

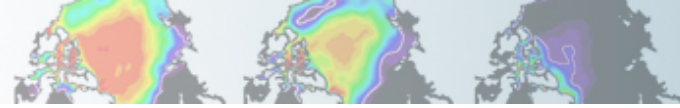
CLM4.5 Tutorial: Basic Modifications

Danica Lombardozzi



U.S. DEPARTMENT OF
ENERGY

Office of
Science



Review: The 4 commands to run CLM

Set of commands to build and run the model on a supported machine: "yellowstone"

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 ~/I1850CLM45_001 -res f19_g16 -compset I1850CLM45 -mach yellowstone
```

go into the case you just created in the last step

```
cd ~/I1850CLM45_001
```

(2) invoke cesm_setup

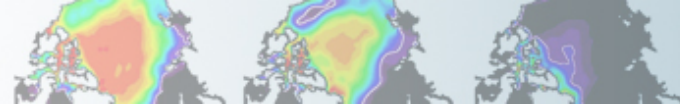
```
./cesm_setup
```

(3) build the executable

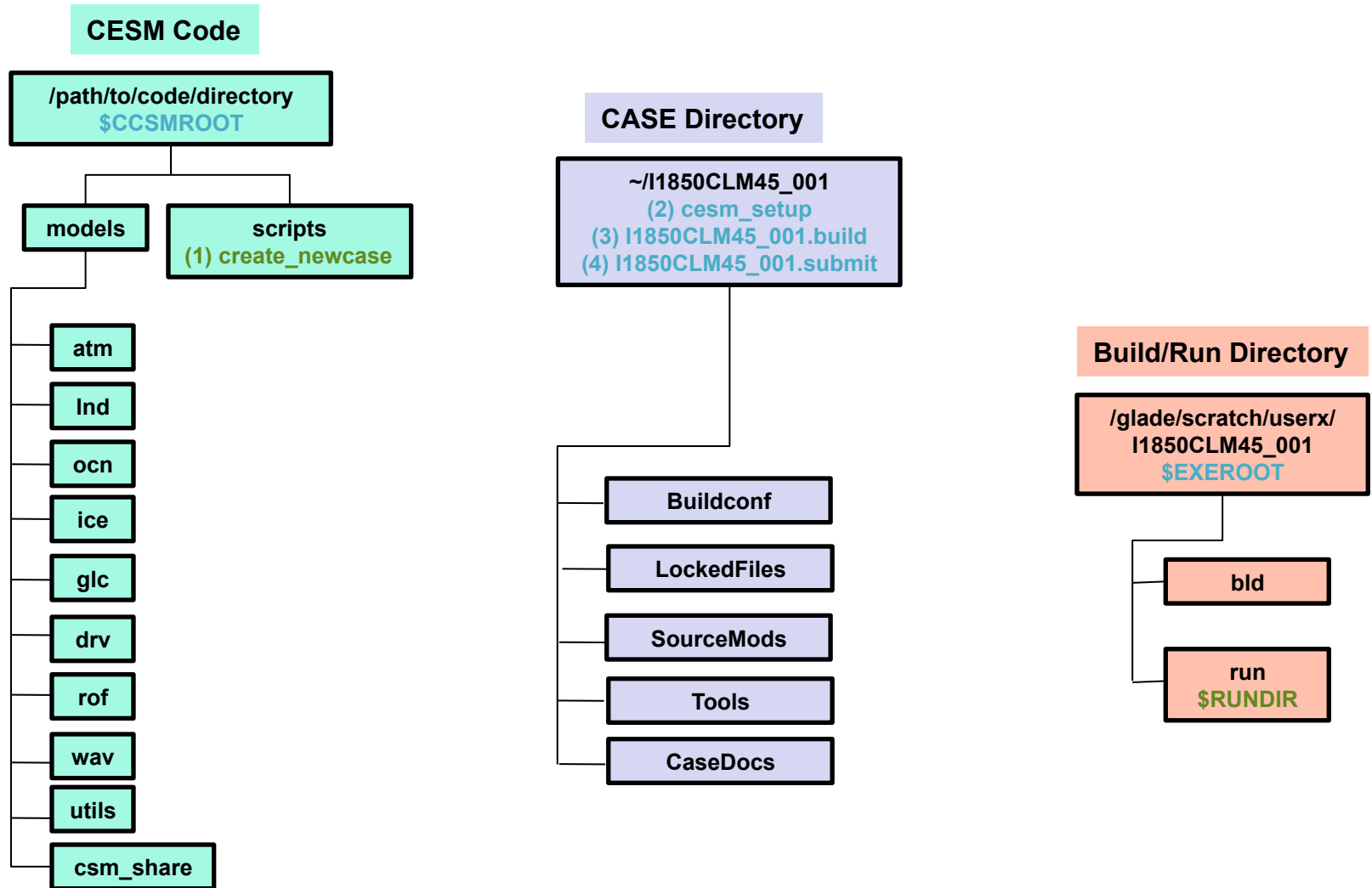
```
./I1850CLM45_001.build
```

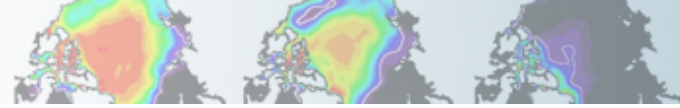
(4) submit your run to the batch queue

