen, type `gnuplot plot_mesh_quality_histogram.gnu`.

## Chapter 4

# Creating the Distributed Databases

After using `xmeshfem3D` or `xdecompose_mesh`, the next step in the workflow is to compile `xgenerate_databases`. This program is going to create all the missing information needed by the SEM solver.

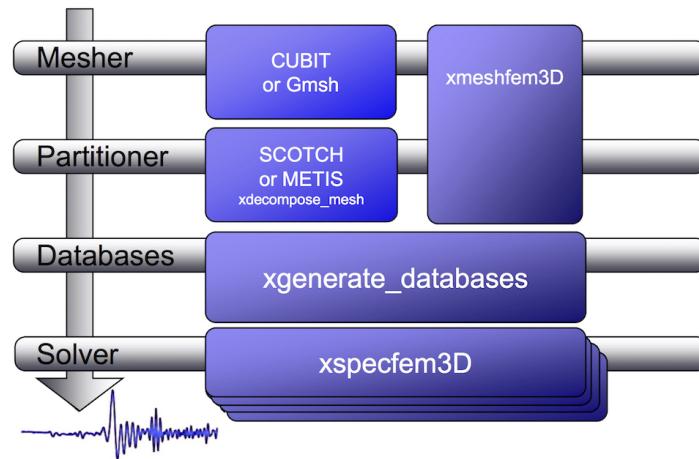


Figure 4.1: Schematic workflow for a SPECFEM3D Cartesian simulation. The executable `xgenerate_databases` creates the GLL mesh points and assigns specific model parameters.

In the main directory, type

```
make xgenerate_databases
```

Input for the program is provided through the main parameter file `Par_file`, which resides in the subdirectory `DATA`. Please note that `xgenerate_databases` must be called directly from the main directory, as most of the binaries of the package.

### 4.1 Main parameter file `Par_file`

Before running `xgenerate_databases`, a number of parameters need to be set in the main parameter `Par_file` located in the subdirectory `DATA`:

`SIMULATION_TYPE` is set to 1 for forward simulations, 2 for adjoint simulations (see Section 7.2) and 3 for kernel simulations (see Section 11.3).

**SAVE\_FORWARD** is only set to `.true.` for a forward simulation with the last frame of the simulation saved, as part of the finite-frequency kernel calculations (see Section 11.3). For a regular forward simulation, leave `SIMULATION_TYPE` and `SAVE_FORWARD` at their default values.

**UTM\_PROJECTION\_ZONE** UTM projection zone in which your model resides, only valid when `SUPPRESS_UTM_PROJECTION` is `.false..`

**SUPPRESS\_UTM\_PROJECTION** set to be `.false.` when your model range is specified in the geographical coordinates, and needs to be `.true.` when your model is specified in a cartesian coordinates. UTM PROJECTION ZONE IN WHICH YOUR SIMULATION REGION RESIDES.

**NPROC** The number of MPI processors, each one is assigned one slice of the whole mesh.

**NSTEP** The number of time steps of the simulation. This controls the length of the numerical simulation, i.e., twice the number of time steps requires twice as much CPU time. This feature is not used at the time of generating the distributed databases but is required for the solver, i.e., you may change this parameter after running `xgenerate_databases`.

**DT** The length of each time step in seconds. This feature is not used at the time of generating the distributed databases but is required for the solver. Please see also Section 4.2 for further details.

**NGNOD** The number of nodes for 2D and 3D shape functions for hexahedra. We use either 8-node mesh elements (bricks) or 27-node elements. If you use the internal mesher, the only option is 8-node bricks (27-node elements are not supported). CUBIT does not support HEX27 elements either (it can generate them, but they are flat, i.e. identical to HEX8). To generate HEX27 elements with curvature properly taken into account, you can use Gmsh <http://geuz.org/gmsh/>

**MODEL** Must be set to one of the following:

Models defined by mesh parameters:

**default** Uses model parameters as defined by meshing procedures described in the previous Chapter 3.

1D models with real structure:

**1D\_prem** Isotropic version of the spherically symmetric Preliminary Reference Earth Model (PREM) [Dziewoński and Anderson, 1981].

**1D\_socal** A standard isotropic 1D model for Southern California.

**1D\_cascadia** Isotropic 1D profile for the Cascadia region.

Fully 3D models:

**aniso** For a user-specified fully anisotropic model. Parameters are set up in routines located in file `model_aniso.f90` in directory `src/generate_databases/`. See Chapter 13 for a discussion on how to specify your own 3D model.

**external** For a user-specified isotropic model which uses externally defined model parameters. Uses external model definitions set up in routines located in file `model_external_values.f90` in directory `src/generate_databases/`. Please modify these generic template routines to use your own model definitions.

**g11** For a user-specified isotropic model which uses external binary files for  $v_p$ ,  $v_s$  and  $\rho$ . Binary files are given in the same format as when outputted by the `xgenerate_databases` executable when using option `SAVE_MESH_FILES`. These binary files define the model parameters on all GLL points which can be used for iterative inversion procedures. Note that for simulation setups with attenuation, it will also read in the external binary mesh files for  $Q_\kappa$  and  $Q_\mu$ . Note that  $Q_{\mu}$  is always equal to  $Q_s$ , but  $Q_{\kappa}$  is in general not equal to  $Q_p$ . To convert one to the other see `doc/note_on_Qkappa_versus_Qp.pdf` and `utils/attenuation/conversion_from_Qkappa_Qmu_to_Qp_Qs_from_Dahlen_T`

**salton\_trough** A 3D  $V_p$  model for Southern California. Users must provide the corresponding data file `regrid3_vel_p.bin` in directory `DATA/st_3D_block_harvard/`.

**tomo** For a user-specified 3D isotropic model which uses a tomographic model file `tomographic_model.xyz` in directory `DATA`. See Section 13.1, for a discussion on how to specify your own 3D tomographic model.

**APPROXIMATE\_OCEAN\_LOAD** Set to `.true.` if the effect of the oceans on seismic wave propagation should be incorporated based upon the (rough) approximate treatment discussed in Komatitsch and Tromp [2002b]. This feature is inexpensive from a numerical perspective, both in terms of memory requirements and CPU time. This approximation is accurate at periods of roughly 20 s and longer. At shorter periods the effect of water phases/reverberations is not taken into account, even when the flag is on. If you want to model the effect of a fluid-solid model at short periods, then set this flag to `.false.` and mesh the fluid layer explicitly in your mesher, so that it is computed accurately and without this approximation.

**TOPOGRAPHY** This feature is only effective if `APPROXIMATE_OCEAN_LOAD` is set to `.true..` Set to `.true.` if topography and bathymetry should be read in based upon the topography file specified in the main constants file `constants.h` found in subdirectory `src/shared/` to evaluate elevations. If not set, elevations will be read from the numerical mesh.

**ATTENUATION** Set to `.true.` if attenuation should be incorporated. Turning this feature on increases the memory requirements significantly (roughly by a factor of 1.5), and is numerically fairly expensive. See Komatitsch and Tromp [1999, 2002a] for a discussion on the implementation of attenuation based upon standard linear solids. Please note that the Vp- and Vs-velocities of your model are given for a reference frequency. To change this reference frequency, you change the value of `ATTENUATION_f0_REFERENCE` in the main constants file `constants.h` found in subdirectory `src/shared/`. The code uses a constant  $Q$  quality factor, `write(IMAIN,*)` "but approximated based on a series of Zener standard linear solids (SLS). The approximation is thus performed in a given frequency band determined based on that `ATTENUATION_f0_REFERENCE` reference frequency. Note that  $Q_{mu}$  is always equal to  $Q_s$ , but  $Q_{kappa}$  is in general not equal to  $Q_p$ . To convert one to the other see `doc/note_on_Qkappa_versus_Qp.pdf` and `utils/attenuation/conversion_from_Qkappa_Qmu_to_Qp_Qs_from_Dahler`.

**ANISOTROPY** Set to `.true.` if you want to use an anisotropy model. Please see the file `model_aniso.f90` in subdirectory `src/generate_databases/` for the current implementation of anisotropic models.

**TOMOGRAPHY\_PATH** Directory in which the tomography files are stored for using external tomographic Earth models (please read Chapter 13 and Section 13.1 ‘Using external tomographic Earth models’ for further details.).

**USE\_OLSEN\_ATTENUATION** Set to `.true.` if you want to use the attenuation model that scaled from the S-wave speed model using Olsen’s empirical relation (see Olsen et al. [2003]).

**OLSEN\_ATTENUATION\_RATIO** Determines the Olsen’s constant in Olsen’s empirical relation (see Olsen et al. [2003]).

**PML\_CONDITIONS** Set to `.true.` to turn on C-PML boundary conditions for a regional simulation. Both fluids and elastic solids are supported. Note that `xmeshfem3d` generated meshes do not support C-PML yet.

**PML\_INSTEAD\_OF\_FREE\_SURFACE** Set to `.true.` to turn on C-PML boundary conditions on the top surface instead of the usual free surface.

**f0\_FOR\_PML** Determines the dominant frequency that will be used in the calculation of PML damping profiles; this should be set to the same (or similar) dominant frequency as that of the source that you will use in your simulation. If you plan to use a Dirac source, then use the dominant frequency of the source wavelet with which you plan to convolve your seismograms later on in post-processing.

**STACEY\_ABSORBING\_CONDITIONS** Set to `.true.` to turn on Clayton-Enquist absorbing boundary conditions (see Komatitsch and Tromp [1999]). In almost all cases it is much better to use CPML absorbing layers (see the options above) and leave this flag to `.false..`

**STACEY\_INSTEAD\_OF\_FREE\_SURFACE** Set to `.true.` to turn on absorbing boundary conditions on the top surface which by default constitutes a free surface of the model.

**BOTTOM\_FREE\_SURFACE** When STACEY\_ABSORBING\_CONDITIONS is set to .true. : absorbing conditions are defined in xmin, xmax, ymin, ymax and zmin this option BOTTOM\_FREE\_SURFACE can be set to .true. to make zmin free surface instead of absorbing condition.

**CREATE\_SHAKEMAP** Set this flag to .true. to create a ShakeMap®, i.e., a peak ground velocity map of the maximum absolute value of the two horizontal components of the velocity vector.

**MOVIE\_SURFACE** Set to .false., unless you want to create a movie of seismic wave propagation on the Earth's surface. Turning this option on generates large output files. See Section 11.2 for a discussion on the generation of movies. This feature is only relevant for the solver.

**MOVIE\_TYPE** Set this flag to 1 to show the top surface (tomography + oceans) only, to 2 to show all external faces of the mesh (i.e. topography + vertical edges + bottom) in shakemaps and surface movies.

**MOVIE\_VOLUME** Set to .false., unless you want to create a movie of seismic wave propagation in the Earth's interior. Turning this option on generates huge output files. See Section 11.2 for a discussion on the generation of movies. This feature is only relevant for the solver.

**SAVE\_DISPLACEMENT** Set this flag to .true. if you want to save the displacement instead of velocity for the movie frames.

**USE\_HIGHRES\_FOR\_MOVIES** Set this flag to .true. if you want to save the values at all the NGLL grid points for the movie frames.

**NTSTEP\_BETWEEN\_FRAMES** Determines the number of timesteps between movie frames. Typically you want to save a snapshot every 100 timesteps. The smaller you make this number the more output will be generated! See Section 11.2 for a discussion on the generation of movies. This feature is only relevant for the solver.

**HDUR\_MOVIE** Determines the half duration of the source time function for the movie simulations. When this parameter is set to be 0, a default half duration that corresponds to the accuracy of the simulation is provided. Otherwise, it adds this half duration to the half duration specified in the source file CMTSOLUTION, thus simulates longer periods to make the movie images look smoother.

**SAVE\_MESH\_FILES** Set this flag to .true. to save ParaView ([www.paraview.org](http://www.paraview.org)) mesh files for subsequent viewing. Turning the flag on generates large (distributed) files in the LOCAL\_PATH directory. See Section 11.1 for a discussion of mesh viewing features.

**LOCAL\_PATH** Directory in which the distributed databases will be written. Generally one uses a directory on the local disk of the compute nodes, although on some machines these databases are written on a parallel (global) file system (see also the earlier discussion of the LOCAL\_PATH\_IS\_ALSO\_GLOBAL flag in Chapter 2). xgenerate\_databases generates the necessary databases in parallel, one set for each of the NPROC slices that constitutes the mesh (see Figure 3.4 and Figure 3.5). After the executable finishes, you can log in to one of the compute nodes and view the contents of the LOCAL\_PATH directory to see the (many) files generated by xgenerate\_databases. Please note that the LOCAL\_PATH directory should already contain the output files of the partitioner, i.e. from xdecompose\_mesh or xmshfem3D.

**NTSTEP\_BETWEEN\_OUTPUT\_INFO** This parameter specifies the interval at which basic information about a run is written to the file system (timestamp\* files in the OUTPUT\_FILES directory). If you have access to a fast machine, set NTSTEP\_BETWEEN\_OUTPUT\_INFO to a relatively high value (e.g., at least 100, or even 1000 or more) to avoid writing output text files too often. This feature is not used at the time of meshing. One can set this parameter to a larger value than the number of time steps to avoid writing output during the run.

**NTSTEP\_BETWEEN\_OUTPUT\_SEISMOS** This parameter specifies the interval at which synthetic seismograms are written in the LOCAL\_PATH directory. If a run crashes, you may still find usable (but shorter than requested) seismograms in this directory. On a fast machine set NTSTEP\_BETWEEN\_OUTPUT\_SEISMOS to a relatively high value to avoid writing to the seismograms too often. This feature is only relevant for the solver.

**USE\_FORCE\_POINT\_SOURCE** Turn this flag on to use a (tilted) FORCESOLUTION force point source instead of a CMTSOLUTION moment-tensor source. When the force source does not fall exactly at a grid point, the solver interpolates the force between grid points using Lagrange interpolants. This can be useful e.g. for oil industry foothills simulations in which the source is a vertical force, normal force, tilted force, or an impact etc. Note that in the FORCESOLUTION file, you will need to edit the East, North and vertical components of an arbitrary (not necessarily unitary, the code will normalize it automatically) direction vector of the force vector; thus refer to Appendix A for the orientation of the reference frame. This vector is made unitary internally in the solver and thus only its direction matters here; its norm is ignored and the norm of the force used is the factor force source times the source time function.

When using this option, by default the code can locate the force source anywhere between mesh points in order to honor its exact location; this is more precise than using the closest GLL mesh point, but it is also a bit slower. If needed, you can change that default behavior and force the code to use the closest GLL mesh point instead by setting flag `USE_BEST_LOCATION` to `.false.` instead of `.true.` in file `src/shared/constants.h.in` and running the `configure` script again and recompiling the code.

**USE\_RICKER\_TIME\_FUNCTION** Turn this flag on to use a Ricker source time function instead of the source time functions set by default to represent a (tilted) FORCESOLUTION force point source or a CMTSOLUTION moment-tensor source. Originally, if a CMTSOLUTION moment-tensor source is used, a (pseudo) Heaviside step function with a very short half duration is defined for elastic cases to represent the permanent slip on the fault while in the acoustic case a Gaussian source time function with a similarly short half duration is defined to physically describe actions within the fluid. Otherwise, if a FORCESOLUTION force source is used, a (pseudo) Dirac delta source time function is defined by default. Any other source-time function may then be obtained by convolution.

**PRINT\_SOURCE\_TIME\_FUNCTION** Turn this flag on to print information about the source time function in the file `OUTPUT_FILES/plot_source_time_function.txt`. This feature is only relevant for the solver.

**NUMBER\_OF\_SIMULTANEOUS\_RUNS** adds the ability to run several calculations (several earthquakes) in an embarrassingly-parallel fashion from within the same run; this can be useful when using a very large supercomputer to compute many earthquakes in a catalog, in which case it can be better from a batch job submission point of view to start fewer and much larger jobs, each of them computing several earthquakes in parallel.

To turn that option on, set parameter `NUMBER_OF_SIMULTANEOUS_RUNS` to a value greater than 1. To implement that, we create `NUMBER_OF_SIMULTANEOUS_RUNS` MPI sub-communicators, each of them being labeled `my_local_mpi_comm_world`, and we use them in all the routines in "src/shared/parallel.f90", except in `MPI_ABORT()` because in that case we need to kill the entire run.

When that option is on, of course the number of processor cores used to start the code in the batch system must be a multiple of `NUMBER_OF_SIMULTANEOUS_RUNS`, all the individual runs must use the same number of processor cores, which as usual is `NPROC` in the `Par_file`, and thus the total number of processor cores to request from the batch system should be `NUMBER_OF_SIMULTANEOUS_RUNS * NPROC`. All the runs to perform must be placed in directories called `run0001`, `run0002`, `run0003` and so on (with exactly four digits).

Imagine you have 10 independent calculations to do, each of them on 100 cores; you have three options:

- 1/ submit 10 jobs to the batch system
- 2/ submit a single job on 1000 cores to the batch, and in that script create a sub-array of jobs to start 10 jobs, each running on 100 cores (see e.g. [http://www.schedmd.com/slurmdocs/job\\_array.html](http://www.schedmd.com/slurmdocs/job_array.html))
- 3/ submit a single job on 1000 cores to the batch, start SPECFEM3D on 1000 cores, create 10 sub-communicators, cd into one of 10 subdirectories (called e.g. `run0001`, `run0002`,... `run0010`) depending on the sub-communicator your MPI rank belongs to, and run normally on 100 cores using that sub-communicator.

The option `NUMBER_OF_SIMULTANEOUS_RUNS` implements 3/.

**BROADCAST\_SAME\_MESH\_AND\_MODEL** if we perform simultaneous runs in parallel, if only the source and receivers vary between these runs but not the mesh nor the model (velocity and density) then we can also read the mesh and model files from a single run in the beginning and broadcast them to all the others; for a large number of simultaneous runs for instance when solving inverse problems iteratively this can DRASTICALLY reduce

I/Os to disk in the solver (by a factor equal to NUMBER\_OF\_SIMULTANEOUS\_RUNS), and reducing I/Os is crucial in the case of huge runs. Thus, always set this option to .true. if the mesh and the model are the same for all simultaneous runs. In that case there is no need to duplicate the mesh and model file database (the content of the DATABASES\_MPI directories) in each of the run0001, run0002,... directories, it is sufficient to have one in run0001 and the code will broadcast it to the others).

**USE\_FAILSAFE\_MECHANISM** if one or a few of these simultaneous runs fail, kill all the runs or let the others finish using a fail-safe mechanism (in most cases, should be set to true).

TODO / future work to do: currently the BROADCAST\_SAME\_MESH\_AND\_MODEL option assumes to have the (master) mesh files in run0001/DATABASES\_MPI or run0001/OUTPUT\_FILES/DATABASES\_MPI. However, for adjoint runs you still need a DATABASES\_MPI/ folder in each of the sub-runs directories, e.g. run0002/DATABASES\_MPI, etc. to store the forward wavefields, kernels etc. of each sub-run. This would not be needed for forward simulations.

TODO / future work to do: the sensitivity kernel summing and smoothing tools in directory src/tomography are currently not ported to this new option to do many runs simultaneously, only the solver (src/specfem3d) is. Thus these tools should work, but in their current version will need to be run for each simulation result independently.

More precisely, the current kernel summing and smoothing routines work fine, with the exception that you need to move out the mesh files (and also the parameters). This works because these routines consider multiple runs by design. You simply have to provide them the directories where the kernels are.

**GPU\_MODE** Turn this flag on to use GPUs.

**ADIOS\_ENABLED** Turn this flag on to enable ADIOS. If set to .false., subsequent ADIOS parameters will not be considered.

**ADIOS\_FOR\_DATABASES** Turn this flag on to use ADIOS for xmshfem3D output and xgenerate\_database input.

**ADIOS\_FOR\_MESH** Turn this flag on to use ADIOS for generated databases.

**ADIOS\_FOR\_FORWARD\_ARRAYS** Turn this flag on to read and write forward arrays using ADIOS.

**ADIOS\_FOR KERNELS** Turn this flag on to produce ADIOS kernels that can later be visualized with the ADIOS version of combine\_vol\_data.

The present version of SPECFEM can handle fully saturated porous simulations, Christina Morency implemented Biot equation. But the code cannot calculate partially saturated cases in its state. Christina Morency is presently working on a dual porosity, dual permeability formulation, type Pride and Berryman, for an other project, but it will not be available for some time.

The way we prescribe material property for porous material in SPECFEM3D is as follow: We use a file name "nummaterial\_poroelastic\_file", which is located in the directory MESH/, the format is as follow:

```
rhos rhof phi c kxx kxy kxz kyy kyz kzz Ks Kf Kfr etaf mufr
```

where

$\rho_s$  = solid density,

$\rho_f$  = fluid density,

$\phi$  = porosity,

$t_{ort}$  = tortuosity,

$k_{xx}$  = xx component of permeability tensor,

$k_{xy}$  = xy,yx components of permeability tensor,

$k_{yy}$  = yy component of permeability tensor,

$k_{xz}$  = xz,zx components of permeability tensor,

$k_{zz}$  = 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_f` = frame shear modulus.

Using an external mesh (for instance coming from CUBIT/TRELIS), poroelastic materials have the ID number 3, while 1 is acoustic and 2 is elastic (see the example in the package: EXAMPLES/homogeneous\_poroelastic).

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 CUBIT/TRELIS or similar), try to have elements inside the PML as regular as possible, i.e. ideally non-deformed cubes obtained by ‘extrusion’ of regular surface 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). **We have two utilities in directory utils/CPML that do that automatically and that are very fast.** To stabilize PMLs it also helps to add a transition layer of geometrically-regular non-PML elements, in which attenuation is also turned off (i.e.  $Q_\kappa = Q_\mu = 9999$  in that layer), as in the red layer of Figure 4.2. Our tools in directory utils/CPML implement that transition layer automatically.

If you use PML and an external tomographic velocity and density model, 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 4.3.

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 3D 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.

## 4.2 Choosing the time step DT

The parameter `DT` sets the length of each time step in seconds. The value of this parameter is crucial for the stability of the spectral-element simulation. Your time step `DT` will depend on the minimum ratio between the distance  $h$  of neighboring mesh points and the wave speeds  $v$  defined in your model. The condition for the time step  $\Delta t$  is:

$$\Delta t < C \min_{\Omega} ( h/v )$$

where  $C$  is the so-called Courant number and  $\Omega$  denotes the model volume. The distance  $h$  depends on the mesh element size and the number of GLL points `NGLL` specified in the main constants file `constants.h` located in the `src/shared/` subdirectory. The wave speed  $v$  is determined based on your model’s P- (or S-) wave speed values.

The database generator `xgenerate_databases`, as well as the internal mesher `xmeshfem3D`, are trying to evaluate the value of  $\Delta t$  for empirically chosen Courant numbers  $C \sim 0.3$ . If you used the mesher `xmeshfem3D` to generate your mesh, you should set the value suggested in `OUTPUT_FILES/output_mesher.txt` file, which is created after the mesher completed. In case you used CUBIT to create the mesh, you might use an arbitrary value when running `xgenerate_databases` and then use the value suggested in the `OUTPUT_FILES/output_mesher.txt` file after the database generation completed. Note that the implemented Newmark time scheme uses this time step globally, thus your simulations become more expensive for very small mesh elements in high wave-speed regions. Please be aware of this restriction when constructing your mesh in Chapter 3.

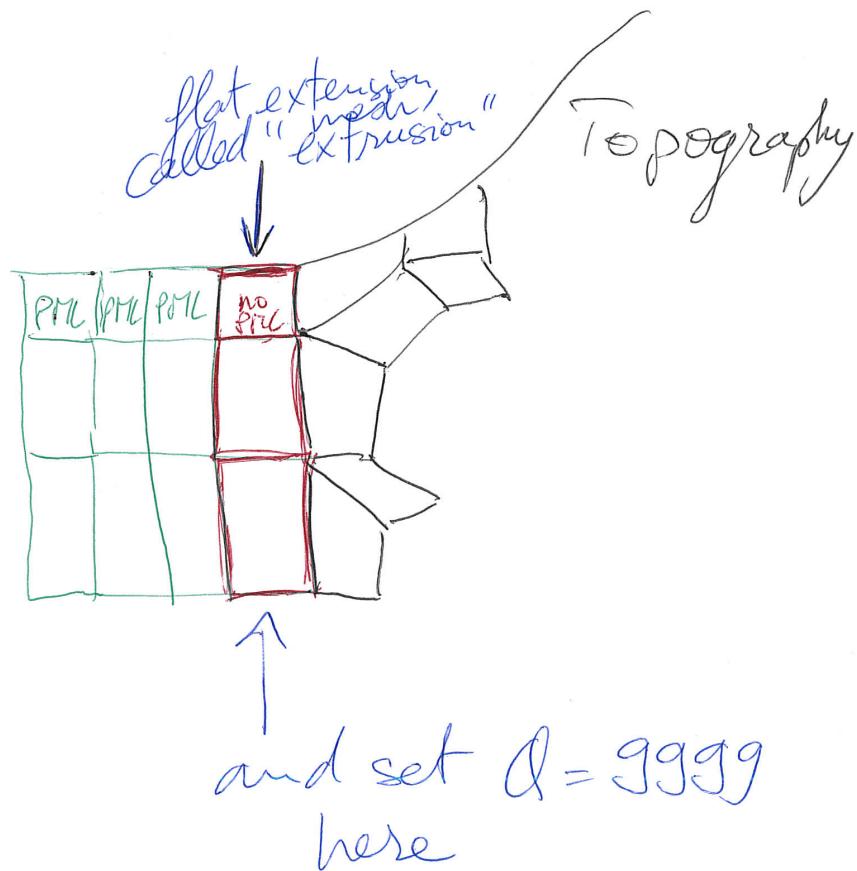


Figure 4.2: Mesh extrusion for PML (green elements) and a non-PML stabilization layer (red elements).

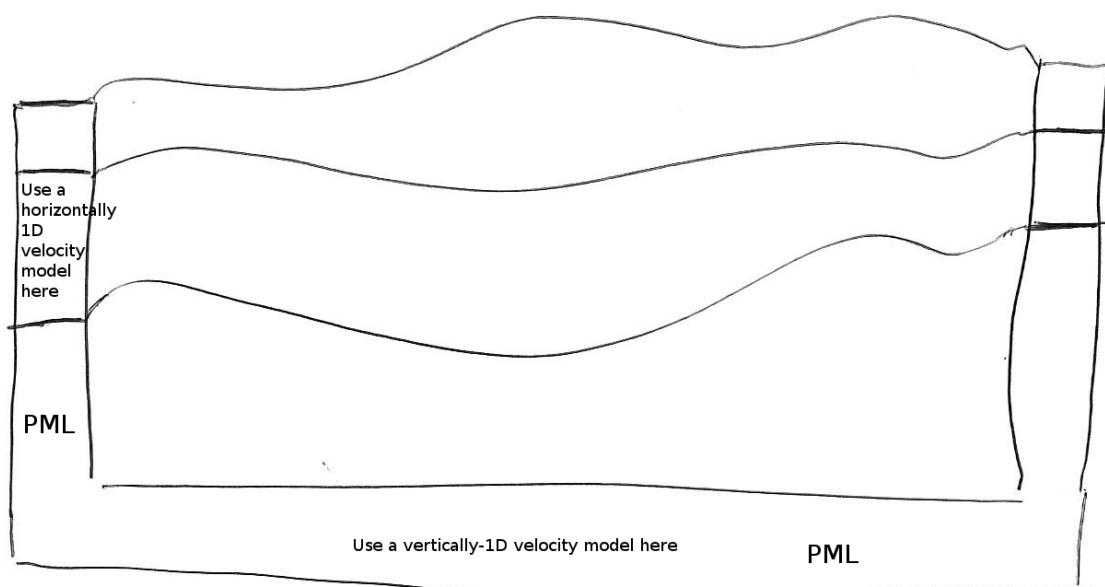


Figure 4.3: How to modify your external 3D velocity and density model in order to use PML. Such a modification is not needed when using Stacey absorbing boundary conditions (but such conditions are significantly less efficient).

