Hands on tutorial #2: Installing LMDZ in paralel mode ; Setting up and runing a simulation

LMDZ team

November 2022

This tutorial shows the steps for setting up a 3D simulation to be run in parallel mode, in particular for a zoomed configuration of LMDZ.

Installing the model to run in parallel mode

Download and run the script `install_lmdz_par.sh` , prepared for you to install the model with the appropriate options, in a folder named LMDZpar :

```
wget https://lmdz.lmd.jussieu.fr/pub/Training/2022FormationMaroc/install_lmdz_par.sh
chmod +x install_lmdz_par.sh
./install_lmdz_par.sh
```

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

- Go to the directory `LMDZpar/modipsl/modeles/LMDZ` , which contains the files `makelmdz_fcm`, `libf` etc. Download the `tutorial.tar` archive, unpack it, then go in the resulting `TUTORIAL` folder :

  ```
  cd LMDZpar/modipsl/modeles/LMDZ
  wget https://lmdz.lmd.jussieu.fr/pub/Training/2022FormationMaroc/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 = 2 both in longitude and latitude (`grossismx=2.`, `grossismy=2.`), with the zoomed area centered at (0E, 45N) : `clon=0.`, `clat=45.`. If you want to set the center of the zoom at a different location, just change `clon` and `clat`.

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

- You will run LMDZ with the simple 'bucket' surface scheme: in the `init.sh` script, check that you have the option `veget=0`. (Note : `veget=1` activates the coupling with the much more complex ORCHIDEE continental surface model).

- As you installed the model in parallel mode, in `init.sh` you must also have `parallel=1`.

- Run the main script `init.sh` :

  ```
  ./init.sh
  ```

The script `init.sh` does the following :

- (re)compiles the model (`gcm.e`) with resolution 48x36x39 ;

- 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 copied in the newly-created directories `SIMU1` and `PROD0` .

Please check that these 3 files have been created in the directories `INITIAL`, `SIMU1` and `PROD0`. If not, ask for our help.

Running a short simulation (a few days of run)

- You can now go in the SIMU1 directory and run the model in parallel on 2x2 processors : `./run_local.sh 2 2 gcm.e`   .

  By default it will run for 1 day : `nday=1` in `DEF/run.def` .

  The simulation should end with the message "Everything is cool" and the output files `histday.nc`, `histmth.nc` and `histhf.nc` should be created. You can start analyzing the output variables in these files. Note that your `histmth.nc` output file is "empty" ; you need at least 5 days of run (`nday=5`) to get average values in this file.

Running a long simulation (months or years)

- You'll keep PROD0 as a reference, and to start other simulations. Make a copy of PROD0 as MyPROD in which you'll run your simulation, then go in MyPROD :

  ```
  cp -pr PROD0 MyPROD
  cd MyPROD
  ```

- You may want to look at the DEF/*.def files, and possibly adjust them, for ex the output frequency and content. NOTE : you should NOT change to the grid parameters in gcm.def because they must be on the same grid as the start* and limit.nc files you have already produced.

- To run the simulation, you'll use the script `enchaine.sh`. By default, the date of beginning of the run is 1 jan 2000. You can see it in the file `etat` as follows : `200001 afaire` (that is : month 01 of year 2000, to do).

  The model runs automatically month after month. Each individual month is executed in a temporary folder with name starting with "WD".

  The run stops at the end of the month before `stopsim` in `enchaine.sh`. By default `stopsim=200101` , so the model stops at the end of december 2000.

- You may want to start by running for only one month, to check if you have all the output you need. Set `stopsim=200002` in `enchaine.sh` (ligne 29), then launch :

  ```
  ./enchaine.sh &
  ```

- At the end of the month, you'll see in the file `etat` : `200001 fait ; 200002 afaire`
  To continue the simulation, you just change `stopsim` in `enchaine.sh` and launch again   `./enchaine.sh &`