```
./I1850CLM45_001.submit
```

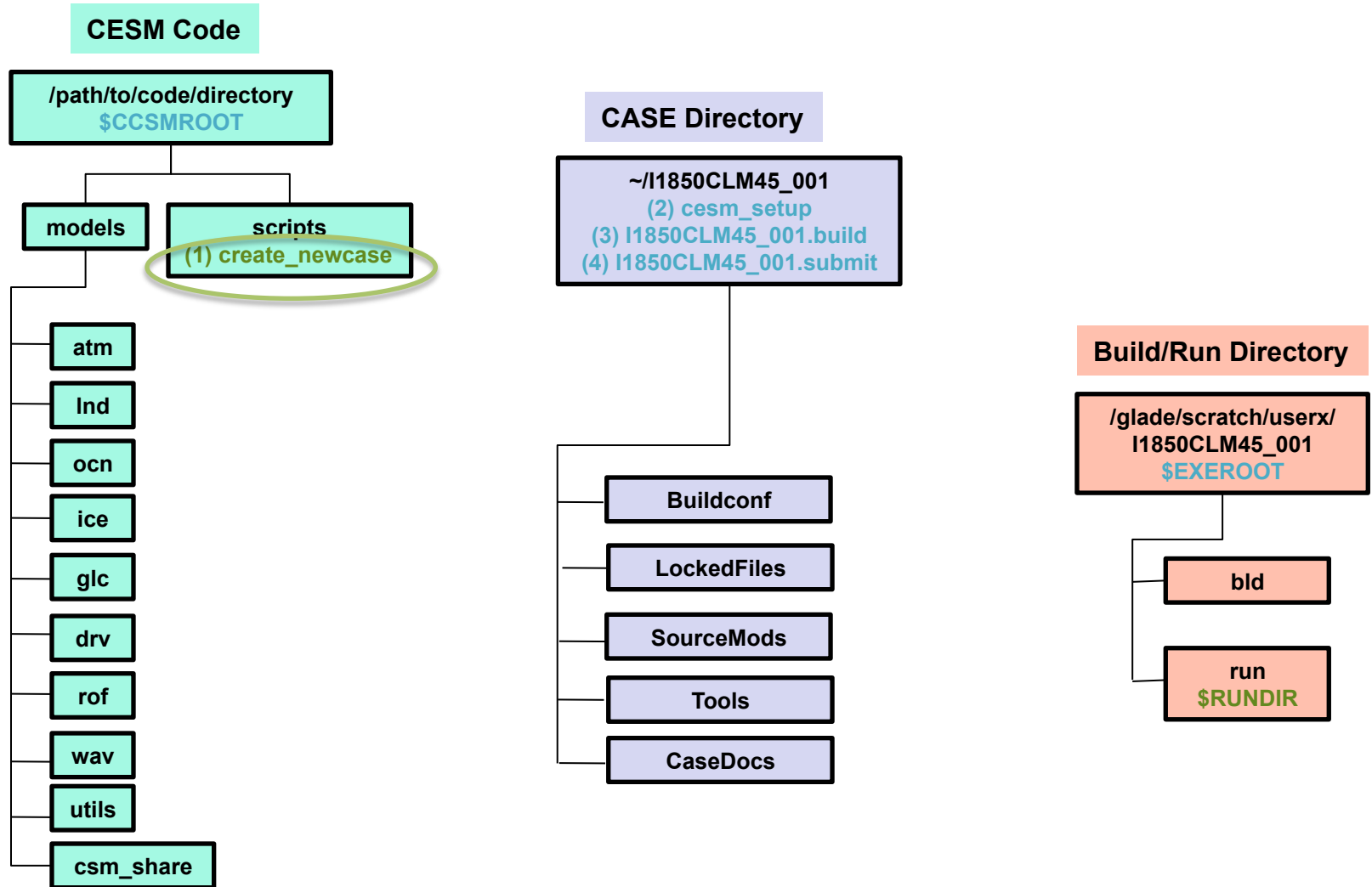


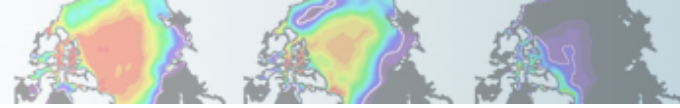
CLM directories & commands



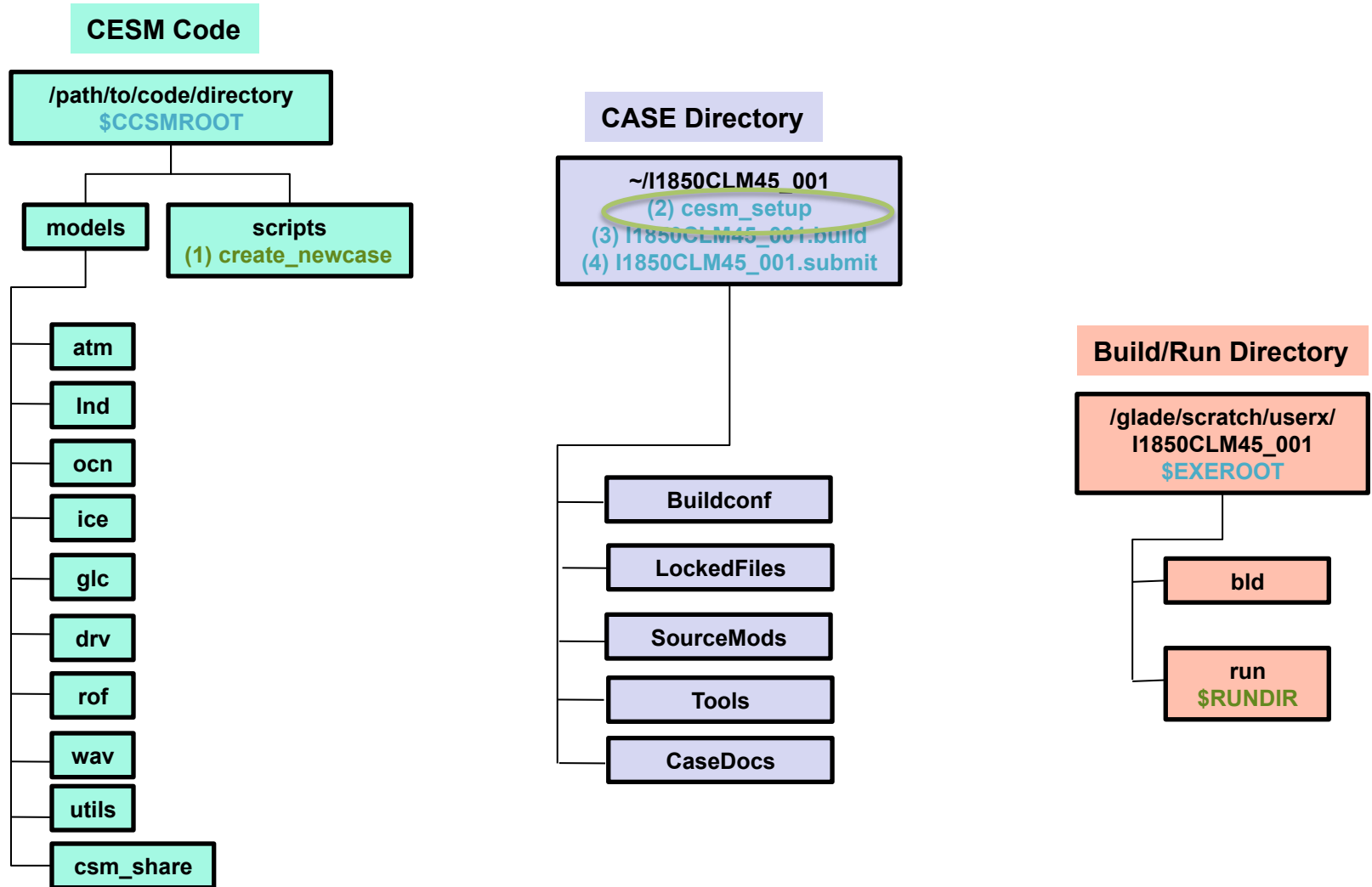


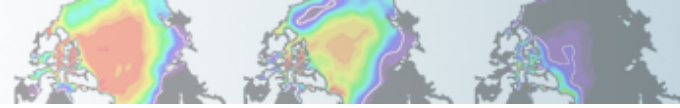
CLM directories & commands



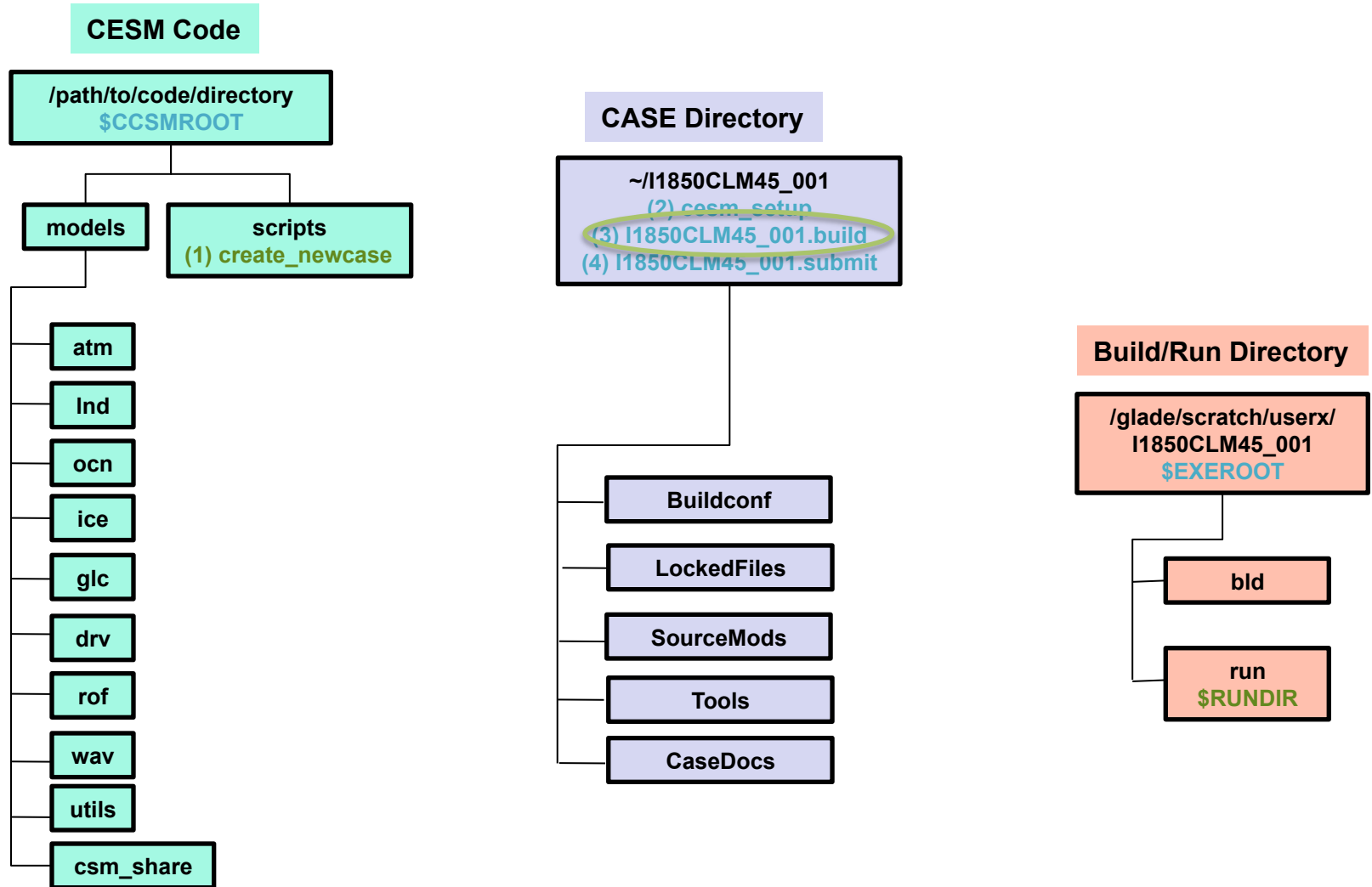


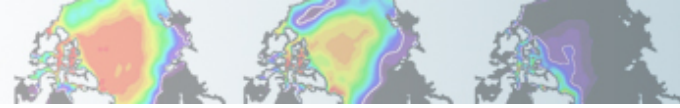
CLM directories & commands



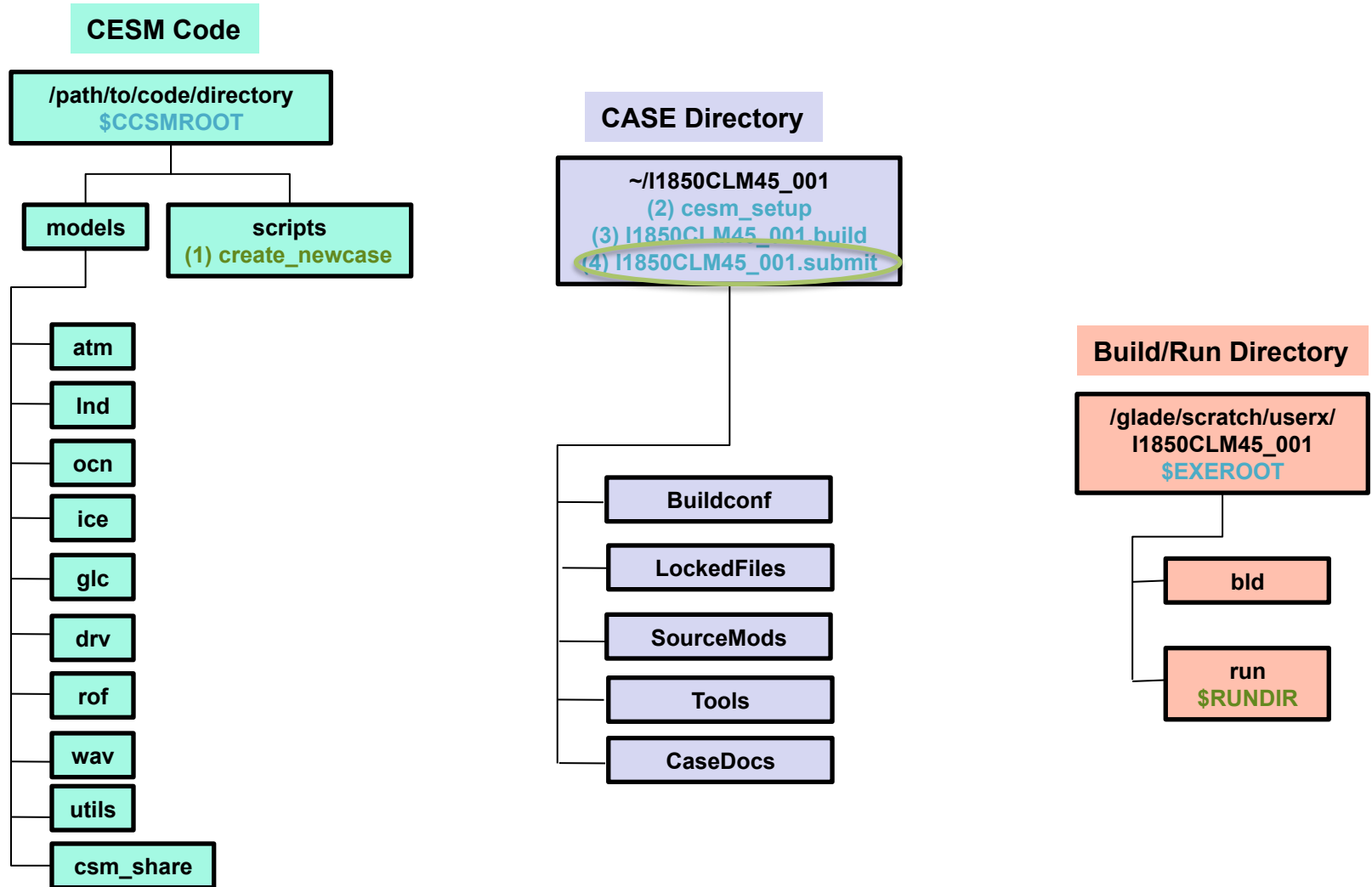


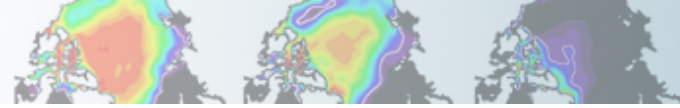
CLM directories & commands





CLM directories & commands





Review: Queues and Jobs

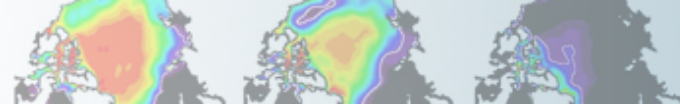
Yellowstone

Checking jobs:

- a. Type *bjobs* or
- b. Type *bjobs -u all* to see everyone's jobs, or

Killing jobs:

- a. Find your JOBID after typing *bjobs*
- b. Type *bkill <JOBID>*

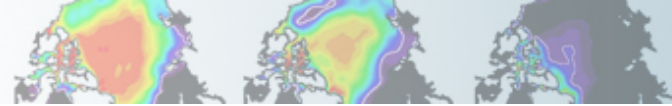


Finding model output

Directory:

Change this to your user name

`/glade/scratch/{userXX}/archive/I850CLM45_001/Ind/hist`



Finding model output

Directory:

Change this to your user name

