

Tutorial #2 – Zoom

LMDZ team

December 17, 2013

Contents

1	Setting up a case with a zoomed grid	1
2	Modifying parameters	2
3	Nudging	3

This tutorial makes use of zoomed configurations of LMDZ. It includes 1) an initialization phase, 2) an example of use of the model (three versions of the tutorial exist for this part), and 3) a last optional phase in which the model can be nudged toward analyzed wind fields.

If you work on a station of the local LMD network, start by creating a directory at your name, then enter this directory. Install the model with the script `install.sh`, if you have not already done so in tutorial #1.

1 Setting up a case with a zoomed grid

- Go to the directory `LMDZtesting/modips1/modeles/LMDZ5`, which contains files `makegcm`, `libf` ... In this directory, download the following tar file :

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

- In the directory `TUTORIAL`, take a look at the extracted files. You should edit file `gcm.def` if you want to place the center of the zoom at your preferred location. For that you just have to change the longitude and latitude of the zoom center, `clon` and `clat`.
- If you are interested in coupling LMDZ with the surface scheme Orchidee, edit the `init.sh` file and set option `veget=1`. If not (`veget=0`), the model will be run with a simplified bucket scheme for surface hydrology.
- Run the script :

```
./init.sh
```

The script first compiles the model (`gcm.e`) again because it uses a different resolution, $48 \times 36 - L39$, than the one used in `install.sh`. `init.sh` also compiles the program `ce01.e`, which creates initial state and boundary conditions. `init.sh` then downloads new NetCDF files which contain the surface orography, sea-surface temperature, and so on, as well as 3D meteorological files taken from ECMWF analyses at a particular date. `init.sh` then runs `ce01.e` which creates files `start.nc`, `startphy.nc` and `limit.nc`. Check that these files have been actually created. If not, please ask for our help. If you set `veget=1`, the model will also be run automatically for one day in order to create also a start file for Orchidee, which will be called `start_sech.nc`.

- If you now have the files `start.nc`, `startphy.nc` and `limit.nc` (and optionally `start_sech.nc` for `veget=1`), you can run the model by executing the command `./gcm.e`.

Note that the run can fail due to a lack of memory. In this case (usually ending up with a segmentation fault), you can increase available memory this way:

```
ulimit -Ss unlimited
```

For further runs, the length of the simulation can be adjusted using keyword `nday=n` in `run.def`, where `n` is a number of days.

2 Modifying parameters

Take files `restart.nc` and `restartphy.nc` created at the end of the first simulation and use them as start files `start.nc` and `startphy.nc` to prepare a new 5-day simulation.

Prepare 2 identical simulations in 2 directories.

In the first one, you may run the model with the same `.def` files as in the first one.

In the second one, you will modify the `.def` files.

For instance, you could change the boundary layer parameterization by deactivating the Mellor and Yamada scheme for the boundary layer, based on a prognostic equation for the Turbulent Kinetic Energy (TKE), by imposing

```
iflag_pbl=1
```

instead of the nominal value 11 (which corresponds to the variant for very stable planetary boundary layer and exact dissipation). You will so switch to the "Standard Physics" (SP, LMDZ5A).

You may also try to deactivate the thermal plume model:

```
iflag_thermals=0
```

instead of the nominal value 15.

... or any test of the fascinating options available in the LMDZ physics ...

Compare the results.

Remember that you could also modify the content or time frequency of the output files if you want to focus on a particular question.

3 Nudging

- You first have to create the file `grilles_gcm.nc` which contains the longitudes and latitudes of the model grid. To do this, add the line:

```
grilles_gcm_netcdf = TRUE
```

in `run.def` and run again `ce01.e`. You can then plot the orography map as seen by the zoomed grid, by opening file `grilles_gcm.nc` with `ferret` or `grads` and plotting the surface geopotential `phis`. You can also easily plot the horizontal resolution of the model as the square root of the grid mesh area (`aire`).

- Then you have to get the reanalysis files for nudging. You will find the script `get_era.sh` in the directory `TUTORIAL`. Run the script. It will interpolate the winds on the model grid (by reading the model grid from file `grilles_gcm.nc`). You should end up with files `u.nc` and `v.nc` in your current directory. Note that for this tutorial we have given open access to a subset of the ERA-interim wind fields. ERA-interim files are stored at IDRIS, CCRT and Climserv, with restricted access. To access these files at IDRIS or on Climserv, you should contact Sophie Bouffies-Cloch  (IPSL). For access at CCRT, contact Anne Cozic (LSCE). `get_era.sh` is a very simplified script for the tutorial, but more general scripts are available on http://forge.ipsl.jussieu.fr/igcmg/svn/CONFIG/LMDZOR/branches/LMDZOR_v4/CREATE/SCRIPT.
- You can then modify file `guide.def`.

```
ok_guide=y
guide_u= y
guide_v= y
guide_T= n
guide_P= n
guide_Q= n
tau_min_u=0.0208333
tau_max_u=0.125
tau_min_v=0.0208333
tau_max_v=0.125
```

Here, in `guide.def`, nudging is activated for variables `u` and `v` only (as is often the case). The relaxation time is set to 3 hours inside the zoomed area (`tau_max=0.125` days) and half an hour outside (`tau_min=0.0208333` days). The smaller the relaxation, the stronger the nudging.

- Add the line:

```
INCLUDEDEF=guide.def

in run.def.
```

- Without overwriting the old `hist*` files, run the model again with nudging:

`./gcm.e`

- Compare the results of the simulations with and without nudging.