Hands on tutorial #2: Setting up a simulation

LMDZ team

January 9-11, 2024

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 (same steps as in Tutorial #1) :

If you don't have a working folder named `LMDZ` in your `home` directory, then you create it:

```
cd
mkdir LMDZ
```

Go in this `~/LMDZ` folder. Download the script `install_lmdz.sh`, and run it to install the model in sequential mode (default option "parallel=none" in install_lmdz.sh), in a folder named LMDZseq. The default resolution is 32x32x39 (equivalent to running `install_lmdz.sh` with the option `"-d 32x32x39"`)

```
cd ~/LMDZ
wget https://lmdz.lmd.jussieu.fr/pub/install_lmdz.sh
chmod +x install_lmdz.sh
./install_lmdz.sh -rad oldrad -name LMDZseq
```

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

- Go to the directory `~/LMDZ/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 :

  ```
  # Normally you are in your  ~/LMDZ 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.

- 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 :

  - (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
  ```

- 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.