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 : prerequisits

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

A.1) Download LMDZ (+ models coupled to LMDZ*)* + IO libraries (**IO**IPSL, X**IO**S)
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&parameters (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 **prerequisits B)**
  - reinterpolate all files grid-dependent (**prerequisits C,D)**

- after code changes: recompile cf **(prerequisits 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, ***O****rchidee*)

ICOLMDZ

Download your choice(s) from :

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