# Chapter 5

## Running the Solver `xspecfem3D`

Now that you have successfully generated the databases, you are ready to compile the solver. In the main directory, type

```
make xspecfem3D
```

Please note that `xspecfem3D` must be called directly from the main directory, as most of the binaries of the package.

The solver needs three input files in the `DATA` directory to run:

**Par\_file** the main parameter file which was discussed in detail in the previous Chapter 4,

**CMTSOLUTION or FORCESOLUTION** the earthquake source parameter file or the force source parameter file, and

**STATIONS** the stations file.

Most parameters in the `Par_file` should be set prior to running the databases generation. Only the following parameters may be changed after running `xgenerate_databases`:

- the simulation type control parameters: `SIMULATION_TYPE` and `SAVE_FORWARD`
- the time step parameters `NSTEP` and `DT`
- the absorbing boundary control parameter `PML_CONDITIONS` on condition that the `PML_INSTEAD_OF_FREE_SURFACE` flag remains unmodified after running the databases generation.
- the movie control parameters `MOVIE_SURFACE`, `MOVIE_VOLUME`, and `NTSTEPS_BETWEEN_FRAMES`
- the ShakeMap® option `CREATE_SHAKEMAP`
- the output information parameters `MOVIE_TYPE`, `NTSTEP_BETWEEN_OUTPUT_INFO` and `NTSTEP_BETWEEN_OUTPUT_SEISMOS`
- the `PRINT_SOURCE_TIME_FUNCTION` flags

Any other change to the `Par_file` implies rerunning both the database generator `xgenerate_databases` and the solver `xspecfem3D`.

For any particular earthquake, the `CMTSOLUTION` file that represents the point source may be obtained directly from the Harvard Centroid-Moment Tensor (CMT) web page ([www.seismology.harvard.edu](http://www.seismology.harvard.edu)). It looks like the example shown in Fig. 5.1. The `CMTSOLUTION` file should be edited in the following way:

- Set the latitude or UTM *x* coordinate, longitude or UTM *y* coordinate, depth of the source (in km).
- Set the time shift parameter equal to 0.0 (the solver will not run otherwise.) The time shift parameter would simply apply an overall time shift to the synthetics, something that can be done in the post-processing (see Section 14.1).

Preliminary Determination of Epicenter										body-wave magnitude	surface-wave magnitude
year	day	min	month	hour	sec	latitude	longitude	depth	mb	Ms	PDE event name
PDE 2001	9	23	59	17.78	34.0745	-118.3792	6.4	4.2	4.2	HOLLYWOOD	
event name:	9703873										
time shift:	0.0000										
half duration:	0.0000										
latutorUM:	34.0745										
longorUM:	-118.3792										
depth:	5.4000										
Mrr:	-0.002000e+23										
Mtt:	-0.064000e+23										
Mpp:	0.066000e+23										
Mrt:	-0.090000e+23										
Mrp:	-0.002000e+23										
Mtp:	0.188000e+23										

$$\mathbf{M} = \begin{bmatrix} M_{rr} & M_{r\theta} & M_{r\phi} \\ M_{r\theta} & M_{\theta\theta} & M_{\theta\phi} \\ M_{r\phi} & M_{\theta\phi} & M_{\phi\phi} \end{bmatrix}$$

$$M_0 = \frac{1}{\sqrt{2}} (\mathbf{M} : \mathbf{M})^{1/2} \approx 2.18 \times 10^{22} \text{ dyne cm}$$

$$M_w = \frac{2}{3} (\log_{10} M_0 - 16.1) \approx 4.19$$

Figure 5.1: CMTSOLUTION file based on the format from the Harvard CMT catalog.  $\mathbf{M}$  is the moment tensor,  $M_0$  is the seismic moment, and  $M_w$  is the moment magnitude.

- For point-source simulations (see finite sources, page 38) we recommend setting the source half-duration parameter `half duration` equal to zero, which corresponds to simulating a step source-time function, i.e., a moment-rate function that is a delta function. If `half duration` is not set to zero, the code will use a Gaussian (i.e., a signal with a shape similar to a ‘smoothed triangle’, as explained in Komatitsch and Tromp [2002a] and shown in Fig 5.2) source-time function with half-width `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 (see Section 14.1). Komatitsch and Tromp [2002a] 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.csh` 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.csh` to the desired half-duration.

- 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  $t = -1.5 * \text{half duration}$  (the 1.5 is to make sure the moment rate function is very close to zero when starting the simulation). To convert to absolute time  $t_{\text{abs}}$ , set

$$t_{\text{abs}} = t_{\text{pde}} + \text{time shift} + t_{\text{synthetic}}$$

where  $t_{\text{pde}}$  is the time given in the first line of the CMTSOLUTION, `time shift` is the corresponding value from the original CMTSOLUTION file and  $t_{\text{synthetic}}$  is the time in the first column of the output seismogram.

If you know the earthquake source in strike/dip/rake format rather than in CMTSOLUTION format, use the C code `SPECFEM3D_GLOBE/utils/strike_dip_rake_to_CMTSOLUTION.c` to convert it. The conversion formulas are given for instance in Aki and Richards [1980]. Note that the Aki and Richards [1980] convention is slightly different from the Harvard CMTSOLUTION convention (the sign of some components is different). The C code outputs both.

Centroid latitude and longitude should be provided in geographical coordinates. The code converts these coordinates to geocentric coordinates [Dahlen and Tromp, 1998]. Of course you may provide your own source representations by designing your own CMTSOLUTION file. Just make sure that the resulting file adheres to the Harvard CMT conventions (see Appendix A). Note that the first line in the CMTSOLUTION file is the Preliminary Determination of Earthquakes (PDE) solution performed by the USGS NEIC, which is used as a seed for the Harvard CMT inversion.

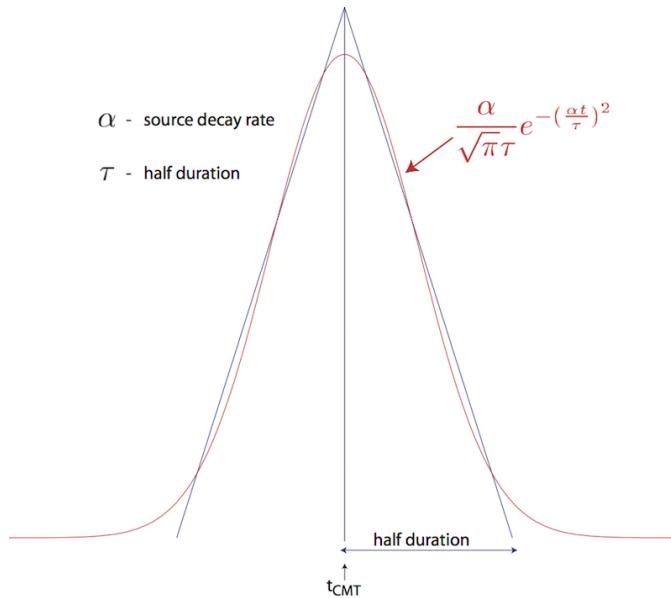


Figure 5.2: Comparison of the shape of a triangle and the Gaussian function actually used.

The PDE solution is based upon P waves and often gives the hypocenter of the earthquake, i.e., the rupture initiation point, whereas the CMT solution gives the ‘centroid location’, which is the location with dominant moment release. The PDE solution is not used by our software package but must be present anyway in the first line of the file.

To simulate a kinematic rupture, i.e., a finite-source event, represented in terms of  $N_{sources}$  point sources, provide a CMTSOLUTION file that has  $N_{sources}$  entries, one for each subevent (i.e., concatenate  $N_{sources}$  CMTSOLUTION files to a single CMTSOLUTION file). At least one entry (not necessarily the first) must have a zero time shift, and all the other entries must have non-negative time shift. Each subevent can have its own half duration, latitude, longitude, depth, and moment tensor (effectively, the local moment-density tensor).

Note that the zero in the synthetics does NOT represent the hypocentral time or centroid time in general, but the timing of the *center* of the source triangle with zero time shift (Fig 5.3).

Although it is convenient to think of each source as a triangle, in the simulation they are actually Gaussians (as they have better frequency characteristics). The relationship between the triangle and the gaussian used is shown in Fig 5.2. For finite fault simulations it is usually not advisable to use a zero half duration and convolve afterwards, since the half duration is generally fixed by the finite fault model.

The FORCESOLUTION file should be edited in the following way:

- Set the time shift parameter equal to 0.0 (the solver will not run otherwise.) The time shift parameter would simply apply an overall time shift to the synthetics, something that can be done in the post-processing (see Section 14.1).
- Set the f0 parameter (the dominant frequency) of the Ricker source time function when USE\_RICKER\_TIME\_FUNCTION is turned on in the main parameter file Par\_file. In case that the solver uses a (pseudo) Dirac delta source time function to represent a force point source, a very short duration of five time steps is automatically set by default.
- Set the latitude or UTM x coordinate, longitude or UTM y coordinate, depth of the source (in km).
- Set the magnitude of the force source.
- Set the components of a (non-unitary) direction vector for the force source in the East/North/Vertical basis (see Appendix A for the orientation of the reference frame).

Where necessary, set a `FORCESOLUTION` file in the same way you configure a `CMTSOLUTION` file with  $N_{\text{sources}}$  entries, one for each subevent (i.e., concatenate  $N_{\text{sources}}$  `FORCESOLUTION` files to a single `FORCESOLUTION` file). At least one entry (not necessarily the first) must have a zero time shift, and all the other entries must have non-negative time shift. Each subevent can have its own half latitude, longitude, depth, half duration and force parameters.

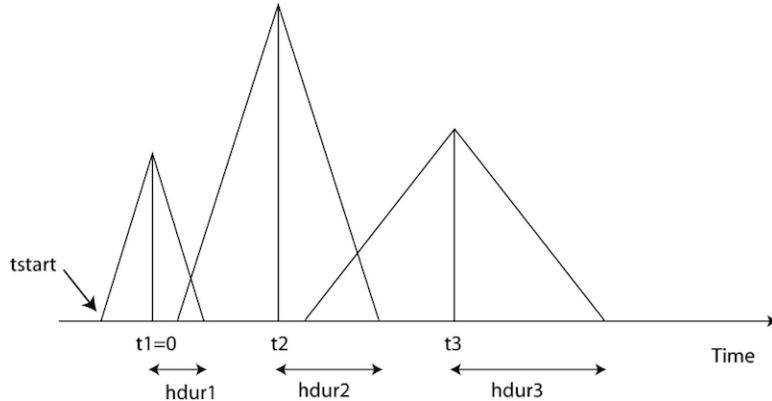


Figure 5.3: Example of timing for three sources. The center of the first source triangle is defined to be time zero. Note that this is NOT in general the hypocentral time, or the start time of the source (marked as `tstart`). The parameter `time shift` in the `CMTSOLUTION` file would be  $t_1(=0)$ ,  $t_2$ ,  $t_3$  in this case, and the parameter `half duration` would be  $hdur1$ ,  $hdur2$ ,  $hdur3$  for the sources 1, 2, 3 respectively.

In addition to inbuild source time function, the solver can also use an external source time function defined by the user. This option can be activated by setting `EXTERNAL_SOURCE_FILE` to true in `Par_File` and by adding the name of the file containing the source time function at the end of `FORCESOLUTION` or `CMTSOLUTION` files. The source time function file must contain 2 columns with the current time and the amplitude of the source function. The time step must be the same as used for the simulation. Note when the `EXTERNAL_SOURCE_FILE` is set to false then the line with the external source time function file must not appear in the files `FORCESOLUTION` and `CMTSOLUTION` otherwise the solver will exit with an error.

The solver can calculate seismograms at any number of stations for basically the same numerical cost, so the user is encouraged to include as many stations as conceivably useful in the `STATIONS` file, which looks like this:

Network		Longitude (deg)		Burial (m)	
Station	Latitude (deg)			Elevation (m)	
ASBS	AZ	33.6208	-116.4664	0.0	0.0
BZN	AZ	33.4915	-116.6670	0.0	0.0
CRY	AZ	33.5654	-116.7373	0.0	0.0
ELKS	AZ	33.5813	-116.4496	0.0	0.0
AGA	CI	33.6384	-116.4011	0.0	0.0
AGO	CI	34.1465	-118.7670	0.0	0.0
ALP	CI	34.6870	-118.2995	0.0	0.0
BAK	CI	35.3444	-119.1044	0.0	0.0
BAR	CI	32.6801	-116.6722	0.0	0.0
BBA	CI	34.1955	-118.3534	0.0	0.0
BBB	CI	33.3526	-115.7332	0.0	0.0
BBR	CI	34.2623	-116.9207	0.0	0.0
BBS	CI	33.9214	-116.9805	0.0	0.0
:	:	:	:	:	:

Figure 5.4: Sample STATIONS file. Station latitude and longitude should be provided in geographical coordinates. The width of the station label should be no more than 32 characters (see MAX\_LENGTH\_STATION\_NAME in the constants.h file), and the network label should be no more than 8 characters (see MAX\_LENGTH\_NETWORK\_NAME in the constants.h file).

Each line represents one station in the following format:

```
Station Network Latitude(degrees) Longitude(degrees) Elevation(m) burial(m)
```

The solver xspecfem3D filters the list of stations in file DATA/STATIONS to exclude stations that are not located within the region given in the Par\_file (between LATITUDE\_MIN and LATITUDE\_MAX and between LONGITUDE\_MIN and LONGITUDE\_MAX). The filtered file is called DATA/STATIONS\_FILTERED.

Elevation and burial are generally applicable to geographical regions. Burial is measured down from the top surface. For other problems in other fields (ultrasonic testing, medical imaging etc...), it may be confusing. We generally follow either of the following procedures for those kind of problems:

Procedure 1:

```
=====
```

- Always put the origin on the top of the model.
- Let's say you want to place two receivers at (x1,y1,z1) and (x2,y2,z2). Your STATIONS file should look like:

```
BONE GR y1 x1 0.00 -z1
BONE GR y2 x2 0.00 -z2
```

Depending on the geometry of the models it may not work in some cases.

Procedure 2: (less recommended)

```
=====
```

- Open setup/constants.h.in file and change the line logical, parameter :: USE\_SOURCES\_RECEIVERS\_Z = .false.

to

```
logical, parameter :: USE_SOURCES_RECEIVERS_Z = .true.
```

configure the package and compile.

- Let's say you want to place two receivers at (x1,y1,z1) and (x2,y2,z2). Your STATIONS file should look like:

```
BONE GR y1 x1 0.00 z1
BONE GR y2 x2 0.00 z2
```

The option set to .true. in 1 will discard the elevation and set burial as the *z* coordinate.

Third column is Y and Fourth is X due to the latitude/longitude convention.

You can replace the station name "BONE" with any word of length less than 32, and the network name "GR" with any word of length less than 8.

You can always plot OUTPUT\_FILES/sr.vtk file in ParaView to check the source/receiver locations after your simulation.

Solver output is provided in the OUTPUT\_FILES directory in the `output_solver.txt` file. Output can be directed to the screen instead by uncommenting a line in `constants.h`:

```
! uncomment this to write messages to the screen
! integer, parameter :: IMAIN = ISTANDARD_OUTPUT
```

On PC clusters the seismogram files are generally written to the local disks (the path LOCAL\_PATH in the `Par_file`) and need to be gathered at the end of the simulation.

While the solver is running, its progress may be tracked by monitoring the 'timestamp\*' files in the OUTPUT\_FILES/ directory. These tiny files look something like this:

```
Time step #      10000
Time:    108.4890      seconds
Elapsed time in seconds =   1153.28696703911
Elapsed time in hh:mm:ss =   0 h 19 m 13 s
Mean elapsed time per time step in seconds =   0.115328696703911
Max norm displacement vector U in all slices (m) =   1.0789589E-02
```

The timestamp\* files provide the Mean elapsed time per time step in seconds, which may be used to assess performance on various machines (assuming you are the only user on a node), as well as the Max norm displacement vector U in all slices (m). If something is wrong with the model, the mesh, or the source, you will see the code become unstable through exponentially growing values of the displacement and fluid potential with time, and ultimately the run will be terminated by the program. You can control the rate at which the timestamp files are written based upon the parameter NTSTEP\_BETWEEN\_OUTPUT\_INFO in the `Par_file`.

Having set the `Par_file` parameters, and having provided the CMTSOLUTION (or the FORCESOLUTION) and STATIONS files, you are now ready to launch the solver! This is most easily accomplished based upon the go\_solver script (See Chapter 12 for information about running through a scheduler, e.g., LSF). You may need to edit the last command at the end of the script that invokes the `mpirun` command. The `runall` script compiles and runs both `xgenerate_databases` and `xspecfem3D` in sequence. This is a safe approach that ensures using the correct combination of distributed database output and solver input.

It is important to realize that the CPU and memory requirements of the solver are closely tied to choices about attenuation (ATTENUATION) and the nature of the model (i.e., isotropic models are cheaper than anisotropic models). We encourage you to run a variety of simulations with various flags turned on or off to develop a sense for what is involved.

For the same model, one can rerun the solver for different events by simply changing the CMTSOLUTION or FORCESOLUTION file, or for different stations by changing the STATIONS file. There is no need to rerun the `xgenerate_databases` executable. Of course it is best to include as many stations as possible, since this does not add to the cost of the simulation.

We have also added the ability to run several calculations (several earthquakes) in an embarrassingly-parallel fashion from within the same run; this can be useful when using a very large supercomputer to compute many earthquakes in a catalog, in which case it can be better from a batch job submission point of view to start fewer and much larger jobs, each of them computing several earthquakes in parallel. To turn that option on, set parameter NUMBER\_OF\_SIMULTANEOUS\_RUNS to a value greater than 1 in file setup/constants.h.in before configuring and compiling the code. When that option is on, of course the number of processor cores used to start the code in the batch system must be a multiple of NUMBER\_OF\_SIMULTANEOUS\_RUNS, all the individual runs must use the same number of processor cores, which as usual is NPROC in the input file DATA/Par\_file, and thus the total number of processor cores to request from the batch system should be NUMBER\_OF\_SIMULTANEOUS\_RUNS  $\times$  NPROC. All the runs to perform must be placed in directories called run0001, run0002, run0003 and so on (with exactly four digits) and you must create a link from the root directory of the code to the first copy of the executable programs by typing `ln -s run0001/bin bin`.

## 5.1 Note on the viscoelastic model used

The model used is a constant  $Q$ , thus with no dependence on frequency ( $Q(f) = \text{constant}$ ). See e.g. Blanc et al. [2016].

However in practice for technical reasons it is approximated based on the sum of different Generalized Zener body mechanisms and thus the code outputs the band in which the approximation is very good, outside of that range it can be less accurate. The logarithmic center of that frequency band is the ATTENUATION\_f0 parameter defined (in Hz) in input file DATA/Par\_file.

# Chapter 6

## Kinematic and dynamic fault sources

SPECFEM3D can handle finite fault sources of two kinds:

1. *Kinematic*: the spatio-temporal distribution of slip rate is prescribed along the fault surface
2. *Dynamic*: a friction law and initial stresses are prescribed on the fault, a spontaneous rupture process is computed.

### 6.1 Mesh Generation with Split Nodes

Faults need to be handled in a special way during mesh generation. A fault surface must lie at the interface between elements (the mesh must honor the fault surfaces). Moreover, a fault is made of two surfaces in contact. Each of these two surfaces needs a separate set of nodes. This approach is known as "split nodes". Currently faults can only be run with `xdecompose_mesh` and CUBIT. `xmeshfem3D` is not yet ready to handle faults.

To facilitate the mesh generation with split nodes in CUBIT, we need to separate the two fault surfaces by a small distance, effectively creating a tiny opening of the fault (Figure 6.1, 6.2). Note that the opening distance should not be too small especially in single precision simulations (For example, if your fault is at  $y = y_0$  plane, then the opening distance should be at least  $10^{-6}y_0$  since single precision can ensure precision within only 7 digits). Note that only the interior of the fault must be opened, its edges must remain closed (except the edge on the free surface). The fault is automatically closed later by SPECFEM3D.

Here is an example CUBIT script to generate a mesh with split nodes for a buried vertical strike-slip fault:

```
reset
brick x 10 y 10 z 10
webcut volume all with plane xplane
webcut volume all with plane yplane
webcut volume all with plane xplane offset 3
webcut volume all with plane zplane offset 3
webcut volume all with plane zplane offset -3
imprint all
merge all
unmerge surf 160
mesh vol all
set node constraint off
node in surf 168 move X 0 Y 0.01 Z 0
node in surf 160 move X 0 Y -0.01 Z 0
```

The CUBIT scripts (\*.jou and \*.py) in the directory EXAMPLES generate more complicated meshes. The \*.py files are Python scripts that execute CUBIT commands and use the CUBIT-python interface for SPECFEM3D (see next section). The Python language allows to define and manipulate variables to parameterize the mesh. Alternatively, the Python script can call a CUBIT journal file (\*.jou), which looks like the example above. Variables can be defined and manipulated there using the APREPRO language built in CUBIT.

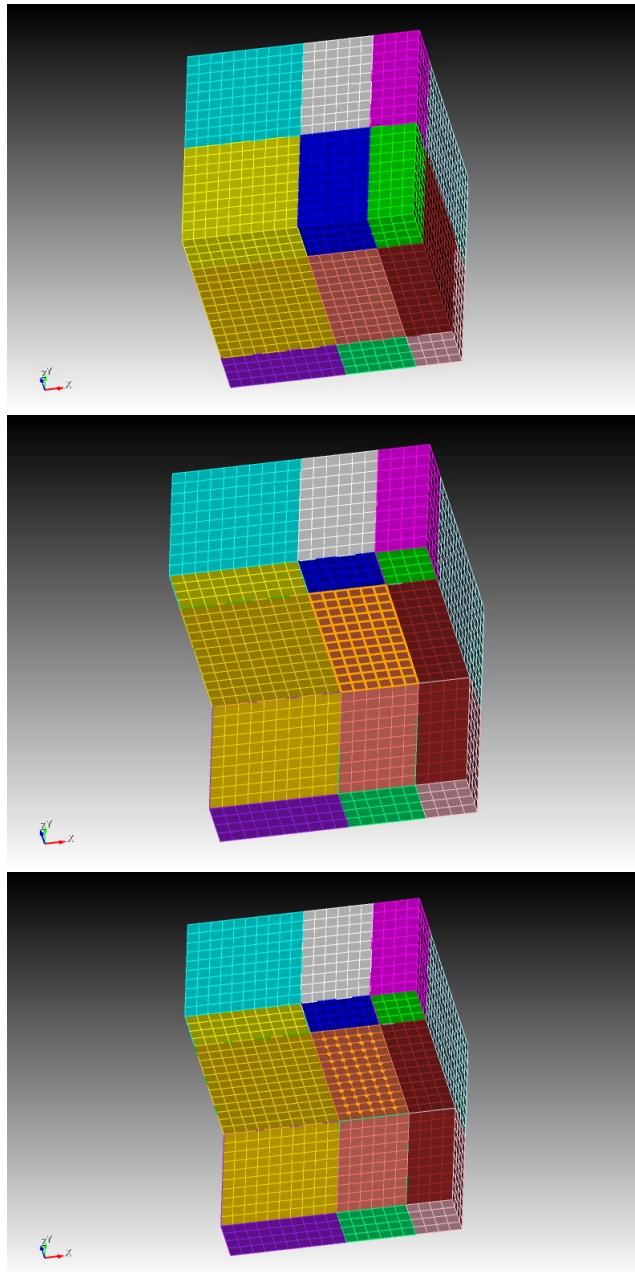


Figure 6.1: Screenshots of the CUBIT example for the embedded fault with split nodes: The entire mesh is decomposed into several volumes bounding the fault (top), the fault surface 168 is shown as orange squares (middle) and split nodes of the fault shown as orange dots (bottom). Note that only interior nodes of the fault are split while those on the edges of the fault surface are touching each other.

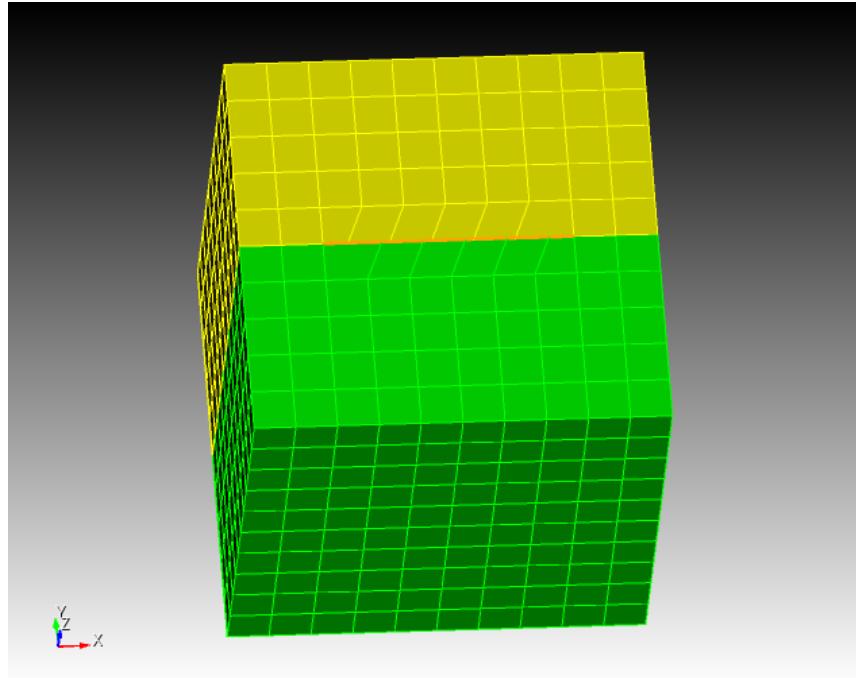


Figure 6.2: Screenshots of a CUBIT example showing split nodes for a fault reaching the surface. Surface trace of the fault is shown in orange. Note that the edges of the fault are not split while the interior nodes are offset by a small distance on either side of the fault

Note that you should avoid gaps in the list of indices of mesh objects with the following CUBIT command:

```
compress ids hex face edge node
```

(otherwise you will get a segmentation fault during domain decomposition)

## 6.2 CUBIT-Python Scripts for Faults

The mesh generated in CUBIT needs to be processed and exported in a format compatible with SPECFEM3D. This is achieved in the Python scripts by calling the Python-CUBIT interface functions defined in the CUBIT directory:

1. Function `define_bc` (or `boundary_definition.py`) must be called to set up the absorbing boundaries database
2. Function `fault_input` must be called once for each fault to set up the fault database
3. Function `cubit2specfem3d.export2SPECFEM3D` must be called at the very end of the script to export the mesh in a SPECFEM3D format.

The functions in #1 and #3 are for the bulk and are documented in Section 3.1.2. We focus here on #2:

**Function:** `fault_input`

**Purpose:** export a fault mesh database from CUBIT to a SPECFEM3D-compliant file

**Syntax:** `fault_input(id_fault, ids_surf_1, ids_surf_2)`

**Inputs:**

- `id_fault` integer index assigned to the fault. (The user must number all the faults, starting at 1, with unit increments)

- `ids_surf_1` list of CUBIT ids of all surfaces that form side 1 of the fault.
- `ids_surf_2` list of CUBIT ids of all surfaces that form side 2 of the fault. (The user must decide which side of the fault is side 1. This choice affects the sign conventions of fault quantities as explained in Section 6.4).