/glade/scratch/{userXX}/archive/I1850CLM45_001/Ind/hist

Files (use “/s” to list them):

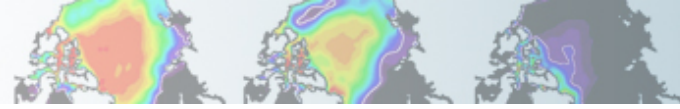
I1850CLM45_001.h0.0001-12.nc

Case Name

Time

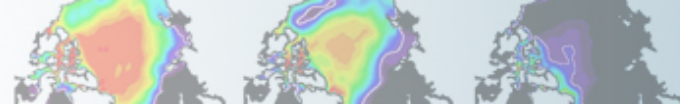
Output Type
(history)

File Type
(netCDF)



3 Types of Basic Modifications

1. Component Sets
2. ENV files (`env_[command]`)
3. Namelist files (`user_nl_[model]`)



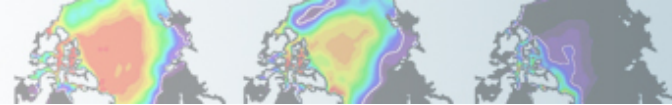
3 Types of Basic Modifications

1. Component Sets

Set up a simulation for 2000

2. ENV files (env_[command])

3. Namelist files (user_nl_[model])



Creating a new case

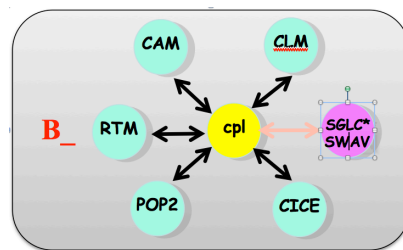
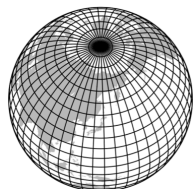
`create_newcase` requires 4 arguments

What is the casename ?

Which resolution?

**Which model configuration ?
Which set of components ?**

Which machine are you running on?



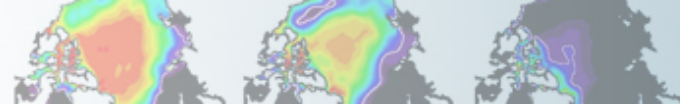
YourCaseName

**f19_g16
(2-degree)**

**I1850CLM45
(I = CLM only, 1850)**

yellowstone

