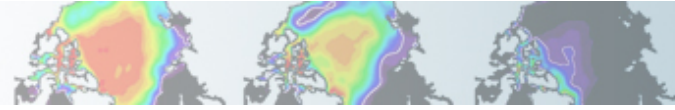


CLM4.5 Tutorial: Running for Single-Point/ Regional Cases



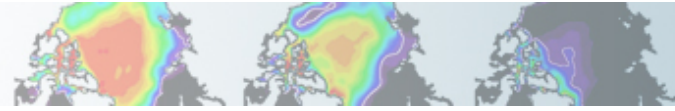
U.S. DEPARTMENT OF
ENERGY

Office of
Science



Objectives for This Session

1. Make you sick of the four steps to run CLM/CESM! (really make you familiar enough with it that you comfortable using them)
2. Give you some familiarity with the CLM datasets that you would customize for your site.
3. Get you familiar with the process on how you change your case to use your own datasets.
4. Give an introduction to the process of making CLM input datasets, so that you can study it on your own later (using the CLM User's Guide).
5. Give an introduction to PTCLM that makes the process of creating datasets easier.
6. Do all this in an incremental fashion so we start with standard out of the box cases, and end on something more complex.



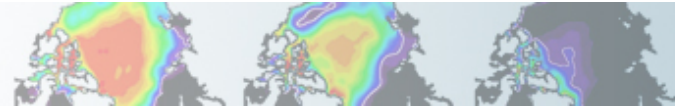
Caveat!

This is our first go of teaching this section.

The way that the PTCLM and getregional scripts work here is very different than the last public version of the model! So if you want to use the public release, you'll want to study the CLM User's Guide for their specific use.

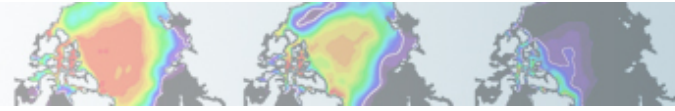
If you really need this version of the model for development – you'll need to ask for access to it after the tutorial.

For the development version, you'll want to study the README files and other documentation inside the distribution.



Three Sections we will Divide Work into

- 1. Work with out of the box single point cases to become familiar with the datasets that you need to change to work on your own site.**
- 2. Work with the getregional script that subsets existing datasets to get you familiar with setting up a case with your own datasets.**
- 3. Introduction to the process of making datasets, and to PTCLM that helps you with it.**
- 4. Gives you two chances to bail early!**



Examples and questions are on yellowstone

1. These slides are at...

- <http://www.cesm.ucar.edu/events/tutorials/20140219/day2-practical2-kluzek.pdf>

1. The examples and questions are under:

- `/glade/p/cesm/lmwg/CLM2014_tutorial_space`

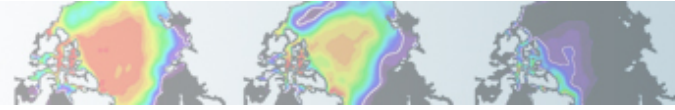
2. In the Day2 subdirectory (and the cases and other examples are in there as well)

3. Three files:

- `SinglePtExercisesI.txt`
- `SinglePtExercisesII.txt`
- `SinglePtExercisesIII.txt`

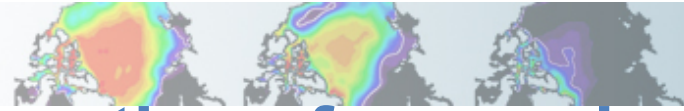


Section I – Running Supported Single Point Resolutions



Objectives for Section I (Running Supported Single Point Resolutions)

1. Continue familiarizing you with the steps to create cases.
2. Start with the closest cases to what you've already worked on.
3. Create some out of the box single point cases.
4. Look at the input datasets that depend on the site location.
5. Discuss what's in those datasets.



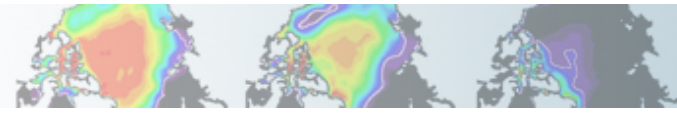
What are the different options for running single point or regional cases?

There are three different options we will address:

1. Run supported resolutions out of the box (there are a few available).
2. Subset global datasets using the “getregional” script
3. Run PTCLM to create datasets for you to use.

The other option that we will give a brief introduction to is...

- Create datasets on your own using CLM tools (PTCLM just makes this simpler).



Creating a new case for a single point

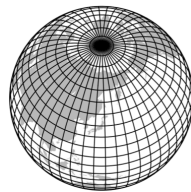
`create_newcase` requires 4 arguments

What is the casename ?



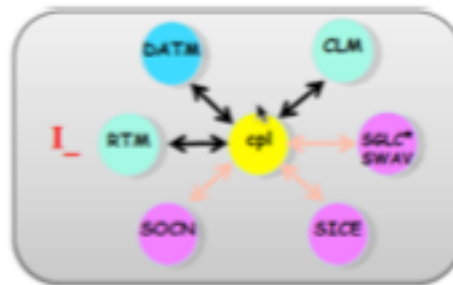
Day2Brazil

Which resolution?
(or location)



1x1_brazil (a single point in brazil)

Which model configuration?
Which set of components?



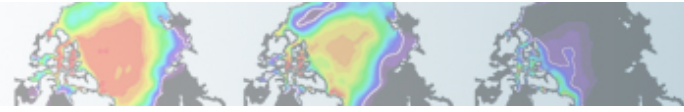
ICLM45
(I = CLM only, 2000)

Which machine are you running on?



yellowstone

```
./create_newcase -case Day2Brazil -res 1x1_brazil -compset ICLM45 -mach yellowstone
```



Remember: The 4 commands to run CLM

1. `./create_newcase`
2. `./cesm_setup`
3. Build the case (`./$CASE.build`)
4. Submit to the batch queue and run the simulation (`./$CASE.submit`)

So for a supported single-point case...

```
# go into scripts directory into the source code download
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/scripts

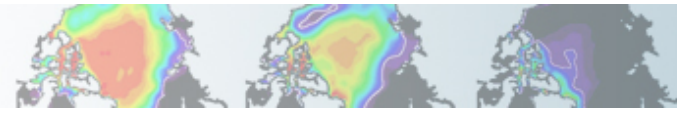
# (1) create a new case in the directory "cases" in your home directory
./create_newcase -case ~/cases/Day2brazil -res 1x1_brazil -compset ICLM45 -mach
yellowstone

# go into the case you just created in the last step
cd ~/cases/Day2brazil

# (2) invoke cesm_setup
./cesm_setup

# (3) build the executable
./Day2brazil.build

# (4) submit your run to the batch queue
./Day2brazil.submit
```



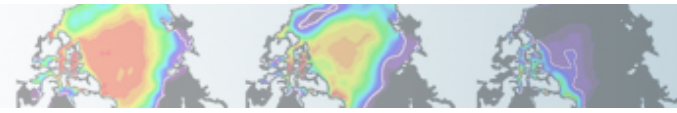
Get a list of the resolutions you can use

1. `./create_newcase -list grids`

```
....  
=====
```

Component	Grids: name (alias)
reg	
null	
CLM_USRDAT	
	1x1_numalA
	1x1_brazil
	1x1_smallvilleIA
	1x1_camdenNJ
	1x1_mexicocityMEX
	1x1_vancouverCAN
	1x1_tropicAtl
	1x1_urbanc_alpha
	5x5_amazon

```
...  
=====
```



Supported Single Point Resolutions

1. create_newcase -list grids

```
....  
=====
```

Component Grids: name (alias)	
=====	
reg	
null	
CLM_USRDAT	
1x1_numalA	
1x1_brazil	← This is the case we just ran (for Novo Progresso Brazil)
1x1_smallvilleIA	
1x1_camdenNJ	
1x1_mexicocityMEX	← This is an urban case that includes Meteorological forcing
1x1_vancouverCAN	
1x1_tropicAtl	
1x1_urbanc_alpha	
5x5_amazon	

```
...
```



Creating a new case for Mexicocity

create_newcase requires 4 arguments

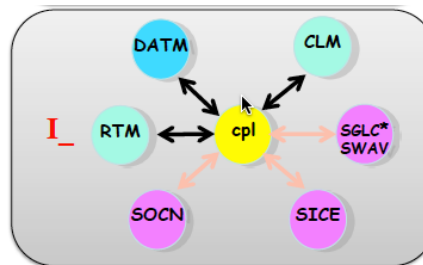
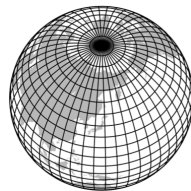
What is the casename ?

Which resolution?
(or location)

Which model configuration ?
Which set of components ?

Which meteorology?

Which machine are you running on?



Day2MexCity

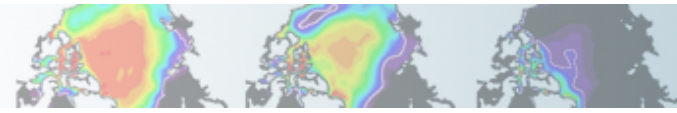
1x1_mexicocityMEX

I1PTCLM45

(with tower Meteorology)

yellowstone

```
./create_newcase -case Day2MexCity -res 1x1_mexicocityMEX -compset I1PTCLM45 -mach yellowstone
```



Let's go to Mexicocity!