**Outputs:** file `fault_file_X.dat`, where X is the fault id (`id_fault`).

**Example:** For the example in Section 6.1:

```
A1 = [168]
A2 = [160]
fault_input(1,A1,A2)
```

## 6.3 Examples

The package includes examples, the SCEC benchmark problems:

- TPV5, a planar vertical strike-slip fault
- TPV14 and TPV15, vertical strike-slip fault system with a fault branch
- TPV102 and TPV103, rate-and-state benchmarks
- TPV22 and TPV23, step-over benchmarks
- TPV16, heterogenous initial stress

and

- Splay fault models from Wendt et al. (2009)

Read the documents in the directory `EXAMPLES/*/description`. They contain a description of the example and additional instructions to run it. Visualize the results with the Matlab scripts in the directory `EXAMPLES/*/post`

## 6.4 Sign Convention for Fault Quantities

During mesh generation, the fault is defined by two surfaces in contact. Let's denote as "*side 1*" the SECOND surface declared by the user in the call to the python function "`fault_input`", and the FIRST surface as "*side 2*". The local coordinate system on the fault is defined as the right-handed coordinate system defined by (strike, dip, normal), where "*normal*" is the normal vector outgoing from side 1, "*dip*" is parallel to the along-dip direction pointing downwards, and "*strike*" is the horizontal along-strike vector such that the system is right-handed.

Slip is defined as displacement on side 2 minus displacement on side 1. In the local coordinate system on the fault, positive along-strike slip is right-lateral and positive along-dip slip is thrust if side 1 is on the hanging wall (normal faulting if side 1 is on the foot wall).

Traction is defined as the stress induced on side 1 by side 2, which is the stress tensor times the normal vector outgoing from side 1. In the local coordinate system on the fault, the normal traction is negative in compression, positive along-strike traction generates right-lateral slip and positive along-dip traction generates thrust slip if side 1 is on the hanging wall (normal faulting if side 1 is on the foot wall).

## 6.5 Input Files

Three additional input files are required in the DATA/ directory for dynamic and kinematic faults. They are Par\_file\_faults, FAULT\_STATIONS and input\_file.txt or rsf\_hete\_input\_file.txt(optional). If the former does not exist, the code assumes that there are no faults.

**DATA/Par\_file\_faults** contains parameters of the fault. The first part of this file has a strict format:

- Line** 1: Number of faults (NF)
- Lines** 2 to NF+1: Kelvin Voigt damping (in seconds) for each fault. (See below how to set this parameter)
- Line** NF+2: Type of simulation (1=dynamic , 2 = kinematic)
- Line** NF+3: Number of time steps between updates of the time series outputs at selected fault points (see DATA/FAULT\_STATIONS), usually a large number (100s or 1000s). Note that the sampling rate of the time series is usually much higher.
- Line** NF+4: Number of time steps between fault snapshot outputs (quantities at every fault point exported at regular times), usually a large number (100s or 1000s).
- Line** NF+5: Slip velocity threshold below which frictional healing is set (friction coefficient is reset to its static value). If this value is negative healing is disabled.
- Line** NF+6: Slip velocity threshold to define the rupture front. Only used for outputs.

The rest of this file is made of namelist input blocks (see "namelist" in a Fortran 9x manual). The input for each fault has the following sequence (arguments in [brackets] are optional):

```
&RUPTURE_SWITCHES / RATE_AND_STATE ,TPV16 ,HETE_RSF ,TPV10X
&BEGIN_FAULT /
&STRESS_TENSOR Sigma =  $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{xz}, \sigma_{yz}$  /
&INIT_STRESS S1, S2, S3 [,n1, n2, n3] /
followed by (n1+n2+n3) &DIST2D blocks
&SWF mus, mud, dc [, nmus, nmud, ndc] /
&RSF V0,f0,a,b,L,V_init,theta_init,C,StateLaw [ nV0,nf0,na,nb,nL,nV_init,ntheta_init,nC ] /
followed by (nV0+nf0+na+nb+nL+nV_init+ntheta_init+nC) &DIST2D blocks
```

**&RUPTURE\_SWITCHES** input block sets some switches in the simulation process.

**RATE\_AND\_STATE** = .TRUE. use rate and state friction,=.FALSE. use slip weakening friction

**TPV16** =.TRUE turn on heterogeneity fault property input for slip weakening friction from DATAinput\_file.txt, =.FALSE. turn off such feature

**HETE\_RSF** =.TRUE. turn on heterogeneity fault property input for rate and state friction using input from DATArsf\_hete\_input\_fir.txt, =.FALSE. turn off such feature

**TPV10X** =.TRUE. turn on some ad hoc features for TPV101-104 simulations, a rate strengthening layer surrounding the rate weakening region. =.FALSE. turn off such feature

**&STRESS\_TENSOR** input block sets the initial fault stresses by projecting a uniform regional stress field onto the fault plane. So that the tractions  $\tau = \sigma : n$  while  $n$  is the local norm of the fault plane. Sigma =  $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{xz}, \sigma_{yz}$  (in Pa)

**&INIT\_STRESS** input block sets the initial fault stresses relative to the foot-wall side of the fault. Initial stresses are composed of a constant background value possibly overwritten in prescribed regions by heterogeneous distributions (see &DIST2D blocks below):

**S1** = constant background value of along-strike shear stress (positive in the usual strike direction)

**S2** = constant background value of along-dip shear (positive is down-dip, normal faulting)

**S3** = constant background value of normal stress (negative in compression)

**n1** = number of heterogeneous items for along-strike shear stress [default is 0]

**n2** = number of heterogeneous items for along-dip shear stress [default is 0]

**n3** = number of heterogeneous items for normal stress [default is 0]

**&SWF** input block sets the slip-weakening friction parameters of the fault:

**mus** = constant background value of static friction coefficient

**mud** = constant background value of dynamic friction coefficient

**dc** = constant background value of critical slip-weakening distance

**nmus** = number of heterogeneous items for static friction coefficient [default is 0]

**nmud** = number of heterogeneous items for dynamic friction coefficient [default is 0]

**ndc** = number of heterogeneous items for critical slip-weakening distance [default is 0]

**&RSF** input block sets the rate and state friction parameters of the fault:

We refer to the well known rate and state friction formula:

$$\mu = f_0 + a * \log\left(\frac{v}{v_0}\right) + b * \log\left(\frac{v_0}{\theta L}\right)$$

**V0** = constant background value of  $V_0$

**f0** = constant background value of  $f_0$

**a** = constant background value of  $a$

**b** = constant background value of  $b$

**L** = constant background value of  $L$

**V\_init** = constant background value of along strike right lateral slip rate at time 0

**theta\_init** = constant background value of state variable  $\theta$  at time 0

**C** = constant background value of cohesion (in Pa)

**StateLaw** = 1 or 2. 1 for aging law , 2 for slip law with strong dynamic weakening.

**n\*\*** = number of heterogeneous items for quantity \*\*

**&DIST2D** input blocks modify (overwrite) the value of a fault parameter by a heterogeneous spatial distribution:

**&DIST2D** shapeval='square', val, xc, yc, zc, 1 /

sets a constant value (val) within a cube with center (xc,yc,zc) and edge size 1.

**&DIST2D** shapeval='rectangle', val, xc, yc, zc, lx, ly, lz /

sets a constant value (val) within a rectangular prism with center (xc,yc,zc) and edge sizes (lx,ly,lz).

**&DIST2D** shapeval='rectangle - taper', val, valh, xc, yc, zc, lx, ly, lz /

sets a vertical linear gradient within a rectangular prism with center (xc,yc,zc) and edge sizes (lx,ly,lz). Values vary linearly as a function of vertical position z between value val at z = zc-lz/2 and value valh at z = zc+lz/2 .

**&DIST2D** shapeval='circular', val, xc, yc, zc, r /

sets a constant value (val) within a sphere with center (xc,yc,zc) and radius r.

**DATA/FAULT\_STATIONS** Stations in the fault plane.

**Line 1** : number of stations.

**Line 2 to end**: 5 columns: X, Y, Z (-depth), station name, fault-id

The fault-id identifies the fault that contains the station. It is the index of appearance in the faults list after line 2 of Par\_file\_faults

**DATA/input\_file.txt** Heterogeneous stresses and friction for linear slip weakening friction parameters are documented in page 10 of

EXAMPLES/tpv16/description/TPV16\_17\_Description\_v03.pdf

To activate this feature, in Par\_file\_faults name list &**RUPTURE\_SWITCHES**, set TPV16=.TRUE..

**DATA/rsf\_hete\_input\_file.txt** Heterogeneous stresses and friction input for rate and state friction. To activate this feature, in Par\_file\_faults name list &**RUPTURE\_SWITCHES**, set RSF\_HETE=.TRUE..

The format of **DATA/rsf\_hete\_input\_file.txt** is as such, in the first line there are four integers that are sequentially documenting the number of divisions along strike, number of divisions along dip, cell size along strike , cell size along dip. Then the following N(N=NumberOfDivisionsAlongStrike \* NumberOfDivisionsAlongDip) lines will document the stress and friction properties on the grid.

There are a total of 13 columns.

1. Column1 = Along strike distance(m)
2. Column2 = Along dip distances(m) The distance should be negative for points at depth
3. Column3 = Normal stress(Pa)
4. Column4 = Horizontal right-lateral shear stress(Pa)
5. Column5 = Vertical up-dip shear stress(Pa)
6. Column6 =  $V_0$  value in RSF(m/s)
7. Column7 =  $f_0$  value in RSF
8. Column8 = a value in RSF
9. Column9 = b value in RSF
10. Column10 = L value in RSF
11. Column11 = Initial slip velocity(m/s)
12. Column12 = State variable  $\theta$
13. Column13 = Cohesion(Pa)

Note that the input grid file do not have to coincide with the mesh. For each fault node on the mesh, they will use the value from the nearest node on the input grid.

## 6.6 Setting the Kelvin-Voigt Damping Parameter

The purpose of the Kelvin-Voigt viscosity in the dynamic fault solver is to damp spurious oscillations generated by the fault slip at frequencies that are too high to be resolved by the mesh. The viscosity **eta** (in seconds) depends on the size of the elements on the fault. Here is how to set it:

1. Determine the average linear size of the elements on the fault plane, **h\_fault**. Usually this value is prescribed by the user during mesh generation. Otherwise it can be found by inspection of the mesh inside the CUBIT GUI.
2. Use the Matlab function `utils/critical_timestep.m` to compute  
`dtc_fault = critical_timestep(c_p, h_fault, ngl1).`  
 This is the critical time step in an elastic medium for a hypothetical element of cubic shape with size equal to **h\_fault**.
3. Set **eta** in **Par\_file\_faults** to  $(0.1 \text{ to } 0.3) \times dtc\_fault$ . A larger **eta** damps high-frequencies more aggressively but it might also affect lower frequencies and rupture speed.

Viscosity reduces numerical stability: the critical timestep in a simulation with Kelvin-Voigt damping needs to be smaller than that in a purely elastic simulation. Here is how to set the time step accordingly:

1. Run a test simulation without viscosity (`eta=0` and only a few time steps)
2. Look for the "maximum suggested time step" in `OUTPUT_FILES/output_mesher.txt`. This is the critical timestep of a purely elastic simulation, `dtc_bulk`.
3. Reset the timestep of the simulation with a Kelvin-Voigt material to a value smaller than  

$$dtc_kv = eta \left( \sqrt{1 + dtc_bulk^2 / eta^2} - 1 \right)$$

Note that in general `dtc_bulk` is smaller than `dtc_fault`, because elements off the fault might be smaller or more distorted than element faces on the fault.

## 6.7 Output Files

Several output files are saved in `OUTPUT_FILES/`:

1. Seismograms for each station on the fault plane given in `DATA/FAUL_STATIONS`. One output file is generated for each station, named after the station. The files are ascii and start with a header (22 lines long) followed by a data block with the following format, one line per time sample:

Column 1 = Time (s)  
Column 2 = horizontal right-lateral slip (m)  
Column 3 = horizontal right-lateral slip rate (m/s)  
Column 4 = horizontal right-lateral shear stress (MPa)  
Column 5 = vertical up-dip slip (m)  
Column 6 = vertical up-dip slip rate (m/s)  
Column 7 = vertical up-dip shear stress (MPa)  
Column 8 = normal stress (MPa)

The stresses are relative to the footwall side of the fault (this convention controls their sign, but not their amplitude). Slip is defined as displacement of the hanging wall relative to the footwall.

2. Seismograms at stations in the bulk (out of the fault plane) given in `DATA/STATIONS`.
3. Rupture time files are named `Rupture_time_FAULT-id`. One file is generated for each fault. The files are ascii and start with a header (12 lines long) followed by a data block with the following format, one line per fault node:  
Column 1 = horizontal coordinate, distance along strike (m)  
Column 2 = vertical coordinate, distance down-dip (m)  
Column 3 = rupture time (s)
4. Fault quantities (slip, slip rate, stresses, etc) at regular times are stored in binary data files called `Snapshot#it#.bin`, where `#it#` is the timestep number. These can be read in Matlab with the function `utils/FSEM3D_snapshot.m`

## 6.8 Post-processing and Visualization

Some Matlab functions for post-processing and visualization are included in directory `utils`. The functions are internally documented (see their matlab help).

`FSEM3D_snapshot` reads a fault data snapshot

The directories `EXAMPLES/*/post` contain additional Matlab scripts to generate figures specific to each example.

# Chapter 7

## Adjoint Simulations

Adjoint simulations are generally performed for two distinct applications. First, they can be used in point source moment-tensor inversions, or source imaging for earthquakes with large ruptures such as the Lander's earthquake [Wald and Heaton, 1994]. Second, they can be used to generate finite-frequency sensitivity kernels that are a critical part of tomographic inversions based upon 3D reference models [Tromp et al., 2005, Liu and Tromp, 2006, Tromp et al., 2008, Liu and Tromp, 2008]. In either case, source parameter or velocity structure updates are sought to minimize a specific misfit function (e.g., waveform or traveltime differences), and the adjoint simulation provides a means of computing the gradient of the misfit function and further reducing it in successive iterations. Applications and procedures pertaining to source studies and finite-frequency kernels are discussed in Sections 7.1 and 7.2, respectively. The two related parameters in the `Par_file` are `SIMULATION_TYPE` (1, 2 or 3) and the `SAVE_FORWARD` (boolean).

### 7.1 Adjoint Simulations for Sources Only (not for the Model)

When a specific misfit function between data and synthetics is minimized to invert for earthquake source parameters, the gradient of the misfit function with respect to these source parameters can be computed by placing time-reversed seismograms at the receivers as virtual sources in an adjoint simulation. Then the value of the gradient is obtained from the adjoint seismograms recorded at the original earthquake location.

#### 1. Prepare the adjoint sources

- (a) First, run a regular forward simulation (`SIMULATION_TYPE = 1` and `SAVE_FORWARD = .false.`). You can automatically set these two variables using the `utils/change_simulation_type.pl` script:

```
utils/change_simulation_type.pl -f
```

and then collect the recorded seismograms at all the stations given in `DATA/STATIONS`.

- (b) Then select the stations for which you want to compute the time-reversed adjoint sources and run the adjoint simulation, and compile them into the `DATA/STATIONS_ADJOINT` file, which has the same format as the regular `DATA/STATIONS` file.

- Depending on what type of misfit function is used for the source inversion, adjoint sources need to be computed from the original recorded seismograms for the selected stations and saved in a sub-directory called `SEM/` in the root directory of the code, with the format `NT.STA.BX?.adj`, where `NT`, `STA` are the network code and station name given in the `DATA/STATIONS_ADJOINT` file, and `BX?` represents the component name of a particular adjoint seismogram. Please note that the band code can change depending on your sampling rate (see Appendix B for further details).
- The adjoint seismograms are in the same format as the original seismogram (`NT.STA.BX?.sem?`), with the same start time, time interval and record length.

- (c) Notice that even if you choose to time reverse only one component from one specific station, you still need to supply all three components because the code is expecting them (you can set the other two components to be zero).

- (d) Also note that since time-reversal is done in the code itself, no explicit time-reversing is needed for the preparation of the adjoint sources, i.e., the adjoint sources are in the same forward time sense as the original recorded seismograms.

## 2. Set the related parameters and run the adjoint simulation

In the DATA/Par\_file, set the two related parameters to be `SIMULATION_TYPE = 2` and `SAVE_FORWARD = .false..` More conveniently, use the scripts `utils/change_simulation_type.pl` to modify the Par\_file automatically (`change_simulation_type.pl -a`). Then run the solver to launch the adjoint simulation.

## 3. Collect the seismograms at the original source location

After the adjoint simulation has completed successfully, collect the seismograms from `LOCAL_PATH`.

- These adjoint seismograms are recorded at the locations of the original earthquake sources given by the DATA/CMTSOLUTION file, and have names of the form `NT.S?????.S???.sem` for the six-component strain tensor (SNN, SEE, SZZ, SNE, SNZ, SEZ) at these locations, and `NT.S?????.BX?.sem` for the three-component displacements (BXN, BXE, BXZ) recorded at these locations.
- `S?????` denotes the source number; for example, if the original CMTSOLUTION provides only a point source, then the seismograms collected will start with `S00001`.
- These adjoint seismograms provide critical information for the computation of the gradient of the misfit function.

## 7.2 Adjoint Simulations for Finite-Frequency Kernels (Kernel Simulation)

Finite-frequency sensitivity kernels are computed in two successive simulations (please refer to Liu and Tromp [2006] and Tromp et al. [2008] for details).

### 1. Run a forward simulation with the state variables saved at the end of the simulation

Prepare the CMTSOLUTION and STATIONS files, set the parameters `SIMULATION_TYPE = 1` and `SAVE_FORWARD = .true..` in the Par\_file (`change_simulation_type -F`), and run the solver.

- Notice that attenuation is not implemented yet for the computation of finite-frequency kernels; therefore set `ATTENUATION = .false..` in the Par\_file.
- We also suggest you modify the half duration of the CMTSOLUTION to be similar to the accuracy of the simulation (see Equation 3.1) to avoid too much high-frequency noise in the forward wavefield, although theoretically the high-frequency noise should be eliminated when convolved with an adjoint wavefield with the proper frequency content.
- This forward simulation differs from the regular simulations (`SIMULATION_TYPE = 1` and `SAVE_FORWARD = .false..`) described in the previous chapters in that the state variables for the last time step of the simulation, including wavefields of the displacement, velocity, acceleration, etc., are saved to the `LOCAL_PATH` to be used for the subsequent simulation.
- For regional simulations, the files recording the absorbing boundary contribution are also written to the `LOCAL_PATH` when `SAVE_FORWARD = .true..`

### 2. Prepare the adjoint sources

The adjoint sources need to be prepared the same way as described in the Section 1.

- In the case of travel-time finite-frequency kernel for one source-receiver pair, i.e., point source from the CMTSOLUTION, and one station in the STATIONS\_ADJOINT list, we supply a sample program in `utils/adjoint_sources/traveltime/xcreate_adjsrc_travelttime` to cut a certain portion of the original displacement seismograms and convert it into the proper adjoint source to compute the finite-frequency kernel.

```
xcreate_adjsrc_travelttime t1 t2 ifile[0-5] E/N/Z-ascii-files [baz]
```

where  $t_1$  and  $t_2$  are the start and end time of the portion you are interested in, `ifile` denotes the component of the seismograms to be used (0 for all three components, 1 for East, 2 for North, and 3 for vertical, 4 for transverse, and 5 for radial component), `E/N/Z-ascii-files` indicate the three-component displacement seismograms in the right order, and `baz` is the back-azimuth of the station. Note that `baz` is only supplied when `ifile = 4 or 5`.

- Similarly, a sample program to compute adjoint sources for amplitude finite-frequency kernels may be found in `utils/adjoint_sources/amplitude` and used in the same way as described for travel-time measurements

```
xcreate_adjsrc_amplitude t1 t2 ifile[0-5] E/N/Z-ascii-files [baz]
```

### 3. Run the kernel simulation

With the successful forward simulation and the adjoint source ready in the `SEM/` directory, set `SIMULATION_TYPE = 3` and `SAVE_FORWARD = .false.` in the `Par_file` (you can use `change_simulation_type.pl -b`), and rerun the solver.

- The adjoint simulation is launched together with the back reconstruction of the original forward wavefield from the state variables saved from the previous forward simulation, and the finite-frequency kernels are computed by the interaction of the reconstructed forward wavefield and the adjoint wavefield.
- The back-reconstructed seismograms at the original station locations are saved to the `LOCAL_PATH` at the end of the kernel simulations, and can be collected to the local disk.
- These back-constructed seismograms can be compared with the time-reversed original seismograms to assess the accuracy of the backward reconstruction, and they should match very well.
- The arrays for density, P-wave speed and S-wave speed kernels are also saved in the `LOCAL_PATH` with the names `proc??????_rho(alpha,beta)_kernel.bin`, where `proc???????` represents the processor number, `rho(alpha,beta)` are the different types of kernels.

### 4. Run the anisotropic kernel simulation

Instead of the kernels for the isotropic wave speeds, you can also compute the kernels for the 21 independent components  $C_{IJ}$ ,  $I, J = 1, \dots, 6$  (using Voigt's notation) of the elastic tensor in the cartesian coordinate system. This is done by setting `ANISOTROPIC_KL = .true.` in `constants.h` before compiling the package. The definition of the parameters  $C_{IJ}$  in terms of the corresponding components  $c_{ijkl}, i, j, k, l = 1, 2, 3$  of the elastic tensor in cartesian coordinates follows Chen and Tromp [2007]. The 21 anisotropic kernels are saved in the `LOCAL_PATH` in one file with the name of `proc??????_cijkl_kernel.bin` (with `proc???????` the processor number). The output kernels correspond to absolute perturbations  $\delta C_{IJ}$  of the elastic parameters and their unit is in  $s/GPa/km^3$ . For consistency, the output density kernels with this option turned on are for a perturbation  $\delta\rho$  (and not  $\frac{\delta\rho}{\rho}$ ) and their unit is in  $s / (kg/m^3) / km^3$ . These 'primary' anisotropic kernels can then be combined to obtain the kernels for different parameterizations of anisotropy. This can be done, for example, when combining the kernel files from slices into one mesh file (see Section 11.3).

If `ANISOTROPIC_KL = .true.` by additionally setting `ANISOTROPIC_KL = .true.` in `constants.h` the package will save anisotropic kernels parameterized as velocities related to transverse isotropy based on the the Chen and Tromp parameters Chen and Tromp [2007]. The kernels are saved as relative perturbations for horizontal and vertical P and S velocities,  $\alpha_v, \alpha_h, \beta_v, \beta_h$ . Explicit relations can be found in appendix B. of Sieminski et al. [2007]

The anisotropic kernels are only currently available for CPU mode.

In general, the three steps need to be run sequentially to assure proper access to the necessary files. If the simulations are run through some cluster scheduling system (e.g., LSF), and the forward simulation and the subsequent kernel simulations cannot be assigned to the same set of computer nodes, the kernel simulation will not be able to access the database files saved by the forward simulation. Solutions for this dilemma are provided in Chapter 12. Visualization of the finite-frequency kernels is discussed in Section 11.3.

# Chapter 8

## Doing tomography, i.e., updating the model based on the sensitivity kernels obtained

UNDER CONSTRUCTION (July 2015). New content will be added soon.

### 8.1 Tomographic full waveform inversion (FWI) / imaging using the sensitivity kernels obtained

One of the fundamental reasons for computing sensitivity kernels (Section 7.2) is to use them within a tomographic inversion, i.e., perform imaging. In other words, use recorded seismograms, make measurements with synthetic seismograms, and use the misfit between the two to iteratively improve the model described by (at least)  $V_p$ ,  $V_s$ , and  $\rho$  as a function of space.

As explained for instance in Monteiller et al. [2015], full waveform inversion means that one considers the observed seismograms (possibly filtered) as the basic observables that one wants to fit. One thus searches for the model that minimizes the mean squared difference between observed and synthetic seismograms. In other words, the goal is to find a structural model that can explain a larger portion of seismological records, and not simply the phase of a few seismic arrivals.

Whatever misfit function you use for the tomographic inversion (you can find several examples in Tromp et al. [2008] and Tromp et al. [2005]), you will weigh the sensitivity kernels with measurements. Furthermore, you will use as many measurements (stations, components, time windows) as possible per event; hence, we call these composite kernels “event kernels”, which are volumetric fields representing the gradient of the misfit function with respect to one of the variables (e.g.,  $V_s$ ). The basic features of an adjoint-based tomographic inversion were illustrated in Tromp et al. [2008] and Tape et al. [2007] using a conjugate-gradient algorithm. Other, more powerful techniques such as the Limited-Memory Broyden-Fletcher-Goldfarb-Shanno (L-BFGS) method can now be used, as illustrated for instance in Monteiller et al. [2015].

#### 8.1.1 Principle

We want to minimize the classical waveform misfit function:

$$\chi(\mathbf{m}) = \sum_{s=1}^N \sum_{r=1}^M \int_0^T \frac{1}{2} \|\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) - \mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; t)\|^2 dt. \quad (8.1)$$

This functional quantifies the  $L^2$  difference between the observed waveforms  $\mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; t)$  at receivers  $\mathbf{x}_r$ ,  $r = 1, \dots, M$  produced by sources at  $\mathbf{x}_s$ ,  $s = 1, \dots, N$ , and the corresponding synthetic seismograms  $\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t)$  computed in model  $\mathbf{m}$ . While this misfit function is indeed classical, it is worth mentioning that in the case of noisy real data other norms could be used, since in the oil industry for instance it is known that the  $L^1$  norm (Crase et al. [1990], Brossier et al. [2010]), hybrid  $L^1 - L^2$  norms (Bube and Langan [1997]), Hubert norm (Ha et al. [2009]), Student-t distribution

(Aravkin et al. [2011], Jeong et al. [2015]) etc... can be more robust than the  $L^2$  norm used here in the context of synthetic data with no noise. In the vicinity of  $\mathbf{m}$ , the misfit function can be expanded into a Taylor series:

$$\chi(\mathbf{m} + \delta\mathbf{m}) \approx \chi(\mathbf{m}) + \mathbf{g}(\mathbf{m}) \cdot \delta\mathbf{m} + \delta\mathbf{m} \cdot \mathbf{H}(\mathbf{m}) \cdot \delta\mathbf{m}, \quad (8.2)$$

where  $\mathbf{g}(\mathbf{m})$  is the gradient of the waveform misfit function:

$$\mathbf{g}(\mathbf{m}) = \frac{\partial \chi(\mathbf{m})}{\partial \mathbf{m}}, \quad (8.3)$$

and  $\mathbf{H}(\mathbf{m})$  the Hessian:

$$\mathbf{H}(\mathbf{m}) = \frac{\partial^2 \chi(\mathbf{m})}{\partial \mathbf{m}^2}. \quad (8.4)$$

In the following, for simplicity the dependence of the gradient and Hessian on the model will be implicitly assumed and omitted in the notations. The nearest minimum of  $\chi$  in (8.2) with respect to the model perturbation  $\delta\mathbf{m}$  is reached for

$$\delta\mathbf{m} = -\mathbf{H}^{-1} \cdot \mathbf{g}. \quad (8.5)$$

The local minimum of (8.1) is thus given by perturbing the model in the direction of the gradient preconditioned by the inverse Hessian.