```
./create_newcase -case ~/I1850CLM45_001 -res f19_g16 -compset I1850CLM45 -mach yellowstone
```



Creating a new case

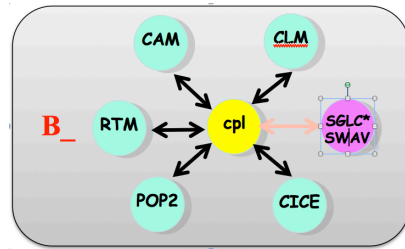
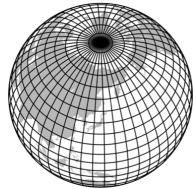
create_newcase requires 4 arguments

What is the casename ?

Which resolution?

Which model configuration ?
Which set of components ?

Which machine are you running on?



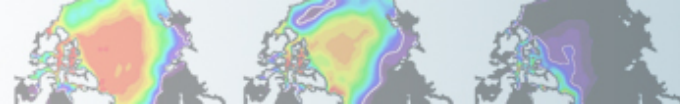
YourCaseName

f19_g16
(2-degree)

I1850CLM45
(I = CLM only, 1850)

yellowstone

```
./create_newcase -case ~/I1850CLM45_001 -res f19_g16 -compset I1850CLM45 -mach yellowstone
```

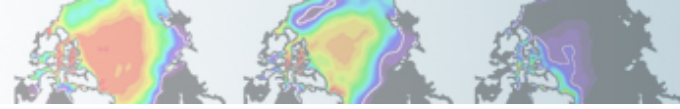


Changing Simulation Components

Compset, or component set:

predefined options for running the model

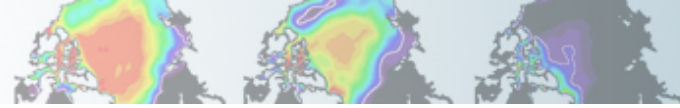
Use compset to change the type of simulation



Changing compsets lets you run different experiments

Component options:

- Year (1850, 2000, transient, etc.)
- Data atmosphere (Qian, CRUNCEP, CPLHIST3HrWx)
- Model options (SP [satellite phenology], BGC [biogeochemistry])
- RCP scenarios



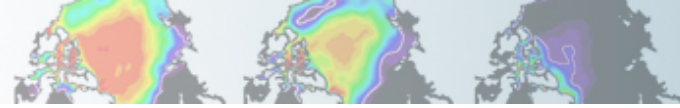
Changing compsets lets you run different experiments

Component options:

- Year (1850, 2000, transient, etc.)
- Data atmosphere (Qian, CRUNCEP, CPLHIST3HrWx)
- Model options (SP [satellite phenology], BGC [biogeochemistry])
- RCP scenarios

Examples of simulations using different compsets:

- Stabilize (“spin up”) a biogeochemistry (includes N & C cycles) simulation for 1850
- Run a transient historical simulation from 1850-2000 based on the 1850 spin up
- Run a transient future simulation from 2000 through 2100 using RCP8.5
- Run a “time slice” simulation for 2000



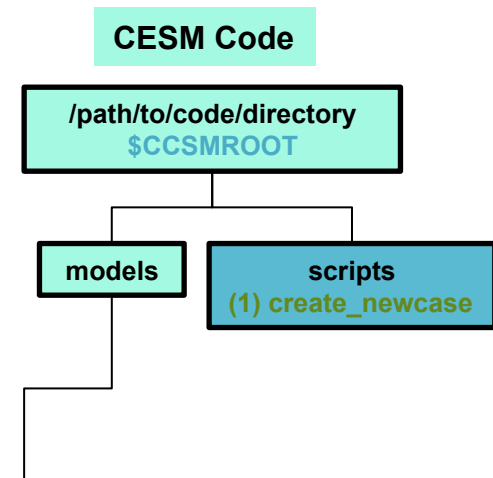
Where to find a list of compsets:

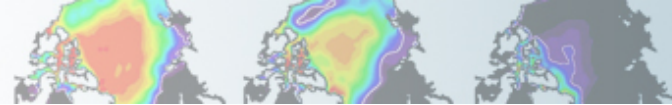
http://www.cesm.ucar.edu/models/cesm1.2/clm/scripts/ccsm_utils/Case.template/config_compsets.xml

- Website lists ALL compsets for CESM. CLM only = “I” compsets

In CESM scripts directory, can run:
`./create_newcase –list compsets`

Tip: Add “ | more” at the end of the command line, then use the spacebar to scroll through the options





Exercise 1: Create & build simulation for 2000

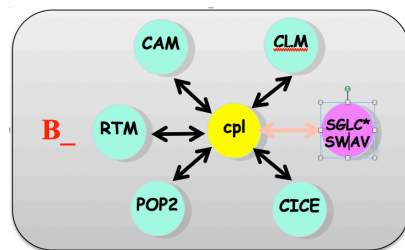
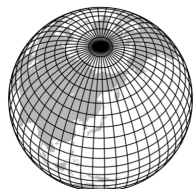
`create_newcase` requires 4 arguments

What is the casename ?

Which resolution?

Which model configuration ?
Which set of components ?

Which machine are you running on?



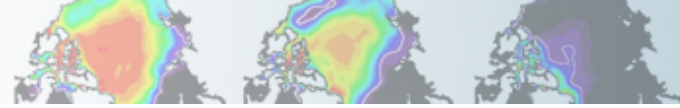
YourCaseName

f19_g16
(2-degree)

ICLM45
(I = CLM only, 2000)

yellowstone