Let's run a case for Mexicocity Mexico using the meteorological forcing that goes with the case.

```
# go into scripts directory into the source code download
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/scripts

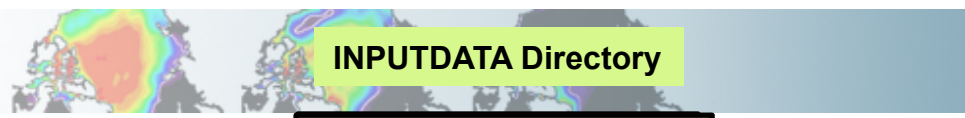
# (1) create a new case in the directory "cases" in your home directory
./create_newcase -case ~/cases/Day2MexCity -res 1x1_mexicocityMEX -compset I1PTCLM45 -
mach yellowstone

# go into the case you just created in the last step
cd ~/cases/Day2MexCity

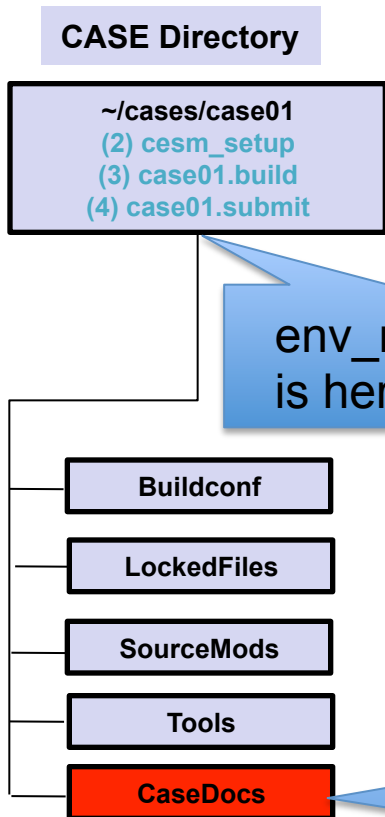
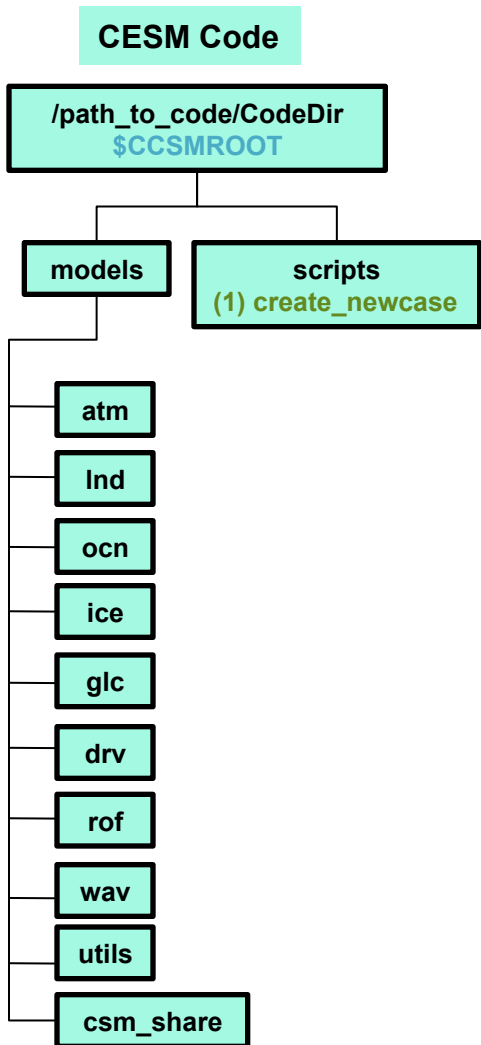
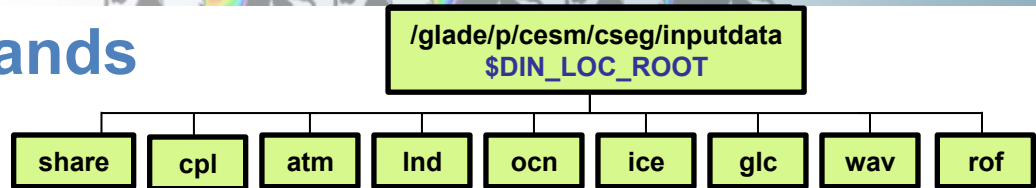
# (2) invoke cesm_setup
./cesm_setup

# (3) build the executable
./Day2MexCity.build

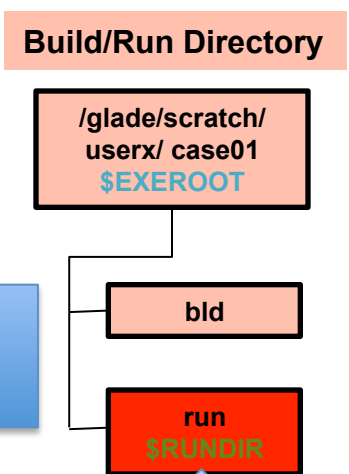
# (4) submit your run to the batch queue
./Day2MexCity.submit
```



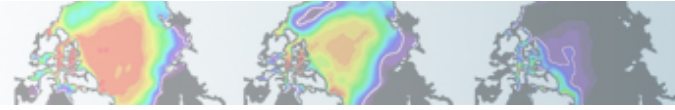
CLM directories & commands



env_run file is here



Namelist files are here (both locations)



What's Different between the two cases?

Let's look at the difference between the env_run.xml files for the two cases
(in ~/cases/Day2brazil and ~/cases/Day2Mexico)

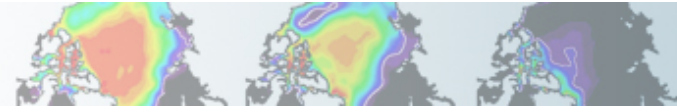
Brazil CASE

```
RUN_STARTDATE = 0001-01-01
STOP_OPTION    = ndays
STOP_N         = 5
DATM_MODE      = CLM_QIAN
ATM_DOMAIN_FILE = domain.Ind.1x1pt-brazil_navy.
090715.nc
```

Mexicocity CASE:

```
RUN_STARTDATE = 1993-12-01
STOP_OPTION    = nsteps
STOP_N         = 158
DATM_MODE      = CLM1PT
ATM_DOMAIN_FILE = domain.Ind.1x1pt-
mexicocityMEX_navy.090715.nc
```

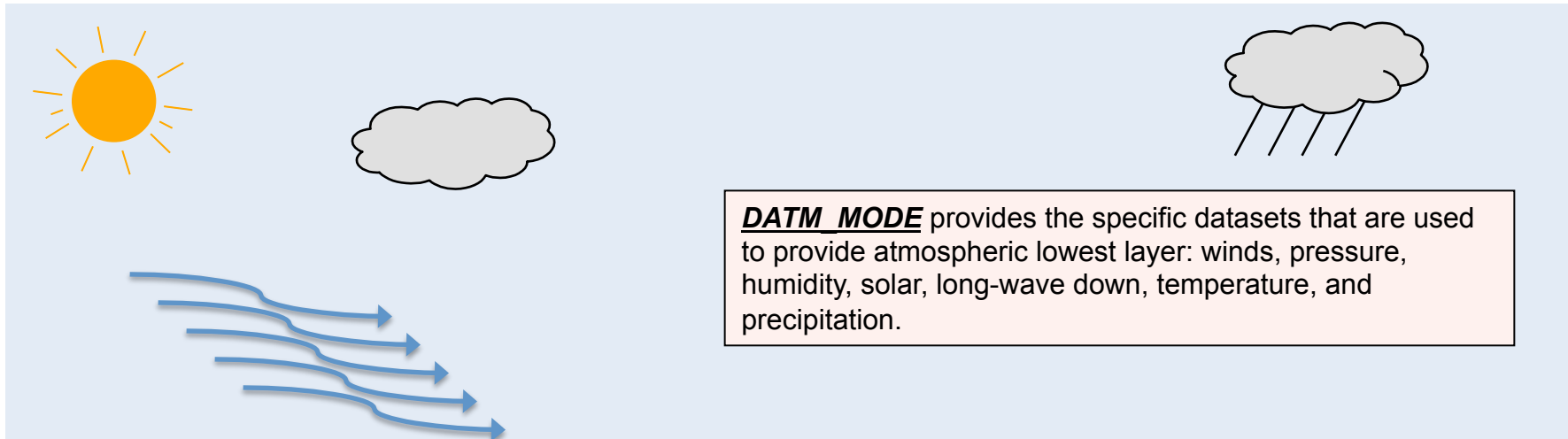
- **The start date and the time to run for is different.**
- **The “DATM_MODE” (type of Meteorology) is also different.**
- **The brazil case runs for Qian forcing**
- **While the Mexicocity case runs for “CLM1PT” forcing (tower Meteorology)**



Aside: What's the deal with DATM_MODE?

