Tutorial I. Installing the LMDZ model

Ionela Musat

Laboratoire de Météorologie Dynamique

LMDZ Training course, December 2018

Tutorial I. Installing and running the LMDZ model

1/ Get source codes: netcdf, IOIPSL, ORCHIDEE, LMDZ

2/ Compile the codes

3/ Run a 1-day bench test

Tutorial I. Installing the LMDZ model

There are 3 ways to install LMDZ.
Right choice depends on the machine you are using and the type of simulation (long, test or development) you run.

1) using the install_lmdz.sh script (⇒ this Tutorial)

- the script will download the source codes needed (IOIPSL, ORCHIDEE, LMDZ) and netcdf library and will compile them
- recommended method for Linux PC and short development or test runs.

2) using modipsl and libIGCM (⇒ IPSL Training course)

- you will need to install one of the configuration defined by modipsl (for example LMDZOR_v6).
- recommended for IDRIS, TGCC, CINES and for long simulations, as it provides tested reference versions and scripts for launching and monitor long simulations.

3) by hand

- get source codes for each component you need (IOIPSL, ORCHIDEE, LMDZ) and link them with the netcdf library installed on your machine.

Tutorial 1. Using install_lmdz.sh – Contents

It will do most of the work for you, using standard shell tools and commands (gcc, wget, gunzip, tar, ...) :

- Download the required codes archives
- Choose adequate compiler options and build a Makefile
- Install ancillary libraries (netcdf, modipsl, IOIPSL)
- Install land surface model ORCHIDEE if needed
- Install LMDZ using makelmdz_fcm (or makelmdz) script
- Run a test bench

Further details on LMDZ version (in French), in particular the main modifications between versions:

`http://www.lmd.jussieu.fr/~lmdz/Distrib/LISMOI.trunk`

Tutorial 1. install_lmdz.sh Options

./install_lmdz.sh **-h**

-v version:	choose a version	**YYYMMDD.trunk**
-r release_nb:	choose a “svn release”	**svn-number/ “last”**
-compiler compiler	gfortran / ifort/ pgf90/ mpif90	**default:gfortran**
-parallel mode:	sequential/mixed parallelism	**none/ mpi_omp**
-d grid_resolution:	choose model grid resolution	**nlon**x**nlat**x**nlev**
-bench:	launch or not a test bench	**1/0**
-name MODEL:	choose model folder name	**LMDZversion-release**
-netcdf PATH:	PATH to an existing netcdf	**netcdfPATH**
-xios	add **with_xios=”y” (need parallel=mpi_omp !)**	
-gprof	compile with **-pg** to enable profiling	
-SCM	install **1D** version automatically	
-opt_makelmdz version	**makelmdz_fcm**/makelmdz	(compile_with_fcm)

Tutorial 1. Download and launch install_lmdz.sh

```
wget http://www.lmd.jussieu.fr/~lmdz/pub/install_lmdz.sh

chmod +x install_lmdz.sh

./install_lmdz.sh -d 32x32x39 -v 20181204.trunk
```

=> modipsl.20181204.trunk.tar.gz

=> bench_lmdz_32x32x39.tar.gz

Tutorial 1. Choosing which LMDZ version to work with

Choose between the different available versions on the LMDZ web site:

`http://www.lmd.jussieu.fr/~lmdz/pub/LISMOI.trunk`

Ask the LMDZ team for more information on which versions are actually used :

lmdz-svp@lmd.jussieu.fr

Tutorial 1. What you need to run the LMDZ GCM (1)

•Executable (LMDZ) file :
- gcm.e

• Parameters files :
run.def, gcm.def, vert.def, physiq.def, traceur.def, config.def, etc

