

1. General notes on linux

If you are not familiar with linux, talk to the instructor before you do anything! Unlike Windows or MacOS, under linux one line of command can kill everything. Be aware that you are sharing an account with others. You should create a subdirectory under /home/student1 and perform all of your work under that directory.

Do not use space (" ") in the filename or the name of a directory. (Linux can handle it but the fix is somewhat tedious.)

2. General notes on compiling and running the CCM3 model

A working "package" has been copied to /home/student1/huei/t01. This directory contains all of the subdirectories that include the Linux scripts for compiling and running the fortran codes, the files for the boundary and initial conditions, and the source codes (in fortran):

Run scripts are in /home/student1/huei/t01/bld

Files for boundary conditions are in /home/student1/huei/t01/data

Source codes are in /home/student1/huei/t01/src

For the first test run, it is recommended that you create your own directory, /home/student1/yourname, then copy the whole package to under that directory. The package will now be in /home/student1/yourname/t01

The files for the run scripts and source codes are small. The total size of the package is only ~ 35 MB. Therefore, it is OK that you create multiple copies of it for different sets of runs. On the other hand, The output of a simulation can easily reach hundreds of MB. Please be careful in managing the output.

For each simulation, it is recommended that the output be redirected to a separate directory that's created specifically for that experiment. For instance, to run the case "t01", you may designate the output directory to be /home/student1/yourname/t01/output. This can be done by modifying a parameter in the run script. See detail below.

For a long simulation (say > 1 hour of CPU time), it is recommended that you submit it as a "background job". This way, you can log out and the simulation will continue. When doing so, you should also arrange to redirect the on-screen output to a designated file. See detail below. This last point is important and must be followed.

3. What's inside the package

Under the t01 directory, you will see several subdirectories. They are:

src, where the fortran source codes are stored. This is the mother directory under which there are more subdirectories that store the fortran subroutines by their functions. For example, the codes for physical parameterization (moist convection, cloud formation, etc.) are stored under src/physics. For more advanced applications, one can modify the codes in one or more subroutines then re-compile the whole package.

data, where the boundary and initial conditions are stored. For example, the files named hybsstc*.nc and citrus_*.nc were SST files created by HPH for various numerical experiments. We will later add the specific files that are needed for your assignment.

build, where the linux scripts are stored. We use them to drive the whole process of setting the environmental parameters, compiling the codes, and running the model.

The other subdirectories are less important and will not be discussed further.

4. Your first "test run"

In this case, we will use the script, run.pc.linux.csh.demo to control the process.

Step 1: Change directory to **build** and modify the script, run.pc.linux.csh.demo

1a. Change all "huei" to "yourname" for the following 4 commands

```
setenv MODEL_SRCDIR /home/student1/huei/t01/src
setenv MODEL_EXEDIR /home/student1/huei/t01/output
setenv MODEL_DATDIR /home/student1/huei/t01/data
setenv MODEL_DEPDIR /home/student1/huei/t01/tools/makdep
```

For example, change

```
setenv MODEL_SRCDIR /home/student1/huei/t01/src
to
setenv MODEL_SRCDIR /home/student1/yourname/t01/src
```

Note that 2nd of the above 4 commands has already designated /home/student1/huei/t01/output as the output directory for the specific run.

1b. Set the appropriate parameter values for your simulation

Many model parameters that control the simulation can be modified or specified without changing the fortran codes. Some of the basic parameters are already given within the segment,

```
cat >! atm.parm << EOF
&CCMEXP
caseid   = 'ccm3bld'
ctitle   = 'ccm3bld'
ncdata   = '$MODEL_DATDIR/SEP1.T42.0198.nc' <--- (A)
bndtvs   = '$MODEL_DATDIR/citrus_c01_control.nc' <--- (B)
bndtvo   = '$MODEL_DATDIR/noaao3.1990.21999.nc' <--- (C)
iradsw   = -1
iradlw   = -1
iradae   = -12
inithist = 'NONE'
nhtfrq   = 0
nrefrq   = 0
dtime    = 1200. <--- (D)
nестep   = -2 <--- (E)
irt      = 0
kmxhdc   = 1
iyear_ad = 1950 <--- (F)
/
&lsmexp
mksrfdir = '$MODEL_DATDIR'
finidat   = '$MODEL_DATDIR/CCM3.5LSMICSEP1.0198.310lsm.nc' <--- (G)
/
EOF
```

For example,

(A) is the filename for the initial condition (in this case, a generic September 1 initial state)

(B) is the SST file

(C) is the file for the prescribed ozone concentration

(D) is the "delta-t", the time step size for numerical integration

(E) is the number of days for the duration of the simulation

(F) is nominally the year when the simulation starts (which is not critical unless one performs a run for paleoclimate)

(G) is the data file for the surface (vegetation) types.

There are many more parameters that one can adjust. If they are not specified, the default values will be used. For example, in the script we did not specify the value of CO2 concentration so the model will use the default value (around 370 ppm). For a simulation with double CO2, one needs to specify the value of CO2 concentration in the above control segment in order to override the default. A master list of the external parameters for the model can be found in CCM3 user's guide (ask the instructor for a copy - serious inquiry only). For our planned experiments, we only need to change (B) and (E).

When the parameter for (E) is set to a negative value, the unit is in days. In the above example, the model will be run for just 2 days. The unit for (D) is second. Thus, the 2-day integration is equivalent to 144 time steps.

1d. Compiling and running the model

The Linux shell script takes care of the end-to-end procedure of setting the environmental parameters, compiling the fortran codes, and running the model. Once the modifications are done in 1b and 1c, change directory to **bld** and simply execute the run script. in our example for the first test, this is by typing

```
run.pc.linux.csh.demo
```

Then, you are done.

While running the script, you will see a few messages being echoed on screen to indicate the status. For a long simulation, it is recommended that you redirect the on-screen output to a file. For example, by typing

```
run.pc.linux.csh.demo > message_t01 ,
```

the on-screen message will be redirected to message_t01. You will see the file under **bld**. It will be updated so you can **cat** or **vi** the file to check progress.

For a long simulation, it is also recommended that you run the model as a background job. This is so the model will keep running even after you log out. This can be done by typing

```
run.pc.linux.csh.demo > message_t01 &!
```

in this case, you will be able to check progress by inspecting the content of message_t01. You may also use the command,

```
ps -u your_username
```

to check the processes that are active. (Here, your_username is just student1.) This will give you an indication of the cumulative CPU time of the job. To check the instantaneous usage of CPU and memory, try

```
top 10
```

You can exit the status report screen by typing "q".

1d. The output files

The standard output (default is monthly mean) is a collection of netCDF files with filenames of xxxx-xx.nc. Their contents can be readily checked by using `ncdump -c` to extract the headers.