There are four modes used with CLM that specify the type of Meteorological data that's used:

- CLMCRUNCEP
- CLM_QIAN
- CLM1PT
- CPLHIST3HrWx



- **CLMCRUNCEP** – Use global NCEP forcing at half-degree resolution from CRU goes from 1900-2010
- **CLM_QIAN** – Use NCEP forcing at T62 resolution corrected by Qian et. al. goes from 1948-2004
- **CLM1PT** – Use the local meteorology from your specific tower site
- **CPLHIST3HrWx** – Use atmospheric data from a previous CESM simulation

What's Different between the two cases? (cont)

Let's look at the difference between the namelist Ind_in files for the two cases
(in ~/cases/Day2brazil/CaseDocs and ~/cases/Day2Mexico/CaseDocs)

Brazil CASE

```
fatmIndfrc = '$DIN_LOC_ROOT/share/domains/domain.clm/  
domain.Ind.1x1pt-brazil_navy.090715.nc'
```

```
fsurdat = '$DIN_LOC_ROOT/Ind/clm2/surfdata_map/  
surfdata_1x1_brazil_simyr2000_c130927.nc'
```

Mexicocity CASE:

```
fatmIndfrc = '$DIN_LOC_ROOT/share/domains/  
domain.clm/domain.Ind.1x1pt-mexicocityMEX_navy.  
090715.nc'
```

```
fsurdat = '$DIN_LOC_ROOT/Ind/clm2/surfdata_map/  
surfdata_1x1_mexicocityMEX_simyr2000_c130927.nc'
```

The two files that are different are:

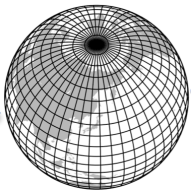
Domain file – Where in the world you are (or for global simulations what specific grid resolution)

Surface dataset – description of surface type, vegetation, and soil and etcetera.



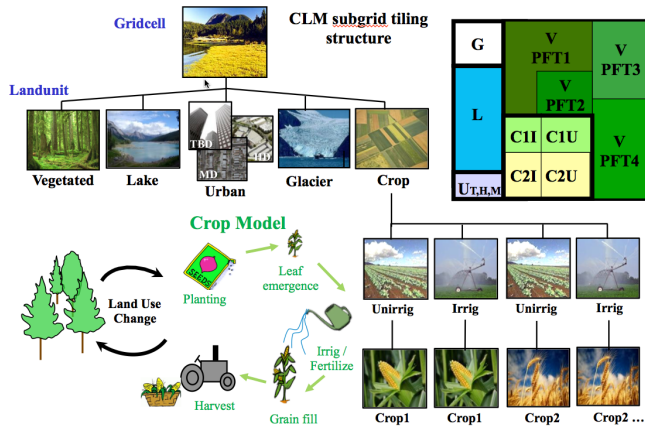
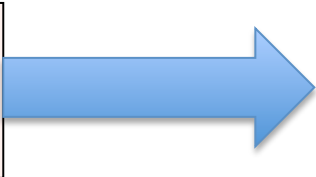
What's in the domain and surface files?

Domain file
Where in the world your grid points are



- Exact grid points
- Corners of the grid points
- Land/Ocean mask
- Land fractional area

Surface file
Description of the land surface information for each grid point.

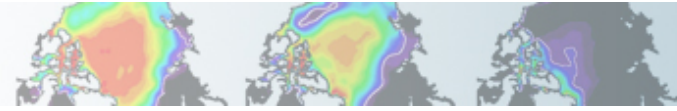


- Percent coverage in the grid cell for each land-unit type.
- Percent coverage of each vegetation type
- Soil type and color
- A whole host of other information that describes the land-cover for each grid cell



Review

1. Steps to create a case are:
 1. `./create_newcase`
 2. `./cesm_setup`
 3. `$CASE.build`
 4. `$CASE.submit`
2. Created an out of the box case for a single point in brazil and one in Mexicocity Mexico
3. We compared the two, the main difference is the domain and surface datasets.
4. Domain datasets have grid information
5. Surface datasets have land-type information for each grid point

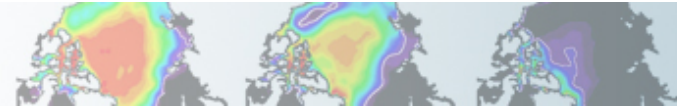


Exercise I

1. Setup a case for brazil
2. Setup a case for Mexicocity
3. In your case directory (for either case) in what env*.xml file is the domain file found?
4. Looking in your case directory for the brazil case in the CaseDocs/Ind_in file figure out the creation date string (_cYYMMDD format) of the surface dataset?
5. Looking in your case directory for the brazil case in the CaseDocs/Ind_in file figure out the creation date string of the domain dataset (.YYMMDD format)?

Extra Credit

1. In which of the namelist files (the files that end in _in) (for the brazil case in your case directory under CaseDocs) are the Meteorology forcing streams files found (this case is with Qian forcing)?
2. For the brazil case look in each of the streams files for meteorological forcing (Solar, Precip and TPQW) , how many total active variables are there (data variables that don't just describe the grid point location) (look for the variableNames field inside the fieldInfo field)?
3. Looking in the " CaseDocs/datm.stream.txt.CLM_QIAN.Solar" file for the brazil case figure out the creation date for the datm forcing data (from the filepath with a .cYYMMDD format)?
4. What resolution is the forcing dataset for the brazil case?
5. In the meteorological forcing streams file for the Mexicocity case – how many active variables are there (as in extra credit question 2 above)?

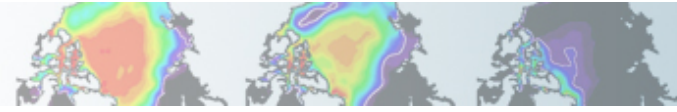


Hints -- Exercise I

- 3. In your case directory (for either case) in what env*.xml file is the domain file found?**
Look in each of the ~/cases/Day2brazil/env_*.xml files for ATM_DOMAIN_FILE. Or use
./xmlquery ATM_DOMAIN_FILE
and it will list the filename.
- 4. Looking in your case directory for the brazil case in the CaseDocs/lnd_in file figure out the creation date string (_cYYMMDD format) of the surface dataset?**
Look for the line in ~/cases/Day2brazil/lnd_in that contains "fsurdat = ". The end of the filename includes the creation date (just before the ".nc" at the very end).
- 5. Looking in your case directory for the brazil case in the CaseDocs/lnd_in file figure out the creation date string of the domain dataset (.YYMMDD format)?**
Look for the line in ~/cases/Day2brazil/lnd_in that contains "fatmIndfrc = ". Also the domain filename is in the env file that you answered in question 3 above.

Hints -- Extra Credit

- 1. In which of the namelist files (the files that end in _in) (for the brazil case in your case directory under CaseDocs) are the Meteorology forcing streams files found (this case is with Qian forcing)?**
The Meteorology forcing defines the atmospheric data from the data atmosphere model (datm). So the namelist files for the datm are the "datm*_in" files. Look for the namelist file that contains the "streams" variable.
- 2. For the brazil case look in each of the streams files for meteorological forcing (Solar, Precip and TPQW) , how many total active variables are there (data variables that don't just describe the grid point location) (look for the variableNames field inside the fieldInfo field)?**
The streams files are the files that start with "datm.streams.txt.". There are a total of four streams files. Ignoring the "presaero" file, look for the <variableNames>" field that is also within the "<fieldInfo>" field and count the variables. Count the variables in the other two files and add them together what is the total?
- 3. Looking in the " CaseDocs/datm.stream.txt.CLM_QIAN.Solar" file for the brazil case figure out the creation date for the datm forcing data (from the filepath with a .cYYMMDD format)?**
Look for the "<filePath>" field what does the creation date string in it correspond to?
- 4. What resolution is the forcing dataset for the brazil case?**
The filename includes the resolution in the field above "T62".
- 5. In the meteorological forcing streams file for the Mexicocity case – how many active variables are there (as in extra credit question 2 above)?**
Here we look at the file "CaseDocs/datm.streams.txt.CLM1PT.1x1_mexicocityMEX" and count the number of "<variableNames>" in the "<fieldInfo>" field.