### 8.1.2 Computation of the gradient based on the adjoint method

A direct method to compute the gradient is to take the derivative of (8.1) with respect to model parameters:

$$\frac{\partial \chi(\mathbf{m})}{\partial \mathbf{m}} = - \sum_{s=1}^N \sum_{r=1}^M \int_0^T \frac{\partial \mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t)}{\partial \mathbf{m}} \cdot [\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) - \mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; t)] dt. \quad (8.6)$$

This equation can be reformulated as the matrix-vector product:

$$\mathbf{g} = -\mathbf{J}^* \cdot \delta\mathbf{d}, \quad (8.7)$$

where  $\mathbf{J}^*$  is the adjoint of the Jacobian matrix of the forward problem that contains the Fréchet derivatives of the data with respect to model parameters, and  $\delta\mathbf{d}$  is the vector that contains the data residuals. The determination of  $\mathbf{J}$  would require computing the Fréchet derivatives for each time step in the time window considered and for all the source-station pairs, which is completely prohibitive on current supercomputers (let us note that this situation may change one day). However, it is possible to obtain this gradient without computing the Jacobian matrix explicitly. The approach to determine the gradient without computing the Fréchet derivatives was introduced in nonlinear optimization by Chavent [1974] working with J. L. Lions, and later applied to seismic exploration problems by Bamberger et al. [1977], Bamberger et al. [1982], Lailly [1983] and Tarantola [1984]. The idea is to resort to the adjoint state, which corresponds to the wavefield emitted and back-propagated from the receivers [e.g., Tromp et al., 2005, 2008, Plessix, 2006, Fichtner et al., 2006].

Let us give an outline of the theory to compute the gradient based on the adjoint method, and refer the reader to e.g. Tromp et al. [2005] and Tromp et al. [2008] for further details. The perturbation of the misfit function can be expressed as:

$$\delta\chi(\mathbf{m}) = \sum_{s=1}^N \sum_{r=1}^M \int_0^T [\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) - \mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; t)] \cdot \delta\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) dt, \quad (8.8)$$

where  $\delta\mathbf{u}$  is the perturbation of displacement given by the first-order Born approximation [e.g., Hudson, 1977]:

$$\begin{aligned} \delta\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) &= - \int_0^t \int_V [\delta\rho(\mathbf{x}) \mathbf{G}(\mathbf{x}_r, \mathbf{x}; t-t') \cdot \partial_{t'}^2 \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t')] \\ &\quad + \nabla \mathbf{G}(\mathbf{x}_r, \mathbf{x}; t-t') : \delta\mathbf{c}(\mathbf{x}) : \nabla \mathbf{u}(\mathbf{x}; t') d^3\mathbf{x} dt'. \end{aligned} \quad (8.9)$$

In this expression,  $\mathbf{G}$  is the Green's tensor,  $\delta\rho$  the perturbation of density,  $\delta c$  the perturbation of the fourth-order elasticity tensor, and a colon denotes a double tensor contraction operation. Inserting (8.9) into (8.8) we obtain

$$\begin{aligned}\delta\chi(\mathbf{m}) &= -\sum_{s=1}^N \sum_{r=1}^M \int_0^T [\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; t) - \mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; t)] \int_0^t \int_V [\delta\rho(\mathbf{x}) \mathbf{G}(\mathbf{x}_r, \mathbf{x}; t-t') \cdot \partial_t^2 \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t')] \\ &\quad + \nabla \mathbf{G}(\mathbf{x}_r, \mathbf{x}; t-t') : \delta \mathbf{c}(\mathbf{x}) : \nabla \mathbf{u}(\mathbf{x}, t')] d^3 \mathbf{x} dt' dt.\end{aligned}\quad (8.10)$$

Defining the waveform adjoint source for each source  $\mathbf{x}_s$

$$\mathbf{f}^\dagger(\mathbf{x}, \mathbf{x}_s; t) = \sum_{r=1}^M [\mathbf{u}(\mathbf{x}_r, \mathbf{x}_s; T-t) - \mathbf{d}(\mathbf{x}_r, \mathbf{x}_s; T-t)] \delta(\mathbf{x} - \mathbf{x}_r), \quad (8.11)$$

and the corresponding adjoint wavefield

$$\mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; t) = \int_0^{t'} \int_V \mathbf{G}(\mathbf{x}, \mathbf{x}'; t'-t) \cdot \mathbf{f}^\dagger(\mathbf{x}', \mathbf{x}_s; t) d^3 \mathbf{x}' dt, \quad (8.12)$$

the perturbation of the misfit function may be expressed as:

$$\begin{aligned}\delta\chi(\mathbf{m}) &= -\sum_{s=1}^N \int_V \int_0^T [\delta\rho \mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; T-t) \cdot \partial_t^2 \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t) \\ &\quad + \nabla \mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; T-t) : \delta \mathbf{c} : \nabla \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t)] d^3 \mathbf{x} dt.\end{aligned}\quad (8.13)$$

At this point, we make some assumptions on the nature of the elasticity tensor. A general fourth-order elasticity tensor is described by 21 elastic parameters, a very large number that makes its complete characterization way beyond the reach of any tomographic approach. For the time being, let us thus consider isotropic elasticity tensors, described by the two Lamé parameters  $\lambda$  and  $\mu$ :

$$c_{ijkl} = \lambda \delta_{ij} \delta_{kl} + \mu (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk}). \quad (8.14)$$

In this case, (8.13) can be written as:

$$\begin{aligned}\delta\chi(\mathbf{m}) &= -\sum_{s=1}^N \int_V [K_\rho(\mathbf{x}, \mathbf{x}_s) \delta \ln \rho(\mathbf{x}) + K_\lambda(\mathbf{x}, \mathbf{x}_s) \delta \ln \lambda(\mathbf{x}) \\ &\quad + K_\mu(\mathbf{x}, \mathbf{x}_s) \delta \ln \mu(\mathbf{x})] d^3 \mathbf{x},\end{aligned}\quad (8.15)$$

where  $\ln()$  is the natural logarithm and where the Fréchet derivatives with respect to the density and Lamé parameters are given by:

$$K_\rho(\mathbf{x}, \mathbf{x}_s) = -\int_0^T \rho(\mathbf{x}) \mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; T-t) \cdot \partial_t^2 \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t) dt \quad (8.16)$$

$$K_\lambda(\mathbf{x}, \mathbf{x}_s) = -\int_0^T \lambda(\mathbf{x}) \nabla \cdot \mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; T-t) \nabla \cdot \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t) dt \quad (8.17)$$

$$K_\mu(\mathbf{x}, \mathbf{x}_s) = -2 \int_0^T \mu(\mathbf{x}) \nabla \mathbf{u}^\dagger(\mathbf{x}, \mathbf{x}_s; T-t) : \nabla \mathbf{u}(\mathbf{x}, \mathbf{x}_s; t) dt. \quad (8.18)$$

Since the propagation of seismic waves mainly depends on compressional wave speed  $\alpha$  and shear wave speed  $\beta$ , but also because these seismic velocities are easier to interpret, tomographic models are usually described based on these two parameters. With this new parametrization, the perturbation of the misfit function may be written as:

$$\begin{aligned}\delta\chi(\mathbf{m}) &= -\sum_{s=1}^N \int_V [K'_\rho(\mathbf{x}, \mathbf{x}_s) \delta \ln \rho(\mathbf{x}) + K'_\alpha(\mathbf{x}, \mathbf{x}_s) \delta \ln \alpha(\mathbf{x}) \\ &\quad + K'_\beta(\mathbf{x}, \mathbf{x}_s) \delta \ln \beta(\mathbf{x})] d^3 \mathbf{x},\end{aligned}\quad (8.19)$$

where

$$K'_\rho(\mathbf{x}, \mathbf{x}_s) = K_\rho(\mathbf{x}, \mathbf{x}_s) + K_\lambda(\mathbf{x}, \mathbf{x}_s) + K_\mu(\mathbf{x}, \mathbf{x}_s) \quad (8.20)$$

$$K'_\alpha(\mathbf{x}, \mathbf{x}_s) = 2 \left( \frac{\lambda + 2\mu}{\lambda} \right) K_\lambda(\mathbf{x}, \mathbf{x}_s) \quad (8.21)$$

$$K'_\beta(\mathbf{x}, \mathbf{x}_s) = 2K_\mu - \frac{4\mu}{\lambda} K_\lambda(\mathbf{x}, \mathbf{x}_s). \quad (8.22)$$

As can be seen from these expressions, the principle of the adjoint-state method is to correlate two wavefields: the direct (i.e. forward) field that propagates from the source to the receivers, and the adjoint field that propagates from all the receivers backward in time. The same approach can be followed for any type of seismic observable (phase, amplitude, envelope, time series...), provided the appropriate adjoint source is used [Tromp et al., 2005, 2008]. For example, for the cross-correlated traveltimes of a seismic phase, the adjoint source is defined as the velocity of that synthetic phase weighted by the travel-time residual.

Computing the gradient based on the adjoint-state method requires performing two simulations per source (forward and adjoint fields) regardless of the type of observable. However, to define the adjoint field one must know the adjoint source, and that source is computed from the results of the forward simulation. One must therefore perform the forward simulation before the adjoint simulation. A straightforward solution for time-domain methods would be to store the whole forward field to disk at each time step during the forward run and then read it back during the adjoint simulation to calculate the interaction of these two fields. In 2-D this is feasible but in the 3-D case for very short seismic periods and without lossy compression, downsampling, or a large amount of disk or memory checkpointing [e.g., Fichtner et al., 2009b, Rubio Dalmau et al., 2014, Cyr et al., 2015] the amount of disk storage required would currently be too large.

However let us note again that this situation will change in the future. In the mean time, a standard possible solution is to perform three simulations per source [Tromp et al., 2008, Peter et al., 2011], i.e., perform the forward calculation twice, once to compute the adjoint sources and once again at the same time as the adjoint simulation to correlate the two fields and sum their interaction on the fly over all the time steps. Doing so for an elastic Earth, one only needs a small amount of disk storage to store the last time step of the forward run, which is then used as an initial condition to redo the forward run backwards, as well as the field on the outer edges of the mesh for each time steps in order to be able to undo the absorbing boundary conditions.

## 8.2 Tomographic tools

Besides the ability to compute adjoint sensitivity kernels, the SPECFEM3D package provides a number of tools useful for post-processing kernels, gradients and conducting tomographic model updates. The tomographic and post-processing tools can be compiled by typing in the main directory:

```
make tomography
make postprocess
```

You can modify specific settings affecting, e.g., the (isotropic) parameterization or directory setup structure in file `setup/constants_tomography.h`. The compiled binaries can be found in directory `bin/`.

The iterative inversion workflow may comprise three main steps for which tools are provided:

1. Summing event kernels, e.g.  $K_{\alpha,\beta,\rho}$ , to build the misfit gradient  $\mathbf{g}(\mathbf{m})$
2. Smoothing and post-processing of the gradient  $\mathbf{g}(\mathbf{m})$
3. Updating the model  $\mathbf{m}_{i+1} = \mathbf{m}_i + \delta\mathbf{m}$ .

Please note that the tomographic routines are very similar for both SPECFEM3D\_Cartesian and SPECFEM3D\_GLOBE versions. Some differences occur, e.g., when reading in mesh files and when the maximum of the gradient is taken for the update step length. Also note that currently only the binary database format is supported, no ADIOS file format support is implemented yet. In future, we plan to extend and merge these into the same set of tools within a common SPECFEM function library.

### 8.2.1 Summing kernels

For summing different event kernels, the executables `xsum_kernels` and `xsum_preconditioned_kernels` can be used. The binaries sum up either a set of isotropic or transversely isotropic kernel files, depending on the setting in file `setup/constants_tomography.h`. The following parameterizations can be chosen:

- isotropic kernels  $K_{\alpha,\beta,\rho}$  or
- isotropic bulk kernels  $K_{c,\beta,\rho}$ , or
- transversely isotropic bulk kernels  $K_{c,\beta_v,\beta_h,\eta}$ .

The tools for summing event kernels can be used in parallel, using the same number of processes (`NPROC`) as the forward/kernel simulations. The summation tools use the following input/output format (see Fig. 8.1 for the default directory structure):

- **Input:** Input is provided by a list of event directories, given in file `kernels_list.txt` (default setting for `KERNEL_FILE_LIST` in `constants_tomography.h`). The file lists all event kernel directories which should be summed together. All event directories have to be located in directory `INPUT_KERNELS/`, which can be setup as links to all individual event kernel directories.
- **Output:** The summed kernels will be stored in corresponding kernel files in output directory `OUTPUT_SUM/`.

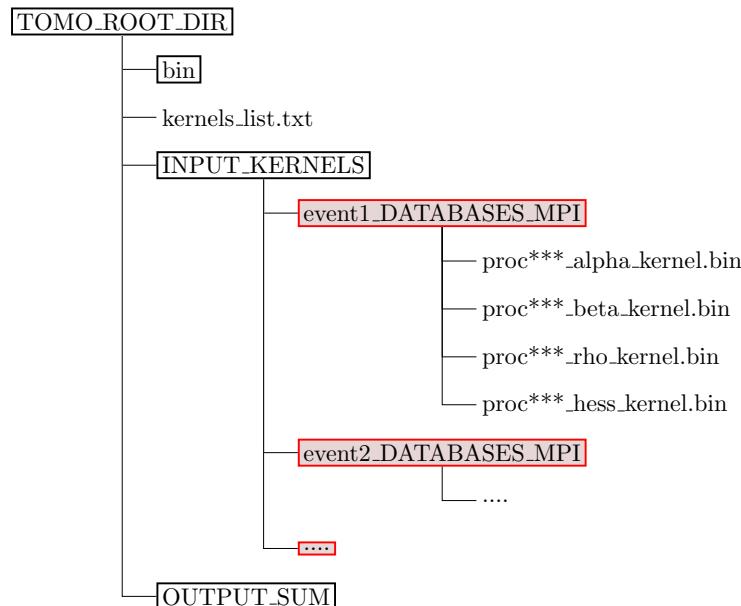


Figure 8.1: Example directory structure when using tomographic tools for summation. Event directory names (red) can be chosen freely with corresponding name entries in file `kernels_list.txt`

**Kernel summation:** As example with a total of 4 MPI processes

```
mpirun -np 4 ./bin/xsum_kernels
```

adds sensitivity kernels  $K$  from different events together, i.e. outputs  $\mathbf{g}$  where

$$\mathbf{g} = \sum_i^N K^i$$

for events  $i = 1, \dots, N$  as specified in file `kernel_list.txt`.

**Kernel Summation with Preconditioning:** As example with a total of 4 MPI processes

```
mpirun -np 4 ./bin/xsum_preconditioned_kernels
```

adds sensitivity kernels  $K$  together from different events and "preconditions" the sum by dividing with the sum of the corresponding approximate Hessians  $\tilde{H}$ , i.e. outputs  $\mathbf{g}$  where

$$\mathbf{g} = \frac{1}{\sum_i \tilde{H}^i} \sum_i K^i$$

In this case, the factor  $\frac{1}{\sum_i \tilde{H}^i}$  acts as preconditioner, approximating the inverse of the Hessian  $\mathbf{H}^{-1}$ .

**Kernel names:** The summation tool will look for kernels with following names:

- for an *isotropic* model parameterization, kernel names are:

<pre>## Cartesian version proc***_alpha_kernel.bin proc***_beta_kernel.bin proc***_rho_kernel.bin</pre>	<pre>## GLOBE version proc***_reg1_alpha_kernel.bin proc***_reg1_beta_kernel.bin proc***_reg1_rho_kernel.bin</pre>
---	--

- for an *isotropic bulk* model parameterization, kernel names are:

<pre>## Cartesian version proc***_bulk_kernel.bin proc***_bulk_beta_kernel.bin proc***_rho_kernel.bin</pre>	<pre>## GLOBE version proc***_reg1_bulk_kernel.bin proc***_reg1_bulk_beta_kernel.bin proc***_reg1_rho_kernel.bin</pre>
---	--

- for a *transversely isotropic* model parameterization, kernel names are:

<pre>## Cartesian version proc***_bulk_c_kernel.bin proc***_bulk_betav_kernel.bin proc***_bulk_betah_kernel.bin proc***_eta_kernel.bin</pre>	<pre>## GLOBE version proc***_reg1_bulk_c_kernel.bin proc***_reg1_bulk_betav_kernel.bin proc***_reg1_bulk_betah_kernel.bin proc***_reg1_eta_kernel.bin</pre>
--	--

Note that these event kernels are stored after a kernel simulation (SIMULATION\_TYPE = 3) in the LOCAL\_PATH directory, which by default points to directory DATABASES\_MPI/. Isotropic kernels will be created by default. To create transversely isotropic kernels, you set ANISOTROPIC\_KL and SAVE\_TRANSVERSE\_KL to .true. in the parameter file DATA/Par\_file.

- for preconditioning, the approximate Hessian kernels taken for summation are:

<pre>## Cartesian version proc***_hess_kernel.bin</pre>	<pre>## GLOBE version proc***_reg1_hess_kernel.bin</pre>
---	--

These Hessian kernels approximate the diagonal elements of the Hessian matrix and can be used as preconditioner in a gradient optimization scheme. To create these kernels, you have to set APPROXIMATE\_HESS\_KL to .true. in the parameter file DATA/Par\_file.

For SPECFEM3D\_GLOBE, by default only kernels in the crust/mantle region ("reg1") will be considered for summation. You can change the default by setting parameter REG in file constants\_tomography.h to the region of interest.

Note that although we provide the preconditioned summation here, we recommend to smooth first both, the summed kernels and summed approximate Hessians, before inverting the summed Hessian and applying it as preconditioner to the gradient. From experience, we prefer doing

$$\mathbf{g} = \frac{1}{\mathcal{F}_{smooth} \left( \sum_i^N \tilde{H}^i \right)} \mathcal{F}_{smooth} \left( \sum_i K^i \right)$$

rather than

$$\mathbf{g} = \mathcal{F}_{smooth} \left( \frac{1}{\sum_i^N \tilde{H}^i} \sum_i K^i \right).$$

Due to the large sensitivity kernel values close to source and receiver locations, the inverse of the thresholded summed Hessian becomes better balanced in the former.

## 8.2.2 Smoothing and post-processing

For additional smoothing and post-processing of kernels, gradients, or models, we provide tools useful for gradient and model preparation. These additional tools for post-processing sensitivity kernels are provided together with the main package. For compilation of these tools, you type in the main directory:

```
make postprocess
```

The following tools are provided:

- `xsmooth_sem` is used for smoothing with a gaussian function.
- `xclip_sem` is used to threshold a kernel, gradient or model, clipping to a minimum and maximum value.
- `xcombine_sem` is used to combine different event kernels, gradient or model files, summing all individually specified files together (similar to `xsum_kernels` but more flexible).

The post-processing tools are primarily intended to be used to process kernel files. They can be used though on any scalar field of dimension (`NGLLX`, `NGLLY`, `NGLLZ`, `NSPEC`). The tools are parallel programs – they must be invoked with `mpirun` or other appropriate utility. Operations are performed in embarrassingly-parallel fashion.

**Smoothing:** To smooth a kernel, gradient or model file, use

```
mpirun -np NPROC bin/xsmooth_sem SIGMA_H SIGMA_V KERNEL_NAME INPUT_DIR OUPUT_DIR
```

with command line arguments: `SIGMA_H` horizontal smoothing radius, `SIGMA_V` vertical smoothing radius, `KERNEL_NAME` kernel name, e.g. `alpha_kernel`, `INPUT_DIR` directory from which kernels are read, `OUTPUT_DIR` directory to which smoothed kernels are written.

Smooths kernels by convolution with a Gaussian. Writes the resulting smoothed kernels to `OUTPUT_DIR`.

Files written to `OUTPUT_DIR` have the suffix '`smooth`' appended, e.g. `proc***alpha_kernel.bin` becomes `proc***alpha_kernel_smooth.bin`.

**Clipping:** Values in a kernel, gradient or model binary file can be clipped with a minimum/maximum threshold value using

```
mpirun -np NPROC bin/xclip_sem MIN_VAL MAX_VAL KERNEL_NAMES INPUT_FILE OUTPUT_DIR
```

with command line arguments: `MIN_VAL` threshold below which array values are clipped, `MAX_VAL` threshold above which array values are clipped, `KERNEL_NAMES` one or more kernel names separated by commas, `INPUT_DIR` directory from which arrays are read, `OUTPUT_DIR` directory to which clipped array are written.

For each name in `KERNEL_NAMES`, reads kernels from `INPUT_DIR`, applies thresholds, and writes the resulting clipped kernels to `OUTPUT_DIR`.

`KERNEL_NAMES` is a comma-delimited list of kernel names, e.g. `alphav_kernel,alphah_kernel`.

For example, in this case type

```
mpirun -np 4 ./bin/xclip_sem -0.1 0.1 alphav_kernel,alphah_kernel DIR1/ DIR1/
to clip the corresponding kernels in directory DIR1 to be in range [-0.1,0.1].
```

Files written to `OUTPUT_DIR` have the suffix 'clip' appended, e.g. `proc***alpha_kernel.bin` becomes `proc***alpha_kernel_clip.bin`

**Combining:** To combine kernel values from different kernel directories, you can use

```
mpirun -np NPROC bin/xcombine_sem KERNEL_NAMES INPUT_FILE OUTPUT_DIR
```

with command line arguments: `KERNEL_NAMES` one or more kernel names separated by commas, `INPUT_FILE` text file containing list of kernel directories, `OUTPUT_PATH` directory to which summed kernels are written.

For each name in `KERNEL_NAMES`, sums kernels from directories specified in `INPUT_FILE`. Writes the resulting sums to `OUTPUT_DIR`.

`INPUT_FILE` is a text file containing a list of absolute or relative paths to kernel directories, one directory per line.

`KERNEL_NAMES` is a comma-delimited list of kernel names, e.g. `alpha_kernel,beta_kernel,rho_kernel`.

### 8.2.3 Model updating

We provide (simple) tools for updating the current model  $\mathbf{m}_i$  using a gradient  $\mathbf{g}$  and steplength  $\alpha$ ; i.e.

$$\mathbf{m}_{i+1} = \mathbf{m}_i + \delta\mathbf{m}$$

with

$$\delta\mathbf{m} = -\alpha \mathbf{g}.$$

The following tomographic tools are provided:

- `xadd_model_iso` can be used to update isotropic model files with a (summed & smoothed) gradient. The gradient files are given for isotropic parameters or isotropic bulk parameters.

Note that instead of using the density kernels  $K_\rho$  for model updates, setting `USE_RHO_SCALING` to `true`. will ignore the density kernel and use density perturbations scaled from isotropic  $V_s$  perturbations.

**Isotropic model update:** The program `xadd_model_iso` can be used to update *isotropic* model files with (smoothed & summed) event kernels:

```
mpirun -np 4 bin/xadd_model_iso step_factor [INPUT-KERNELS-DIR/] [OUTPUT-MODEL-DIR/]
```

with command line arguments: `step_factor` the step length to scale the gradient, e.g. 0.03 for a  $\pm 3\%$  update, `INPUT-KERNELS-DIR/` (optional) directory which holds summed kernels (e.g. `proc***alpha_kernel.bin`..., by default directory `INPUT_GRADIENT/` is used), `OUTPUT-MODEL-DIR/` (optional) directory which will hold new model files (e.g. `proc***vp_new.bin`..., by default directory `OUTPUT_MODEL/` is used).

The gradients are given for isotropic parameters (`alpha,beta,rho`) or (`bulk_c,beta,rho`).

The algorithm uses a steepest descent method with a step length determined by the given maximum update percentage.

By default, the directory and file setup used is:

- directory INPUT\_MODEL/ contains:
 

```
proc***_vs.bin
proc***_vp.bin
proc***_rho.bin
```
- directory INPUT\_GRADIENT/ contains:
 

```
proc***_bulk_c_kernel_smooth.bin
proc***_bulk_beta_kernel_smooth.bin
proc***_rho_kernel_smooth.bin
```

or

```
proc***_alpha_kernel_smooth.bin
proc***_beta_kernel_smooth.bin
proc***_rho_kernel_smooth.bin
```

depending on the model parameterization.
- directory topo/ contains:
 

```
proc***_external_mesh.bin
```

for the Cartesian version, and

```
proc***_solver_data.bin
```

for the GLOBE version.

The new model files are stored in directory OUTPUT\_MODEL/ as

```
proc***_vp_new.bin
proc***_vs_new.bin
proc***_rho_new.bin
```

Note that additional post-processing and model update tools are provided for the SPECFEM3D\_GLOBE version and will be provided for the Cartesian version as well in future.

### 8.3 OLD VERSION OF THE SECTION, WILL SOON BE IMPROVED AND ENRICHED

There are many other versions of gradient-based inversion algorithms that could alternatively be used (see e.g. Virieux and Operto [2009], Monteiller et al. [2015] for a list). The tomographic inversion of Tape et al. [2009, 2010] used SPECFEM3D\_Cartesian as well as several additional components which are also stored on the CIG svn server, described next.

The directory containing external utilities for tomographic inversion using SPECFEM3D\_Cartesian (or other packages that evaluate misfit functions and gradients) is in directory utils/ADJOINT\_TOMOGRAPHY\_TOOLS/:

```
flexwin/ -- FLEXTWIN algorithm for automated picking of time windows
measure_adj/ -- reads FLEXTWIN output file and makes measurements,
               with the option for computing adjoint sources
iterate_adj/ -- various tools for iterative inversion
               (requires pre-computed "event kernels")
```

This directory also contains a brief README file indicating the role of the three subdirectories, flexwin [Maggi et al., 2009], measure\_adj, and iterate\_adj. The components for making the model update are there; however, there are no explicit rules for computing the model update, just as with any optimization problem. There are options for computing a conjugate gradient step, as well as a source subspace projection step.

The best single file to read is probably: ADJOINT\_TOMO/iterate\_adj/cluster/README.

# Chapter 9

## Noise Cross-correlation Simulations

Besides earthquake simulations, SPECFEM3D Cartesian includes functionality for seismic noise tomography as well. In order to proceed successfully in this chapter, it is critical that you have already familiarized yourself with procedures for meshing (Chapter 3), creating distributed databases (Chapter 4), running earthquake simulations (Chapters 5) and adjoint simulations (Chapter 7). Also, make sure you read the article ‘Noise cross-correlation sensitivity kernels’ [Tromp et al., 2010b], in order to understand noise simulations from a theoretical perspective.

### 9.1 Input Parameter Files

As usual, the three main input files are crucial: `Par_file`, `CMTSOLUTION` and `STATIONS`. Unless otherwise specified, those input files should be located in directory `DATA/`.

`CMTSOLUTION` is required for all simulations. At a first glance, it may seem unexpected to have it here, since the noise simulations should have nothing to do with the earthquake – `CMTSOLUTION`. However, for noise simulations, it is critical to have no earthquakes. In other words, the moment tensor specified in `CMTSOLUTION` must be set to zero manually!

`STATIONS` remains the same as in previous earthquake simulations, except that the order of receivers listed in `STATIONS` is now important. The order will be used to determine the ‘master’ receiver, i.e., the one that simultaneously cross correlates with the others.

`Par_file` also requires careful attention. A parameter called `NOISE_TOMOGRAPHY` has been added which specifies the type of simulation to be run. `NOISE_TOMOGRAPHY` is an integer with possible values 0, 1, 2 and 3. For example, when `NOISE_TOMOGRAPHY` equals 0, a regular earthquake simulation will be run. When it is 1/2/3, you are about to run step 1/2/3 of the noise simulations respectively. Should you be confused by the three steps, refer to Tromp et al. [2010b] for details.

Another change to `Par_file` involves the parameter `NSTEP`. While for regular earthquake simulations this parameter specifies the length of synthetic seismograms generated, for noise simulations it specifies the length of the seismograms used to compute cross correlations. The actual cross correlations are thus twice this length, i.e.,  $2 \text{NSTEP} - 1$ . The code automatically makes the modification accordingly, if `NOISE_TOMOGRAPHY` is not zero.