• Start files v:
- start.nc, startphy.nc

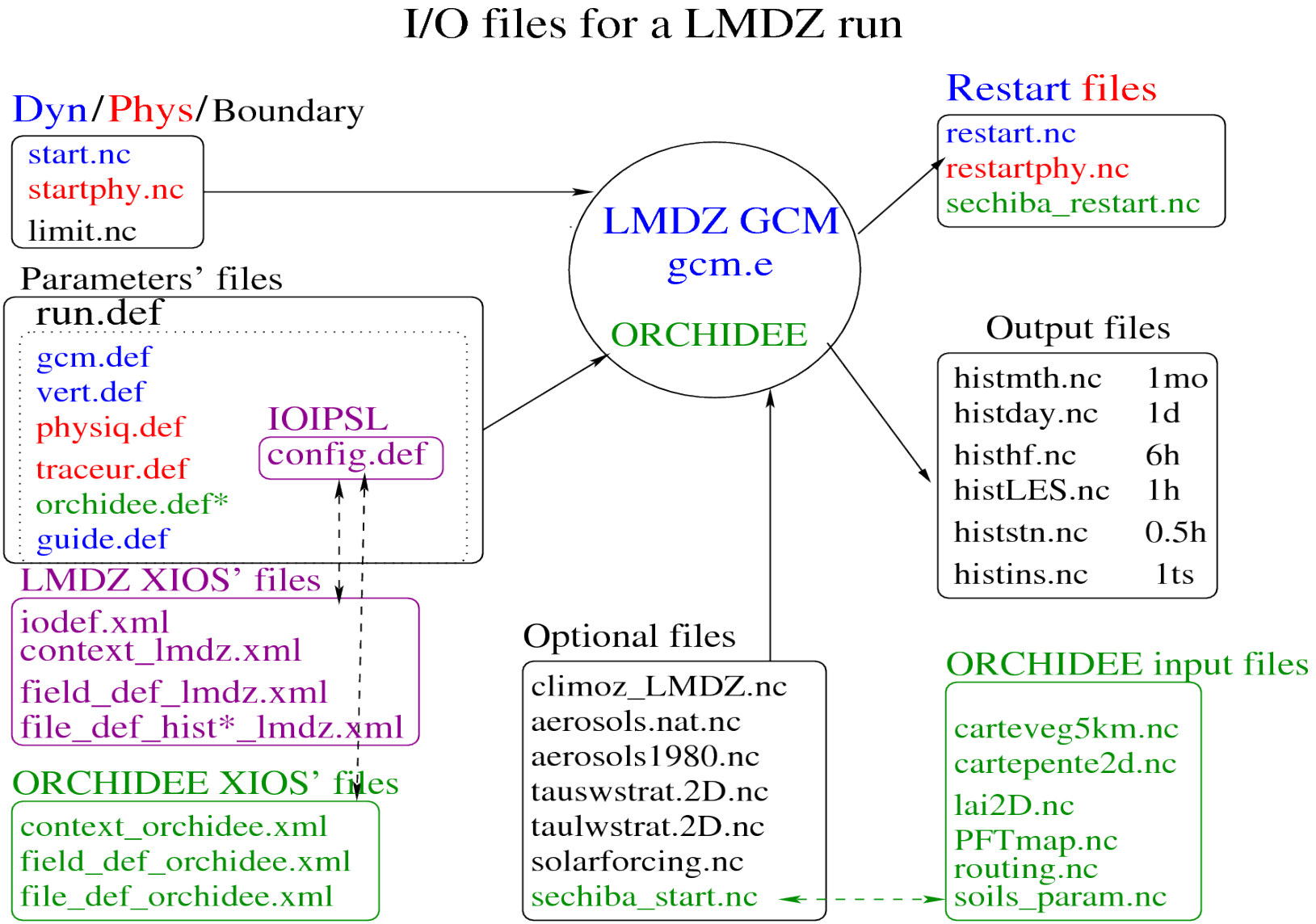
These files are created by the ce0l.e program or may be the result of previous runs

• Boundary conditions file v:
limit.nc
Created by ce0l.e

• Some optional input files v (depending on the simulation) :
- aerosols.nc, climoz_LMDZ.nc, nudging input files (u.nc, v.nc,..), etc

v : these files have to be interpolated on the horizontal grid of the model

Tutorial 1. What you need to run the LMDZ GCM (2)



Tutorial 1. Running the model

```
ls
```

```
gcm.e  start.nc startphy.nc limit.nc config.def gcm.def orchidee.def
physiq.def  run.def  traceur.def  vert.def
```

```
./gcm.e
```

or

```
./gcm.e > lmdz.out 2>&1
```

To carry on a simulation that has been run, you have to copy the restart files obtained at the end of the previous run as new initial start files:

```
mv restart.nc start.nc
mv restartphy.nc startphy.nc

./gcm.e
```

Tutorial 1. Has your run completed successfully ?

YES

► you will then have a message saying ***Everything is cool*** on the standard output or in the output text file.

► The code will have created 2 restart files

restart.nc and restartphy.nc

needed to carry on your run

► and some output diagnostic files

histday.nc, histmth.nc, etc. ...

to explore/view using ferret, grads, ...

Tutorial 1. Has your run completed successfully ?

NO

You must find out what the problem is...

Look for an error message in the output text file.
Search for one of the following key words/phrase: ***Houston, we have a problem***, **STOP**, ***hgardfou***, ***integrd: negative surface pressure,*** etc.

Different typical errors :

- technical problem : a missing input file, an error in one of the *.def file

- problem with the model's stability.

Instability in the physics are likely to be detected by hgardfou, which checks the model temperature has realistic values.

Instability in the dynamics most often end up the run with a ***negative surface pressure*** error message.

► **In any of these cases you will most probably have to adjust some flags in the .def files**.

- you have some source code modifications that might not have been thoroughly tested or validated.

Tutorial 1. Take-off infos

Re-compile LMDZ : use compile.sh

cd ~/LMDZ20181204.trunk/modipsl/modeles/LMDZ
./compile.sh

Re-run a LMDZ simulation: use bench.sh

cd~/LMDZ20181204.trunk/modipsl/moeles/LMDZ
./bench.sh