```
./create_newcase -case ~/I2000CLM45_001 -res f19_g16 -compset ICLM45 -mach yellowstone
```



Exercise 1: Create & build simulation for 2000

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 ~/I2000CLM45_001 -res f19_g16 -compset ICLM45 -mach yellowstone
```

go into the case you just created in the last step

```
cd ~/I2000CLM45_001
```

(2) invoke cesm_setup

```
./cesm_setup
```

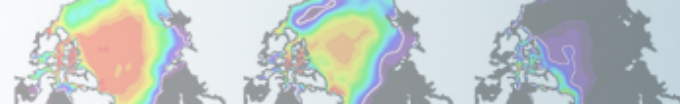
(3) build the executable

```
./I2000CLM45_001.build
```

Stop Here

(4) submit your run to the batch queue

```
./I2000CLM45_001.submit
```



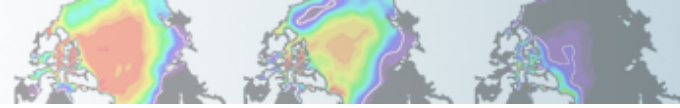
3 Types of Basic Modifications

1. Component Sets

2. ENV files (`env_[command].xml`)

Changing the length of the run

3. Namelist files (`user_nl_[model]`)



Review: The 4 commands to run CLM

Set of commands to build and run the model on a supported machine: "yellowstone"

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 ~/I2000CLM45_001 -res f19_g16 -compset ICLM45 -mach yellowstone
```

go into the case you just created in the last step

```
cd ~/I2000CLM45_001
```

(2) invoke cesm_setup

```
./cesm_setup
```

(3) build the executable

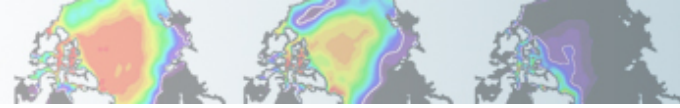
```
./I2000CLM45_001.build
```



Change the run length BEFORE submitting

(4) submit your run to the batch queue