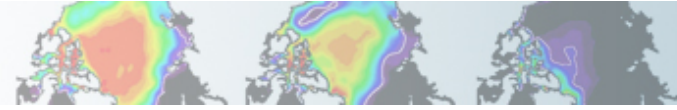


Exercise I

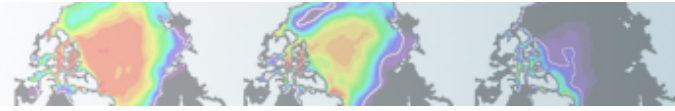
1. Setup a case for brazil
2. Setup a case for Mexicocity
3. In your case directory (for either case) in what env*.xml file is the domain file found?
 - **env_run.xml** file
4. Looking in your case directory for the brazil case in the CaseDocs/Ind_in file figure out the creation date string (_cYYMMDD format) of the surface dataset?
 - **c130927 Sep/27/2013**
5. Looking in your case directory for the brazil case in the CaseDocs/Ind_in file figure out the creation date string of the domain dataset (.YYMMDD format)?
 - **090715 July/15/2009**

Extra Credit

1. In which of the namelist files (the files that end in _in) (for the brazil case in your case directory under CaseDocs) are the Meteorology forcing streams files found?
 - **datm_atm_in**
2. For the brazil case look in each of the streams files for meteorological forcing (Solar, Precip and TPQW) , how many total active variables are there?
 - **6 (Precip, Solar, Temperature, Pressure, Humidity, and Wind-speed)**
3. Looking in the " CaseDocs/datm.stream.txt.CLM_QIAN.Solar" file for the brazil case figure out the creation date for the datm forcing data?
 - **c080727 Jul/27/2008**
4. What resolution is the forcing dataset for the brazil case?
 - **T62 94x192 points, at equator (210km or 1.875 degree's)**
5. In the meteorological forcing streams file for the Mexicocity case – how many active variables are there?
 - **10 (additionally includes LW down, height, and Solar split into direct and diffuse)**

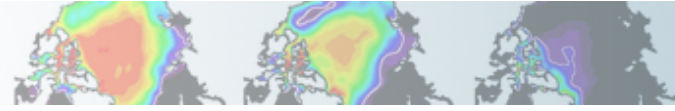


Section II – Using getregional to subset global datasets



Objectives for Section II (Using `getregional` to subset global datasets)

1. Start introducing the directory with tools to make CLM input files.
2. Examine the CLM tool: `getregional_datasets.pl`
3. Use it to create a dataset over Alaska (domain and surface datasets).
4. Create a case that uses the Alaska datasets.
5. Thus we demonstrate how to create a case that uses your own datasets in the simplest way possible.



CLM Tools for creating Datasets

Review: the two datasets that are specific to the resolution are these two files:

- Domain file
- Surface dataset

These files can be created using various tools provided by the CLM distribution

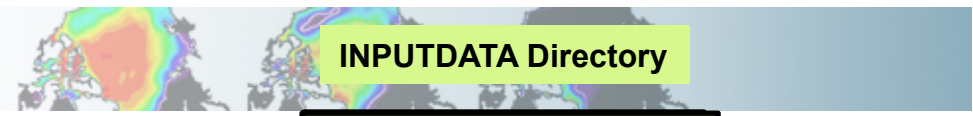
To create the domain file you use “gen_domain”.

To create the surface dataset you use “mksurfddata_map”.

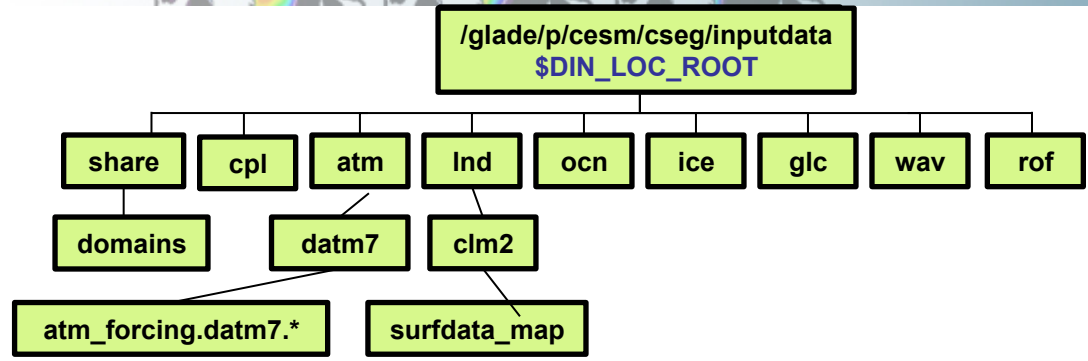
Here we will use the script “getregional_datasets.pl” to create these two files by pulling out a sub-region from an existing 2-degree resolution datasets.



