Community Earth System Model (CESM 1) developed at the National Center for Atmospheric Research (NCAR) in Boulder, CO (formerly the Community Climate System Model (CCSM)).

Ocean component comes in two grid resolutions:

gx1v6 – approximately 1 degree resolution horizontally (320x384x60) “by 1”

gx3v7 – approximately 3 degree resolution horizontally (100x116x60) “by 3”

Both models have the same vertical resolution with 60 vertical levels.

These levels are 10m thick in the upper 150m, and then increase in thickness with depth.

The latitudinal resolution varies, with finer resolution near the equator.

Both grids also have a “displaced pole”, that is the longitudinal lines do not converge at the geographic north pole, but instead converge in central Greenland. This improves Arctic ocean circulation, and provides higher resolution in the waters around Greenland where North Atlantic Deep Water forms.

Model output is generated using netcdf files (ending in .nc). Netcdf allows multiple variables to be stored in one output file, along with ancillary information (i.e. var names, dimensions, etc…).

Some key files in the plotting directories: many are also variables in the output files

kmt – an integer array (320x384, or 100x116) giving the number of ocean levels (depth) at each location.

For a given location, say x=30 and y=20, if kmt(30,20) = 0, this is a land point, no ocean,

If kmt(30,20) = 5, then the ocean is 5 levels, or 50m deep at that location.

(In my IDL plotting routines, I often call this popkmt or inpopkmt.)

Arrays in idl are typically defined to hold integer or floating point data.

temp = intarr(320,384)

temp = fltarr(100,116,60)

varnames\_ccsm4 or cesm\_gx3v7\_varnames - lists the variables typically present in the model ocean output files

zw\_pop and zw\_pop\_m gives the depths of the ocean levels in cm and m, respectively.

tlats – gives the latitude at each point on the 320x384 or 100x116 grid.

tlongs – gives the longitude at each point on the 320x384 or 100x116 grid.

tarea – gives the area of each grid cell in cm^2.

dz – gives the thickness of each model layer in cm.

/DFS-L/DATA/moore/jkmoore/plotannx3

/DFS-L/DATA/moore/jkmoore/plotannx1

- subdirectories with IDL routines for extracting and plotting output from the main model output files, the name format for these is jkm.001.pop.h.0001.nc for annual output files, where jkm.001 is the name of the job/simulation/run, .0001.nc indicates this is year 1 output, …0002.nc for year 2, etc…

Often I will average the annual output files for the last 20 years of a simulation, to remove the short-term variability (ENSO, etc…) and get something closer to a climatology for comparing with WOA, etc… These types of files will have format …. jkm.001.pop.h.0291\_0310.nc mean of years 291-310.

Subdirectories include:

/plotannx3 – routines for extracting plotting model output from annual output files.

/plottrendx3 – routines for extracting and plotting time series of key fluxes and tracer concentrations from multiple annual output files. (i.e., time series how does primary production vary over time..).

/plotmonthx3 – routines for plotting output from saved monthly CESM files.

Commands for Use on Greenplanet (typing man cp will give information about the copy command).

cp – copy files

rm – remove files

rmdir – remove directory

mkdir – create directory

ls – list files in current directory

pwd – list current directory

cd – change current directory

cp /path1/file1 /path2 copies file1 to directory /path2

cp /path1/file1 /path2/file2 copies file1 to directory /path2 and renames it file2

? – a wildcard that can replace one character in these commands

\* - a wildcard that can replace multiple characters in these commands

rm \*.ps will delete all files that end in .ps

rm temp?.ps will delete all files with format temp?.ps where ? can be any letter

Setting up directories on greenplanet, create the following directories on greenplanet

cd /DFS-L/DATA/moore/USERID/

mkdir plotannx3

mkdir cesm\_runs

mkdir archive

/cesm\_runs is the scripts directory, we put files here when running the model to tell CESM which non-standard files to use, how many years to run, etc…

/archive is the directory where the model output files are copied at the end of simulation

/DFS-L/SCRATCH/moore/USERID/jkm.001/run - this is the run directory, where the code actually executes and where the output files are initially written during execution, these are copied to /DFS-L/SCRATCH/moore/USERID/archive/jkm.001/ocn/hist

copy my plotting routines, etc.. to your plotting directory

cd /DFS-L/DATA/moore/jkmoore/plotannx3

cp \* /DFS-L/DATA/moore/USERID/plotannx3

Setting up directories on your PC or laptop

mkdir plotannx3

mkdir cesm\_runs

mkdir output\_files

with subdirectory /jkm.001.yr291-310

Running IDL and the plotting routines

After SSH connecting to greenplanet, copy and paste the following lines:

Load necessary libraries, compilers, licences, etc.. by entering

module load intel/2018.2 openmpi/3.0.1 netcdf/4.6.1

ml idl

These point to the license file for IDL and various libraries needed by code

Then enter “idl” to start IDL

Idl

You can still execute Unix/Linux commands from within IDL by prefacing with $ for example

$ls - lists the contents of the current directory.