```
./I2000CLM45_001.submit
```



ENV files

Example 2. Simulations for a different length of time

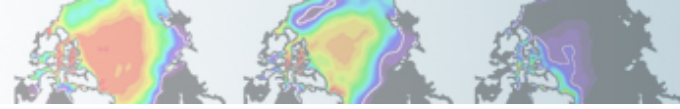
Two steps to change run length:

- 1) Modify `env_run.xml` to set desired simulated length
- 2) Modify `I2000CLM45_001.run` to tell computer how much computer time is needed to complete the simulation (Wall Clock Time)

When modifying files, use an editor of your choice

Examples:

emacs
nedit
vi



“xmlchange” command

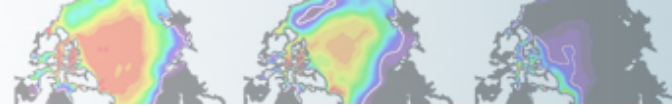
Use when modifying “xml” files (e.g. env_run.xml)

1. Benefit: Won’t let you mess up the syntax!
2. For help, type *./xmlchange -help*
3. Use *./xmlquery list* to list all variables and their values in all the .xml files

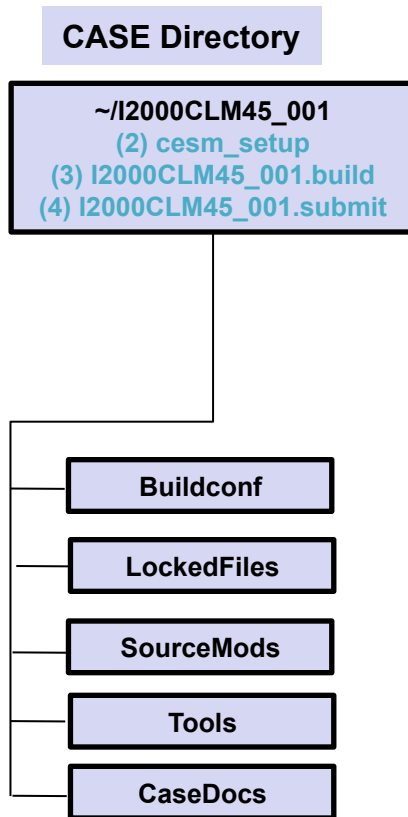
Example: editing env_run.xml via the xmlchange tool

```
./xmlchange {variable to be changed}={value to change to}
```

* We won’t use xml commands right now, but you will during the next section.



1) Modify `env_run.xml` to set desired simulated length



In a text editor*, open `env_run.xml`

* If you don't have a preferred editor, emacs is more user friendly.

Type "emacs `env_run.xml`" (or "emacs anyfilename")

```

<!-- ===== -->
<!-- -->
<!-- These variables MAY BE CHANGED ANYTIME during a run. -->
<!-- Additional machine specific variables that can be changed -->
<!-- during a run are contained in the env_mach_specific file -->
<!-- -->
<!-- Note1: users SHOULD NOT modify BUILD_COMPETE in env_build.xml -->
<!-- this is done automatically by the scripts -->
<!-- ===== -->

<!--"case run directory (by default will be set to $EXEROOT/./run) (char) " -->
<entry id="RUNDIR" value="/glade/scratch/$CCSMUSER/$CASE/run" />

<!--"CCSM tag (char) " -->
<entry id="CCSM_REPOTAG" value="" />

<!--"case description (char) " -->
<entry id="CASESTR" value="UNSET" />

<!-- ===== -->

<!--"Run initialization type, valid values: startup,hybrid,branch (char) " -->
<entry id="RUN_TYPE" value="startup" />

<!--"Run start date (yyyy-mm-dd). Only used for startup or hybrid runs (char) " -->
<entry id="RUN_STARTDATE" value="0001-01-01" />

<!--"start time-of-day (integer) " -->
<entry id="START_TOD" value="0" />

<!--"Reference case for hybrid or branch runs (char*256) " -->
<entry id="RUN_REFCASE" value="case.std" />

<!--"Reference date for hybrid or branch runs (yyyy-mm-dd) (char*10) " -->
<entry id="RUN_REFDATE" value="0001-01-01" />

<!--"Reference time of day (seconds) for hybrid or branch runs (sssss) (char) " -->
<entry id="RUN_REFTOD" value="00000" />

<!--"allow same branch casename as reference casename, valid values: TRUE,FALSE (logical) " -->
<entry id="BRNCH_RETAIN_CASENAME" value="FALSE" />

<!--"flag for automatically prestaging the refcase restart dataset, valid values: TRUE,FALSE (logical) " -->
<entry id="GET_REFCASE" value="FALSE" />

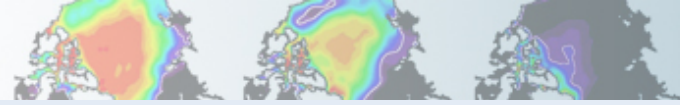
<!-- ===== -->

<!--"sets the run length with STOP_N and STOP_DATE (must be nyear(s) for _GLC compsets for restarts to work properly), valid values: none,never,nsteps,nstep,
nseconds,nsecond,nminutes,nminute,nhours,nhour,ndays,nday,nmonths,nmonth,nyears,nyear,date,ifdays0,end (char) " -->
<entry id="STOP_OPTION" value="nyears" />

<!--"sets the run length with STOP_OPTION and STOP_DATE (integer) " -->
<entry id="STOP_N" value="20" />

