

# Hands on tutorial #2: Setting up a simulation

LMDZ team

December 13th, 2018

This tutorial focuses on the various steps required to set up a 3D simulation, and in particular for a zoomed configuration of LMDZ.

If you have already installed the model with the script `install_lmdz.sh` in tutorial #1, go directly below to "**Setting up a simulation...**". Otherwise, start by installing the model as follows :

If you don't have a working folder named LMDZ, then you create it :

```
mkdir LMDZ
cd LMDZ
```

then download and run the script `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
```

## Setting up a simulation with a (regular or) zoomed grid

- Go to the directory `LMDZ20181204.trunk/modips1/modeles/LMDZ`, which contains the files `make_lmdz_fcm`, `libf` etc. In this directory, download and unpack the following tar file, then go in the resulting `TUTORIAL` folder :

```
wget http://www.lmd.jussieu.fr/~lmdz/pub/Training/tutorial.tar
tar -xf tutorial.tar
cd TUTORIAL
```

- Examine the content of the `TUTORIAL` folder : there are some scripts and a `DEF` directory, all briefly described in there `Readme` file. In the `DEF` directory, edit the file `gcm.def` and examine the different parameters defining the grid.

By default, the defined grid has a zoom factor = 3 both in longitude and latitude

(`grossismx=3.`, `grossismy=3.`), with the zoomed area centered at (0E,45N) (`c1on=0.`, `c1at=45.`).

In order to place the center of the zoom at your preferred location, you just need to change the longitude and latitude of the zoom center, `c1on` and `c1at`.

If you want to use a regular grid, set `grossismx=1.` and `grossismy=1.` .

- For the time being, you will run LMDZ without the surface scheme Orchidee: in the `init.sh` file, check that you have the option `veget=0`. The model will be run with a simplified scheme for surface hydrology: the "bucket" scheme.
- As you installed the model in sequential mode, in `init.sh` you must also have `parallel=0`.
- Skip this step if you run locally :

You may increase the stack memory you can use by typing the following command:

```
ulimit -Ss unlimited
```

or, even better, add this command in you shell start-up file (if you use Bash, your start-up file can be `.profile` or `.bash_profile`).

- Run the script :

```
./init.sh
```

The script `init.sh` does the following :

- compiles the model (`gcm.e`) again, with a different resolution than before : 48x36x39 ;
- compiles the program `ce01.e`, needed to create initial state and boundary conditions ;
- downloads input files for `ce01.e` (NetCDF files containing surface orography, sea-surface temperature etc.), as well as 3D meteorological files from ECMWF analyses at a particular date, to be used with the nudging option;
- runs `ce01.e` , which creates files `start.nc`, `startphy.nc` and `limit.nc` in a directory called `INITIAL`. These files will be used to initialize a new simulation in a newly-created directory called `SIMU1`.

Please check that these 3 files have been created in the directory `SIMU1`. If not, ask for our help.

- Now edit the file `config.def`, and look at the section "Controle des sorties" (Eng. "Output control"). In the high-frequency NetCDF output file #3, `hsthf.nc`., you'll want to include the variable `pres`, containing the pressure at model levels. To do that, add the following line :

```
flag_pres__00003 =          4
```

- You can now go in the `SIMU1` directory and run the model by executing the command `./gcm.e` (output on screen) or `./gcm.e > listing1` (output in file `listing1`). The simulation should end with the message "Everything is cool" and the output files `histday.nc`, `histmth.nc` and `hsthf.nc` should be created. Make some plots from one of these files.