**Key IDL programs (in files in plotting directories: extraction.V3 and pop4\_coords :**

**Enter:**

**.run extraction.V3 and**

**.run pop4\_coords to compile these routines for use within IDL.**

poplatlong\_x3 – finds location on ocean grid for a specific lat/long

poplatlong\_x3,lat, lon, x, y

poplatlong\_x3, -10.0, 140.0, x, y will return x grid location in the variable x, and y location in y.

print,x,y

extract\_var\_file – extracts a variable from a netcdf file.

extract\_var\_file,’jkm.001.pop.h.010.nc’,’NO3’,nitrate

after execution, nitrate will be a fltarr(100,116,60) that has the nitrate concentration from the simulation.

To execute these in IDL you have to first “compile” them with the .run command, i.e.

.run extraction.V3

There is a second type of plotting routine that runs from the command line. These you don’t have to compile first with the .run command. The routines ending in ….run in the arun.. file are this type. They compile and run when you call them.

To use these, copy the model output files to either plotannx3 then copy and paste to run just some routines, or @arun\_idl\_min\_601 to run all the plotting routines in this file.

A quirk of IDL is that array index notation starts with 0, not 1, as in Fortran, so to access all values of the nitrate array, you need to scroll through x=0 to 99, and y= 0,115, and z=0,59.

**Key files in the CESM Fortran code:**

ecosys\_parms.F90 – sets the values of key parameters in the model.

namelist\_defaults\_pop2.xml – sets the value of some parameters at runtime, OVERRIDES VALUES in ecosys\_parms.F90 (will replace ecosys\_parms in future model versions)

ecosys\_mod.F90 – ecosystem/biogeochemical model code, key subroutine is ecosys\_set\_interior

which updates the biogeochemical sink/source terms

ocn.ecosys.setup.csh – sets up the variables to be saved in output files

We can run just the ocean component, or ocean-sea ice components, with forcing values from the atmosphere from NCEP/NCAR reanalysis data, or form full coupled model output. When forcing with the NCEP/NCAR reanalysis data, the model repeatedly cycles through all available years

(current 1948-2009). Five time through this 62 year cycle, or 310 years is enough to fully spin-up upper ocean biogeochemistry and physics.

Guidelines for Running CESM1.X Ocean Simulations on Greenplanet

Types of CESM simulations

Startup – brand new run starting from built initialization files (i.e. WOA nutrients, etc…)

Branch – initializes the model using output from another simulation.

Hybrid - some mix of the two, maybe sea ice is in startup mode but the ocean tracers branch from another run.

For now you’ll be running startup simulations.

Key directories on the greenplanet cluster:

USERID = jkmoore

/DFS-L/DATA/moore/jkmoore/cesm1\_2\_2/ - contains the standard CESM 1.2.2 model code downloaded from NCAR. We never edit/modify the files in this directory. Everyone can access from this location.

/DFS-L/DATA/moore/jkmoore/gx3v7\_inputs/ - contains input files for the gx3v7 ocean model

/DFS-L/DATA/moore/jkmoore/gx1v6\_inputs/ - contains input files for the gx1v6 ocean model

/DFS-L/DATA/moore/jkmoore/CESM1.98/SourceCode\_gx3v7/ - contains non-standard CESM1.2.2 code for my locally modified code version, this is the version you will use for now. The biogeochemistry is nearly identical to the code going into the CESM2.0, in terms of the scientific equations, but the software architecture will change quite a bit in CESM2.0 (to be released in 2018).

/DFS-L/SCRATCH/moore/USERID/ - this is the scratch space where the model will run and output will be stored. Output that you want to keep long term should be moved to /DFS-L/DATA/moore/….

/DFS-L/DATA/moore/jkmoore/CESM1.98/SourceCode\_gx3v7/SampleNotes/ - sample NOTES and .slurm files for you to use setting up simulations, and an updated version of the IntroCESMoceans file for use with the new system and code. This replaces any earlier version you received.