<!--"date in yyyyymmdd format, sets the run length with STOP_OPTION and STOP_N (integer) " -->
<entry id="STOP_DATE" value="-999" />

```



env_run.xml

Runtime variables can be changed in `env_run.xml` *at any point* during the run and control the mechanics of the run (length, resubmits, and archiving).

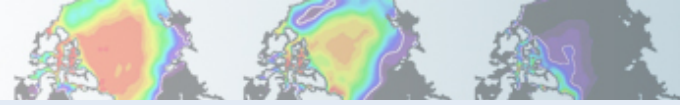
Common variables to change include

1. **STOP_OPTION** → sets the run time interval type, i.e. nmonths, ndays, nyears

2. **STOP_N** → sets the number of intervals to run the model during the specified wallclock* time.

* Wallclock time is set in your `YourCaseName.run` file and is a measure of the actual time.

3. **RESUBMIT** → sets the number of times to resubmit the run



env_run.xml

Exercise 2: Run simulation for 5 years (Part 1)

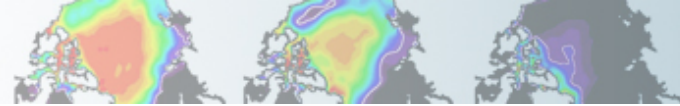
1. **STOP_OPTION** → change to “nyears”

2. **STOP_N** → change to “5”

3. **RESUBMIT** → sets the number of times to resubmit the run

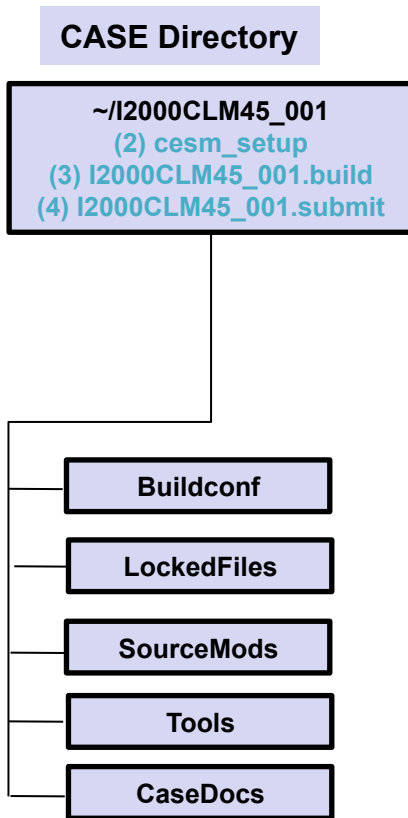
To use resubmit, can set “STOP_N” to 1, then set RESUBMIT to “4”.

** This will run 5 different simulations for 1 year each **

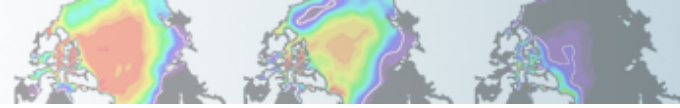


Run Scripts: Wall clock time

2) Modify **I2000CLM45_001.run** to tell computer how much computer time is needed to complete the simulation (Wall Clock Time)



Using a text editor,
open I2000CLM45_001.run



Run Scripts: Wall clock time

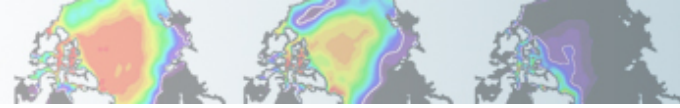
Review run script: *I2000CLM45_001.run*

Common BSUB command to change:

#BSUB -q regular	queue type (also: economy, premium, etc.)
#BSUB -o cesm.stdout.%J	machine standard out
#BSUB -e cesm.stderr.%J	machine standard error
#BSUB -J ICLM200045_001	job name
#BSUB -W 2:00	wallclock time requested*
#BSUB -P UCGD0001	project number

Files written to case directory

*Note: Maximum allowable wall clock time on Yellowstone is 12 hours.
Submissions requesting under an hour typically have shorter wait times in the queue.



Exercise 2: Run for different length of time

```
# 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 ~/I2000CLM45_001 -res f19_g16 -compset ICLM45 -mach yellowstone

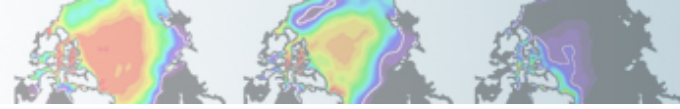
# go into the case you just created in the last step
cd ~/I2000CLM45_001

# (2) invoke cesm_setup
./cesm_setup

# (3) build the executable
./I2000CLM45_001.build

After modifying env_run.xml and I2000CLM45_001.run,
START HERE

# (4) submit your run to the batch queue
./I2000CLM45_001.submit
```



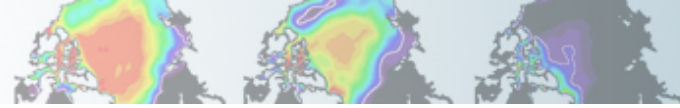
3 Types of Basic Modifications

1. Component Sets

2. ENV files (env_[command])

3. Namelist files (user_nl_[model])

* Going back to I1850CLM45_001 case, changing data record frequency



Review: The 4 commands to run CLM

Set of commands to build and run the model on a supported machine: "yellowstone"

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 ~/I1850CLM45_001-res f19_g16 -compset I1850CLM45 -mach yellowstone
```

go into the case you just created in the last step

```
cd ~/I1850CLM45_001
```

(2) invoke cesm_setup

```
./cesm_setup
```

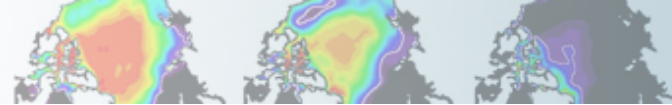
**This is when you modify the namelists
(and Source Code – Friday practicum)**