There are other parameters in `Par_file` which should be given specific values. For instance, since the first two steps for calculating noise sensitivity kernels correspond to forward simulations, `SIMULATION_TYPE` must be 1 when `NOISE_TOMOGRAPHY` equals 1 or 2. Also, we have to reconstruct the ensemble forward wavefields in adjoint simulations, therefore we need to set `SAVE_FORWARD` to `.true.` for the second step, i.e., when `NOISE_TOMOGRAPHY` equals 2. The third step is for kernel constructions. Hence `SIMULATION_TYPE` should be 3, whereas `SAVE_FORWARD` must be `.false..`

Finally, for most system architectures, please make sure that LOCAL\_PATH in Par\_file is in fact local, not globally shared. Because we have to save the wavefields at the earth's surface at every time step, it is quite problematic to have a globally shared LOCAL\_PATH, in terms of both disk storage and I/O speed.

## 9.2 Noise Simulations: Step by Step

Proper parameters in those parameter files are not enough for noise simulations to run. We have more parameters to specify: for example, the ensemble-averaged noise spectrum, the noise distribution etc. However, since there are a few ‘new’ files, it is better to introduce them sequentially. In this section, standard procedures for noise simulations are described.

### 9.2.1 Pre-simulation

- As usual, we first configure the software package using:

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

Use the following if SCOTCH is needed:

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

- Next, we need to compile the source code using:

```
make xgenerate_databases
make xspecfem3D
```

- Before we can run noise simulations, we have to specify the noise statistics, e.g., the ensemble-averaged noise spectrum. Matlab scripts are provided to help you to generate the necessary file:

```
EXAMPLES/noise_tomography/NOISE_TOMOGRAPHY.m (main program)
EXAMPLES/noise_tomography/PetersonNoiseModel.m
```

In Matlab, simply run:

```
NOISE_TOMOGRAPHY(NSTEP, DT, Tmin, Tmax, NOISE_MODEL)
```

DT is given in Par\_file, but **NSTEP is NOT the one specified in Par\_file. Instead, you have to feed 2 NSTEP – 1 to account for the doubled length of cross correlations.** Tmin and Tmax correspond to the period range you are interested in, whereas NOISE\_MODEL denotes the noise model you will be using (‘NLNM’ for New Low Noise Model or ‘NHNM’ for New High Noise Model). Details can be found in the Matlab script.

After running the Matlab script, you will be given the following information (plus a figure in Matlab):

```
*****
the source time function has been saved in:
.... /S_squared (note this path must be different)
S_squared should be put into directory:
./NOISE_TOMOGRAPHY/ in the SPECFEM3D Cartesian package
```

In other words, the Matlab script creates a file called S\_squared, which is the first ‘new’ input file we encounter for noise simulations.

One may choose a flat noise spectrum rather than Peterson’s noise model. This can be done easily by modifying the Matlab script a little.

- Create a new directory in the SPECFEM3D Cartesian package, name it as `./NOISE_TOMOGRAPHY/`. We will add some parameter files later in this folder.
- Put the Matlab-generated-file `S_squared` in `./NOISE_TOMOGRAPHY/`.

That's to say, you will have a file `./NOISE_TOMOGRAPHY/S_squared` in the example provided in the SPECFEM3D Cartesian package.

- Create a file called `./NOISE_TOMOGRAPHY/irec_master_noise`. Note that this file is located in directory `./NOISE_TOMOGRAPHY/` as well. In general, all noise simulation related parameter files go into that directory. `irec_master_noise` contains only one integer, which is the ID of the ‘master’ receiver. For example, if this file contains 5, it means that the fifth receiver listed in `DATA/STATIONS` becomes the ‘master’. That's why we mentioned previously that the order of receivers in `DATA/STATIONS` is important.

Note that in some simulations, the `DATA/STATIONS` might contain receivers which are outside of our computational domains. Therefore, the integer in `irec_master_noise` is actually the ID in `DATA/STATIONS_FILTERED` (which is generated by `bin/xgenerate_databases`).

- Create a file called `./NOISE_TOMOGRAPHY/nu_master`. This file holds three numbers, forming a (unit) vector. It describes which component we are cross-correlating at the ‘master’ receiver, i.e.,  $\hat{v}^\alpha$  in Tromp et al. [2010b]. The three numbers correspond to E/N/Z components respectively. Most often, the vertical component is used, and in those cases the three numbers should be 0, 0 and 1.
- Describe the noise direction and distributions in `src/specfem3d/noise_tomography.f90`. Search for a subroutine called `noise_distribution_direction` in `noise_tomography.f90`. It is actually located at the very beginning of `noise_tomography.f90`. The default assumes vertical noise and a uniform distribution across the whole free surface of the model. It should be quite self-explanatory for modifications. Should you modify this part, you have to re-compile the source code.

## 9.2.2 Simulations

With all of the above done, we can finally launch our simulations. Again, please make sure that the `LOCAL_PATH` in `DATA/Par_file` is not globally shared. It is quite problematic to have a globally shared `LOCAL_PATH`, in terms of both disk storage and speed of I/O (we have to save the wavefields at the earth’s surface at every time step).

As discussed in Tromp et al. [2010b], it takes three steps/simulations to obtain one contribution of the ensemble sensitivity kernels:

- Step 1: simulation for generating wavefields

```
SIMULATION_TYPE = 1
NOISE_TOMOGRAPHY = 1
SAVE_FORWARD (not used, can be either .true. or .false.)
```

- Step 2: simulation for ensemble forward wavefields

```
SIMULATION_TYPE = 1
NOISE_TOMOGRAPHY = 2
SAVE_FORWARD = .true.
```

- Step 3: simulation for ensemble adjoint wavefields and sensitivity kernels

```
SIMULATION_TYPE = 3
NOISE_TOMOGRAPHY = 3
SAVE_FORWARD = .false.
```

Note Step 3 is an adjoint simulation, please refer to previous chapters on how to prepare adjoint sources and other necessary files, as well as how adjoint simulations work.

It is better to run the three steps continuously within the same job on a cluster, otherwise you have to collect the saved surface movies from the old nodes to the new nodes. This process varies from cluster to cluster and thus cannot be discussed here. Please ask your cluster administrator for information/configuration of the cluster you are using.

### 9.2.3 Post-simulation

After those simulations, you have all stuff you need, either in the `OUTPUT_FILES/` or in the directory specified by `LOCAL_PATH` in `DATA/Par_file` (which are most probably on local nodes). Collect whatever you want from the local nodes to your workstation, and then visualize them. This process is the same as what you may have done for regular earthquake simulations. Refer to other chapters if you have problems.

Simply speaking, two outputs are the most interesting: the simulated ensemble cross correlations and one contribution of the ensemble sensitivity kernels.

The simulated ensemble cross correlations are obtained after the second simulation (Step 2). Seismograms in `OUTPUT_FILES/` are actually the simulated ensemble cross correlations. Collect them immediately after Step 2, or the Step 3 will overwrite them. Note that we have a ‘master’ receiver specified by `irec_master_noise`, the seismogram at one station corresponds to the cross correlation between that station and the ‘master’. Since the seismograms have three components, we may obtain cross correlations for different components as well, not necessarily the cross correlations between vertical components.

One contribution of the ensemble cross-correlation sensitivity kernels are obtained after Step 3, stored in the `DATA/LOCAL_PATH` on local nodes. The ensemble kernel files are named the same as regular earthquake kernels.

You need to run another three simulations to get the other contribution of the ensemble kernels, using different forward and adjoint sources given in Tromp et al. [2010b].

## 9.3 Example

In order to illustrate noise simulations in an easy way, one example is provided in `EXAMPLES/noise_tomography/`. See `EXAMPLES/noise_tomography/README` for explanations.

Note, however, that they are created for a specific workstation (`CLOVER@PRINCETON`), which has at least 4 cores with ‘mpif90’ working properly.

If your workstation is suitable, you can run the example in `EXAMPLES/noise_tomography/` using:

```
./pre-processing.sh
```

Even if this script does not work on your workstation, the procedure it describes is universal. You may review the whole process described in the last section by following the commands in `pre-processing.sh`, which should contain enough explanations for all the commands.

## Chapter 10

# Gravity integral calculations for the gravity field of an Earth model

SPECFEM can now compute the gravity field as well as its derivatives (i.e., gravity gradiometry) generated by any given 3D Earth model at a given elevation, i.e., on a given observation surface. That feature is still experimental but should work just fine.

To see how it is implemented and to use it, type this in the root directory of the code:

```
grep -i GRAVITY_INTEGRALS src/*/*
```

# Chapter 11

## Graphics

### 11.1 Meshes

In case you used the internal mesher `xmeshfem3D` to create and partition your mesh, you can output mesh files in ABAQUS (.INP) and DX (.dx) format to visualize them. For this, you must set either the flag `CREATE_DX_FILES` or `CREATE_ABAQUS_FILES` to `.true.` in the mesher's parameter file `Mesh_Par_file` prior to running the mesher (see Chapter 3.2 for details). You can then use AVS ([www.avs.com](http://www.avs.com)) or OpenDX ([www.opendx.org](http://www.opendx.org)) to visualize the mesh and MPI partition (slices).

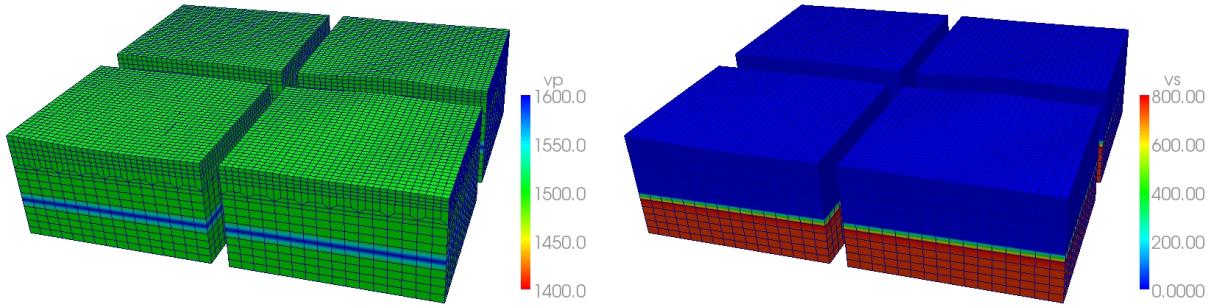


Figure 11.1: Visualization using Paraview of VTK files created by `xgenerate_databases` showing P- and S-wave velocities assigned to the mesh points. The mesh was created by `xmeshfem3D` for 4 processors.

You have also the option to visualize the distributed databases produced by `xgenerate_databases` using Paraview ([www.paraview.org](http://www.paraview.org)). For this, you must set the flag `SAVE_MESH_FILES` to `.true.` in the main parameter file `Par_file` (see Chapter 4.1 for details). This will create VTK files for each single partition. You can then use Paraview ([www.paraview.org](http://www.paraview.org)) to visualize these partitions.

### 11.2 Movies

To make a surface or volume movie of the simulation, set parameters `MOVIE_SURFACE`, `MOVIE_VOLUME`, `MOVIE_TYPE`, and `NTSTEP_BETWEEN_FRAMES` in the `Par_file`. Turning on the movie flags, in particular `MOVIE_VOLUME`, produces large output files. `MOVIE_VOLUME` files are saved in the `LOCAL_PATH` directory, whereas `MOVIE_SURFACE` output files are saved in the `OUTPUT_FILES` directory. We save the displacement field if the parameter `SAVE_DISPLACEMENT` is set, otherwise the velocity field is saved. The look of a movie is determined by the half-duration of the source. The

half-duration should be large enough so that the movie does not contain frequencies that are not resolved by the mesh, i.e., it should not contain numerical noise. This can be accomplished by selecting a CMT HALF\_DURATION  $> 1.1 \times$  smallest period (see figure 5.1). When MOVIE\_SURFACE = .true., the half duration of each source in the CMTSOLUTION file is replaced by

$$\sqrt{(\text{HALF\_DURATION}^2 + \text{HDUR\_MOVIE}^2)}$$

**NOTE:** If HDUR\_MOVIE is set to 0.0, the code will select the appropriate value of  $1.1 \times$  smallest period. As usual, for a point source one can set half duration in the CMTSOLUTION file to be 0.0 and HDUR\_MOVIE = 0.0 in the Par\_file to get the highest frequencies resolved by the simulation, but for a finite source one would keep all the half durations as prescribed by the finite source model and set HDUR\_MOVIE = 0.0.

### 11.2.1 Movie Surface and Shakemaps

When running xspecfem3D with the MOVIE\_SURFACE flag turned on, the code outputs moviedata?????? files in the OUTPUT\_FILES directory. There are several flags in the main parameter file Par\_file that control the output of these movie data files, please see section 4.1 for more details. Especially, the following parameters affect the output:

- SAVE\_DISPLACEMENT: to save displacement instead of velocity,
- NTSTEP\_BETWEEN\_FRAMES: to set the timesteps between frames,
- USE\_HIGHRES\_FOR\_MOVIES: to save values at all GLL point instead of element edges.

You can also output shakemaps independant of the MOVIE\_SURFACE flag. Shakemaps show the peak-ground values of the simulation. For creating shakemaps (at the end of the simulation), you would set the parameter CREATE\_SHAKEMAP to .true.. For shakemaps, both the parameters MOVIE\_TYPE and USE\_HIGHRES\_FOR\_MOVIES affect the output. The following setting has an additional affect:

- MOVIE\_TYPE: if set to 1, the horizontal peak-ground values of displacement/velocity/acceleration are output. if set to 2, the maximum length of the particle displacement/velocity/acceleration vector is output. Please be aware that these peak-ground values can differ from each other.

The movie and shakemap output files are in a custom binary format, but there is a program provided to convert the output into more user friendly formats:

**xcreate\_movie\_shakemap\_AVG\_DX\_GMT** From create\_movie\_shakemap\_AVG\_DX\_GMT.f90, it outputs data in ASCII, OpenDX, or AVS format (also readable in ParaView). Before compiling the code, make sure you have the file surface\_from\_mesher.h in the OUTPUT\_FILES/ directory. This file will be created by the solver run. Then type

```
make xcreate_movie_shakemap_AVG_DX_GMT
```

and run the executable xcreate\_movie\_shakemap\_AVG\_DX\_GMT in the main directory. It will create visualization files in your format of choice. The code will prompt the user for input parameters.

The SPECFEM3D Cartesian code is running in near real-time to produce animations of southern California earthquakes via the web; see Southern California ShakeMovie®([www.shakemovie.caltech.edu](http://www.shakemovie.caltech.edu)).

### 11.2.2 Movie Volume

When running xspecfem3D with the MOVIE\_VOLUME flag turned on, the code outputs several files in LOCAL\_PATH specified in the main Par\_file, e.g. in directory OUTPUT\_FILES/DATABASES\_MPI. The output is saved by each processor at the time interval specified by NTSTEP\_BETWEEN\_FRAMES. For all domains, the velocity field is output to files:

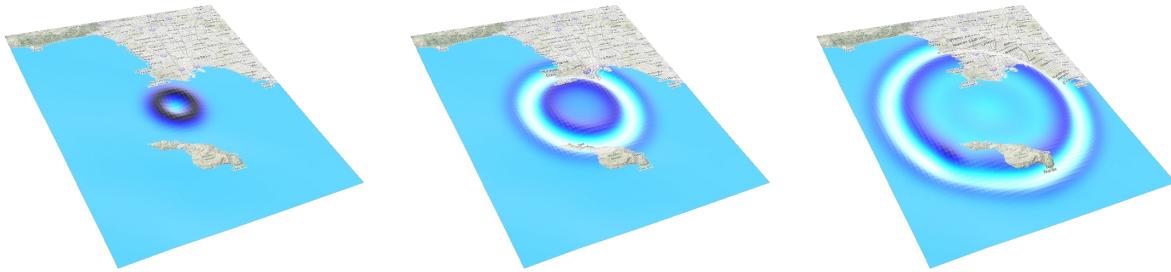


Figure 11.2: Visualization using AVS files created by `xcreate_movie_shakemap_AVSDX_GMT` showing movie snapshots of vertical velocity components at different times.

```
proc??????_velocity_X_it?????.bin
proc??????_velocity_Y_it?????.bin
proc??????_velocity_Z_it?????.bin
```

For elastic domains, the divergence and curl taken from the velocity field, i.e.  $\nabla \cdot \mathbf{v}$  and  $\nabla \times \mathbf{v}$ , get stored as well:

```
proc??????_div_it?????.bin
proc??????_curl_X_it?????.bin
proc??????_curl_Y_it?????.bin
proc??????_curl_Z_it?????.bin
```

The files denoted `proc??????_div_glob_it?????.bin` and `proc??????_curl_glob_it?????.bin` are stored on the global points only, all the other arrays are stored on all GLL points. Note that the components X/Y/Z can change to E/N/Z according to the `SUPPRESS_UTM_PROJECTION` flag (see also Appendix A and B).

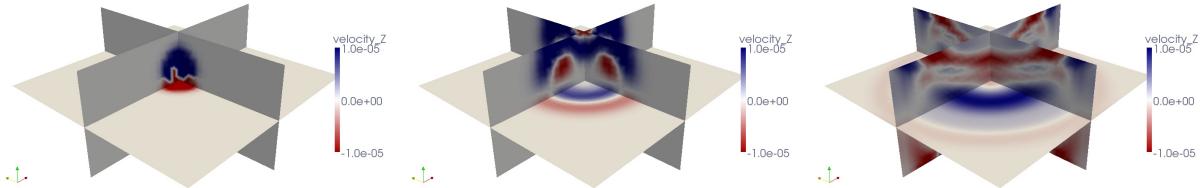


Figure 11.3: Paraview visualization using movie volume files (converted by `xcombine_vol_data` and `mesh2vtu.pl`) and showing snapshots of vertical velocity components at different times.

To visualize these files, we use an auxiliary program `combine_vol_data.f90` to combine the data from all slices into one mesh file. To compile it in the root directory, type:

```
make xcombine_vol_data
```

which will create the executable `bin/xcombine_vol_data`. To output the usage of this executable, type

```
./bin/xcombine_vol_data
```

without arguments.

`xcombine_vol_data` will combine the data on the different processors (located in `OUTPUT_FILES/DATABASES_MPI`), for a given quantity and a given iteration, to one file. For example, if you want to combine `velocity_Z`, for the iteration 400 on 4 processors, i.e. if you want to combine these files :

```
proc000000_velocity_Z_it000400.bin
proc000001_velocity_Z_it000400.bin
proc000002_velocity_Z_it000400.bin
proc000003_velocity_Z_it000400.bin
```

You have to go in the directory of the concerned example (where are the directories DATA, OUTPUT\_FILES, etc.). Then, you can launch `xcombine_vol_data`, specifying the path where it is located. As an example, if the DATA and OUTPUT\_FILES directories of the concerned example are in the root specfem3d directory, `xcombine_vol_data` is in `./bin`, and you have to type:

```
./bin/xcombine_vol_data 0 3 velocity_Z_it000400 ./OUTPUT_FILES/DATABASES_MPI ./OUTPUT_FILES 0
```

Here, 0 is the number of the first processor, 3 the number of the last one, `velocity_Z_it000400` the name of the files we want to combine without the prefix `proc000000*_`, `./OUTPUT_FILES/DATABASES_MPI` the directory where the files are located, `./OUTPUT_FILES` the directory where the combined file will be stored, and 0 is the parameter to create a low-resolution mesh file (1 for a high-resolution).

At the beginning of the source file `combine_vol_data.f90` (located in `src/auxiliaries/`), there is a logical constant parameter `USE_VTK_OUTPUT` whose default value is `.true.`, that which will provide the output combined file directly in the VTK format and you can use Paraview to visualize it.

If the value of this parameter is `.false.`, the output mesh file will have the name `velocity_Z_it000400.mesh`. Then we next have to convert the `.mesh` file into the VTU (Unstructured grid file) format which can be viewed in ParaView.

For this task, you can use and modify the script `mesh2vtu.pl` located in directory `utils/Visualization/Paraview/`, for example:

```
mesh2vtu.pl -i velocity_Z_it000400.mesh -o velocity_Z_it000400.vtu
```

Notice that this Perl script uses a program `mesh2vtu` in the `utils/Visualization/Paraview/mesh2vtu` directory, which further uses the VTK (<http://www.vtk.org/>) run-time library for its execution. Therefore, make sure you have them properly set in the script according to your system.

Then, to do a movie with several iterations, you have to repeat this process for each iteration you want to put in your movie.

## 11.3 Finite-Frequency Kernels

The finite-frequency kernels computed as explained in Section 7.2 are saved in the LOCAL\_PATH at the end of the simulation. Therefore, we first need to collect these files on the front end, combine them into one mesh file, and visualize them with some auxiliary programs.

### 1. Create slice files

We will only discuss the case of one source-receiver pair, i.e., the so-called banana-doughnut kernels. Although it is possible to collect the kernel files from all slices on the front end, it usually takes up too much storage space (at least tens of gigabytes). Since the sensitivity kernels are the strongest along the source-receiver great circle path, it is sufficient to collect only the slices that are along or close to the great circle path.

A Perl script `slice_number.pl` located in directory `utils/Visualization/Paraview/` can help to figure out the slice numbers that lie along the great circle path. It applies to meshes created with the internal mesher `xmeshfem3D`.

- (a) On machines where you have access to the script, copy the `Mesh_Par_file`, and `output_solver` files, and run:

```
slice_number.pl Mesh_Par_file output_solver.txt slice_file
```

which will generate a `slices_file`.

- (b) For cases with multiple sources and multiple receivers, you need to provide a slice file before proceeding to the next step.

### 2. Collect the kernel files

After obtaining the slice files, you can collect the corresponding kernel files from the given slices.

- (a) You can use or modify the script `utils/copy_basin_database.pl` to accomplish this:

```
utils/copy_database.pl slice_file lsf_machine_file filename [jobid]
```

where `lsf_machine_file` is the machine file generated by the LSF scheduler, `filename` is the kernel name (e.g., `rho_kernel`, `alpha_kernel` and `beta_kernel`), and the optional `jobid` is the name of the subdirectory under `LOCAL_PATH` where all the kernel files are stored.

- (b) After executing this script, all the necessary mesh topology files as well as the kernel array files are collected to the local directory of the front end.

### 3. Combine kernel files into one mesh file

We use an auxiliary program `combine_vol_data.f90` to combine the kernel files from all slices into one mesh file.

- (a) Compile it in the root directory:

```
make xcombine_vol_data
./bin/xcombine_vol_data slice_list filename input_dir output_dir high/low-resolution
```

where `input_dir` is the directory where all the individual kernel files are stored, and `output_dir` is where the mesh file will be written.

- (b) Use 1 for a high-resolution mesh, outputting all the GLL points to the mesh file, or use 0 for low resolution, outputting only the corner points of the elements to the mesh file.  
(c) The output mesh file will have the name `filename_rho(alpha,beta).mesh`

### 4. Convert mesh files into .vtu files

- (a) We next convert the `.mesh` file into the VTU (Unstructured grid file) format which can be viewed in ParaView. For this task, you can use and modify the script `mesh2vtu.pl` located in directory `utils/Visualization/Paraview/`, for example:

```
mesh2vtu.pl -i file.mesh -o file.vtu
```

- (b) Notice that this Perl script uses a program `mesh2vtu` in the `utils/Visualization/Paraview/mesh2vtu` directory, which further uses the VTK (<http://www.vtk.org/>) run-time library for its execution. Therefore, make sure you have them properly set in the script according to your system.

### 5. Copy over the source and receiver .vtk file

In the case of a single source and a single receiver, the simulation also generates the file `sr.vtk` located in the `OUTPUT_FILES/` directory to describe the source and receiver locations, which can also be viewed in Paraview in the next step.

### 6. View the mesh in ParaView

Finally, we can view the mesh in ParaView ([www.paraview.org](http://www.paraview.org)).

- (a) Open ParaView.
- (b) From the top menu, File → Open data, select `file.vtu`, and click the Accept button.
  - If the mesh file is of moderate size, it shows up on the screen; otherwise, only the bounding box is shown.
- (c) Click Display Tab → Display Style → Representation and select wireframe of surface to display it.
- (d) To create a cross-section of the volumetric mesh, choose Filter → cut, and under Parameters Tab, choose Cut Function → plane.
- (e) Fill in center and normal information given by the `global_slice_number.pl` script (either from the standard output or from `normal_plane.txt` file).
- (f) To change the color scale, go to Display Tab → Color → Edit Color Map and reselect lower and upper limits, or change the color scheme.

- (g) Now load in the source and receiver location file by File → Open data, select `sr.vtk`, and click the Accept button. Choose Filter → Glyph, and represent the points by ‘spheres’.
- (h) For more information about ParaView, see the ParaView Users Guide ([www.paraview.org/files/v1.6/ParaViewUsersGuide.PDF](http://www.paraview.org/files/v1.6/ParaViewUsersGuide.PDF)).

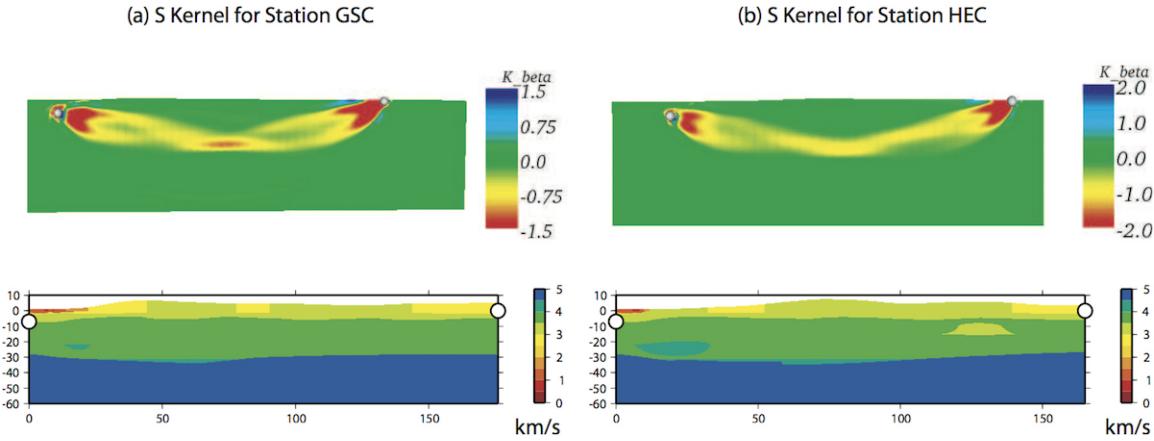


Figure 11.4: (a) Top Panel: Vertical source-receiver cross-section of the S-wave finite-frequency sensitivity kernel  $K_\beta$  for station GSC at an epicentral distance of 176 km from the September 3, 2002, Yorba Linda earthquake. Lower Panel: Vertical source-receiver cross-section of the 3D S-wave speed model used for the spectral-element simulations [Komatitsch et al., 2004]. (b) The same as (a) but for station HEC at an epicentral distance of 165 km [Liu and Tromp, 2006].

## Chapter 12

# Running through a Scheduler

The code is usually run on large parallel machines, often PC clusters, most of which use schedulers, i.e., queuing or batch management systems to manage the running of jobs from a large number of users. The following considerations need to be taken into account when running on a system that uses a scheduler:

- The processors/nodes to be used for each run are assigned dynamically by the scheduler, based on availability. Therefore, in order for the `xgenerate_databases` and the `xspecfem3D` executables (or between successive runs of the solver) to have access to the same database files (if they are stored on hard drives local to the nodes on which the code is run), they must be launched in sequence as a single job.
- On some systems, the nodes to which running jobs are assigned are not configured for compilation. It may therefore be necessary to pre-compile both the `xgenerate_databases` and the `xspecfem3D` executables.
- One feature of schedulers/queuing systems is that they allow submission of multiple jobs in a “launch and forget” mode. In order to take advantage of this property, care needs to be taken that output and intermediate files from separate jobs do not overwrite each other, or otherwise interfere with other running jobs.

