

# Rayleigh

## Notes on Building and Running the Code

Nick Featherstone  
feathern@gmail.com

### 1. Required Libraries

Rayleigh requires that a few external libraries are installed before you can compile it:

1. BLAS
2. LAPack
3. FFTW3
4. MPI

If you are using Intel's 2012 (or later) compiler, the Math Kernel Library (MKL) is included and provides interfaces to FFTW, LAPack, and BLAS routines. Examples of MKL and non-MKL compiles are included in the Makefiles directory. Specifically, Makefile\_Nick provides an example for linking items 1-3 and compiling with gfortran. Makefile\_LCD, Makefile\_Pleiades, and Makefile\_Discover illustrate compilation using modern Ifort and MKL.

### 2. Building the code

Before building Rayleigh, you will likely need to copy one of the existing makefiles and modify it for your own machine. Each makefile refers to another file called `object_list`. That file should not be modified and is intended to be machine-independent. Place the new makefile in the Makefiles directory, and name it `Makefile_machinename`, where `machinename` indicates the machine you are using.

Once you have a makefile, building the code is straightforward. Within the `rayleigh_dev` directory, you will find an executable script named *build\_rayleigh*. To run this script, type

```
./build_rayleigh machinename
```

This build script copies all relevant source files to the build directory, along with the relevant Makefile from the *Makefiles* directory. Each Makefile is tagged with a machine name – the same machine name you would use in the command above. For example, to build the code on my personal laptop, I type

```
./build_rayleigh Nick
```

so that `Makefile_Nick` is copied to the build directory. To build the code on Pleiades, I type:

```
./build_rayleigh Pleiades ,
```

and `Makefile_Pleiades` is copied to the build directory. Following these copies, the script executes a “make rayleigh” command, and an executable named *rayleigh* is built within the build directory.

`Build_rayleigh` also accepts three optional arguments following the machine name. Each of these is assigned to an environment variable named `RAYLEIGH_OPT1`, `RAYLEIGH_OPT2`, and `RAYLEIGH_OPT3`. These variables may be seen by the Makefile and can be used to control the flow

of the makefile (using debugging flags instead of optimized flags, for instance).

As an example, running  
`./build_rayleigh Pleiades debug`

would compile the code with Intel's debugging flags (see `Makefile_Pleiades`).

**Note:** Depending on how your environment is set up, you might run into issues with the build script trying to use the Makefile. If that's the case, run the `build_script` (so that it copies the source files over), and then just move to the build directory and make the code directly from the command line (*make rayleigh*).

### 3. Preparing to run the Code

Each simulation run using Rayleigh should have its own directory. The code is run from within that directory, and code output is stored in various subdirectories. This means that, wherever you create your simulation directory, you need to have sufficient space (/nobackupp on Pleiades, for example) to store the output.

After you create your run directory, you will want to copy (`cp`) or softlink (`ln -s`) the executable from `rayleigh_dev/build` to your run directory. I like to soft-link, so that the executable is always up-to-date. Next, you should copy the script named `make_dirs` to your run directory. Execute this script from within your run directory. This will create the various subdirectories needed by the code (will discuss what these are later).

Finally, you need a `main_input` file. This file contains all the information that describes how your simulation is run. Rayleigh always looks for a file named `main_input` in the directory that it is launched from. For now, copy the `main_input_bench` file from the `input_examples/anelastic_benchmarks` directory to your run directory. Rename it to `main_input`. This input file is setup to run a Boussinesq dynamo benchmark (see section 6 for details).

### 4. Running the Code

We will examine the `main_input` file in detail shortly but, first, we want to just run the code. While an OpenMP/MPI hybridization is built into the code, it hasn't been thoroughly tested yet. Thus, Rayleigh is effectively parallelized using pure MPI and a 2-D domain decomposition. The pure MPI bit means that if you have any OpenMP environment variables set for FFTW, LAPack, or BLAS, you should probably disable them so as not to inadvertently trip any OpenMP flags used by the external libraries mentioned in section 1. The 2-D domain decomposition bit means that we envision the MPI Ranks as being distributed in rows and columns. The number of MPI ranks within a row is *nprow* and the number of MPI ranks within a column is *npcol*. When Rayleigh is run with N MPI ranks, the following relation must be satisfied:

$$N = npcol \times nprow$$

If this relation is not satisfied, the code will complain and exit.

If you look at the `main_input` file, you will see that it is divided into Fortran namelists. The first namelist is `problemsize_namelist`. Within this namelist, you will see a place to specify *nprow* and *npcol*. Edit the makefile so that *nprow* and *npcol* agree with the N you intend to use. The O(0) effect

on scalability is the number of messages sent per iteration. This means that nprow and npcol should be as close to one another in value as possible. The  $O(1)$  effect is cache latency/cache bandwidth. This means that if nprow and npcol cannot be equal, then nprow should be largest. As nprow increases, fewer Legendre polynomial matrices have to be pulled into/out of memory during each iteration. In summary:

1.  $N = \text{nrow} \times \text{ncol}$ .
2. nprow and npcol should be equal.
3. If they cannot be equal, increase nprow first.