STARTING a NEW CESM SIMULATION

There are samples directories to with files to setup and run the model on the 12 of the 12cpu nodes in the new2.8 queue at /DFS-L/DATA/moore/jkmoore/CESM1.98/SourceCode\_gx3v7/SampleNotes/

make a new local directory on your laptop called jkm.001, except replace jkm with your initials

copy the following files from this directory to your local /jkm.001/ directory for editing:

NOTES\_12\_12 - a copy and paste script for starting a new job,

You must edit USERID and jobname, other options available (i.e. length of run, etc…)

jkm.001\_12\_12.slurm – this is the batch execution file you submit to the scheduler to run the model

with command: sbatch ./jkm.001\_12\_12.slurm,

You must edit job name (twice) and directory path.

AAverage\_gdev\_001 – script to cut and paste average the last 20 years of the simulation,

model years 291-310 corresponds to NCEP forcings for 1990-2009

(after five times through the (1948-2009) NCEP reanalysis forcings)

arun\_idl\_gdev\_001 – IDL file with IDL plotting routines that work on annual

model output files, can copy and paste to run individual routines in IDL, or entering

@arun\_idl\_gdev\_001 – will execute all the commands in this file, as if they were copied and pasted to command line.

arun\_idl\_min\_001 – IDL scripts, has just the most commonly used plotting routines.

Copy all the files in one of these directories to your local desktop machine.

/DFS-L/SCRATCH/moore/USERID/cesm\_runs/ - scripts directory containing files specific for setting up and starting particular simulations, one subdirectory for each model run,i.e. jkm.001. This directory will be created automatically during model build and compilation:

/DFS-L/SCRATCH/moore/USERID/cesm\_runs/jkm.001/

/DFS-L/SCRATCH/moore/USERID/jkm.001/run – run directory, this is where the model actually runs, initially all output files are written to this directory. This directory created automatically during build and compilation of code.

/DFS-L/SCRATCH/moore/USERID/archive/jkm.001/ – long term storage of the model output and the restart files, output files from simulations are automatically copied here at the end of the simulation. This directory created automatically at end of model run.

here the subdirectory would be

/DFS-L/SCRATCH/moore/USERID/archive/jkm.001/ with subdirectories:

/rest/ - files for restarting the model, or starting a branch run from end of this run.

/ocn/hist/ - ocean output files will copied to here

You will mainly just use the output files with format jkm.001.pop.h.0001.nc (standard annually averaged ocean model output files) or jkm.001.pop.h.0001-01.nc (standard monthly averaged ocean model output files)

/ice/hist – sea ice model output files… etc…

Create under your user account the directories:

/DFS-L/SCRATCH/moore/USERID/plotannx3

/DFS-L/SCRATCH/moore/USERID/popoutx3 and /DFS-L/SCRATCH/moore/USERID/popoutx1 - these you’ll use to store the output plotting files generated in the plotdirs, with one subdirectory for each simulation, for a given year of simulation, i.e. jkm.001.yr30

On your local machine create a directory, /pophome/jkm.001, that contains the files from my sample directory.. You‘ll need to copy to the scripts directory on greenplanet when setting up a new simulation. In this local copy of the scripts directory, will be the following files

/arun\_idl\_gdev\_001 – text file for running annual model output plotting routines

/s\_extract\_gdev\_001 – text file for extracting the global fluxes/tracer concentrations from multiple annual output files (to look at trends).

/s\_plot\_gdev\_001 – text file for plotting the time series data using input files generated by s\_extract\_gdev\_001

/jkm.001.slurm – this is the main file that actually runs the model, it is what you submit to the job scheduler on greenplanet. The number of processors listed in this file must match what you used in the NOTES file to set up and compile the simulation. When setting up new runs you’ll need to edit the jobname and the name of this file to match the new simulation name, i..e. jkm.001 to jkm.002, and the directory path that currently says. (At NCAR this is jkm.001.run). changing jkmoore to you userid and updating the dir name jkm.001

/DFS-L/SCRATCH/moore/jkmoore/cesm\_runs/jkm.001

/ecosys\_mod.F90 – main ecosystem module fortran code

/ecosys\_parms.F90 – parameter values for use by the code in ecosys\_mod.F90

There are other files that will be copied to your scripts directory that you might need to edit in the future:

/ocn.ecosys.setup.csh – this file tells the model what variables to save in the output files, normally you won’t edit this file.

/env\_run – tells whether this is a CONTINUE\_RUN or not, main things to edit here would be the number of total months to run the model for each submission, the number of times for the model to automatically re-submit itself to run again (RESUBMIT variable).