Examples of job scripts can be found in the `utils/Cluster/` directory and can straightforwardly be modified and adapted to meet more specific running needs.

# Chapter 13

## Changing the Model

In this section we explain how to change the velocity model used for your simulations. These changes involve contributing specific subroutines that replace existing subroutines in the SPECFEM3D Cartesian package. Note that SPECFEM3D Cartesian can handle Earth models with material properties that vary within each spectral element.

### 13.1 Using external tomographic Earth models

To implement your own external tomographic model(s), you must provide your own external tomography file(s), and choose between two possible options:

(1) set in the `Par_file` the model parameter `MODEL = tomo`,  
or for more user control:

(2) set in the `Par_file` the model parameter `MODEL = default`, define the negative `material_ID` identifier for each element in the file `MESH/materials_file` and use the following format in the file `MESH/nummaterial_velocity` when using CUBIT to construct your mesh (see also section 3.1.2):

```
domain_ID material_ID tomography elastic file_name 1
```

where:

`domain_ID` is 1 for acoustic or 2 for elastic materials,  
`material_ID` a negative, unique identifier (i.e., -1,-2,...),  
`tomography` keyword for tomographic material definition,  
`elastic` keyword for elastic material definition,  
`file_name` the name of the tomography file and 1 a positive unique identifier.

The external tomographic model is represented by a grid of points with assigned material properties and homogeneous resolution along each spatial direction x, y and z. The ASCII file `file_name` that describe the tomography should be located in the `TOMOGRAPHY_PATH` directory, set in the `Par_file`. The format of the file, as read from `model_tomography.f90` located in the `src/generate_databases` directory, looks like Figure 13.1, starting with a header information where

**ORIG\_X, END\_X** are, respectively, the coordinates of the initial and final tomographic grid points along the x direction (in the mesh units, e.g., m);

**ORIG\_Y, END\_Y** respectively the coordinates of the initial and final tomographic grid points along the y direction (in the mesh units, e.g., m);

**ORIG\_Z, END\_Z** respectively the coordinates of the initial and final tomographic grid points along the z direction (in the mesh units, e.g., m);

**SPACING\_X, SPACING\_Y, SPACING\_Z** the spacing between the tomographic grid points along the x, y and z directions, respectively (in the mesh units, e.g., m);

**NX, NY, NZ** the number of grid points along the spatial directions x, y and z, respectively; NX is given by [(END\_X - ORIG\_X)/SPACING\_X]+1; NY and NZ are the same as NX, but for the y and z directions, respectively;

**VP\_MIN, VP\_MAX, VS\_MIN, VS\_MAX, RHO\_MIN, RHO\_MAX** the minimum and maximum values of the wave speed vp and vs (in  $m s^{-1}$ ) and of the density rho (in  $kg m^{-3}$ ); these values could be the actual limits of the tomographic parameters in the grid or the minimum and maximum values to which we force the cut of velocity and density in the model.

After the first four lines, the tomography file file\_name lists the data record where all tomographic grid points are listed with the corresponding values of vp, vs and rho (and optionally  $Q_p$  and  $Q_s$ ), scanning the grid along the x coordinate (from ORIG\_X to END\_X with step of SPACING\_X) for each given y (from ORIG\_Y to END\_Y, with step of SPACING\_Y) and each given z (from ORIG\_Z to END\_Z, with step of SPACING\_Z).

Each data record line provides the velocity model values in a format like:

```
# data record format: purely isotropic model
x y z    vp      vs      rho
..
or

# data record format: anelastic isotropic model
x y z    vp      vs      rho Qp   Qs
..
```

where x, y, z are the grid point position, vp, vs and rho the P- and S-wave speeds and density, and  $Q_p$  and  $Q_s$  the quality factors for P- and S-wave speeds. The quality factors are optional and can be omitted for purely elastic models (the tomography routine will recognize both formats). Internally, the quality factors will be converted to bulk and shear attenuation values,  $Q_\kappa$  and  $Q_\mu$  respectively [Anderson and Hart, 1978]. Note that  $Q_{\mu}$  is always equal to  $Q_s$ , but  $Q_{\kappa}$  is in general not equal to  $Q_p$ . To convert one to the other see doc/note\_on\_Qkappa\_versus\_Qp.pdf and utils/attenuation/conversion\_from\_Qkappa\_Qmu\_to\_Qp\_Qs\_from\_Dahlen\_Tromp\_959\_960.f90. For simulations with attenuation, please note that the Vp- and Vs-velocities of your model are given for a reference frequency. To change this reference frequency, you change the value of ATTENUATION\_f0\_REFERENCE in the main constants file constants.h found in subdirectory src/shared/.

The code uses a constant  $Q$  quality factor, write(IMAIN,\*) "but approximated based on a series of Zener standard linear solids (SLS). The approximation is thus performed in a given frequency band determined based on that ATTENUATION\_f0\_REFERENCE reference frequency.

The user can implement his own interpolation algorithm for the tomography model by changing the routine model\_tomography.f90 located in the src/generate\_databases/ directory. Moreover, for models that involve both fully defined materials and a tomography description, the nummaterial\_velocity\_file has multiple lines each with the corresponding suitable format described above.

### Example: External tomography file with variable discretization intervals

The example in Figure 13.1 is for an external tomography file with uniform spacing in the x, y, and z directions. (Note that  $\Delta x$ ,  $\Delta y$ , and  $\Delta z$  need not be the same as in the example.) In that example, the nummaterial\_velocity\_file is

```
2 -1 tomography elastic tomography_model.xyz 1
```

and the file tomography\_model.xyz will need to reside in DATA/. All entries in the second column of materials\_file will be -1, which means that each element in the mesh will be interpolated according to the values in tomography\_model.xyz.

In some cases it may be desirable to have an external tomography model that is described in more than one file. For example, in cases like southern California, the length scale of variation in the structure of the wave speed model is much shorter in the sedimentary basin models within the upper 15 km. Therefore one might want to use an external tomography file that is sampled with, say,  $\Delta x = 1000$  m,  $\Delta y = 1000$  m, and  $\Delta z = 250$  m in the uppermost 15 km, and then use  $\Delta x = 2000$  m,  $\Delta y = 2000$  m, and  $\Delta z = 1000$  m below a depth of 15 km. If these intervals are chosen appropriately, then it will result in a pair of external tomography file that is much smaller than the alternative of having a single file with the fine discretization. In this case nummaterial\_velocity\_file is

ORIG_X	ORIG_Y	ORIG_Z	END_X	END_Y	END_Z
255066.763887	4551257.50051	-32000.0	455066.763887	4751257.50051	2717.0
<b>SPACING_X</b> <b>SPACING_Y</b> <b>SPACING_Z</b> <b>NX</b> <b>NY</b> <b>NZ</b> <b>VP_MIN</b> <b>VP_MAX</b> <b>VS_MIN</b> <b>VS_MAX</b> <b>RHO_MIN</b> <b>RHO_MAX</b>					
1000.0	1000.0	1000.0			
201	201	34			
1782.	7930.	1000.	4361.11	2000.	2785.46
255066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
256066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
257066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
258066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
259066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
260066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
261066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
262066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
263066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
264066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
265066.763887	4551257.50051	-32000.0	7800.0	4333.3333333	2070.90976436
:	:	:	:	:	:
x	y	z	vp m/s	vs m/s	rho kg/m <sup>3</sup>

Figure 13.1: Tomography file `file_name` that describes an external, purely elastic Earth model. The coordinates x, y and z of the grid, their limits ORIG\_X, ORIG\_Y, ORIG\_Z, END\_X, END\_Y, END\_Z and the grid spacings SPACING\_X, SPACING\_Y, SPACING\_Z should be in the units of the constructed mesh (e.g., UTM coordinates in meters).

```
2 -1 tomography elastic file_above_15km.xyz 1
2 -2 tomography elastic file_below_15km.xyz 1
```

and the files `file_above_15km.xyz` and `file_below_15km.xyz` will need to reside in `DATA/`. All entries in the second column of `materials_file` will need to be either -1 or -2, depending on whether the element is above or below 15 km depth. In this sense, the construction of the mesh and the discretization of the external model will generally be done in tandem.

Other more complicated discretizations can be used following the same procedure. (In particular, there is no limit on the number of different external files that are used.)

## 13.2 External (an)elastic Models

To use your own external model, you can set in the `Par_file` the model parameter `MODEL = external`. Three-dimensional acoustic and/or (an)elastic (attenuation) models may then be superimposed onto the mesh based upon your subroutine in `model_external_values.f90` located in directory `src/generate_databases`. The call to this routine would be as follows

```
call model_external_values(xmesh, ymesh, zmesh, &
                           rho, vp, vs, qkappa_atten, qmu_atten, iflag_aniso, idomain_id)
```

Input to this routine consists of:

**xmesh, ymesh, zmesh** location of mesh point

Output to this routine consists of:

**rho, vp, vs** isotropic model parameters for density  $\rho$  ( $kg/m^3$ ),  $v_p$  ( $m/s$ ) and  $v_s$  ( $m/s$ )

**qkappa\_atten, qmu\_atten** Bulk and shear wave quality factor:  $0 < Q_{\kappa,\mu} < 9000$ . Note that  $Q_{\mu}$  is always equal to  $Q_s$ , but  $Q_{\kappa}$  is in general not equal to  $Q_p$ . To convert one to the other see `doc/note_on_Qkappa_versus_Qp.pdf` and `utils/attenuation/conversion_from_Qkappa_Qmu_to_Qp_Qs_from_Dahlen_Tromp_959_960.f90`.

**iflag\_aniso** anisotropic model flag, 0 indicating no anisotropy or 1 using anisotropic model parameters as defined in routine file `model_aniso.f90`

**idomain\_id** domain identifier, 1 for acoustic, 2 for elastic, 3 for poroelastic materials.

Note that the resolution and maximum value of anelastic models are truncated. This speeds the construction of the standard linear solids during the meshing stage. To change the resolution, currently at one significant figure following the decimal, or the maximum value (9000), consult `shared/constants.h`.

### 13.3 Anisotropic Models

To use your anisotropic models, you can either set in the `Par_file` the model parameter `ANISOTROPY` to `.true.` or set the model parameter `MODEL = aniso`. Three-dimensional anisotropic models may be superimposed on the mesh based upon the subroutine in file `model_aniso.f90` located in directory `src/generate_databases`. The call to this subroutine is of the form

```
call model_aniso(iflag_aniso, rho, vp, vs, &
                 c11,c12,c13,c14,c15,c16,c22,c23,c24,c25,c26, &
                 c33,c34,c35,c36,c44,c45,c46,c55,c56,c66)
```

Input to this routine consists of:

**iflag\_aniso** flag indicating the type of the anisotropic model, i.e. 0 for no superimposed anisotropy, 1 or 2 for generic pre-defined anisotropic models.

**rho, vp, vs** reference isotropic model parameters for density  $\rho$ ,  $v_p$  and  $v_s$ .

Output from the routine consists of the following non-dimensional model parameters:

**c11, ..., c66** 21 dimensionalized anisotropic elastic parameters.

You can replace the `model_aniso.f90` file by your own version *provided you do not change the call structure of the routine*, i.e., the new routine should take exactly the same input and produce the required relative output.

### 13.4 Using external SEP models

Stanford Exploration Project (SEP) models consists of two files per variables, one header file (`.H`) and one binary file (`.H@`). Among other information, the header file indicates, for each dimension, the offset (`o1`, `o2` and `o3` fields), the spatial step size (`d1`, `d2` and `d3` fields) and the number of samples (`n1`, `n2` and `n3`) necessary to read the linked binary file (`in` field).

SEP model reading occurs during database generation. It is the user responsibility to ensure that proper interfaces between acoustic and elastic domains are set up during the meshing phase. For more information, refer to section 3.2.

In `DATA/Par_file`, `MODEL` should be set to `sep`, and `SEP_MODEL_DIRECTORY` should point to wherever your sep model files reside.

Note that in order not to overload the parameter file, SEP header files should be named `vp.H`, `vs.H` and `rho.H`. The `in` field in these binary files can be any path as long as it is relative to the previously set `SEP_MODEL_DIRECTORY`. We also assume that any dimensional value inside SEP header files is given in meters and that binary files are single precision floating point numbers stored in little endian format. There is currently only support for `vp`, `vs` and `rho` variables.

An example demonstrating the use SEP models is given in: `EXAMPLES/meshfem3D_examples/sep_bathymetry`.

# Chapter 14

## Post-Processing Scripts

Several post-processing scripts/programs are provided in the `utils/` directory, and most of them need to be adjusted when used on different systems, for example, the path of the executable programs. Here we only list a few of the available scripts and provide a brief description, and you can either refer to the related sections for detailed usage or, in a lot of cases, type the script/program name without arguments for its usage.

### 14.1 Process Data and Synthetics

In many cases, the SEM synthetics are calculated and compared to data seismograms recorded at seismic stations. Since the SEM synthetics are accurate for a certain frequency range, both the original data and the synthetics need to be processed before a comparison can be made.

For such comparisons, the following steps are recommended:

1. Make sure that both synthetic and observed seismograms have the correct station/event and timing information.
2. Convolve synthetic seismograms with a source time function with the half duration specified in the `CMTSOLUTION` file, provided, as recommended, you used a zero half duration in the SEM simulations.
3. Resample both observed and synthetic seismograms to a common sampling rate.
4. Cut the records using the same window.
5. Remove the trend and mean from the records and taper them.
6. Remove the instrument response from the observed seismograms (recommended) or convolve the synthetic seismograms with the instrument response.
7. Make sure that you apply the same filters to both observed and synthetic seismograms. Preferably, avoid filtering your records more than once.
8. Now, you are ready to compare your synthetic and observed seismograms.

We generally use the following scripts provided in the `utils/seis_process/` directory:

#### 14.1.1 Data processing script `process_data.pl`

This script cuts a given portion of the original data, filters it, transfers the data into a displacement record, and picks the first P and S arrivals. For more functionality, type ‘`process_data.pl`’ without any argument. An example of the usage of the script:

```
process_data.pl -m CMTSOLUTION -s 1.0 -l 0/4000 -i -f -t 40/500 -p -x bp DATA/1999.330*.BH?.SAC
```

which has resampled the SAC files to a sampling rate of 1 seconds, cut them between 0 and 4000 seconds, transferred them into displacement records and filtered them between 40 and 500 seconds, picked the first P and S arrivals, and added suffix ‘bp’ to the file names.

Note that all of the scripts in this section actually use the SAC and/or IASP91 to do the core operations; therefore make sure that the SAC and IASP91 packages are installed properly on your system, and that all the environment variables are set properly before running these scripts.

### 14.1.2 Synthetics processing script `process_syn.pl`

This script converts the synthetic output from the SEM code from ASCII to SAC format, and performs similar operations as ‘`process_data.pl`’. An example of the usage of the script:

```
process_syn.pl -m CMTSOLUTION -h -a STATIONS -s 1.0 -l 0/4000 -f -t 40/500 -p -x bp SEM/*.BX?.semd
```

which will convolve the synthetics with a triangular source-time function from the CMTSOLUTION file, convert the synthetics into SAC format, add event and station information into the SAC headers, resample the SAC files with a sampling rate of 1 seconds, cut them between 0 and 4000 seconds, filter them between 40 and 500 seconds with the same filter used for the observed data, pick the first P and S arrivals, and add the suffix ‘bp’ to the file names.

More options are available for this script, such as adding time shift to the origin time of the synthetics, convolving the synthetics with a triangular source time function with a given half duration, etc. Type `process_syn.pl` without any argument for a detailed usage.

In order to convert between SAC format and ASCII files, useful scripts are provided in the subdirectories `utils/sac2000_alpha_convert/` and `utils/seis_process/asc2sac/`.

### 14.1.3 Script `rotate.pl`

The original data and synthetics have three components: vertical (BHZ resp. BXZ), north (BHN resp. BXN) and east (BHE resp. BXE). However, for most seismology applications, transverse and radial components are also desirable. Therefore, we need to rotate the horizontal components of both the data and the synthetics to the transverse and radial direction, and `rotate.pl` can be used to accomplish this:

```
rotate.pl -l 0 -L 180 -d DATA/*.BHE.SAC.bp
rotate.pl -l 0 -L 180 SEM/*.BXE.semd.sac.bp
```

where the first command performs rotation on the SAC data obtained through Seismogram Transfer Program (STP) (<http://www.data.scec.org/STP/stp.html>), while the second command rotates the processed SEM synthetics.

## 14.2 Collect Synthetic Seismograms

The forward and adjoint simulations generate synthetic seismograms in the `OUTPUT_FILES/` directory by default. For the forward simulation, the files are named like `NT.STA.BX?.semd` for two-column time series, or `NT.STA.BX?.semd.sac` for ASCII SAC format, where NT and STA are the network code and station name, and BX? stands for the component name. Please see the Appendix A and B for further details.

The adjoint simulations generate synthetic seismograms with the name `NT.S?????.S???.sem` (refer to Section 7.1 for details). The kernel simulations output the back-reconstructed synthetic seismogram in the name `NT.STA.BX?.semd`, mainly for the purpose of checking the accuracy of the reconstruction. Refer to Section 7.2 for further details.

You do have further options to change this default output behavior, given in the main constants file `constants.h` located in `src/shared/` directory:

**SEISMOGRAMS\_BINARY** set to `.true.` to have seismograms written out in binary format.

**WRITE\_SEISMOGRAMS\_BY\_MASTER** Set to `.true.` to have only the master process writing out seismograms.

This can be useful on a cluster, where only the master process node has access to the output directory.

**USE\_OUTPUT\_FILES\_PATH** Set to `.false.` to have the seismograms output to `LOCAL_PATH` directory specified in the main parameter file `DATA/Par_file`. In this case, you could collect the synthetics onto the frontend using the `collect_seismo_lsf_multi.pl` script located in the `utils/Cluster/lsm/` directory. The usage of the script would be e.g.:

```
collect_seismo.pl machines DATA/Par_file
```

where `machines` is a file containing the node names and `DATA/Par_file` the parameter file used to extract the `LOCAL_PATH` directory used for the simulation.

## 14.3 Clean Local Database

After all the simulations are done, the seismograms are collected, and the useful database files are copied to the frontend, you may need to clean the local scratch disk for the next simulation. This is especially important in the case of kernel simulation, where very large files are generated for the absorbing boundaries to help with the reconstruction of the regular forward wavefield. A sample script is provided in `utils/`:

```
cleanbase.pl machines
```

where `machines` is a file containing the node names.

## 14.4 Plot Movie Snapshots and Synthetic Shakemaps

### 14.4.1 Script `movie2gif.gmt.pl`

With the movie data saved in `OUTPUT_FILES/` at the end of a movie simulation (`MOVIE_SURFACE=.true.`), you can run the '`create_movie_shakemap_AVG_DX_GMT`' code to convert these binary movie data into GMT xyz files for further processing. A sample script `movie2gif.gmt.pl` is provided to do this conversion, and then plot the movie snapshots in GMT, for example:

```
movie2gif.gmt.pl -m CMTSOLUTION -g -f 1/40 -n -2 -p
```

which for the first through the 40th movie frame, converts the `moviedata` files into GMT xyz files, interpolates them using the '`nearneighbor`' command in GMT, and plots them on a 2D topography map. Note that '`-2`' and '`-p`' are both optional.

### 14.4.2 Script `plot_shakemap.gmt.pl`

With the shakemap data saved in `OUTPUT_FILES/` at the end of a shakemap simulation (`CREATE_SHAKEMAP=.true.`), you can also run '`create_movie_shakemap_AVG_DX_GMT`' code to convert the binary shakemap data into GMT xyz files. A sample script `plot_shakemap.gmt.pl` is provided to do this conversion, and then plot the shakemaps in GMT, for example:

```
plot_shakemap.gmt.pl data_dir type(1,2,3) CMTSOLUTION
```

where `type=1` for a displacement shakemap, 2 for velocity, and 3 for acceleration.

## 14.5 Map Local Database

A sample program `remap_database` is provided to map the local database from a set of machines to another set of machines. This is especially useful when you want to run mesher and solver, or different types of solvers separately through a scheduler (refer to Chapter 12).

```
run_lsf.bash --gm-no-shmem --gm-copy-env remap_database old_machines 150
```

where `old_machines` is the LSF machine file used in the previous simulation, and 150 is the number of processors in total.

## Chapter 15

# Information for developers of the code, and for people who want to learn how the technique works

You can get a very simple 1D version of a demo code (there is one in Fortran and one in Python):

```
git clone --recursive https://github.com/geodynamics/specfem1d.git
```

We also have simple 3D demo source codes that implement the SEM in a single, small program, in directory `utils/small_SEM_solvers_in_Fortran_and_C_without_MPI_to_learn` of the `specfem3d` package. They are useful to learn how the spectral-element method works, and how to write or modify a code to implement it. Also useful to test new ideas by modifying these simple codes to run some tests. We also have a similar, even simpler, demo source code for the 2D case in directory `utils/small_SEM_solver_in_Fortran_without_MPI_to_learn` of the `specfem2d` package.

For information on how to contribute to the code, i.e., for how to make your modifications, additions or improvements part of the official package, see <https://github.com/geodynamics/specfem3d/wiki>.

# Bug Reports and Suggestions for Improvements

To report bugs or suggest improvements to the code, please send an e-mail to the CIG Computational Seismology Mailing List ([cig-seismo@geodynamics.org](mailto:cig-seismo@geodynamics.org)).

# Notes & Acknowledgments

In order to keep the software package thread-safe in case a multithreaded implementation of MPI is used, developers should not add modules or common blocks to the source code but rather use regular subroutine arguments (which can be grouped in “derived types” if needed for clarity).

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).

OpenDX (<http://www.opendx.org>) is open-source based on IBM Data Explorer, AVS (<http://www.avs.com>) is a trademark of Advanced Visualization Systems, and ParaView (<http://www.paraview.org>) is an open-source visualization platform.

Please e-mail your feedback, 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

This program is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation (see Appendix D).

Please note that by contributing to this code, the developer understands and agrees that this project and contribution are public and fall under the open source license mentioned above.

## Evolution of the code:

MPI v. 3.0, December 2014: many developers. Convolutional PML, LDDRK time scheme, bulk attenuation support, simultaneous MPI runs, ADIOS file I/O support, new seismogram names, Deville routines for additional GLL degrees, tomography tools, unit/regression test framework, improved CUDA GPUs performance, additional GEOCUBIT support, better make compilation, git versioning system.

MPI v. 2.1, July 2012: Max Rietmann, Peter Messmer, Daniel Peter, Dimitri Komatitsch, Joseph Charles, Zhinan Xie: support for CUDA GPUs, better CFL stability for the Stacey absorbing conditions.

MPI v. 2.0, November 2010: Dimitri Komatitsch, Nicolas Le Goff, Roland Martin and Pieyre Le Loher, University of Pau, France, Daniel Peter, Jeroen Tromp and the Princeton group of developers, Princeton University, USA, and Emanuele Casarotti, INGV Roma, Italy: support for CUBIT meshes decomposed by SCOTCH; much faster solver using Michel Deville's inlined matrix products.

MPI v. 1.4 Dimitri Komatitsch, University of Pau, Qinya Liu and others, Caltech, September 2006: better adjoint and kernel calculations, faster and better I/Os on very large systems, many small improvements and bug fixes.

MPI v. 1.3 Dimitri Komatitsch, University of Pau, and Qinya Liu, Caltech, July 2005: serial version, regular mesh, adjoint and kernel calculations, ParaView support.

MPI v. 1.2 Min Chen and Dimitri Komatitsch, Caltech, July 2004: full anisotropy, volume movie.

MPI v. 1.1 Dimitri Komatitsch, Caltech, October 2002: Zhu's Moho map, scaling of  $V_s$  with depth, Hauksson's regional model, attenuation, oceans, movies.

MPI v. 1.0 Dimitri Komatitsch, Caltech, USA, May 2002: first MPI version based on global code.

Dimitri Komatitsch, IPG Paris, France, December 1996: first 3-D solver for the CM-5 Connection Machine, parallelized on 128 processors using Connection Machine Fortran.

# Bibliography

- C. A. Acosta Minolia and D. A. Kopriva. Discontinuous Galerkin spectral element approximations on moving meshes. *J. Comput. Phys.*, 230(5):1876–1902, 2011. doi: 10.1016/j.jcp.2010.11.038.
- M. Ainsworth and H. Wajid. Dispersive and dissipative behavior of the spectral element method. *SIAM Journal on Numerical Analysis*, 47(5):3910–3937, 2009. doi: 10.1137/080724976.
- M. Ainsworth and H. Wajid. Optimally blended spectral-finite element scheme for wave propagation and nonstandard reduced integration. *SIAM Journal on Numerical Analysis*, 48(1):346–371, 2010. doi: 10.1137/090754017.
- M. Ainsworth, P. Monk, and W. Muniz. Dispersive and dissipative properties of discontinuous Galerkin finite element methods for the second-order wave equation. *Journal of Scientific Computing*, 27(1):5–40, 2006. doi: 10.1007/s10915-005-9044-x.
- K. Aki and P. G. Richards. *Quantitative seismology, theory and methods*. W. H. Freeman, San Francisco, USA, 1980. 700 pages.
- D. L. Anderson and R. S. Hart.  $Q$  of the earth. *J. Geophys. Res.*, 83(B12), 1978.
- A. Aravkin, T. van Leeuwen, and F. Herrmann. Robust full-waveform inversion using the Student's  $t$ -distribution. In *SEG Technical Program Expanded Abstracts 2011*, pages 2669–2673, 2011. doi: 10.1190/1.3627747.
- D. N. Arnold. An interior penalty finite element method with discontinuous elements. *SIAM Journal on Numerical Analysis*, 19(4):742–760, 1982. doi: 10.1137/0719052.
- A. Bamberger, G. Chavent, and P. Lailly. Une application de la théorie du contrôle à un problème inverse sismique. *Ann. Geophys.*, 33:183–200, 1977.
- A. Bamberger, G. Chavent, C. Hemons, and P. Lailly. Inversion of normal incidence seismograms. *Geophysics*, 47(5):757–770, 1982.
- M. Benjemaa, N. Glinsky-Olivier, V. M. Cruz-Atienza, J. Virieux, and S. Piperno. Dynamic non-planar crack rupture by a finite volume method. *Geophys. J. Int.*, 171(1):271–285, 2007. doi: 10.1111/j.1365-246X.2006.03500.x.
- M. Benjemaa, N. Glinsky-Olivier, V. M. Cruz-Atienza, and J. Virieux. 3D dynamic rupture simulation by a finite volume method. *Geophys. J. Int.*, 178(1):541–560, 2009. doi: 10.1111/j.1365-246X.2009.04088.x.
- M. Bernacki, S. Lanteri, and S. Piperno. Time-domain parallel simulation of heterogeneous wave propagation on unstructured grids using explicit, nondiffusive, discontinuous Galerkin methods. *J. Comput. Acoust.*, 14(1):57–81, 2006.
- C. Bernardi, Y. Maday, and A. T. Patera. A new nonconforming approach to domain decomposition: the Mortar element method. In H. Brezis and J. L. Lions, editors, *Nonlinear partial differential equations and their applications*, Séminaires du Collège de France, pages 13–51, Paris, 1994. Pitman.
- E. Blanc, D. Komatitsch, E. Chaljub, B. Lombard, and Z. Xie. Highly accurate stability-preserving optimization of the Zener viscoelastic model, with application to wave propagation in the presence of strong attenuation. *Geophys. J. Int.*, 205(1):427–439, 2016. doi: 10.1093/gji/ggw024.