Once you have picked  $N$ , nprow, and npcol, you are ready to run the code. This will probably be using either mpiexec or mpirun. To run the code, you just type (or put in your job script):

```
mpiexec -np N ./rayleigh
```

Alternatively, you can specify nprow and npcol at run time (as opposed to main\_input). This is useful because you can modify the job script without modifying the main\_input file. In this case, suppose  $X$  and  $Y$  were the values for nprow and npcol, then you would type/insert into your jobscript:

```
mpiexec -np N ./rayleigh -nrow X -ncol Y
```

$X$  and  $Y$  overwrite whatever values of nprow and npcol were specified in main\_input.

If everything worked, you should see the code run for 10 time steps and print some timing information before exiting.

## 5. MPI Environment Variables

It is usually a good idea to set the following variables in your MPI job scripts – otherwise you may not have enough buffer space set aside for MPI. This can cause slowdowns and/or hangups as MPI tries to allocated sufficient buffer space on the fly.

```
setenv MPI_BUFS_PER_PROC 64  
setenv MPI_BUFS_PER_HOST 256
```

## 6. The main\_input File

Rayleigh simulations are controlled by the main\_input file. We'll go over a few of the important options in that file here. At the moment, Rayleigh can be run in a non-dimensional Boussinesq mode, or in a dimensional anelastic mode. There is currently no non-dimensional anelastic mode, though this is on the “to-do” list. In dimensional mode, inputs are provided dimensionally. If you have worked with ASH before, this setup will be very familiar to you.

An example of each type of run is provided:

```
main_input_sun (non-magnetic, anelastic)  
main_input_bench (magnetic, Boussinesq)
```

Main\_input\_bench is set up to run the benchmark (case 1) from Christensen et al. , 2001, Physics of Earth and Planetary Interiors, 128, 25. You can set magnetism = .false. In this file to run case 0 (hydro

benchmark) from that paper. Main\_input\_sun is setup to run the simulation used to generate data for the violet curve of Figure 5 in Greer, et al. , 2015, ApJ, 803, L17.

#### problemsize namelist

Rmin and rmax specify the inner and outer radius of the spherical shell (rmin should never be zero). N\_r and n\_theta specify the number of radial and latitudinal colocation points.

*N\_r and n\_theta should always be even. You may have incorrect results/crashes if they are not. This is not a bug. The transforms exploit parity, and so assume that N\_r and n\_theta are even.*

#### numerical\_controls namelist

Chebyshev set to .true. Set this to .false. to use finite-differences in radius.

#### physical\_controls namelist

Rotation and/or magnetism can be switched on and off here.

#### temporal\_controls namelist

max\_time\_step -- The time step taken is controlled by the CFL condition. Set this to enforce an even more stringent time step. time\_step = min(CFL delta\_t, max\_time\_step).

max\_iterations – the maximum number of time steps for which you want to integrate the simulation (usually set this to something big).

check\_frequency – Number of time steps to take between checkpoint dumps.

#### io\_controls namelist

The main input file needs this namelist, but it can be empty. When Rayleigh runs, most messages are written to stdout, and so you typically see a log file at the end of each run. That log file is generated by the supercomputer, which redirects stdout, unless you are in interactive mode. Alternatively, an output file with a name of your choosing can be specified. When this is done, MOST (but not all) messages are written to this output file. Messages are saved in a string buffer and flushed at specified intervals. This prevents the file from being written to during every time step.

stdout\_flush\_interval = 1000 ! This means flush the buffer to disk after 1000 lines are stored  
stdout\_file = 'routput' ! Name of the file to be flushed to

#### output namelist

The output namelist controls governs the output of different diagnostics. Diagnostics are computed in physics/Diagnostics.F90. At the top of that file, you will see a menu of available diagnostics and their corresponding quantity codes. The different diagnostics are computed in the subroutine PS\_Output.

Each diagnostic quantity may be output in 6 different formats in Rayleigh

Shell Slices:	Lat/Lon slices through the simulation of predefined radial indices
Azimuthal Averages:	Radius/Latitude profiles of diagnostics averaged in longitude (phi).

Shell Averages:	Radial profiles of diagnostics averaged over the sphere at each radius
Global Averages:	Diagnostics averaged over the full spherical shell
Shell Spectra:	Spherical harmonic spectra of diagnostics taken at various radial indices
Full 3D:	Full 3-D dumps of the specified diagnostic.

All outputs use MPI-IO and/or a non-blocking send/receive pattern that cascades across rows and down the first column of processors. The only exception is Full-3D output, which has a blocking component to it's communication prior to the MPI-IO calls. This will be fixed in a future release.