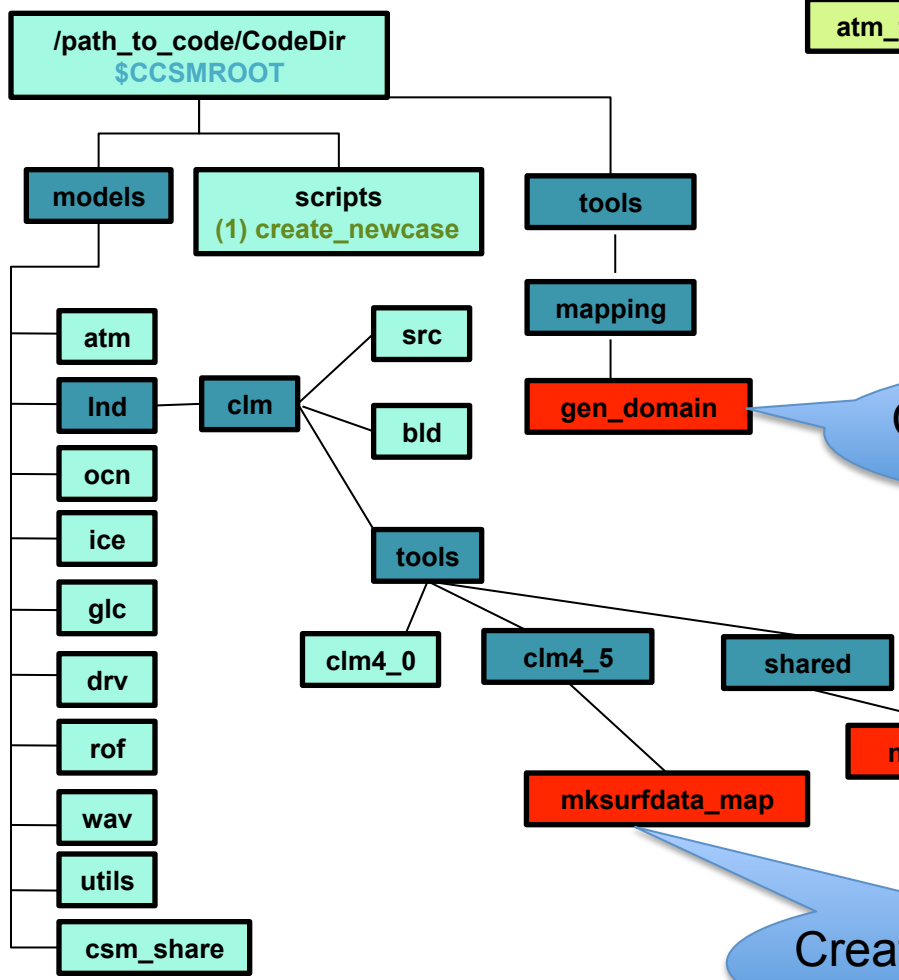
CLM tools directories



INPUTDATA Directory



CESM Code

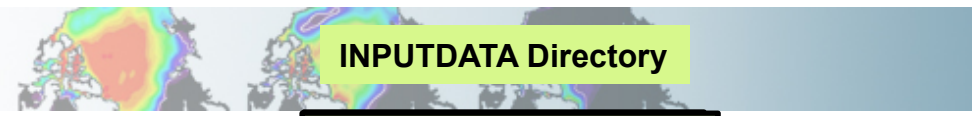


My Own DATA Directory

Creates domain files

getregional script is here

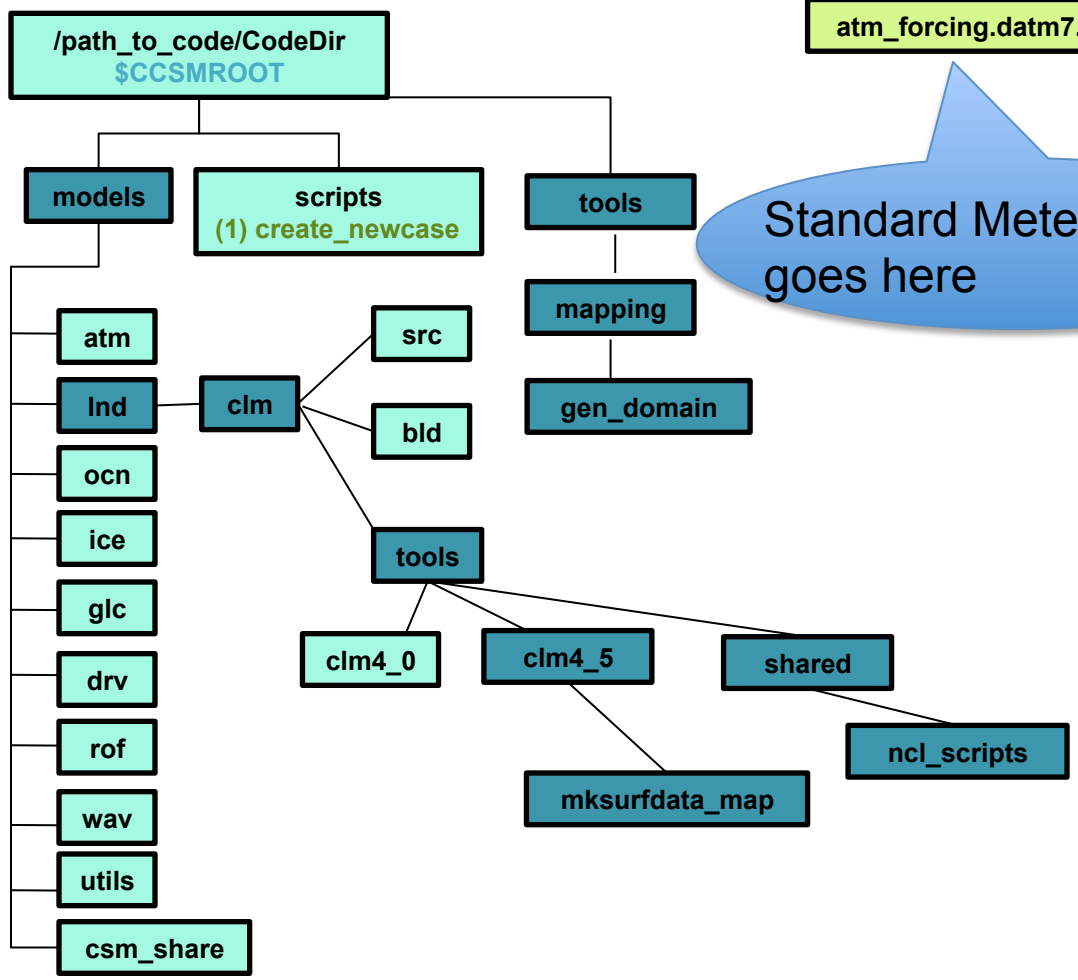
Creates surface datasets



CLM tools directories

Standard domain files go here

CESM Code

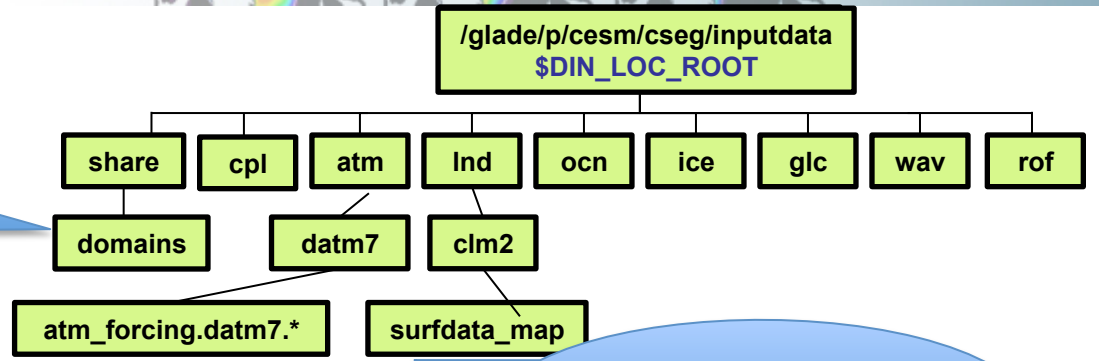


Standard Meteorology goes here

Standard surface datasets go here

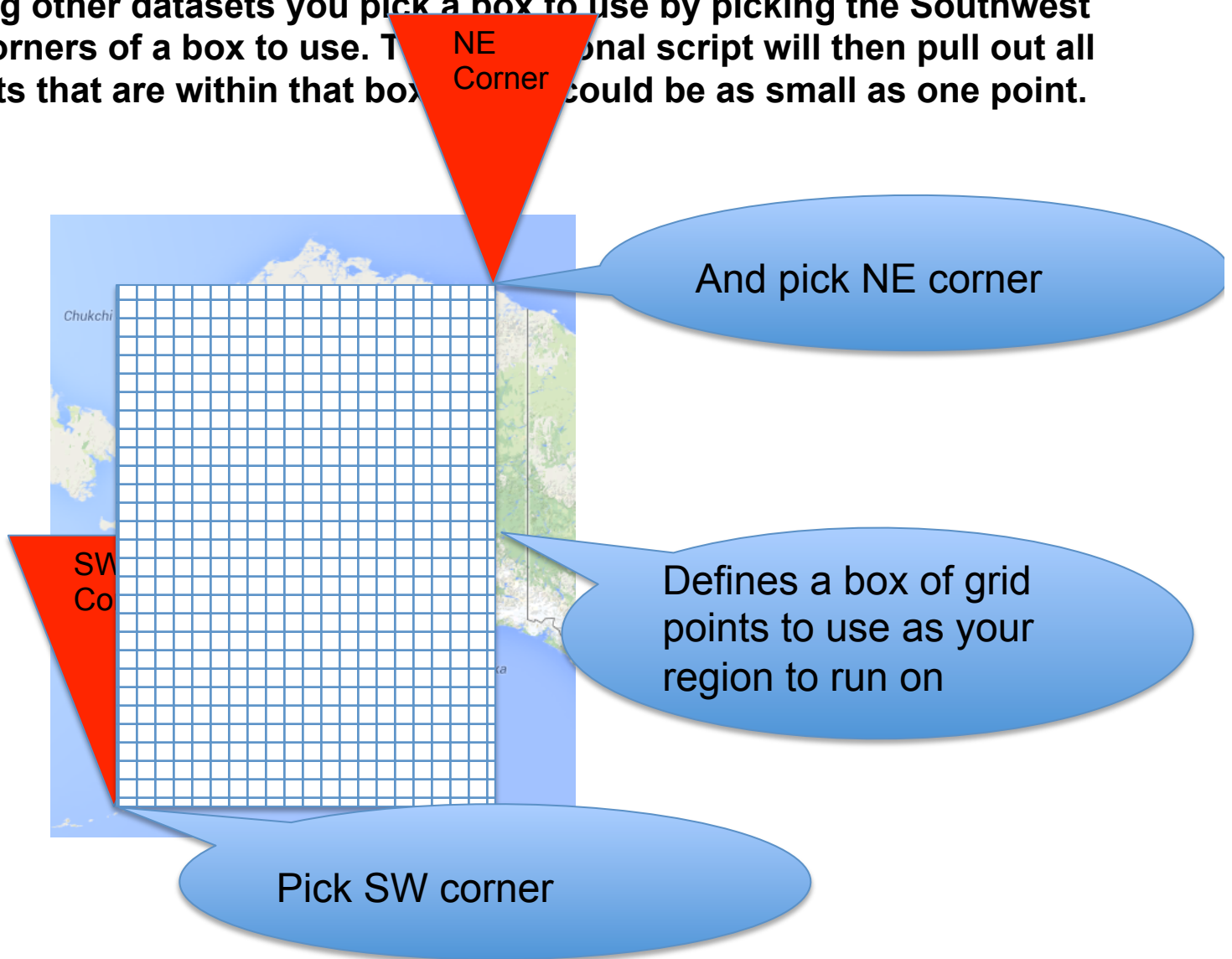
My Own DATA Directory

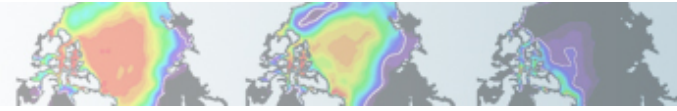
We'll put the files we create in our own directory



Sub-setting global datasets

When sub-setting other datasets you pick a box to use by picking the Southwest and Northeast corners of a box to use. The regional script will then pull out all of the grid points that are within that box. The box could be as small as one point.





Sub-setting global datasets

You can run over a smaller region (or a single point) by creating files that are a subset of the global files for these and run with the resultant files. We do that using the script in the:
models/Ind/clm/tools/shared/ncl_scripts

directory: getregional_datasets.pl

The script requires as input a list of input files, the accompanying output files for the region, and the latitude and longitude for the Southwest (SW) and Northeast (NE) corners to extract.

```
# go into the source code download
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62

# Save the location of the getregional script
cd models/Ind/clm/tools/shared/ncl_scripts
setenv GETREGDIR `pwd`

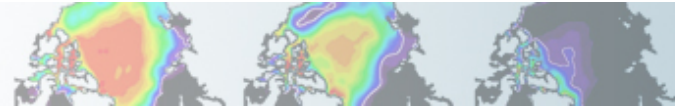
# Load NCL and NCO so we can use them
module load ncl nco

# Create a location to make datasets (here we make it in scratch space –
# IN GENERAL YOU WOULD MAKE IT IN A LOCATION YOU WOULDN'T LOSE IT RATHER THAN SCRATCH)
setenv MYDATADIR /glade/scratch/$USER/mygetregionaldata
mkdir -p $MYDATADIR
cd $MYDATADIR

# Copy the input and output lists (and in general you would edit them)
cp $GETREGDIR/sample_*.

# Make sure NCL and NetCDF-Operators are in your path
module load nco ncl

# Run getregional for a region over Alaska
$GETREGDIR/getregional_datasets.pl -ne 74.0,221.0 -sw 51.0,189.0 -i sample_inlist -o sample_outlist
```



Creating a case based on getregional files

We use the CLM_USRDAT as the resolution

We use the `-user_mods_dir` option to “./create_newcase”

This is the same mechanism you would use to create a case based on your own datasets that you create yourself (by any means).

