Homework 7: Modifying the Barotropic Model

This document is a the second part of a two-part series on running the GFDL FMS barotropic model. This document will discuss the steps required for *modifying* the barotropic model code. Please refer to "Homework 6: Running the Default Barotropic Model" for details about compiling and running the model once modifications are complete. **Don't forget** to replace all instances of "bbanner" with your username.

Overview of the model code

The GFDL FMS code is written in Fortran[1]. Even if you have never programmed in Fortran before, the code is relatively easy to read/understand, and it is straightforward to make basic modifications since you can use the already written code as examples for how things are done.

The code is setup as a series of small files/routines that call each other. They are also (thankfully) nicely organized. I will step you through where to find certain aspects of the code so that they can be modified. Before this, however, two important concepts need to be discussed:

Source Code: A large group of files that contain the meat of the program. These are the files that are *compiled* during the compilation process (see previous handout) and saved in the fms.x executable file. What is most important to remember is that *if you modify any of the source code you must recompile the code!* If you don't recompile, the fms.x won't be updated, and thus, you won't run the updated version of the model. Again, when you run the model you are technically running the fms.x file - so you must update this file after you've modified the source code by recompiling. The source code for this model is located within your directory at: */home/ats601/bbanner/gfdl_spectral_core/src*

Some advice - use commenting liberally so that you can go back and see what you changed. This is especially helpful for debugging. Also, Fortran will ignore files that you have renamed so feel free to make copies of the source code (e.g. barotropic_dynamics_original_code_from_libby) to save a copy of the code.

Namelist: There are some parameters, e.g. the damping coefficient or the latitude of the initial vorticity perturbation, that the writers of the model expected people to want to modify easily. If these parameters were only specified in the source code, then one would need to recompile every time they changed one of these values. Thus, there is something called the *namelist* which is a file where you can specify the values you want certain parameters to be given. *If you modify the namelist file only, you do not need to recompile*, this is because the runscript reads this file separately (i.e. it is not in fms.x), and thus, no recompiling is needed. The namelist file "namelist_default" is located at:
/home/ats601/bbanner/gfdl_spectral_core/exp/spectral_barotropic/namelistsdir

[1] yes, Fortran...

Runscript

The length of the time integration (e.g. 25 days by default) is set in the runscript. If you want to change this, open the file "runscript_default", or make a copy by typing

>> cp runscript_default runscript_newname

Open the file. If using vim, put the editor into text edit mode by hitting the letter "i" and then navigate down to where you see (line 13)

set num_days_list = 25

You can now change the number 25 to whatever you want.[2] When you have modified the code, hit the "esc" to get out of edit mode, and then to save type ":w" (for write) and hit enter. To exit type ":q" (for quit) and hit enter. To write and quit immediately afterward hit ":wq" and hit enter.

Every time you want to run a new simulation, you don't want to overwrite all of your files from the previous simulation. In order to avoid this, you want to make a new copy of the runscript for each simulation (with a descriptive name). In addition, you need to modify the runscript so that it saves the new data in a new directory. To do this, follow the steps above for opening and editing the runscript. Specifically, for each new simulation you want to change the line that says (line 11)

set name = 'default'

and replace the word "default" with the new name that you want for your simulation.

Furthermore, if you have updated the namelist and renamed it (currently titled "namelist_default"), you need to change this name in the runscript (line 22).

Namelist parameters

The namelist contains a list of often modified parameters that are defined in the barotropic.pdf document you all read for HW 6. Of specific interest may be:

- **m_0**: the wavenumber of the initial vorticity disturbance
- ζ**_0**: the amplitude of the initial vorticity disturbance
- **eddy_lat**: the center latitude of the initial vorticity disturbance
- **eddy_width**: the latitudinal width of the initial vorticity disturbance

[2] Don't run the model for too long, as I have your run exiting if it takes more than 2 minutes of wall time per simulation

Source Code: constants

If you want to change parameters that are fixed constants (e.g. radius of the earth), you can do that *within the source code*[3]. To do this, navigate to the Fortran file *constants.f90* by doing the following:
>> cd /home/ats601/bbanner/gfdl_spectral_core/src/shared/constants
and then opening the Fortran file (denoted by the .f90) using an editor.

