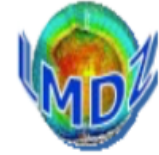


LMDZ Tutorial 2 Presentation



Tutorial 2 has a mandatory part and then a choice of exercises.
Could be a starting point for discussion of your particular projects ...

Mandatory part :

https://lmdz.lmd.jussieu.fr/pub/Training/Tutorials/Tutorial_2.pdf

explains how to setup your own experiment, create start files and boundary files, setup a zoom, ...

Exercises :

Choice of : 1D, parallel, XIOS, nudging, aerosols, tracers,
ORCHIDEE, ICOLMDZ

Setting up a simulation : prerequisites

A) Download and compile model(s) and IO libraries

A.1) Download LMDZ (+ models coupled to LMDZ) + IO libraries (IOIPSL, XIOS)

in a dedicated folder : usually *modipsl/modeles*

A.2) Compile the IO libraries once, in the beginning

A.3) Compile the model(s) with the options of your choice:

IO treatment, parallel/sequential mode, resolution, radiative code...) => [gcm.e](#)

B) Get/create initial and boundary files [start*.nc](#) and [limit.nc](#) for **YOUR** grid

(**grid= horizontal resolution (+ zoom parameters) + vertical grid**)

- generally available for regular grids at standard resolutions,

- to be produced for non-standard grids by compiling and running [ce0l.e](#)

C) Get forcing files

some need **interpolation on YOUR grid** (Ex : aerosols, ...)

some others don't (Ex : GHG=global means)

D) Get additional files if needed : ex. : for nudging

(and guess what...?) **interpolate them on YOUR grid !**

Setting up a simulation : general steps

1) Create a simulation folder

(installation scripts may automatically create a 1st one : BENCH* or TUTORIAL/SIMU1)

2) Copy in it the *.def files

Copy or create links to :

-> **start*.nc** and **limit.nc** for the chosen grid (resolution, zoom)

-> other files for forcing, nudging (for the chosen grid)...

-> executable **gcm.e** compiled for the chosen resolution

-> scripts for post-processing (ex : combining output files from run in parallel mode)

3) Set your desired flags¶meters (other than zoom ones !) in *.def files

4) Run the model (and the post-processing scripts)

For a different simulation :

- with the same gcm.e and grid (resolution, zoom) : redo **Steps 1 to 4**

- with different resolution/zoom parameters:

- recreate **start*** and **limit** (cf **prerequisites B**)

- reinterpolate all files grid-dependent (**prerequisites C,D**)

- after code changes: recompile cf (**prerequisites A3**) => **new gcm.e**

Steps for setting up a simulation using “all-in-one” **tutorial.tar**

(see : https://lmdz.lmd.jussieu.fr/pub/Training/Tutorials/Tutorial_2.pdf)

Prerequisite : **install_lmdz.sh** has **automatically**

- downloaded and compiled model(s)&libraries,
- run a 1st simulation in modipsl/models/LMDZ/BENCH32x32x39

Here : **tutorial.tar** via its **main script init.sh** allows you to **automatically** :

- recompile LMDZ for a different resolution
- create initial and boundary files (start* and limit) for a chosen grid
(NOTE : *grid = resolution+zoom factor*)
- set up simulation folders : SIMU1 (multi-day runs), PROD0 (multi-month runs)
with all you need (*.def files, start* and limit.nc files, gcm.e)

You'll “only” have to : -

- a) Download and unpack tutorial.tar in modeles/LMDZ => folder TUTORIAL
- b) Check/modify *.def files in TUTORIAL/DEF/ (zoom parameters are in **gcm.def**)
- c) Check/modify **init.sh** script : **grid_resolution, veget, parallel**
- d) run : **./init.sh** (NOTE : pay attention at its final instructions about how to run the model)
- e) check the results of **./init.sh** :
 - TUTORIAL/INITIAL : grille.pdf, start.nc, startphy.nc and limit.nc
 - TUTORIAL/SIMU1 : check *def files and links to gcm.e and start+limit.nc files

And finally :

- f) **run the model and visualize the results**

Let's have a look at the tutorial.tar content :

(interactively...)

Proposed exercises :

Nudging

1D

Parallelism

XIOS

Aerosols

Tracers

Coupling with surface schemes (**b**ucket, *Orchidee*)

ICOLMDZ

Download your choice(s) from :

<https://lmdz.lmd.jussieu.fr/pub/Training/Tutorials>