(0) go into the scripts directory of the source code download

```
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/scripts
```

(1) create a new case in the directory “cases” in your home directory

```
./create_newcase -res CLM_USRDAT -user_mods_dir $MYDATADIR -mach yellowstone \  
-case ~/cases/Day2Alaska -compset ICLM45BGC
```

go into the case you just created in the last step

```
cd ~/cases/Day2Alaska
```

(2) invoke cesm_setup

```
./cesm_setup
```

(3) build the executable

```
./Day2Alaska.build
```

(4) submit your run to the batch queue

```
./Day2Alaska.submit
```

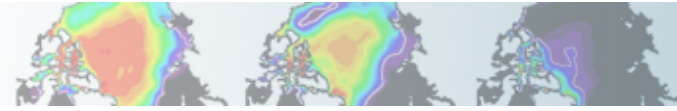


Exercise II

1. Use `getregional_datasets.pl` in the CLM tools directory to create a domain and surface dataset from the 0.9x1.25 global datasets for a region over Alaska
2. What is the creation date string (ending `_cYYMMDD` form) of the files you just created?
3. What are the name of the files created by `getregional` that you can use to run a case from (in the `$MYDATADIR` directory)?

Extra Credit

1. Create and run a case using `create_newcase` for the datasets you created
2. Why might you want to subset a region of the global datasets?
3. In what two directories in the model code tree is `gen_domain` found?
4. In what two CLM tools directories is there a program to create surface datasets?



Hints Exercise II

- 2. What is the creation date string (ending _cYYMMDD form) of the files you just created?**

Look in the \$MYDATADIR directory for the NetCDF files (*.nc) that were created by getregional_datasets.pl.
The output filenames are determined by the "sample_outlist" file and thus by the file creation dates in that file.
- 3. What are the name of the files created by getregional that you can use to run a case from (in the \$MYDATADIR directory)?**

Besides the NetCDF files created, and the "sample_*" files you copied in there are two files that getregional_datasets.pl creates in order to help setup a case from the files created.

Hints Extra Credit

- 2. Why might you want to subset a region of the global datasets?**

Sub-setting a region from global datasets, reduces the total number of grid points. What resources might you save by reducing the number of grid points?
- 3. In what two directories in the model code tree is gen_domain found?**

The two places tools are found is under: tools and models/Ind/clm/tools
Look in both places for the directory to make domain files.
- 4. In what two CLM tools directories is there a program to create surface datasets?**

The CLM tools are in the "models/Ind/clm/tools" directory.

```
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/models/Ind/clm/tools
```

Look in the README file in that directory, to see what the directory names are to create surface datasets.

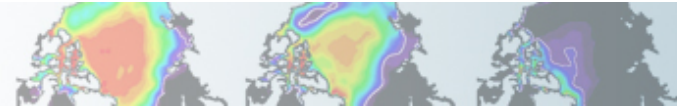


Exercise II

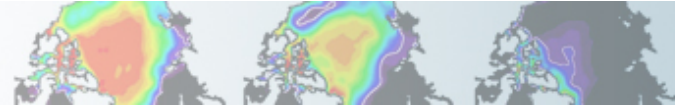
1. Use `getregional_datasets.pl` in the CLM tools directory to create a domain and surface dataset from the 0.9x1.25 global datasets for a region over Alaska
2. What is the creation date string (ending `_cYYMMDD` form) of the files you just created?
 - **c131122 Nov/22/2013**
3. What are the name of the files created by `getregional` that you can use to run a case from (in the `$MYDATADIR` directory)?
 - **user_nl_clm and xmlchange_cmnds**

Extra Credit

1. Create and run a case using `create_newcase` for the datasets you created
2. Why might you want to subset a region of the global datasets?
 - **Save time and computing resources. Customize the data. We show it here to show dataset creation.**
3. In what two directories in the model code tree is `gen_domain` found?
 - **./models/Ind/clm/tools/shared/gen_domain**
 - **./tools/mapping/gen_domain_files/**
3. In what two CLM tools directories is there a program to create surface datasets?
 - **./models/Ind/clm/tools/clm4_5/mksurfddata_map**
 - **./models/Ind/clm/tools/clm4_0/mksurfddata_map**

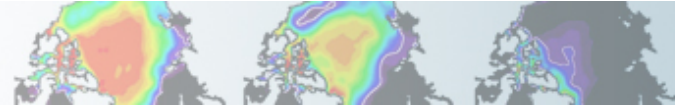


Section III – Using PTCLMmkdata



Objectives for Section III (Using PTCLMmkdata)

1. Give an introduction to the tools that make CLM input files.
2. Give an overview of the process to create input files.
3. Go over the steps that PTCLMmkdata uses to create datasets.
4. Create some datasets using PTCLMmkdata.
5. Create a case based on the datasets created.



What is PTCLM?

PTCLM is a script that helps you:

- Create
- Manage
- Run

Datasets for running CLM based on tower site data for a single-point.

It works similar to getregional to create datasets and files that point to them, that can then be put into a case.

The meteorology that you force the sites with can either be from:

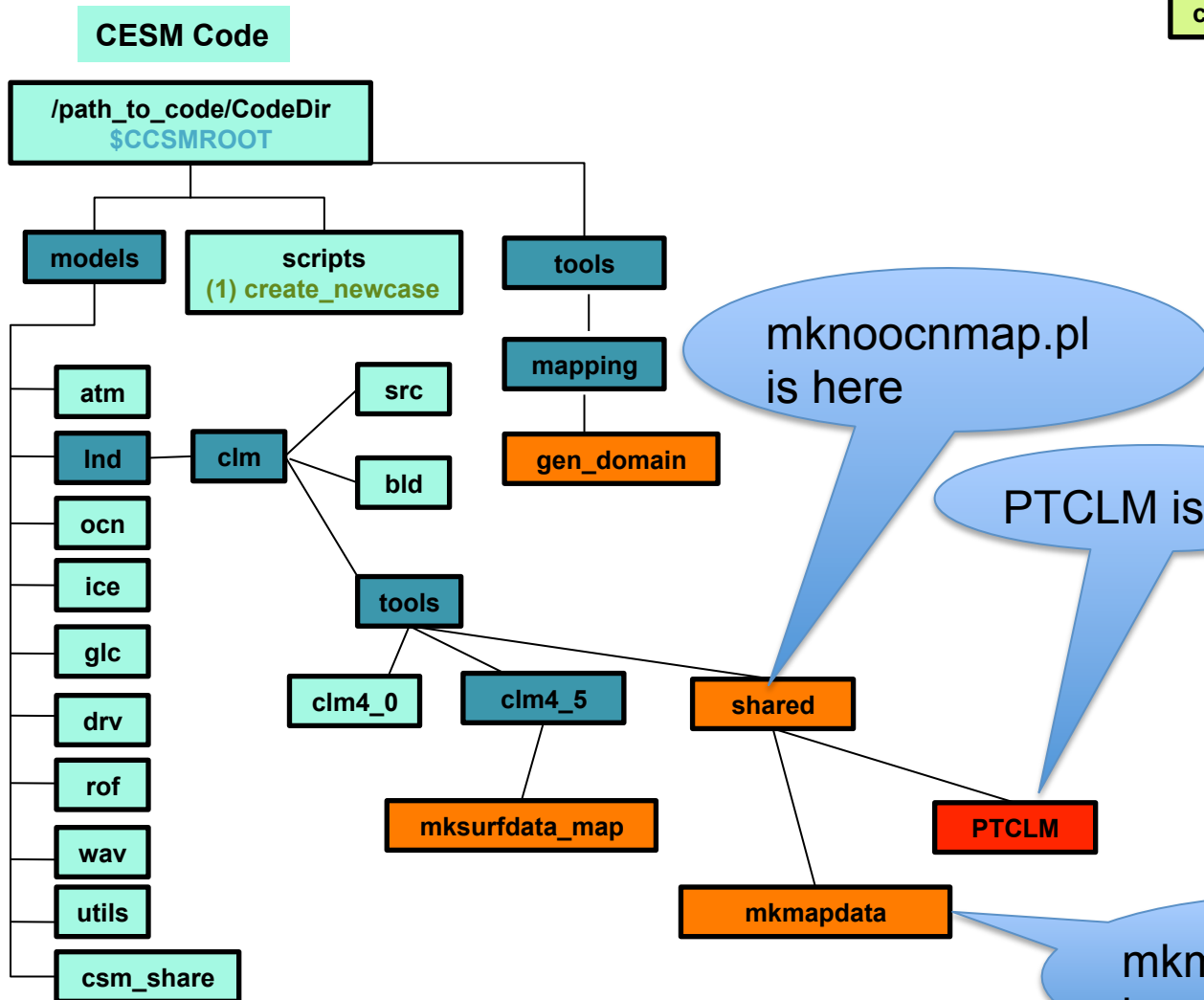
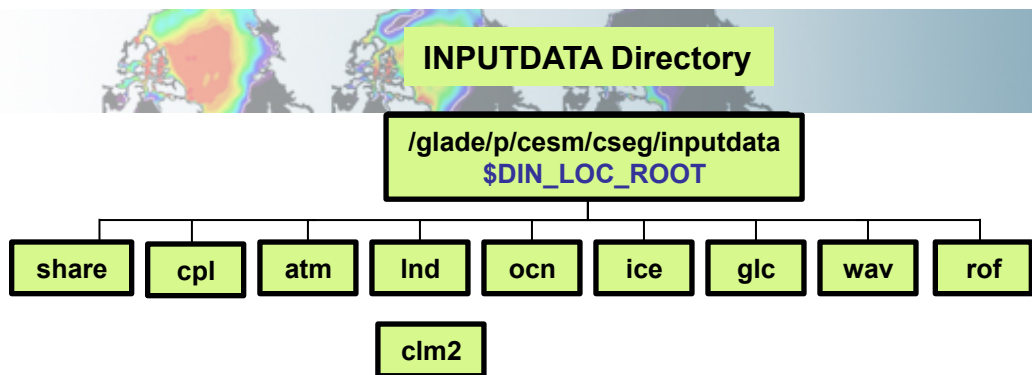
Tower site

Standard global forcing

Depending on the compset (DATM_MODE) you choose.