All outputs types share three common namelist elements:

output\_values : The quantity codes for the diagnostics that are to be output this way  
 output\_frequency: The number of time steps taken between dumps of this output type  
 output\_nrec: The number of dumps (individual timesteps) stored in the output file

Shell slices and shell spectra use an additional namelist element:

output\_levels: radial indices (1 through n\_r) of the slices or spectra to be output

Each output-type is stored in a directory (made by make\_dirs) with a similar name. Files are numbered by the time step of the last output in the file. For example, suppose

```
globalavg_frequency = 3
globalavg_nrec = 2
```

Time steps 3 and 6 would be stored in a file named G\_Avgs/00000006 and time steps 9 and 12 would be stored in file named G\_Avgs/00000012

### Boundary Conditions Namelist

The boundary conditions are set up for fixed entropy (T\_top and T\_Bottom) at the upper and lower boundaries. For now, anything that says T or Tvar should be interpreted as the thermal variable, which is entropy if you are running an anelastic simulation. If you're running a Boussinesq system, you can think of tvar as temperature. You can alternatively run with fixed flux at the top. To do so, you have to first turn off fix\_tvar\_top and then turn on fix\_dtdr\_top:

```
fix_tvar_top = .false.
fix_dtdr_top = .true.
```

Stress-free boundaries are the default. To change this, add this line to the boundary\_conditions\_namelist:

```
no_slip_boundaries = .false.
```

### Initial Conditions Namelist

See the comments in main\_input\_bench but two important things. init\_type controls the hydro init and magnetic\_init\_type controls the initialization of the magnetic stream functions.

```
init_type, magnetic_init_type = -1 : restart from checkpoint
init_type, magnetic_init_type = 7 : random thermal and magnetic perturbations
init_type, magnetic_init_type = 1 : case 0 and case 1 from Christensen et al. 2001
init_type = 6 : hydro anelastic benchmark init
restart_iter = 1330000 : If restarting from a checkpoint, the iteration to restart from
temp_amp = 1.0d1 : The approximate amplitude of random thermal perturbations
```

conductive\_profile = .false.

: Set this to .true. To initialize the  $\ell = 0$  thermal variable to have a conductive profile in radius. This is typically used in conjunction with `init_Type = 7`. All other modes are given random thermal perturbations. This is very important when fixing temperature on both boundaries.

#### Test Namelist

Leave this alone

#### Reference Namelist

This controls the setup of the reference state. See comments. You can set up a polytropic atmosphere, a Boussinesq reference state, or read in a custom reference state. This is also where `dimensional = .true.` or `.false.` is set to distinguish between anelastic and Boussinesq runs.

#### Non-dimensional Namelist

Leave this alone. In development.

#### Transport Namelist

This is where the transport coefficients  $\nu$ ,  $\kappa$ , and  $\eta$  are specified. By default, these coefficients are constant in radius. Alternatively, they can be made to vary as a function of density. To change  $\nu$  so that  $\nu \sim 1/\sqrt{\text{density}}$ , you would add:

`nu_type = 2`

`nu_power = -0.5d0`

`nu_top`, `kappa_top`, and `eta_top` determine the value of these coefficients at the outer boundary.

## **7. Reading & Plotting Output**

The `rayleigh_dev` directory contains a subdirectory named `reading_routines`, which in turn contains `idl` and `python` subdirectories. The easiest thing to do is download the `sample_output` tarball from the website, untar it, and copy all the `.py` routines into that directory. There are four examples of plotting that demonstrate how to read in the output:

`plot_spectrum.py`

`plot_shell.py`

`plot_shellavg.py`

`plot_gavg.py`

`plot_azavg.py` is only partially functional at this time..

## **8. Load Balancing: Choosing `nprow` and `npcol`**

When running `rayleigh`, it is important to consider the load balancing. Radial levels are distributed across across the process columns. This means that  $N_r = n \times \text{npcol}$  for some integer  $n$ . Ideal  $n$  and `npcol` would be factors of  $N_r$ , but the code will run even if they are not (but `npcol` must be no greater than  $N_R$ )

Spherical harmonic modes are distributed in low- $m$ , high- $m$  pairs across a process row. This means that, ideally,  $(l_{\text{max}}+1)/2 = m \times \text{nprow}$  for some integer  $m$ .

The relation between  $l_{\max}$  and  $n_{\text{theta}}$  is that  $n_{\text{theta}} = 3/2 \times (l_{\max} + 1)$ .

Here's a concrete example, since maybe that's confusing. Suppose  $N_R = 128$  and  $n_{\text{theta}} = 192$ . This means that  $l_{\max} + 1 = 128$ . If we are running with 256 cores, a good choice is  $\text{nprow} = \text{npcol} = 16$ . If we run with 512 cores, good choices are  $\text{nprow} = 32$ ,  $\text{npcol} = 16$  and  $\text{nprow} = 16$ ,  $\text{npcol} = 32$ . Though the first choice is better.

You can increase  $\text{nprow}$  as high as  $l_{\max} + 1$ , but the load balancing is bad in that case. It's much better to keep  $\text{nprow}$  at  $(l_{\max} + 1)/2$  or less. In practice, the communication time grows to outweigh the workload somewhere around  $1/4$  of the maximum number of cores for all but the largest problems. That's still a lot of cores. You should be able to run this sample case pretty efficiently at 1024 cores, for instance, with  $\text{npcol} = 64$  and  $\text{nprow} = 32$ .