Hands on tutorial #2: Setting up a simulation

LMDZ team

January 13-15, 2026

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 indicated in Tutorial #1 (Sections 1 and 2).

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

- Go to the directory `LMDZTraining/LMDZseq/modipsl/modeles/LMDZ`, which contains the files `makelmdz_fcm`, `libf` etc. In this directory, download and unpack the `tutorial.tar` file, then go in the resulting `TUTORIAL` folder :

```
# If you are in your LMDZTraining folder :
cd LMDZseq/modipsl/modeles/LMDZ
wget http://lmdz.lmd.jussieu.fr/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 the `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 the zoomed area centered at (0E,45N)

  (`clon=0., clat=45.`) and a zoom factor = 2 both in longitude and latitude :

  (`grossismx=2., grossismy=2. `).

  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, `clon` and `clat`.

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

- Check the following options in `init.sh`, and change the default value if needed :

∗ As you installed the model in sequential mode, you must have `parallel=0`.

∗ The option for radiative code must be `rad=oldrad`.

∗ For the time being, you will run LMDZ **without** the surface scheme Orchidee: you must have the option `veg=NONE` (instead of the default `"CMIP6"`). The model will be run with a simplified scheme for surface hydrology: the “bucket” scheme.

- Increase the stack memory 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 :

  - (re)compiles the model (`gcm.e`) with resolution 48x36x39 (remember that in tutorial 1 you've used 32x32x39);

  - compiles the program `ce0l.e`, needed to create initial state and boundary conditions for the chosen grid;

  - downloads input files for `ce0l.e` (NetCDF files containing surface orography, sea-surface temperature etc.);

  - runs `ce0l.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 `SIMU1/config.def`, and look at the section "Controle des sorties" (Eng. "Output control"). In the high-frequency NetCDF output file #3, `histhf.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
  ```

- make sure that `iflag_rrtm` is set to 0 and that `NSW` is set to 2 in `physiq.def` (`oldrad` mode)

- 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" (on screen, or at the end of `listing1` file), and the output files `histday.nc`, `histmth.nc` and `histhf.nc` should be created. Make some plots from one of these files.