PTCLM directory



My Own DATA Directory

mknoocnmap.pl is here

PTCLM is here

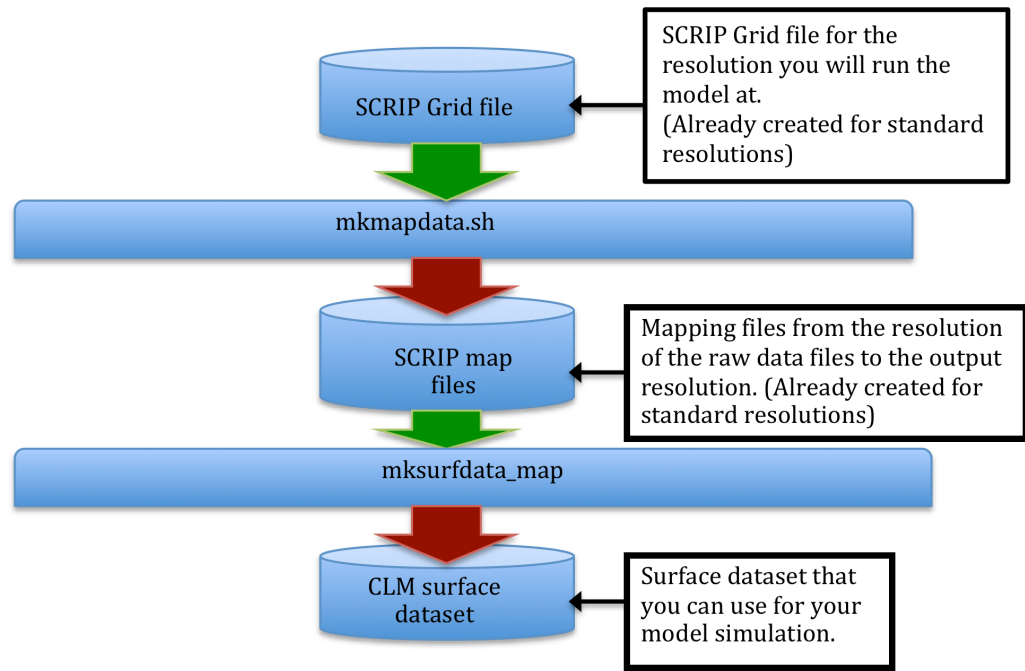
PTCLM runs the programs:

- mknoocnmap.pl
- gen_domain
- mkmapdata.sh
- mk surfdata_map

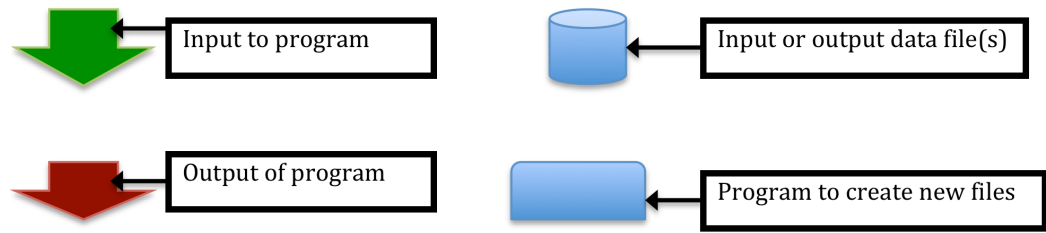
mkmapdata.sh is here

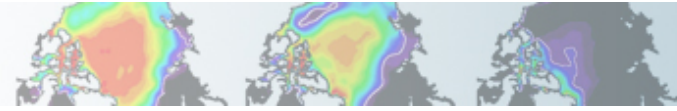


What is the data flow for creating datasets?

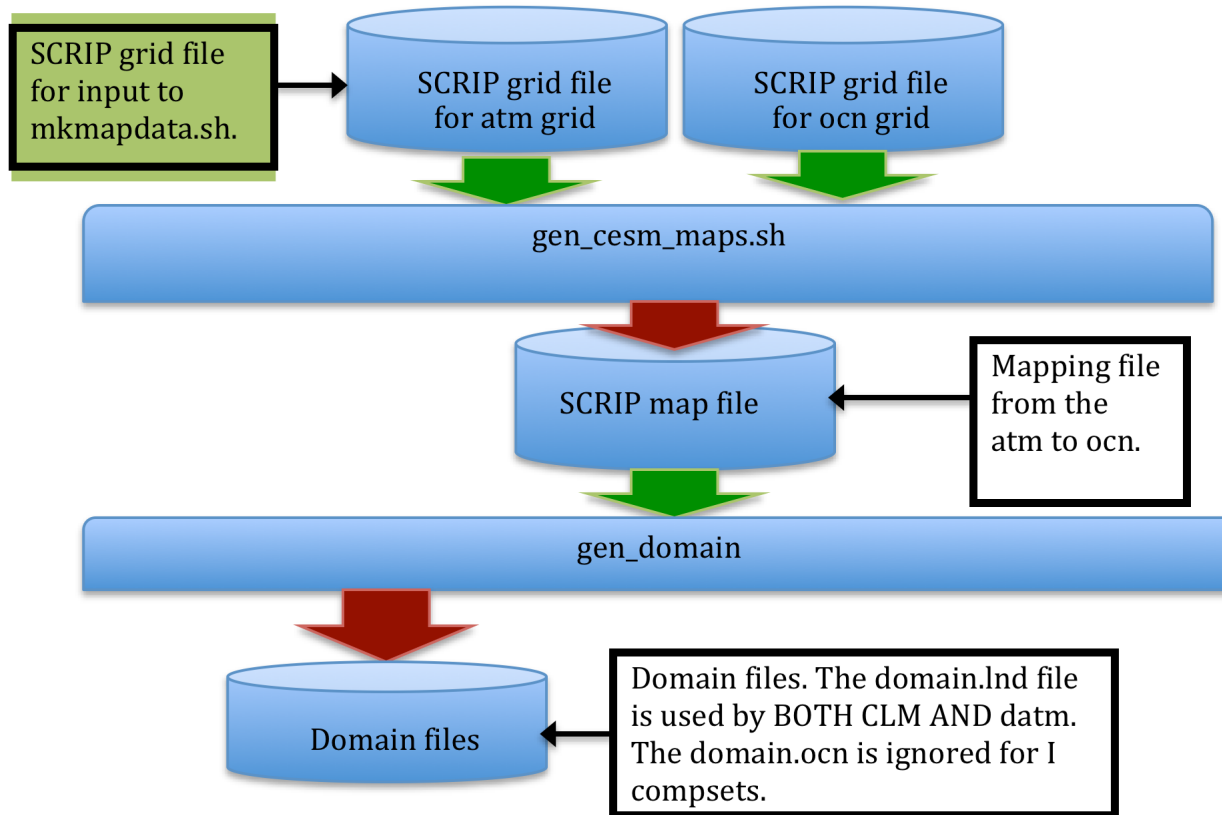


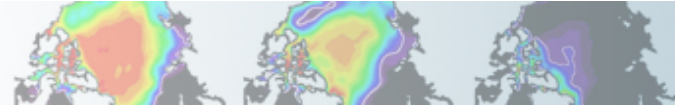
LEGEND





What is the data flow for creating the domain file?





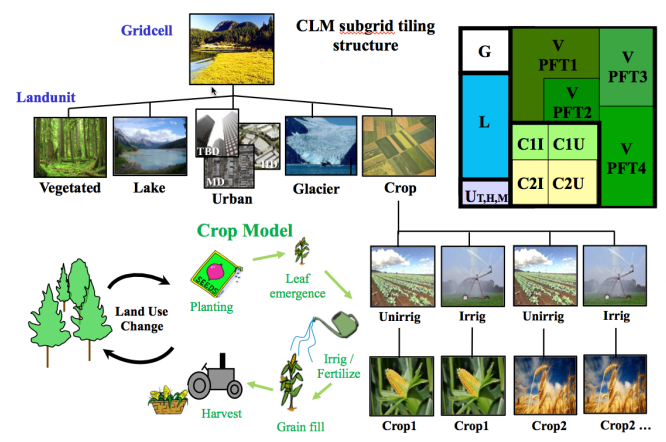
What are the steps that PTCLMmkdata does for you to create datasets?

1. Runs `mknoocnmap.pl` to create the mapping file that is required by CESM.
2. Runs `gen_domain` to create your domain file.
3. Runs `mkmapdata.sh` to create mapping files between your point and the raw datasets that `mksurfddata_map` uses.
4. Runs `mksurfddata.pl` to create your surface dataset.
5. Puts the datasets into a directory that also has files to help setup your case and use the datasets just created (`user_nl` files and `xmlchange_cmnds` file).