/env\_conf – RUN\_TYPE tells cesm whether this is branch, startup, or hybrid run

RUN\_STARTDATE – gives start date for this simulation

RUN\_REFCASE – name of previous run for branch or hybrid runs

RUN\_REFDATE – date to branch from previous run

CCSM\_CO2\_PPMV – tells model what constant atmospheric CO2 concentration to use when calculating the air-sea CO2 flux.

Key directories on your local desktop/laptop machine

/ccsm4/pophomex3 – keep your local copies of the script files here with one subdirectory for each run.

/ccsm4/plotannx3 – keep local copies of annual plotting routines here if you need to edit them, or are developing new plotting routines, Make your own /plotannx3 directory on greenplanet, copying my dir.

/ccsm4/popoutx3 – one subdirectory for each simulation, to store the output files from the plotting routines, i.e. jkm.001.yr30,

/ccsm4/plotmonthx3 – same as plotmonthx3

/ccsm4/plottrendx3 – same as plottrendx3

aLogsCESMRuns – keep a log file where you note what you modified in the code for each run, and

make some notes on the results.

Commands to use on greenplanet

squeue – lists all jobs currently running on greenplanet

squeue –u jkmoore lists all jobs submitted by jkmoore currently running (u is for user)

squeue –q moore\_fast6 lists all jobs in the moore\_fast6 queue.

sinfo –long gives current status of all the nodes and queues on greenplanet

squeue result looks like:

JobID USER QUEUE Jobname … NDS … ElapTime

99999.greenplane jkmoore moore qdev.ell.001 8 1:05

To stop, or kill a job that is running, enter

scancel 99999

– using the job ID you get from qstat –u youruserid

executing jkm.001.slurm – submit this job to run on Greenplanet

sbatch ./jkm.001.slurm

Starting a new CESM Ocean simulation

ssh to gplogin1.ps.uci.edu or to gplogin3.ps.uci.edu (from off campus you have to connect using a VPN,

(see [www.libraries.uci.edu](http://www.libraries.uci.edu), click on button on upper left of page that says “Connect from Off Campus” and install the vpn software on your home PC. Note don’t download any illegal content while on the VPN, music, movies, etc…. UCI will notice.

Load necessary libraries, compilers, licences, etc.. by entering

module load intel/2018.2 openmpi/3.0.1 netcdf/4.6.1

ml idl

Example starting job jkm.001

Cut and paste from the modified NOTES file to build, compile, and submit your run to the appropriate queue on Greenplanet.

There are two steps, copy and paste everywhere above the line beginning with # STOP at the GP command line. This will set up a new simulation, creating the necessary directories, etc…

Establish a sftp connection to Greenplanet and copy your edited .slurm file to the scripts directory.

Here copying jkm.001\_12\_12.slurm to

/DFS-L/SCRATCH/moore/jkmoore/cesm\_runs/jkm.001/

Then, copy any source files that you have edited/changed to the sub-directory, /SourceMods/src.pop2/

/DFS-L/SCRATCH/moore/jkmoore/cesm\_runs/jkm.001/SourceMods/src.pop2/

During compilation, any files in this directory replace the standard CESM1.2.2 versions. When you copy and paste the first part of this file above, my non-standard CESM1.98 files are copied to this directory. If you want to make changes, you need to edit these files and then overwrite my version in this directory with your edited file.

Now you can copy and paste the remaining lines from the NOTES file. These set the simulation length, builds and compiles the code, and submits the job to the scheduler. The lines are:

# STOP, copy over edited ecosys\_mod file to /SourceMods/src.pop2

./xmlchange -file env\_run.xml -id STOP\_OPTION -val nmonth

./xmlchange -file env\_run.xml -id STOP\_N -val 120

./xmlchange -file env\_run.xml -id RESUBMIT -val 1

./xmlchange -file env\_run.xml -id DOUT\_L\_MS -val FALSE

./${CASE\_DST}.build

sbatch ./${CASE\_DST}.slurm

exit

STOP\_N – tells the model how many months to run, here set for a 10 year simulation.

RESUBMIT – here set = 1, means that at the end of the 10 year simulation, the model will write out all the output files, etc.. and then the last thing it does will be to automatically resubmit itself to the queue to run for another 10 year period. So setting RESUBMIT to 5 would give you a 50 simulation total. You can use values up to 744 (62 years, one time through our NCEP forcing dataset) for STOP\_N.

For publication, I usually run the model through the NCEP forcings 5 times, for a 310 year simulation, then average output from the last 20 years to get a “climatology”. During model development, I use much shorter simulations, of a couple of decades.

./${CASE\_DST}.build - This builds and compiles the model, if compilation is not successful, error messages will point to the part of code causing problem.