WACCM quickstart guide for NCAR's bluefire and NASA's pleiades

1) Registered for access to the code repository. Find the registration form from the CESM homepage by clicking on CESM1.0 Code Releases, which brings up the following in the right column:

Acquiring the CESM1.0 Release Code

CESM source code is distributed through a public Subversion code repository. This code can be checked out using Subversion client software, such as the command tool svn, or simply view the latest version with a web browser.

A short registration is required to access the repository. After registering, you will receive an email containing a user name and password that is necessary to gain access to the repository.

Acquistion of the code is more fully described in the CESM1.0 User's Guide.

Click on the registration, and fill out the form. Be sure not to use any parentheses or other special characters which are not allowed. You should receive a confirmation email very shortly with your login and password.

2) Create a "cesm" directory in your home directory. You will store whatever versions of the code you download there, as well as case directories for each run.

```
mkdir ~/cesm
cd ~/cesm
```

Now you can look at which model versions are available for download using the subversion command (svn):

```
svn list https://svn-ccsm-release.cgd.ucar.edu/model_versions
```

Alternately, you can paste the same address into a web browser and see the same list, i.e.:

```
ccsm4_0/
ccsm4_0_a01/
ccsm4_0_a02/
cesm1_0/
cesm1_0_1/
cesm1_0_2/
cesm1_0_3/
```

This shows release versions of CCSM4 and CESM1. We want the latest CESM release, cesm1_0_3 here.

To download this to your directory, use the subversion "checkout" command:

```
svn co https://svn-ccsm-release.cgd.ucar.edu/model_versions/
cesm1_0_3 cesm1_0_3
```

This will create a new directory cesm1_0_3 under the current (~/cesm), and put the whole source code tree.

3) Now choose a component set and resolution for your model run. Once the download has finished, go into the new directory's scripts subdirectory:

```
cd cesm1_0_3/scripts
```

In this directory, we will use the create_newcase command to create a case for each model run. create_newcase is invoked with 4 arguments, as described in the user guide Quick Start:

```
./create_newcase -case $CASEROOT -mach $MACH -compset $COMPSET \
        -res $RES
```

- case is the new directory where you want to put the case for this run. Put this in a stable place (i.e. somewhere under your home directory, under a subdirectory called "case"). Each case must have a unique directory, which will be created by the create_newcase command.

- mach is the machine you are on, which may be:
  - bluefire
  - pleiades (Harpertown processors)
  - pleaides_wes (Westmere processors)

- compset is the component set for this particular run (i.e. data or full ocean; present day, pre-industrial, or transient conditions).

- res is the model resolution. The three most common options for WACCM4 are:
  - 1.9°x2.5° with full ocean (B-compsets): use f19_g16
  - 1.9°x2.5° with data ocean (F-compsets): use f19_f19
  - 4°x5° with data ocean (F-compsets): use f45_f45

To see a full list of all of these options, type:

```
./create_newcase -list | less
```

The list is quite long, but if you search for WACCM, you will see the WACCM compsets.

4) Chose a name for your case, and create it using the create_newcase command with the 4 arguments above. Example:

```
    ‣ ./create_newcase -case ~/case/b40.1955-2005.2deg.wcm.001
      -compset B55TRWCN -res f19_g16 -mach pleiades_wes
```

5) Change directories to your new case directory. From here you will configure and build the model. Prior to those steps, you may want to edit some configure options, such as the start date for the run or namelist options.

Changing the start date can be done by editing the file env_conf.xml in your case directory. For example, we might want to start our transient simulation (above) from

restart data we have from a previous run in the year 1980. In this case, we would change both the RUN_STARTDATE and RUN_REFDATE from 1955-01-01 to 1980-01-01.

Changing namelist options may be done by creating a file called user_nl_cam in your case directory with changes you want to make to the namelist, in the same section divisions you see in the namelist.

6) On Pleiades, you will want to edit the env_run.xml script so that you use the input data we already have there. Change the definition for environment variables DIN_LOC_ROOT_CSMDATA and DIN_LOC_ROOT_CLMQIAN, replacing $CCSMUSER in these 2 lines with mjmills2.

7) Configure the model with the command "./configure -case"

8) Build the model with the command "./*.build"

9) If you have not had the build prestage your restart and IC files, copy them to your run directory, which you will find in your scratch space:

- bluefire: /ptmp/$USER/$case/run

- pleiades: /nobackup/$USER/$case/run

10) Edit the env_run.xml file to make sure you have the run length, restart, and archiving options you want.

11) Edit the *.run and *.l_archive (long-term archiving) scripts to make sure you are charging to the correct account number. Change the line:

```
#PBS -W group_list=group_list
```

to replace the "group_list" that follows the "=" with your group ID.

12) If you wish to be notified via email when the run and/or archive jobs begin and end execution, you can add the following options to the *.run and/or *.l_archive scripts:

bluefire	pleiades

<pre># mail when job begins: 	
#BSUB -B
# mail when job terminates: 	
#BSUB -N</pre>
	<pre># My email address:
	#PBS -M me@email.com
	# Email on abort, begin, end
	#PBS -m abe</pre>

13) Submit the run, by executing the ./*.submit script, which will check your case configuration and submit the run script to the batch system as appropriate for either machine.

14) Check the status of submitted jobs with batch commands. On bluefire, use "bjobs" or "batchview". On pleiades, use "qstat", or "qstat -u $USER"

15) Once the run has begun, check progress in the ccsm.log.* and atm.log.* files in the run directory. The "tail -f" command may be useful in following the end of a log file.

Send suggestions for improving this document to Mike Mills at mmills@ucar.edu