What the deal with the mapping files?

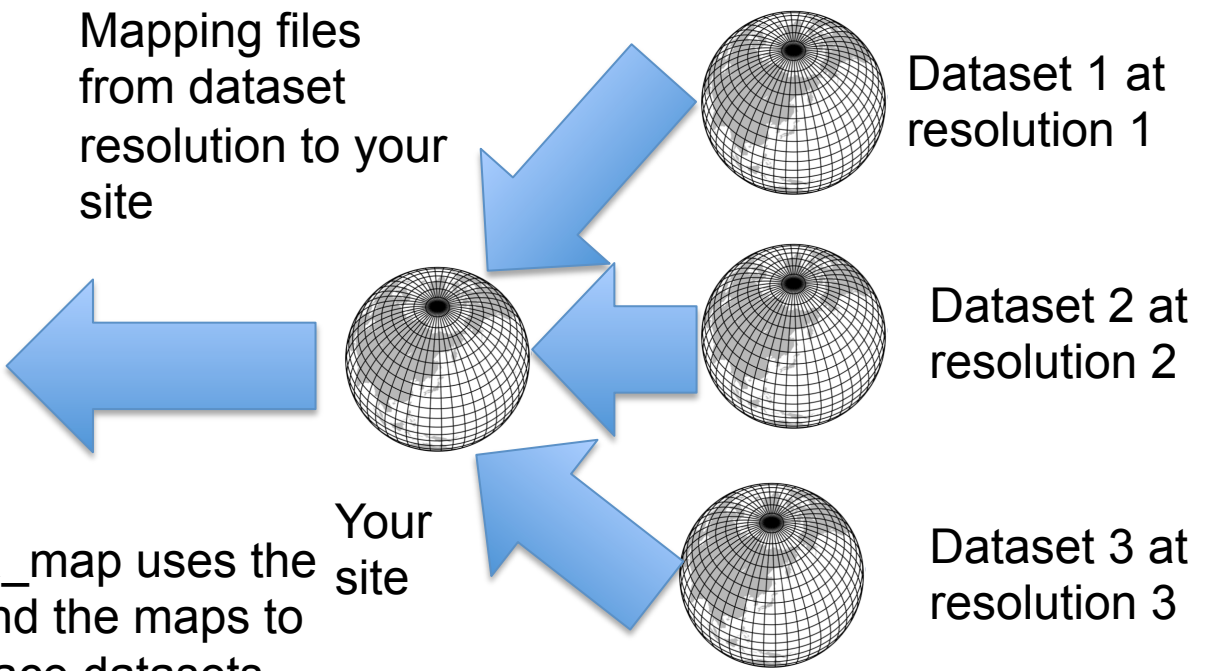
To create surface datasets you read in various “raw” datasets (for PFT, lake, glacier, etc.) at different resolutions and regrid them to your site. The mapping files regrid from the “raw” dataset resolution to your output site.

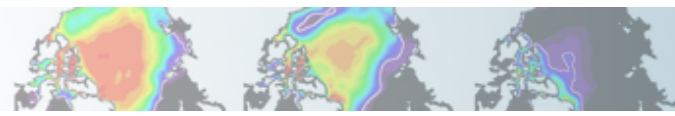


Surface Dataset

Mksurpdata_map uses the datasets and the maps to create surface datasets

Mapping files from dataset resolution to your site





OK so how do I run PTCLM now?

PTCLMmkdata as we said before runs the other file creation scripts to create datasets for you. This is easier than learning each of the previous programs. **However, if something goes wrong you may have to look into the programs that PTCLMmkdata calls to figure out what is up.**

Setup the names that we'll use...

```
setenv CESM_ROOT /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62
setenv CSMDATA /glade/p/cesmdata/cseg/inputdata
setenv MYSITE 1x1pt_US-UMB
```

Setup your data directory (NOTE: NORMALLY THIS WOULD NOT BE IN SCRATCH SPACE!!!!)

```
setenv MYDATADIR /glade/scratch/$USER/mydatadir
mkdir -p $MYDATADIR/$MYSITE
```

Copy the mapping files over so they won't have to be created again

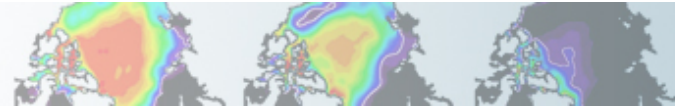
```
cp -p $CSMDATA/Ind/clm2/PTCLMmydatafiles/$MYSITE/map_*_c140121.nc $MYDATADIR/$MYSITE/
```

Make sure ncl is in your path

```
module load ncl
```

Run PTCLM giving it the creation date string for the mapping files, the site name,
where the standard input files are located and the output directory to put your files
They will be put into a subdirectory with the site-name under this directory

```
$CESM_ROOT/models/Ind/clm/tools/shared/PTCLM/PTCLMmkdata \  
--map_gdate 140121 -s US-UMB -d $CSMDATA --mydatadir $MYDATADIR
```



Creating a case based on PTCLMmkdata files

(0) go into the scripts directory of the source code download

```
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/scripts
```

(1) create a new case in the directory “cases” in your

home directory

```
./create_newcase -res CLM_USRDAT -user_mods_dir \  
$MYDATADIR/$MYSITE -mach yellowstone \  
-case ~/cases/Day2UMB -compset I1PTCLM45
```

go into the case you just created in the last step

```
cd ~/cases/Day2UMB
```

(2) invoke cesm_setup

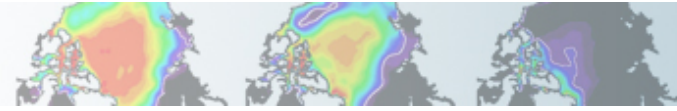
```
./cesm_setup
```

(3) build the executable

```
./Day2UMB.build
```

(4) submit your run to the batch queue

```
./Day2UMB.submit
```

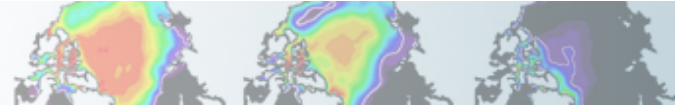


Exercise III

- 1. Run PTCLMmkdata for the US-UMB site**
- 2. Create a case from it**
- 3. Try out the informational options to PTCLMmkdata (-help etc.)**

Extra Credit

- 1. How many sites is PTCLM currently setup for?**
- 2. In what subdirectory beneath the PTCLM tool directory would you add new sites to work on?**
- 3. What files in that subdirectory would you need to edit to add a new site?**
- 4. What options to PTCLMmkdata will result in changes to your surface dataset you create?**
- 5. What options to PTCLMmkdata will change how your case is setup (but not change datasets)?**



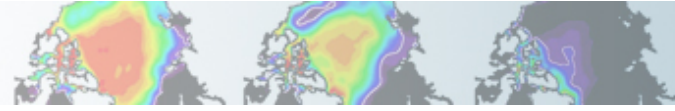
Hints Exercise III

- 3. Try out the informational options to PTCLMmkdata (-help etc.)**
The informational options are: --help, --version, --list

Try each of those out in turn.

Hints Extra Credit

- 1. How many sites is PTCLM currently setup for?**
Use the "--list" option to list all the sites. Or you can look at the list of sites in the site data file: "PTCLM_sitedata/PTCLMDATA_sitedata.txt".
- 2. In what subdirectory beneath the PTCLM tool directory would you add new sites to work on?**
The README file in PTCLM tells you about this directory.
- 3. What files in that subdirectory would you need to edit to add a new site?**
Look for the files in the directory with the names: PTCLMDATA_*.txt
- 4. What options to PTCLMmkdata will result in changes to your surface dataset you create?**
Use the "--help" option to figure out which options change the surface datasets created.
- 5. What options to PTCLMmkdata will change how your case is setup (but not change datasets)?**
Use the "--help" option to figure out which options don't change the datasets, but change the case setup.

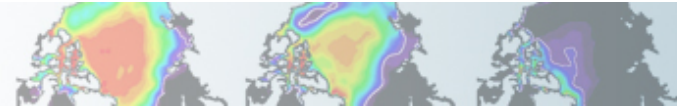


Exercise III

1. Run PTCLMmkdata for the US-UMB site
2. Create a case from it
3. Try out the informational options to PTCLMmkdata (--help etc.)
 - Try the options: --help, --list, and --version

Extra Credit

1. How many sites is PTCLM currently setup for?
 - 38
2. In what subdirectory beneath the PTCLM tool directory would you add new sites to work on?
 - PTCLM_sitedata
3. What files in that subdirectory would you need to edit to add a new site?
 - PTCLMDATA_pftdata.txt PTCLMDATA_sitedata.txt PTCLMDATA_soildata.txt
4. What options to PTCLMmkdata will result in changes to your surface dataset you create?
 - --site, --phys, --pftgrid, --soilgrid, --mksurfddata_opts
5. What options to PTCLMmkdata will change how your case is setup (but not change datasets)?
 - --cycle_forcing, --do_not_use_tower_yrs, --clmmlusecase, and --phys (changes both)



CESM1 Tutorial: Single Point / Regional Cases

Have a Singularly Fun Time!!!

The more you become familiar with these tools by playing around with them the more you'll be able to use them.