- F. Bourdel, P.-A. Mazet, and P. Helluy. Resolution of the non-stationary or harmonic Maxwell equations by a discontinuous finite element method: Application to an E.M.I. (electromagnetic impulse) case. In *Proceedings of the 10th international conference on computing methods in applied sciences and engineering*, pages 405–422, Commack, NY, USA, 1991. Nova Science Publishers, Inc.
- R. Brossier, S. Operto, and J. Virieux. Which data residual norm for robust elastic frequency-domain full waveform inversion? *Geophysics*, 75(3):R37–R46, 2010.
- K. P. Bube and R. T. Langan. Hybrid  $L_1/L_2$  minimization with applications to tomography. *Geophysics*, 62(4): 1183–1195, 1997. doi: 10.1190/1.1444219.
- L. Carrington, D. Komatitsch, M. Laurenzano, M. Tikir, D. Michéa, N. Le Goff, A. Snavely, and J. Tromp. High-frequency simulations of global seismic wave propagation using SPECFEM3D\_GLOBE on 62 thousand processor cores. In *Proceedings of the SC'08 ACM/IEEE conference on Supercomputing*, pages 60:1–60:11, Austin, Texas, USA, Nov. 2008. IEEE Press. doi: 10.1145/1413370.1413432. Article #60, Gordon Bell Prize finalist article.
- F. Casadei and E. Gabellini. Implementation of a 3D coupled Spectral Element solver for wave propagation and soil-structure interaction simulations. Technical report, European Commission Joint Research Center Report EUR17730EN, Ispra, Italy, 1997.
- E. Chaljub. *Modélisation numérique de la propagation d'ondes sismiques en géométrie sphérique : application à la sismologie globale (Numerical modeling of the propagation of seismic waves in spherical geometry: application to global seismology)*. PhD thesis, Université Paris VII Denis Diderot, Paris, France, 2000.
- E. Chaljub, Y. Capdeville, and J. P. Vilotte. Solving elastodynamics in a fluid-solid heterogeneous sphere: a parallel spectral-element approximation on non-conforming grids. *J. Comput. Phys.*, 187(2):457–491, 2003.
- E. Chaljub, D. Komatitsch, J. P. Vilotte, Y. Capdeville, B. Valette, and G. Festa. Spectral element analysis in seismology. In R.-S. Wu and V. Maupin, editors, *Advances in wave propagation in heterogeneous media*, volume 48 of *Advances in Geophysics*, pages 365–419. Elsevier - Academic Press, London, UK, 2007.
- G. Chavent. *Identification of parameter distributed systems*, chapter Identification of function parameters in partial differential equations, pages 31–48. American Society of Mechanical Engineers, 1974.
- M. Chen and J. Tromp. Theoretical and numerical investigations of global and regional seismic wave propagation in weakly anisotropic earth models. *Geophys. J. Int.*, 168(3):1130–1152, 2007. doi: 10.1111/j.1365-246X.2006.03218.x.
- S. Chevrot, N. Favier, and D. Komatitsch. Shear wave splitting in three-dimensional anisotropic media. *Geophys. J. Int.*, 159(2):711–720, 2004. doi: 10.1111/j.1365-246X.2004.02432.x.
- B. Cockburn, G. E. Karniadakis, and C.-W. Shu. *Discontinuous Galerkin Methods: Theory, Computation and Applications*. Springer-Verlag, Heidelberg, Germany, 2000.
- G. Cohen. *Higher-order numerical methods for transient wave equations*. Springer-Verlag, Berlin, Germany, 2002. 349 pages.
- G. Cohen, P. Joly, and N. Tordjman. Construction and analysis of higher-order finite elements with mass lumping for the wave equation. In R. Kleinman, editor, *Proceedings of the second international conference on mathematical and numerical aspects of wave propagation*, pages 152–160. SIAM, Philadelphia, Pennsylvania, USA, 1993.
- E. Crase, A. Pica, M. Noble, J. McDonald, and A. Tarantola. Robust elastic non-linear waveform inversion: application to real data. *Geophysics*, 55:527–538, 1990.
- E. C. Cyr, J. N. Shadid, and T. Wildey. Towards efficient backward-in-time adjoint computations using data compression techniques. *Comput. Methods Appl. Mech. Eng.*, 288:24–44, 2015. doi: 10.1016/j.cma.2014.12.001.
- F. A. Dahlen and J. Tromp. *Theoretical Global Seismology*. Princeton University Press, Princeton, New-Jersey, USA, 1998. 944 pages.

- J. D. De Basabe and M. K. Sen. Grid dispersion and stability criteria of some common finite-element methods for acoustic and elastic wave equations. *Geophysics*, 72(6):T81–T95, 2007. doi: 10.1190/1.2785046.
- J. D. De Basabe and M. K. Sen. Stability of the high-order finite elements for acoustic or elastic wave propagation with high-order time stepping. *Geophys. J. Int.*, 181(1):577–590, 2010. doi: 10.1111/j.1365-246X.2010.04536.x.
- J. D. De Basabe, M. K. Sen, and M. F. Wheeler. The interior penalty discontinuous Galerkin method for elastic wave propagation: grid dispersion. *Geophys. J. Int.*, 175(1):83–93, 2008. doi: 10.1111/j.1365-246X.2008.03915.x.
- J. de la Puente, J. P. Ampuero, and M. Käser. Dynamic rupture modeling on unstructured meshes using a discontinuous Galerkin method. *J. Geophys. Res.*, 114:B10302, 2009. doi: 10.1029/2008JB006271.
- M. O. Deville, P. F. Fischer, and E. H. Mund. *High-Order Methods for Incompressible Fluid Flow*. Cambridge University Press, Cambridge, United Kingdom, 2002. 528 pages.
- D. S. Dreger and D. V. Helmberger. Broadband modeling of local earthquakes. *Bull. seism. Soc. Am.*, 80:1162–1179, 1990.
- M. Dumbser and M. Käser. An arbitrary high-order discontinuous Galerkin method for elastic waves on unstructured meshes-II. The three-dimensional isotropic case. *Geophys. J. Int.*, 167(1):319–336, 2006. doi: 10.1111/j.1365-246X.2006.03120.x.
- A. M. Dziewoński and D. L. Anderson. Preliminary reference Earth model. *Phys. Earth planet. Inter.*, 25(4):297–356, 1981.
- V. Étienne, E. Chaljub, J. Virieux, and N. Glinsky. An  $hp$ -adaptive discontinuous Galerkin finite-element method for 3-D elastic wave modelling. *Geophys. J. Int.*, 183(2):941–962, 2010. doi: 10.1111/j.1365-246X.2010.04764.x.
- E. Faccioli, F. Maggio, R. Paolucci, and A. Quarteroni. 2D and 3D elastic wave propagation by a pseudo-spectral domain decomposition method. *J. Seismol.*, 1:237–251, 1997.
- R. S. Falk and G. R. Richter. Explicit finite element methods for symmetric hyperbolic equations. *SIAM Journal on Numerical Analysis*, 36(3):935–952, 1999. doi: 10.1137/S0036142997329463.
- N. Favier, S. Chevrot, and D. Komatitsch. Near-field influences on shear wave splitting and travelttime sensitivity kernels. *Geophys. J. Int.*, 156(3):467–482, 2004. doi: 10.1111/j.1365-246X.2004.02178.x.
- A. Fichtner, H.-P. Bunge, and H. Igel. The adjoint method in seismology: I. Theory. *Phys. Earth planet. Inter.*, 157 (1-2):86–104, 2006. doi: 10.1016/j.pepi.2006.03.016.
- A. Fichtner, H. Igel, H.-P. Bunge, and B. L. N. Kennett. Simulation and inversion of seismic wave propagation on continental scales based on a spectral-element method. *Journal of Numerical Analysis, Industrial and Applied Mathematics*, 4(1-2):11–22, 2009a.
- A. Fichtner, B. L. N. Kennett, H. Igel, and H. P. Bunge. Full seismic waveform tomography for upper-mantle structure in the Australasian region using adjoint methods. *Geophys. J. Int.*, 179(3):1703–1725, 2009b.
- C. Geuzaine and J. F. Remacle. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. *Int. J. Numer. Methods Eng.*, 79(11):1309–1331, 2009.
- F. X. Giraldo, J. S. Hesthaven, and T. Warburton. Nodal high-order discontinuous Galerkin methods for the spherical shallow water equations. *J. Comput. Phys.*, 181(2):499–525, 2002. doi: 10.1006/jcph.2002.7139.
- L. Godinho, P. A. Mendes, A. Tadeu, A. Cadena-Isaza, C. Smerzini, F. J. Sánchez-Sesma, R. Madec, and D. Komatitsch. Numerical simulation of ground rotations along 2D topographical profiles under the incidence of elastic plane waves. *Bull. seism. Soc. Am.*, 99(2B):1147–1161, 2009. doi: 10.1785/0120080096.
- W. Gropp, E. Lusk, and A. Skjellum. *Using MPI, portable parallel programming with the Message-Passing Interface*. MIT Press, Cambridge, USA, 1994.

- M. J. Grote, A. Schneebeli, and D. Schötzau. Discontinuous Galerkin finite element method for the wave equation. *SIAM Journal on Numerical Analysis*, 44(6):2408–2431, 2006. doi: 10.1137/05063194X.
- T. Ha, W. Chung, and C. Shin. Waveform inversion using a back-propagation algorithm and a Huber function norm. *Geophysics*, 74(3):R15–R24, 2009. doi: 10.1190/1.3112572.
- E. Hauksson. Crustal structure and seismicity distribution adjacent to the Pacific and North America plate boundary in Southern California. *J. Geophys. Res.*, 105:13875–13903, 2000.
- F. Q. Hu, M. Y. Hussaini, and P. Rasetarinera. An analysis of the discontinuous Galerkin method for wave propagation problems. *J. Comput. Phys.*, 151(2):921–946, 1999. doi: 10.1006/jcph.1999.6227.
- J. A. Hudson. Scattered waves in the coda of P. *J. Geophys. Res.*, 43:359–374, 1977.
- F. B. Jensen, W. Kuperman, M. Porter, and H. Schmidt. *Computational Ocean Acoustics*. Springer-Verlag, Berlin, Germany, 2nd edition, 2011. 794 pages.
- W. Jeong, M. Kang, S. Kim, D.-J. Min, and W.-K. Kim. Full waveform inversion using Student's  $t$  distribution: a numerical study for elastic waveform inversion and simultaneous-source method. *Pure Appl. Geophys.*, pages 1–19, 2015. doi: 10.1007/s00024-014-1020-7.
- C. Ji, S. Tsuboi, D. Komatitsch, and J. Tromp. Rayleigh-wave multipathing along the west coast of North America. *Bull. seism. Soc. Am.*, 95(6):2115–2124, 2005. doi: 10.1785/0120040180.
- C. Johnson and J. Pitkänta. An analysis of the discontinuous Galerkin method for a scalar hyperbolic equation. *Math. Comp.*, 46:1–26, 1986. doi: 10.1090/S0025-5718-1986-0815828-4.
- G. Karypis and V. Kumar. A fast and high-quality multilevel scheme for partitioning irregular graphs. *SIAM Journal on Scientific Computing*, 20(1):359–392, 1998a.
- G. Karypis and V. Kumar. Multilevel  $k$ -way partitioning scheme for irregular graphs. *Journal of Parallel and Distributed Computing*, 48(1):96–129, 1998b.
- G. Karypis and V. Kumar. A parallel algorithm for multilevel graph partitioning and sparse matrix ordering. *Journal of Parallel and Distributed Computing*, 48:71–85, 1998c.
- D. Komatitsch. *Méthodes spectrales et éléments spectraux pour l'équation de l'élastodynamique 2D et 3D en milieu hétérogène (Spectral and spectral-element methods for the 2D and 3D elastodynamics equations in heterogeneous media)*. PhD thesis, Institut de Physique du Globe, Paris, France, May 1997. 187 pages.
- D. Komatitsch. Fluid-solid coupling on a cluster of GPU graphics cards for seismic wave propagation. *C. R. Acad. Sci., Ser. IIb Mec.*, 339:125–135, 2011. doi: 10.1016/j.crme.2010.11.007.
- D. Komatitsch and R. Martin. An unsplit convolutional Perfectly Matched Layer improved at grazing incidence for the seismic wave equation. *Geophysics*, 72(5):SM155–SM167, 2007. doi: 10.1190/1.2757586.
- D. Komatitsch and J. Tromp. Introduction to the spectral-element method for 3-D seismic wave propagation. *Geophys. J. Int.*, 139(3):806–822, 1999. doi: 10.1046/j.1365-246x.1999.00967.x.
- D. Komatitsch and J. Tromp. Spectral-element simulations of global seismic wave propagation-I. Validation. *Geophys. J. Int.*, 149(2):390–412, 2002a. doi: 10.1046/j.1365-246X.2002.01653.x.
- D. Komatitsch and J. Tromp. Spectral-element simulations of global seismic wave propagation-II. 3-D models, oceans, rotation, and self-gravitation. *Geophys. J. Int.*, 150(1):303–318, 2002b. doi: 10.1046/j.1365-246X.2002.01716.x.
- D. Komatitsch and J. P. Vilotte. The spectral-element method: an efficient tool to simulate the seismic response of 2D and 3D geological structures. *Bull. seism. Soc. Am.*, 88(2):368–392, 1998.
- D. Komatitsch, R. Martin, J. Tromp, M. A. Taylor, and B. A. Wingate. Wave propagation in 2-D elastic media using a spectral element method with triangles and quadrangles. *J. Comput. Acoust.*, 9(2):703–718, 2001. doi: 10.1142/S0218396X01000796.

- D. Komatitsch, J. Ritsema, and J. Tromp. The spectral-element method, Beowulf computing, and global seismology. *Science*, 298(5599):1737–1742, 2002. doi: 10.1126/science.1076024.
- D. Komatitsch, S. Tsuboi, C. Ji, and J. Tromp. A 14.6 billion degrees of freedom, 5 teraflops, 2.5 terabyte earthquake simulation on the Earth Simulator. In *Proceedings of the SC'03 ACM/IEEE conference on Supercomputing*, pages 4–11, Phoenix, Arizona, USA, Nov. 2003. ACM. doi: 10.1145/1048935.1050155. Gordon Bell Prize winner article.
- D. Komatitsch, Q. Liu, J. Tromp, P. Süss, C. Stidham, and J. H. Shaw. Simulations of ground motion in the Los Angeles basin based upon the spectral-element method. *Bull. seism. Soc. Am.*, 94(1):187–206, 2004. doi: 10.1785/0120030077.
- D. Komatitsch, J. Labarta, and D. Michéa. A simulation of seismic wave propagation at high resolution in the inner core of the Earth on 2166 processors of MareNostrum. *Lecture Notes in Computer Science*, 5336:364–377, 2008.
- D. Komatitsch, D. Michéa, and G. Erlebacher. Porting a high-order finite-element earthquake modeling application to NVIDIA graphics cards using CUDA. *Journal of Parallel and Distributed Computing*, 69(5):451–460, 2009. doi: 10.1016/j.jpdc.2009.01.006.
- D. Komatitsch, G. Erlebacher, D. Göddeke, and D. Michéa. High-order finite-element seismic wave propagation modeling with MPI on a large GPU cluster. *J. Comput. Phys.*, 229(20):7692–7714, 2010a. doi: 10.1016/j.jcp.2010.06.024.
- D. Komatitsch, L. P. Vinnik, and S. Chevrot. SHdiff/SVdiff splitting in an isotropic Earth. *J. Geophys. Res.*, 115(B7):B07312, 2010b. doi: 10.1029/2009JB006795.
- D. Komatitsch, Z. Xie, E. Bozdağ, E. Sales de Andrade, D. Peter, Q. Liu, and J. Tromp. Anelastic sensitivity kernels with parsimonious storage for adjoint tomography and full waveform inversion. *Geophys. J. Int.*, 206(3):1467–1478, 2016. doi: 10.1093/gji/ggw224.
- D. A. Kopriva. Metric identities and the discontinuous spectral element method on curvilinear meshes. *Journal of Scientific Computing*, 26(3):301–327, 2006. doi: 10.1007/s10915-005-9070-8.
- D. A. Kopriva, S. L. Woodruff, and M. Y. Hussaini. Computation of electromagnetic scattering with a non-conforming discontinuous spectral element method. *Int. J. Numer. Methods Eng.*, 53(1):105–122, 2002. doi: 10.1002/nme.394.
- S. Krishnan, C. Ji, D. Komatitsch, and J. Tromp. Case studies of damage to tall steel moment-frame buildings in Southern California during large San Andreas earthquakes. *Bull. seism. Soc. Am.*, 96(4A):1523–1537, 2006a. doi: 10.1785/0120050145.
- S. Krishnan, C. Ji, D. Komatitsch, and J. Tromp. Performance of two 18-story steel moment-frame buildings in Southern California during two large simulated San Andreas earthquakes. *Earthquake Spectra*, 22(4):1035–1061, 2006b. doi: 10.1193/1.2360698.
- P. Lailly. The seismic inverse problem as a sequence of before-stack migrations. In J. B. Bednar, R. Redner, E. Robinson, and A. Weglein, editors, *Proceedings of the Conference on Inverse Scattering, Theory and Application Expanded Abstracts*, pages 206–220, Philadelphia, PA, USA, 1983. Society of Industrial and Applied Mathematics.
- S. J. Lee, H. W. Chen, Q. Liu, D. Komatitsch, B. S. Huang, and J. Tromp. Three-dimensional simulations of seismic wave propagation in the Taipei basin with realistic topography based upon the spectral-element method. *Bull. seism. Soc. Am.*, 98(1):253–264, 2008. doi: 10.1785/0120070033.
- S. J. Lee, Y. C. Chan, D. Komatitsch, B. S. Huang, and J. Tromp. Effects of realistic surface topography on seismic ground motion in the Yangminshan region of Taiwan based upon the spectral-element method and LiDAR DTM. *Bull. seism. Soc. Am.*, 99(2A):681–693, 2009a. doi: 10.1785/0120080264.
- S. J. Lee, D. Komatitsch, B. S. Huang, and J. Tromp. Effects of topography on seismic wave propagation: An example from northern Taiwan. *Bull. seism. Soc. Am.*, 99(1):314–325, 2009b. doi: 10.1785/0120080020.
- A. Legay, H. W. Wang, and T. Belytschko. Strong and weak arbitrary discontinuities in spectral finite elements. *Int. J. Numer. Methods Eng.*, 64(8):991–1008, 2005. doi: 10.1002/nme.1388.

- P. Lesaint and P. A. Raviart. On a finite element method for solving the neutron transport equation (Proc. Symposium, Mathematical Research Center). In U. of Wisconsin-Madison, editor, *Mathematical aspects of finite elements in partial differential equations*, volume 33, pages 89–123, New York, USA, 1974. Academic Press.
- Q. Liu and J. Tromp. Finite-frequency kernels based on adjoint methods. *Bull. seism. Soc. Am.*, 96(6):2383–2397, 2006. doi: 10.1785/0120060041.
- Q. Liu and J. Tromp. Finite-frequency sensitivity kernels for global seismic wave propagation based upon adjoint methods. *Geophys. J. Int.*, 174(1):265–286, 2008. doi: 10.1111/j.1365-246X.2008.03798.x.
- Q. Liu, J. Logan, Y. Tian, H. Abbasi, N. Podhorszki, J. Y. Choi, S. Klasky, R. Tchoua, J. Lofstead, R. Oldfield, M. Parashar, N. Samatova, K. Schwan, A. Shoshani, M. Wolf, K. Wu, and W. Yu. Hello ADIOS: the challenges and lessons of developing leadership class I/O frameworks. *Concurrency and Computation: Practice and Experience*, pages n/a–n/a, Aug. 2013. doi: 10.1002/cpe.3125.
- P. Lovely, J. Shaw, Q. Liu, and J. Tromp. A structural model of the Salton Trough and its implications for seismic hazard. *Bull. seism. Soc. Am.*, 96:1882–1896, 2006.
- Y. Maday and A. T. Patera. Spectral-element methods for the incompressible Navier-Stokes equations. In *State of the art survey in computational mechanics*, pages 71–143, 1989. A. K. Noor and J. T. Oden editors.
- R. Madec, D. Komatitsch, and J. Diaz. Energy-conserving local time stepping based on high-order finite elements for seismic wave propagation across a fluid-solid interface. *Comput. Model. Eng. Sci.*, 49(2):163–189, 2009.
- A. Maggi, C. Tape, M. Chen, D. Chao, and J. Tromp. An automated time-window selection algorithm for seismic tomography. *Geophys. J. Int.*, 178:257–281, 2009.
- R. Martin and D. Komatitsch. An unsplit convolutional perfectly matched layer technique improved at grazing incidence for the viscoelastic wave equation. *Geophys. J. Int.*, 179(1):333–344, 2009. doi: 10.1111/j.1365-246X.2009.04278.x.
- R. Martin, D. Komatitsch, C. Blitz, and N. Le Goff. Simulation of seismic wave propagation in an asteroid based upon an unstructured MPI spectral-element method: blocking and non-blocking communication strategies. *Lecture Notes in Computer Science*, 5336:350–363, 2008a.
- R. Martin, D. Komatitsch, and A. Ezziani. An unsplit convolutional perfectly matched layer improved at grazing incidence for seismic wave equation in poroelastic media. *Geophysics*, 73(4):T51–T61, 2008b. doi: 10.1190/1.2939484.
- R. Martin, D. Komatitsch, and S. D. Gedney. A variational formulation of a stabilized unsplit convolutional perfectly matched layer for the isotropic or anisotropic seismic wave equation. *Comput. Model. Eng. Sci.*, 37(3):274–304, 2008c. doi: 10.3970/cmes.2008.037.274.
- R. Martin, D. Komatitsch, S. D. Gedney, and E. Bruthiaux. A high-order time and space formulation of the unsplit perfectly matched layer for the seismic wave equation using Auxiliary Differential Equations (ADE-PML). *Comput. Model. Eng. Sci.*, 56(1):17–42, 2010. doi: 10.3970/cmes.2010.056.017.
- T. Melvin, A. Staniforth, and J. Thuburn. Dispersion analysis of the spectral-element method. *Quarterly Journal of the Royal Meteorological Society*, 138(668):1934–1947, 2012. doi: 10.1002/qj.1906.
- E. D. Mercerat, J. P. Vilotte, and F. J. Sánchez-Sesma. Triangular spectral-element simulation of two-dimensional elastic wave propagation using unstructured triangular grids. *Geophys. J. Int.*, 166(2):679–698, 2006.
- D. Michéa and D. Komatitsch. Accelerating a 3D finite-difference wave propagation code using GPU graphics cards. *Geophys. J. Int.*, 182(1):389–402, 2010. doi: 10.1111/j.1365-246X.2010.04616.x.
- P. Monk and G. R. Richter. A discontinuous Galerkin method for linear symmetric hyperbolic systems in inhomogeneous media. *Journal of Scientific Computing*, 22-23(1-3):443–477, 2005. doi: 10.1007/s10915-004-4132-5.

- V. Monteiller, S. Chevrot, D. Komatitsch, and Y. Wang. Three-dimensional full waveform inversion of short-period teleseismic wavefields based upon the SEM-DSM hybrid method. *Geophys. J. Int.*, 202(2):811–827, 2015. doi: 10.1093/gji/ggv189.
- C. Morency, Y. Luo, and J. Tromp. Finite-frequency kernels for wave propagation in porous media based upon adjoint methods. *Geophys. J. Int.*, 179:1148–1168, 2009. doi: 10.1111/j.1365-246X.2009.04332.
- S. P. Oliveira and G. Seriani. Effect of element distortion on the numerical dispersion of spectral-element methods. *Communications in Computational Physics*, 9(4):937–958, 2011.
- K. B. Olsen, S. M. Day, and C. R. Bradley. Estimation of  $Q$  for long-period ( $>2$  sec) waves in the Los Angeles basin. *Bull. seism. Soc. Am.*, 93(2):627–638, 2003.
- P. S. Pacheco. *Parallel programming with MPI*. Morgan Kaufmann Press, San Francisco, USA, 1997.
- A. T. Patera. A spectral element method for fluid dynamics: laminar flow in a channel expansion. *J. Comput. Phys.*, 54:468–488, 1984.
- F. Pellegrini and J. Roman. SCOTCH: A software package for static mapping by dual recursive bipartitioning of process and architecture graphs. *Lecture Notes in Computer Science*, 1067:493–498, 1996.
- D. Peter, D. Komatitsch, Y. Luo, R. Martin, N. Le Goff, E. Casarotti, P. Le Loher, F. Magnoni, Q. Liu, C. Blitz, T. Nissen-Meyer, P. Basini, and J. Tromp. Forward and adjoint simulations of seismic wave propagation on fully unstructured hexahedral meshes. *Geophys. J. Int.*, 186(2):721–739, 2011. doi: 10.1111/j.1365-246X.2011.05044.x.
- R. E. Plessix. A review of the adjoint-state method for computing the gradient of a functional with geophysical applications. *Geophys. J. Int.*, 167(2):495–503, 2006.
- E. Priolo, J. M. Carcione, and G. Seriani. Numerical simulation of interface waves by high-order spectral modeling techniques. *J. Acoust. Soc. Am.*, 95(2):681–693, 1994.
- W. H. Reed and T. R. Hill. Triangular mesh methods for the neutron transport equation. Technical Report LA-UR-73-479, Los Alamos Scientific Laboratory, Los Alamos, USA, 1973.
- J. Ritsema, L. A. Rivera, D. Komatitsch, J. Tromp, and H. J. van Heijst. The effects of crust and mantle heterogeneity on PP/P and SS/S amplitude ratios. *Geophys. Res. Lett.*, 29(10):1430, 2002. doi: 10.1029/2001GL013831.
- B. Rivière and M. F. Wheeler. Discontinuous finite element methods for acoustic and elastic wave problems. *Contemporary Mathematics*, 329:271–282, 2003.
- F. Rubio Dalmau, M. Hanzich, J. de la Puente, and N. Gutiérrez. Lossy data compression with DCT transforms. In *Proceedings of the EAGE Workshop on High Performance Computing for Upstream*, page HPC30, Chania, Crete, Greece, 2014. doi: 10.3997/2214-4609.20141939.
- B. Savage, D. Komatitsch, and J. Tromp. Effects of 3D attenuation on seismic wave amplitude and phase measurements. *Bull. seism. Soc. Am.*, 100(3):1241–1251, 2010. doi: 10.1785/0120090263.
- G. Seriani and S. P. Oliveira. Optimal blended spectral-element operators for acoustic wave modeling. *Geophysics*, 72(5):SM95–SM106, 2007. doi: 10.1190/1.2750715.
- G. Seriani and S. P. Oliveira. Dispersion analysis of spectral-element methods for elastic wave propagation. *Wave Motion*, 45:729–744, 2008. doi: 10.1016/j.wavemoti.2007.11.007.
- G. Seriani and E. Priolo. A spectral element method for acoustic wave simulation in heterogeneous media. *Finite Elements in Analysis and Design*, 16:337–348, 1994.
- G. Seriani, E. Priolo, and A. Pregarz. Modelling waves in anisotropic media by a spectral element method. In G. Cohen, editor, *Proceedings of the third international conference on mathematical and numerical aspects of wave propagation*, pages 289–298. SIAM, Philadelphia, PA, 1995.

