Hands on tutorial #2: Setting up a simulation

LMDZ team

December 8th, 2019

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 20191106.trunk
```

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

- Go to the directory `LMDZ20191106.trunk/modipsl/modeles/LMDZ`, which contains the files `makelmdz_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 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 a zoom factor = 3 both in longitude and latitude (`grossismx=3., grossismy=3.`), with the zoomed area centered at (0E,45N) (`clon=0., clat=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, `clon` and `clat`.

  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 (horizontal) resolution than before : 48x36x39 instead of 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.), as well as 3D meteorological files from ECMWF analyses at a particular date, to be used with the nudging option;

  - 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 `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
  ```
- 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 `histhf.nc` should be created. Make some plots from one of these files.