Inside the file (around line 59) you will be able to change:

- **RADIUS**: radius of the earth
- **OMEGA**: rotation rate of the earth

Note that there are many other constants that you can change, however, they won't make any difference for the barotropic model (they are for the full aquaplanet simulations, or the dry dynamical core model). Follow the same steps for modifying the text using the vim editor, and then saving and quitting.

Remember to recompile if you change anything inside constants.f90!

Source Code: flow setup

The one other source file you might want to change is the file where you can modify the full form of the vorticity perturbation (add another perturbation in the Southern Hemisphere, for example) or modify the initial background flow[4]. The file is called *barotropic_dynamics.f90* and can be found by doing the following:
>> cd /home/ats601/bbanner/gfdl_spectral_core/src/atmos_spectral_barotropic
and then opening barotropic_dynamics.f90 using an editor (e.g. vim).

This file is much longer than the rest and contains a lot of information about how the dynamics (e.g. damping, advection) actually is implemented. You will most likely be interested in only modifying two of these aspects, the initial background flow and the initial vorticity perturbation.[5]

[3] Remember that you must recompile for your changes to get saved in the fms.x file!

[4] This is where I removed the initial background jet-stream in the simulation I showed in class

[5] Remember that particular aspects of the initial vorticity perturbation are already defined in the namelist file, so you can just change them there and not have to recompile. Otherwise, if you change them in the barotropic_dynamics.f90 file, you will have to recompile for the changes to take effect.

The initial zonal flow is specified starting at line 236 and looks like:[6].

```
Dyn%Grid%u(:,j,1) =   25.0*cos_lat(j)                          &
                    - 30.0*(cos_lat(j)**3)                     &
                    + 300.0*(sin_lat(j)**2)*(cos_lat(j)**6)
```

Say you want to modify this to have no zonal flow. The first step is to comment out these three lines of code. The comment symbol is an exclamation mark, and thus, you should have:

```
!    Dyn%Grid%u(:,j,1) =   25.0*cos_lat(j)                          &
!                        - 30.0*(cos_lat(j)**3)                     &
!                        + 300.0*(sin_lat(j)**2)*(cos_lat(j)**6)
```

Then, you could set the zonal and meridional winds to zero by typing right underneath the newly commented code.[7]:

```
Dyn%Grid%u(:,j,1) = 0.0
Dyn%Grid%v(:,j,1) = 0.0
```

Similarly, the initial vorticity perturbation is defined starting at line 251.

```
  Dyn%Grid%vor(i,j,1) = Dyn%Grid%vor(i,j,1) + &
          0.5*zeta_0*cos_lat(j)*exp(-yy*yy)*cos(m_0*rad_lon(i))
```

You can comment these lines out and redefine the perturbation as you wish.

Remember to recompile if you change anything inside barotropic_dynamics.f90!

Bringing it home...

OK, let's assume you have made modifications to the source code. In that case, recompile the code following the previous handout, and then go and make a new runscript (as described above) with a new descriptive name for the simulation you are about to run. Don't forget to modify the *set name = 'default'* in your new runscript, otherwise your default directory will be overwritten! Once you've made these changes, you can resubmit your new runscript to the queue following the steps outlined in the previous handout.

[6]To see line numbers in vim, open the file in vim, then *before hitting the "i" key*, just type ":set nu" and hit enter. The line numbers will then appear along the left-hand-side of the terminal

[7]Make sure non-integers always have decimal points, otherwise Fortran will think it is an integer and things will go wonky fast! That is, Fortran treats integers differently than floating point numbers.