- A. Sieminski, Q. Liu, J. Trampert, and J. Tromp. Finite-frequency sensitivity of body waves to anisotropy based upon adjoint methods. *Geophys. J. Int.*, 171:368–389, 2007. doi: 10.1111/j.1365-246X.2007.03528.x.
- M. Stupazzini, R. Paolucci, and H. Igel. Near-fault earthquake ground-motion simulation in the grenoble valley by a high-performance spectral element code. *Bull. Seismol. Soc. Am.*, 99(1):286–301, 2009.
- M. P. Süss and J. H. Shaw. P wave seismic velocity structure derived from sonic logs and industry reflection data in the Los Angeles basin, California. *J. Geophys. Res.*, 108(B3):2170, 2003. doi: 10.1029/2001JB001628.
- J. Tago, V. M. Cruz-Atienza, V. Étienne, J. Virieux, M. Benjemaa, and F. J. Sánchez-Sesma. 3D dynamic rupture with anelastic wave propagation using an hp-adaptive Discontinuous Galerkin method. In *Abstract S51A-1915 presented at 2010 AGU Fall Meeting*, San Francisco, California, USA, December 2010. www.agu.org/meetings/fm10/waisfm10.html.
- C. Tape, Q. Liu, and J. Tromp. Finite-frequency tomography using adjoint methods - Methodology and examples using membrane surface waves. *Geophys. J. Int.*, 168(3):1105–1129, 2007. doi: 10.1111/j.1365-246X.2006.03191.x.
- C. Tape, Q. Liu, A. Maggi, and J. Tromp. Adjoint tomography of the southern California crust. *Science*, 325:988–992, 2009.
- C. Tape, Q. Liu, A. Maggi, and J. Tromp. Seismic tomography of the southern California crust based on spectral-element and adjoint methods. *Geophys. J. Int.*, 180:433–462, 2010.
- A. Tarantola. Inversion of seismic reflection data in the acoustic approximation. *Geophysics*, 49:1259–1266, 1984.
- M. A. Taylor and B. A. Wingate. A generalized diagonal mass matrix spectral element method for non-quadrilateral elements. *Appl. Num. Math.*, 33:259–265, 2000.
- S. A. Teukolsky. Short note on the mass matrix for Gauss-Lobatto grid points. *J. Comput. Phys.*, 283:408–413, 2015. doi: 10.1016/j.jcp.2014.12.012.
- J. Tromp and D. Komatitsch. Spectral-element simulations of wave propagation in a laterally homogeneous Earth model. In E. Boschi, G. Ekström, and A. Morelli, editors, *Problems in Geophysics for the New Millennium*, pages 351–372. INGV, Roma, Italy, 2000.
- J. Tromp, C. Tape, and Q. Liu. Seismic tomography, adjoint methods, time reversal and banana-doughnut kernels. *Geophys. J. Int.*, 160(1):195–216, 2005. doi: 10.1111/j.1365-246X.2004.02453.x.
- J. Tromp, D. Komatitsch, and Q. Liu. Spectral-element and adjoint methods in seismology. *Communications in Computational Physics*, 3(1):1–32, 2008.
- J. Tromp, D. Komatitsch, V. Hjörleifsdóttir, Q. Liu, H. Zhu, D. Peter, E. Bozdağ, D. McRitchie, P. Friberg, C. Trabant, and A. Hutko. Near real-time simulations of global CMT earthquakes. *Geophys. J. Int.*, 183(1):381–389, 2010a. doi: 10.1111/j.1365-246X.2010.04734.x.
- J. Tromp, Y. Luo, S. Hanasoge, and D. Peter. Noise cross-correlation sensitivity kernels. *Geophys. J. Int.*, 183: 791–819, 2010b. doi: 10.1111/j.1365-246X.2010.04721.x.
- S. Tsuboi, D. Komatitsch, C. Ji, and J. Tromp. Broadband modeling of the 2002 Denali fault earthquake on the Earth Simulator. *Phys. Earth planet. Inter.*, 139(3-4):305–313, 2003. doi: 10.1016/j.pepi.2003.09.012.
- R. Vai, J. M. Castillo-Covarrubias, F. J. Sánchez-Sesma, D. Komatitsch, and J. P. Vilotte. Elastic wave propagation in an irregularly layered medium. *Soil Dynamics and Earthquake Engineering*, 18(1):11–18, 1999. doi: 10.1016/S0267-7261(98)00027-X.
- K. van Wijk, D. Komatitsch, J. A. Scales, and J. Tromp. Analysis of strong scattering at the micro-scale. *J. Acoust. Soc. Am.*, 115(3):1006–1011, 2004. doi: 10.1121/1.1647480.
- J. Virieux and S. Operto. An overview of full-waveform inversion in exploration geophysics. *Geophysics*, 74(6): WCC1–WCC26, 2009. doi: 10.1190/1.3238367.

- D. J. Wald and T. H. Heaton. Spatial and temporal distribution of slip for the 1992 Landers, California earthquake. *Bull. seism. Soc. Am.*, 84:668–691, 1994.
- L. C. Wilcox, G. Stadler, C. Burstedde, and O. Ghattas. A high-order discontinuous Galerkin method for wave propagation through coupled elastic-acoustic media. *J. Comput. Phys.*, 229(24):9373–9396, 2010. doi: 10.1016/j.jcp.2010.09.008.
- B. A. Wingate and J. P. Boyd. Spectral element methods on triangles for geophysical fluid dynamics problems. In A. V. Il'in and L. R. Scott, editors, *Proceedings of the Third International Conference on Spectral and High-order Methods*, pages 305–314, Houston, Texas, 1996. Houston J. Mathematics.
- Z. Xie, D. Komatitsch, R. Martin, and R. Matzen. Improved forward wave propagation and adjoint-based sensitivity kernel calculations using a numerically stable finite-element PML. *Geophys. J. Int.*, 198(3):1714–1747, 2014. doi: 10.1093/gji/ggu219.
- Z. Xie, R. Matzen, P. Cristini, D. Komatitsch, and R. Martin. A perfectly matched layer for fluid-solid problems: Application to ocean-acoustics simulations with solid ocean bottoms. *J. Acoust. Soc. Am.*, 140(1):165–175, 2016. doi: 10.1121/1.4954736.
- L. Zhu and H. Kanamori. Moho depth variation in southern California from teleseismic receiver functions. *J. Geophys. Res.*, 105:2969–2980, 2000.

## Appendix A

# Reference Frame Convention

The code uses the following convention for the Cartesian reference frame:

- the  $x$  axis points East
- the  $y$  axis points North
- the  $z$  axis points up

Note that this convention is different from both the Aki and Richards [1980] convention and the Harvard Centroid-Moment Tensor (CMT) convention. The Aki & Richards convention is

- the  $x$  axis points North
- the  $y$  axis points East
- the  $z$  axis points down

and the Harvard CMT convention is

- the  $x$  axis points South
- the  $y$  axis points East
- the  $z$  axis points up

## Source and receiver locations

The SPECFEM3D Cartesian software code internally uses Cartesian coordinates. The given locations for sources and receiver locations thus may get converted. Sources and receiver locations are read in from the CMTSOLUTION (or FORCESOLUTION) and STATIONS files. Note that e.g. the CMTSOLUTION and FORCESOLUTION files denotes the location by "longitude/latitude/depth". We read in longitude as  $x$  coordinate, latitude as  $y$  coordinate.

In case the flag SUPPRESS\_UTM\_PROJECTION is set to `.false.` in the main parameter file (see Chapter 4), the  $x/y$  coordinates have to be given in degrees and are converted to Cartesian coordinates using a UTM conversion with the specified UTM zone.

The value for depth (given in  $km$  in CMTSOLUTION and FORCESOLUTION) or burial depth (given in  $m$  in STATIONS) is evaluated with respect to the surface of the mesh at the specified  $x/y$  location to find the corresponding  $z$  coordinate. It is possible to use this depth value directly as  $z$  coordinate by changing the flag USE\_SOURCES\_RECVS\_Z to `.true.` in the file `constants.h` located in the `src/shared/` subdirectory.

## Seismogram outputs

The seismogram output directions are given in Cartesian  $x/y/z$  directions and not rotated any further. Changing flags in `constants.h` in the `src/shared/` subdirectory only rotates the seismogram outputs if receivers are forced to be located at the surface (`RECVS_CAN_BE_BURIED_EXT_MESH` set to `.false.`) and the normal to the surface at the receiver location should be used (`EXT_MESH_RECV_NORMAL` set to `.true.`) as vertical. In this case, the outputs are rotated to have the vertical component normal to the surface of the mesh,  $x$  and  $y$  directions are somewhat arbitrary orthogonal directions along the surface.

For the labeling of the seismogram channels, see Appendix B. Additionally, we add labels to the synthetics using the following convention: For a receiver station located in an

### **elastic domain:**

- `semd` for the displacement vector
- `semv` for the velocity vector
- `sema` for the acceleration vector

### **acoustic domain:**

(please note that receiver stations in acoustic domains must be buried. This is due to the free surface condition which enforces a zero pressure at the surface.)

- `semd` for the displacement vector
- `semv` for the velocity vector
- `sema` for pressure which will be stored in the vertical component  $z$ . Note that pressure in the acoustic media is isotropic and thus the pressure record would be the same in the other two component directions. We therefore use the other two seismogram components to store the acoustic potential in component  $X$  (or  $N$ ) and the first time derivative of the acoustic potential in component  $Y$  (or  $E$ ).

The seismograms are by default written out in ASCII-format to the `OUTPUT_FILES/` subdirectory by each parallel process. You can change this behavior by changing the following flags in the `constants.h` file located in the `src/shared/` subdirectory:

**SEISMOGRAMS\_BINARY** Set to `.true.` to have seismograms written out in binary format.

**WRITE\_SEISMOGRAMS\_BY\_MASTER** Set to `.true.` to have only the master process writing out seismograms. This can be useful on a cluster, where only the master process node has access to the output directory.

**USE\_OUTPUT\_FILES\_PATH** Set to `.false.` to have the seismograms output to `LOCAL_PATH` directory specified in the main parameter file `DATA/Par_file`.

## Appendix B

# Channel Codes of Seismograms

Seismic networks, such as the Global Seismographic Network (GSN), generally involve various types of instruments with different bandwidths, sampling properties and component configurations. There are standards to name channel codes depending on instrument properties. IRIS (<http://www.iris.edu>) uses SEED format for channel codes, which are represented by three letters, such as LHN, BHZ, etc. In older versions of the SPECFEM3D Cartesian package, a common format was used for the channel codes of all seismograms, which was BHE/BHN/BHZ for three components. To avoid confusion when comparison are made to observed data, we are now using the FDSN convention (<http://www.fdsn.org/>) for SEM seismograms. In the following, we give a brief explanation of the FDSN convention used by IRIS, and how it is adopted in SEM seismograms. Please visit [http://www.iris.edu/manuals/SEED\\_appA.htm](http://www.iris.edu/manuals/SEED_appA.htm) for further information.

**Band code:** The first letter in the channel code denotes the band code of seismograms, which depends on the response band and the sampling rate of instruments. The list of band codes used by IRIS is shown in Figure B.1. The sampling rate of SEM synthetics is controlled by the resolution of simulations rather than instrument properties. However, for consistency, we follow the FDSN convention for SEM seismograms governed by their sampling rate. For SEM synthetics, we consider band codes for which  $dt \leq 1$  s. IRIS also considers the response band of instruments. For instance, short-period and broad-band seismograms with the same sampling rate correspond to different band codes, such as S and B, respectively. In such cases, we consider SEM seismograms as broad band, ignoring the corner period ( $\geq 10$  s) of the response band of instruments (note that at these resolutions, the minimum period in the SEM synthetics will be less than 10 s). Accordingly, when you run a simulation the band code will be chosen depending on the resolution of the synthetics, and channel codes of SEM seismograms will start with either L, M, B, H, C or F, shown by red color in the figure.

**Instrument code:** The second letter in the channel code corresponds to instrument codes, which specify the family to which the sensor belongs. For instance, H and L are used for high-gain and low-gain seismometers, respectively. The instrument code of SEM seismograms will always be X, as assigned by FDSN for synthetic seismograms.

**Orientation code:** The third letter in channel codes is an orientation code, which generally describes the physical configuration of the components of instrument packages. SPECFEM3D Cartesian uses the traditional orientation code E/N/Z (East-West, North-South, Vertical) for three components when a UTM projection is used. If the UTM conversion is suppressed, i.e. the flag `SUPPRESS_UTM_PROJECTION` is set to `.true.`, then the three components are labelled X/Y/Z according to the Cartesian reference frame.

**EXAMPLE:** The sampling rate is given by DT in the main parameter file `Par_file` located in the DATA/ subdirectory. Depending on the resolution of your simulations, if you choose a sampling rate greater than 0.01 s and less than 1 s, a seismogram recording displacements on the vertical component of a station ASBS (network AZ) will be named `AZ.ASBS.MXZ.semd.sac`, whereas it will be `AZ.ASBS.BXZ.semd.sac`, if the sampling rate is greater than 0.0125 and less equal to 0.1 s.

Band code	Band type	Sampling rate (sec)	Corner period (sec)
F	...	> 0.0002 to <= 0.001	$\geq 10 \text{ sec}$
G	...	> 0.0002 to <= 0.001	< 10 sec
D	...	> 0.001 to <= 0.004	< 10 sec
C	...	> 0.001 to <= 0.004	$\geq 10 \text{ sec}$
E	Extremely Short Period	$\leq 0.0125$	< 10 sec
S	Short Period	$\leq 0.1$ to $> 0.0125$	< 10 sec
H	High Broad Band	$\leq 0.0125$	$\geq 10 \text{ sec}$
B	Broad Band	$\leq 0.1$ to $> 0.0125$	$\geq 10 \text{ sec}$
M	Mid Period	$< 1$ to $> 0.1$	
L	Long Period	1	
V	Very Long Period	10	
U	Ultra Long Period	100	
R	Extremely Long Period	1000	
P	On the order of 0.1 to 1 day	$\leq 100000$ to $> 10000$	
T	On the order of 1 to 10 days	$\leq 1000000$ to $> 100000$	
Q	Greater than 10 days	$> 1000000$	
A	Administrative Instrument Channel	variable	NA
O	Opaque Instrument Channel	variable	NA

Figure B.1: The band code convention is based on the sampling rate and the response band of instruments. Please visit [http://www.iris.edu/manuals/SEED\\_appA.htm](http://www.iris.edu/manuals/SEED_appA.htm) for further information. Grey rows show the relative band-code range in SPECFEM3D Cartesian, and the band codes used to name SEM seismograms are denoted in red.

## Appendix C

# Troubleshooting

## FAQ

### **configuration fails:**

Examine the log file 'config.log'. It contains detailed informations. In many cases, the paths to these specific compiler commands F90, CC and MPIF90 will not be correct if './configure' fails.

Please make sure that you have a working installation of a Fortran compiler, a C compiler and an MPI implementation. You should be able to compile this little program code:

```
program main
include 'mpif.h'
integer, parameter :: CUSTOM_MPI_TYPE = MPI_REAL
integer ier
call MPI_INIT(ier)
call MPI_BARRIER(MPI_COMM_WORLD,ier)
call MPI_FINALIZE(ier)
end
```

### **compilation fails stating:**

```
...
obj/program_generate_databases.o: In function 'MAIN__':
program_generate_databases.f90:(.text+0x14): undefined reference to '_gfortran_set_std'
...

```

Make sure you're pointing to the right 'mpif90' wrapper command.

Normally, this message will appear when you are mixing two different Fortran compilers. That is, using e.g. gfortran to compile non-MPI files and mpif90, wrapper provided for e.g. ifort, to compile MPI-files.

fix: e.g. specify > ./configure FC=gfortran MPIF90=/usr/local/openmpi-gfortran/bin/mpif90

**after executing xmshfem3D I've got elements with skewness of 81% percent, what does this mean:** Look at the skewness table printed in the output\_mesher.txt file after executing xmshfem3D for the example given in EXAMPLES/meshfem3D\_examples/simple\_model/:

```
...
histogram of skewness (0. good - 1. bad):
0.000000E+00 - 5.0000001E-02 27648 81.81818 %
5.0000001E-02 - 0.1000000 0 0.0000000E+00 %
...

```

The first line means that you have 27,648 elements with a skewness value between 0 and 0.05 (which means the element is basically not skewed, just plain regular hexahedral element). The total number of elements you have in this mesh is (see in the output\_mesher.txt file a bit further down):

```
...
total number of elements in entire mesh: 33792
...
```

which gives you that:  $27,648 / 33,792 \sim 81.8\%$  of all elements are not skewed, i.e. regular elements. a fantastic value :)

The histogram lists for this mesh also some stronger skewed elements, for example the worst ones belong to:

```
...
0.6000000 - 0.6500000 2048 6.060606 %
...
...
```

about 6 % of all elements have distortions with a skewness value between 0.6 and 0.65. The skewness values give you a hint of how good your mesh is. In an ideal world, you would want to have no distortions, just like the 81% from above. Those elements give you the best approximate values by the GLL quadrature used in the spectral-element method. However, having weakly distorted elements is still fine and the solutions are still accurate enough. So empirically, values up to around 0.7 are tolerable, above that you should consider remeshing...

To give you an idea why some of the elements are distorted, see the following figure C.1 of the mesh you obtain in the example EXAMPLES/meshfem3D\_examples/simple\_model/.

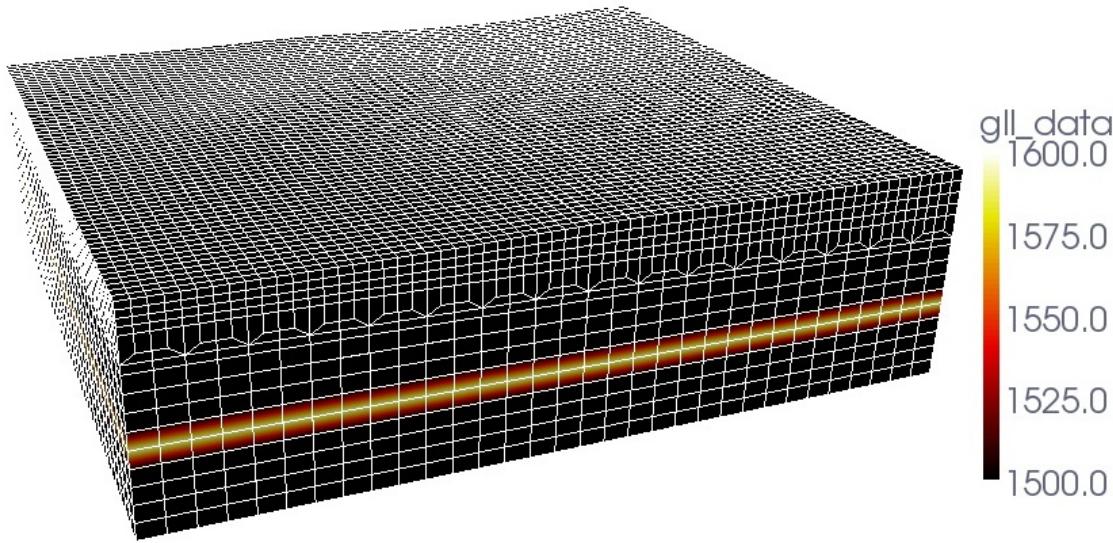


Figure C.1: Paraview visualization using the mesh vtk-files for the example given in EXAMPLES/meshfem3D\_examples/simple\_model/.

You will see that the mesh contains a doubling layer, where we stitch elements together such that the size of two elements will transition to the size of one element (very useful to keep the ratio of wavespeed / element\_size about constant). Those elements in this doubling layer have higher skewness values and make up those 6 % in the histogram.

**the code gives following error message "need at least one receiver":** This means that no stations given in the input file DATA/STATIONS could be located within the dimensions of the mesh. This can happen for example when the mesh was created with the in-house mesher xmshfem3D while using the Universal Transverse Mercator (UTM) projection but the simulation with xspecfem3D was suppressing this projection from latitude/longitude to x/y/z coordinates.

In such cases, try to change your DATA/Par\_file and set e.g.:

```
SUPPRESS_UTM_PROJECTION = .false.
```

to be the same in `Mesh_Par_file` and `Par_file`. This flag should be identical when using the in-house mesher `xmeshfem3D`, `xgenerate_databases` and `xspecfem3D` together to run simulations.

The flag determines if the coordinates you specify for your source and station locations are given as lat/lon degrees and must be converted to UTM coordinates. As an example, if you use `.false.` within `Mesh_Par_file` then you create a mesh with `xmeshfem3D` using the UTM projection from lat/lon as input format to UTM projected coordinates to store the mesh point positions, which is fine. The error then may occur if in the `Par_file` you have this set to `.true.` so that the `xgenerate_databases` and `xspecfem3D` suppress the UTM projection and assume that all coordinates you use now for source and receiver locations are given in meters (that is, converted) already. So it won't find the specified locations in the used mesh. As a solution, just change the flag in `Par_file` to be the same as in `Mesh_Par_file` and rerun `xgenerate_databases` and `xspecfem3D` to make sure that your simulation works fine.

**I get the following error message "forward simulation became unstable and blew up":** In most cases this means that your time step size `DT` is chosen too big. Look at your files `output_mesher.txt` or `output_solver.txt` created in the folder `OUTPUT_FILES`. In these output files, find the section:

```
...
***** Verification of simulation parameters ****
...
*** Minimum period resolved = 4.308774
*** Maximum suggested time step = 6.8863556E-02
...
...
```

then change `DT` in the `DATA/Par_file` to be somewhere close to the maximum suggested time step. In the example above:

`DT = 0.05d0`

would (most probably) work fine. It could be also bigger than the 0.068 s suggested. This depends a bit on the distortions of your mesh elements. The more regular they are, the bigger you can choose `DT`. Just play with this value a bit and see when the simulation becomes stable ...

# Appendix D

## License

**GNU GENERAL PUBLIC LICENSE Version 2, June 1991. Copyright (C) 1989, 1991 Free Software Foundation, Inc. 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA**

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

### Preamble

The licenses for most software are designed to take away your freedom to share and change it. By contrast, the GNU General Public License is intended to guarantee your freedom to share and change free software – to make sure the software is free for all its users. This General Public License applies to most of the Free Software Foundation's software and to any other program whose authors commit to using it. (Some other Free Software Foundation software is covered by the GNU Library General Public License instead.) You can apply it to your programs, too.

When we speak of free software, we are referring to freedom, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for this service if you wish), that you receive source code or can get it if you want it, that you can change the software or use pieces of it in new free programs; and that you know you can do these things.

To protect your rights, we need to make restrictions that forbid anyone to deny you these rights or to ask you to surrender the rights. These restrictions translate to certain responsibilities for you if you distribute copies of the software, or if you modify it.

For example, if you distribute copies of such a program, whether gratis or for a fee, you must give the recipients all the rights that you have. You must make sure that they, too, receive or can get the source code. And you must show them these terms so they know their rights.

We protect your rights with two steps:

1. Copyright the software, and
2. Offer you this license which gives you legal permission to copy, distribute and/or modify the software.

Also, for each author's protection and ours, we want to make certain that everyone understands that there is no warranty for this free software. If the software is modified by someone else and passed on, we want its recipients to know that what they have is not the original, so that any problems introduced by others will not reflect on the original authors' reputations.

Finally, any free program is threatened constantly by software patents. We wish to avoid the danger that redistributors of a free program will individually obtain patent licenses, in effect making the program proprietary. To prevent this, we have made it clear that any patent must be licensed for everyone's free use or not licensed at all.

The precise terms and conditions for copying, distribution and modification follow.

## GNU GENERAL PUBLIC LICENSE TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

- 0. This License applies to any program or other work which contains a notice placed by the copyright holder saying it may be distributed under the terms of this General Public License. The “Program” below refers to any such program or work, and a “work based on the Program” means either the Program or any derivative work under copyright law: that is to say, a work containing the Program or a portion of it, either verbatim or with modifications and/or translated into another language. (Hereinafter, translation is included without limitation in the term “modification.”) Each licensee is addressed as “you.”

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running the Program is not restricted, and the output from the Program is covered only if its contents constitute a work based on the Program (independent of having been made by running the Program). Whether that is true depends on what the Program does.

1. You may copy and distribute verbatim copies of the Program’s source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and give any other recipients of the Program a copy of this License along with the Program.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Program or any portion of it, thus forming a work based on the Program, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

- (a) You must cause the modified files to carry prominent notices stating that you changed the files and the date of any change.
- (b) You must cause any work that you distribute or publish, that in whole or in part contains or is derived from the Program or any part thereof, to be licensed as a whole at no charge to all third parties under the terms of this License.
- (c) If the modified program normally reads commands interactively when run, you must cause it, when started running for such interactive use in the most ordinary way, to print or display an announcement including an appropriate copyright notice and a notice that there is no warranty (or else, saying that you provide a warranty) and that users may redistribute the program under these conditions, and telling the user how to view a copy of this License. (Exception: if the Program itself is interactive but does not normally print such an announcement, your work based on the Program is not required to print an announcement.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Program, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Program, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Program.

In addition, mere aggregation of another work not based on the Program with the Program (or with a work based on the Program) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may copy and distribute the Program (or a work based on it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you also do one of the following:

- (a) Accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
- (b) Accompany it with a written offer, valid for at least three years, to give any third party, for a charge no more than your cost of physically performing source distribution, a complete machine-readable copy of the corresponding source code, to be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
- (c) Accompany it with the information you received as to the offer to distribute corresponding source code. (This alternative is allowed only for noncommercial distribution and only if you received the program in object code or executable form with such an offer, in accord with Subsection b above.)

The source code for a work means the preferred form of the work for making modifications to it. For an executable work, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the executable. However, as a special exception, the source code distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

If distribution of executable or object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place counts as distribution of the source code, even though third parties are not compelled to copy the source along with the object code.

4. You may not copy, modify, sublicense, or distribute the Program except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense or distribute the Program is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.
5. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Program or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Program (or any work based on the Program), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Program or works based on it.
6. Each time you redistribute the Program (or any work based on the Program), the recipient automatically receives a license from the original licensor to copy, distribute or modify the Program subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties to this License.
7. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Program at all. For example, if a patent license would not permit royalty-free redistribution of the Program by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Program.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system, which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

8. If the distribution and/or use of the Program is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Program under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.
9. The Free Software Foundation may publish revised and/or new versions of the General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Program specifies a version number of this License which applies to it and “any later version,” you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of this License, you may choose any version ever published by the Free Software Foundation.

10. If you wish to incorporate parts of the Program into other free programs whose distribution conditions are different, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

## NO WARRANTY

- 11.BECAUSE THE PROGRAM IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM “AS IS” WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.
- 12.IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

## END OF TERMS AND CONDITIONS

### How to Apply These Terms to Your New Programs

If you develop a new program, and you want it to be of the greatest possible use to the public, the best way to achieve this is to make it free software which everyone can redistribute and change under these terms.

To do so, attach the following notices to the program. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the “copyright” line and a pointer to where the full notice is found. For example:

One line to give the program’s name and a brief idea of what it does. Copyright © (year) (name of author)  
This program is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.

This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License along with this program; if not, write to the Free Software Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA

Also add information on how to contact you by electronic and paper mail.

If the program is interactive, make it output a short notice like this when it starts in an interactive mode:

Gnomovision version 69, Copyright © year name of author Gnomovision comes with ABSOLUTELY NO WARRANTY; for details type ‘show w’. This is free software, and you are welcome to redistribute it under certain conditions; type ‘show c’ for details.

The hypothetical commands ‘show w’ and ‘show c’ should show the appropriate parts of the General Public License. Of course, the commands you use may be called something other than ‘show w’ and ‘show c’; they could even be mouse-clicks or menu items – whatever suits your program.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a “copyright disclaimer” for the program, if necessary. Here is a sample; alter the names:

Yoyodyne, Inc., hereby disclaims all copyright interest in the program ‘Gnomovision’ (which makes passes at compilers) written by James Hacker.

(signature of Ty Coon)

1 April 1989

Ty Coon, President of Vice

This General Public License does not permit incorporating your program into proprietary programs. If your program is a subroutine library, you may consider it more useful to permit linking proprietary applications with the library. If this is what you want to do, use the GNU Library General Public License instead of this License.