Tutorial #2 – Tracers

LMDZ team

December 12, 2013

Contents

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/modipsl/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.

- 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 (`.bash_profile` if you use Bash).

- Run the script :

  ```
  ./init.sh
  ```

  The script first compiles the model (`gcm.e`) again because it uses a different resolution, 48×36 – L39, than the one used in `install.sh`. `init.sh` also compiles the program `ce0l.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 `ce0l.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`.

2 Tracers

2.1 Radon and lead

In the file `traceur.def`, you can see that two trace species, named RN, for radon, and PB, for lead, are already defined in addition to vapor and condensed water. Visualize these two tracers in the file `histhf.nc`. The NetCDF variables RN and PB are numbers of atoms per unit mass, in kg^{-1}. Usually, activities are preferred for comparison with observations. The activities per unit mass, in $\mathrm{Bq\ kg}^{-1}$, are:

$$A_{\mathrm{Rn}} = \frac{\mathrm{RN}}{4{,}765 \cdot 10^5}$$
$$A_{\mathrm{Pb}} = \frac{\mathrm{PB}}{1{,}028 \cdot 10^9}$$

(The numerical values that appear in these fractions are the lifetimes of radon and lead, in s.) You can take a look at LMDZ info number 7, figure 19, or the plot in the introductory course (§ 4 “Modes d’utilisation”) to check that you have sensible results. If you want to see activities per unit volume, you should divide by the mass density, using NetCDF variables mass and zhalf in the file `histhf.nc`.

2.2 Inserting new tracers

We will now explain how to modify the code in order to add idealized tracers. As an example, we will add two tracers, and call them `NewTr1` and `NewTr2`. We will define a domain of the horizontal grid in which the two tracers will initially have

the same constant value. The first tracer will be transported by the boundary layer and convective sub-grid-scale motion, in addition to large-scale advection. The second tracer will only be transported by large-scale advection.

We will have to modify the Fortran program but let us first consider the run-time parameters that we have to modify. At run-time, we decide to include the tracers `NewTr1` and `NewTr2` in the simulation by changing the file `traceur.def`. Change the number of tracers at the first line of `traceur.def` and append one line for each tracer:

```
6
14 14 H2Ov
10 10 H2Ol
10 10 RN
10 10 PB
10 10 NewTr1
10 10 NewTr2
```

Now let us turn to the Fortran program. The only file we need to modify is `phylmd/traclmdz_mod.F90`. Here are the changes you should make in that file.

- Declare two new module variables, `id_NewTr1` and `id_NewTr2`, wih type integer. These are the identifying numbers of the tracers in the program. You can take a previous declaration (`id_pcsat`, `id_pcocsat` ...) as a template.

- All remaining changes will be made in the procedure `traclmdz_init`, which is inside the module `traclmdz_mod`. In order to define `id_NewTr1` and `id_NewTr2`, the program will scan the file `traceur.def`, looking for NewTr1 and NewTr2. So you should initialize `id_NewTr1` and `id_NewTr2` to 0 before the loop beginning at line 175. There is a comment just above, saying "Recherche des traceurs connus", which means "looking for known tracers".

- In the body of this loop, set `id_NewTr1` to the value of index `it` if tname(iiq) equals NewTr1. You can add the test near line 265, for instance, after the test for pcq0. You can take another tracer as a template. Do the same for `id_NewTr2`.

- For NewTr2, just after setting `id_NewTr2`, deactivate convective and boundary layer transport by setting `conv_flg(it)` and `pbl_flg(it)` to 0.

- Finally, we will initialize the tracers. There is a loop on tracers, beginning at line 294, which tests whether the intial tracer field, read from the file `start.nc`, is zero everywhere. At this point, the tracer field would also be zero if it was not found in `start.nc`. There is a comment just above the test which says "Initalize tracer that was not found in restart file". In the body of the test, for our two tracers, change the value at the surface in some horizontal domain. The index of the surface in the vertical dimension is 1. Use variables `xlat` (latitudes) and `xlon` (longitudes) to choose the horizontal domain.

In summary, after making those changes, svn diff should give you something like this:

```
$ svn diff traclmdz_mod.F90
Index: traclmdz_mod.F90
===================================================================
--- traclmdz_mod.F90 (revision 1910)
+++ traclmdz_mod.F90 (working copy)
@@ -58,6 +58,8 @@
   LOGICAL,SAVE :: rnpb=.FALSE. ! Presence du couple Rn222, Pb210
 !$OMP THREADPRIVATE(rnpb)

+  INTEGER, SAVE:: id_newtr1, id_newtr2
+  !$OMP THREADPRIVATE(id_newtr1, id_newtr2)

 CONTAINS

@@ -172,6 +174,8 @@
 ! ----------------------------------------
     id_rn=0; id_pb=0; id_aga=0; id_be=0; id_o3=0
     id_pcsat=0; id_pcocsat=0; id_pcq=0; id_pcs0=0; id_pcos0=0; id_pcq0=0
+    id_newtr1 = 0
+    id_newtr2 = 0
     DO it=1,nbtr
        iiq=niadv(it+2)
        IF ( tname(iiq) == "RN" ) THEN
@@ -262,6 +266,12 @@
        ELSE IF ( tname(iiq) == "pcq0" .OR. tname(iiq) == "Pcq0" ) THEN
          id_pcq0=it
          conv_flg(it)=0 ! No transport by convection for this tracer
+       else if (tname(iiq) == "NewTr1") then
+          id_newtr1 = it
+       else if (tname(iiq) == "NewTr2") then
+          id_newtr2 = it
+          conv_flg(it) = 0
+          pbl_flg(it) = 0
        ELSE
          WRITE(lunout,*) 'This is an unknown tracer in LMDZ : ', trim(tname(iiq))
        END IF
@@ -325,6 +335,9 @@
                    tr_seri(i,:,it) = 100.
                 END IF
              END DO
+          else if (it == id_newtr1 .or. it == id_newtr2) then
+             where (xlat >= 40. .and. xlat <= 45. .and. xlon >=0. &
+                  .and. xlon <= 5.) tr_seri(:, 1, it) = 1.
           ELSE
             ! No specific initialization exist for this tracer
             tr_seri(:,:,it) = 0.
```

Re-compile the program:

```
cd some_path/LMDZtesting/modipsl/modeles/LMDZ5/TUTORIAL
rm gcm.e
```

```
cd ..
```

If you ran `init.sh` with veget=0:

```
./makelmdz -d 48x36x39 gcm
```

If you set veget=1 in `init.sh`:

```
./makelmdz -d 48x36x39 -v true gcm
```

Rename restart files:

```
mv restart.nc start.nc
mv restartphy.nc startphy.nc
mv restart_sech.nc start_sech.nc
```

Run the model:

```
cd TUTORIAL
../gcm.e
```

(could take about 15 mn). Visualize the two new tracers in `histhf.nc` and the difference between them.

3 Nudging

- You first have to create the file `grilles_gcm.nc` wich 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 `ce0l.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-Cloché (IPSL). For access at CCRT, contact Anne Cozic (LSCE). `get_era.sh` is a very simplified script for the tutorial, but more general scrips are available on `http://forge.ipsl.jussieu.fr/igcmg/svn/CONFIG/LMDZOR/branches/LMDZOR_v4/CREATE/SCRIPT`.

- Take a look at file `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. You can change parameters in this file if you want.

- Add the line:

  `INCLUDEDEF=guide.def`

  in `run.def`.

- Rename your `hist*` files so that they are not overwritten by the next run, and delete the file `restart_sech.nc`.

- Run the model again with nudging:

  `./gcm.e`

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