(3) build the executable

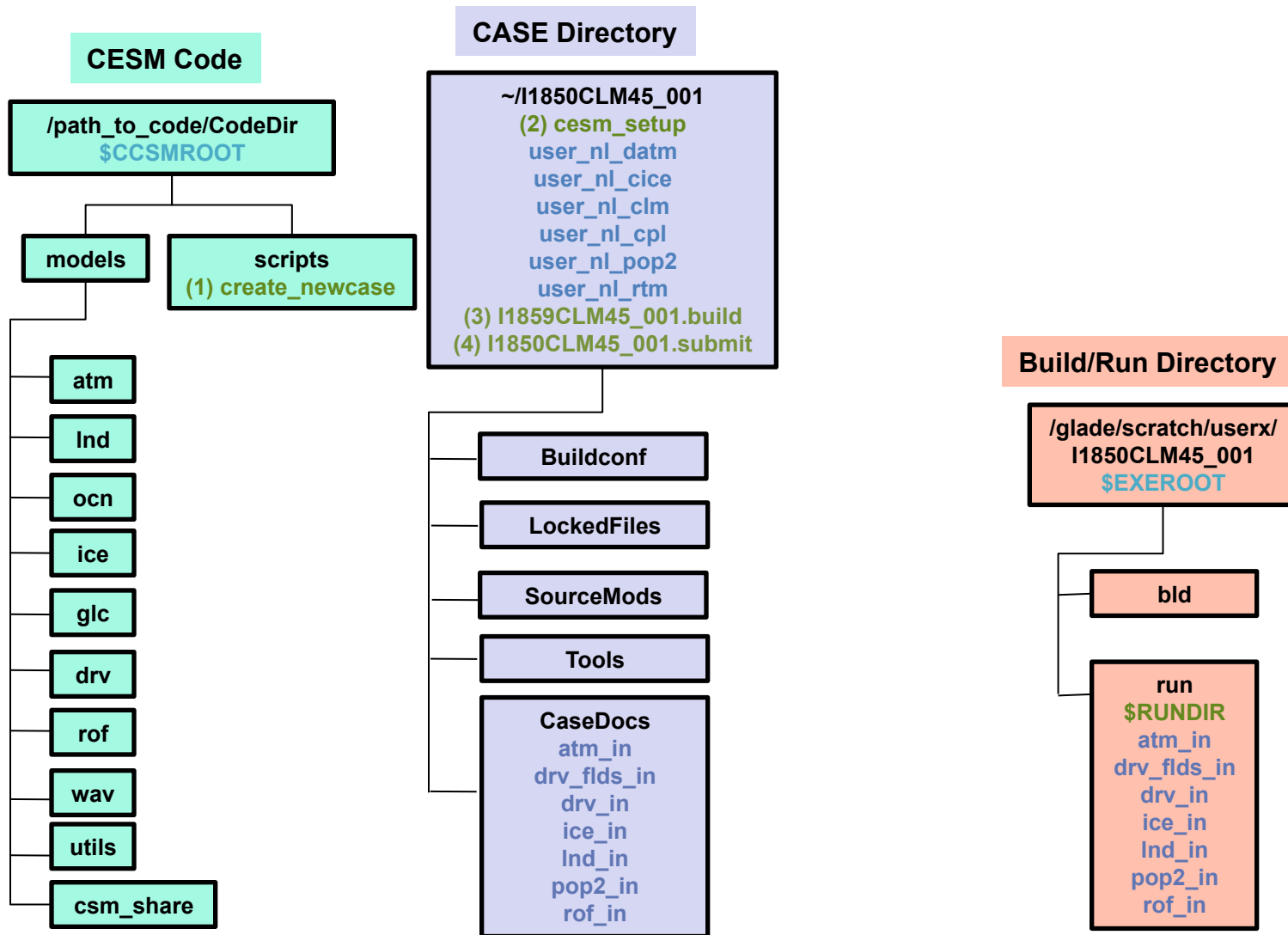
```
./I1850CLM45_001.build
```

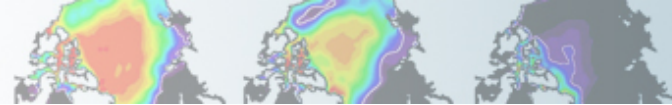
(4) submit your run to the batch queue

```
./I1850CLM45_001.submit
```

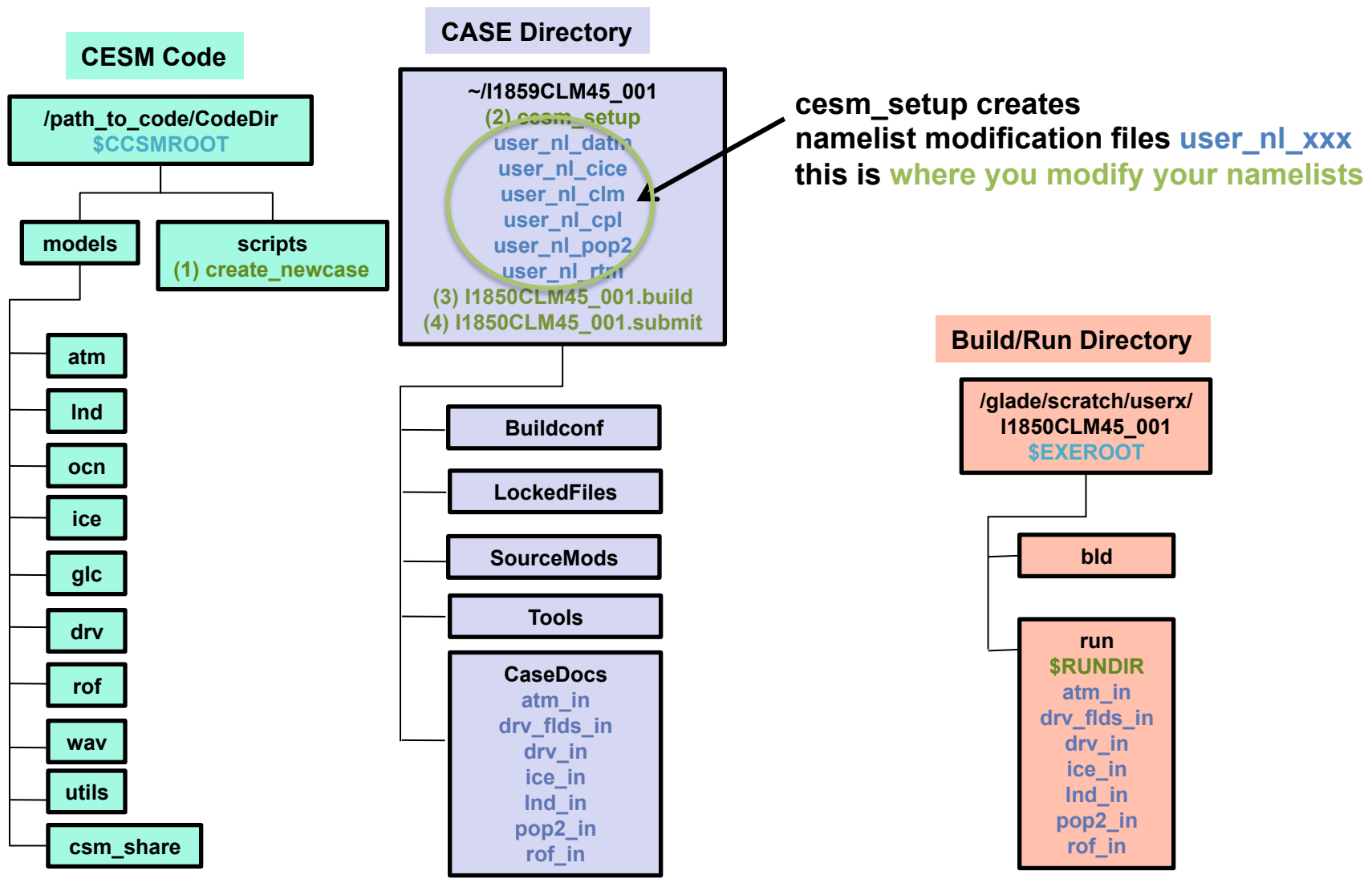


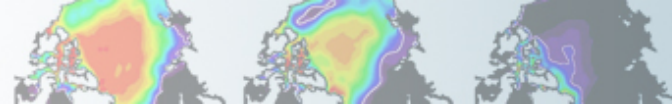
CLM directories & commands



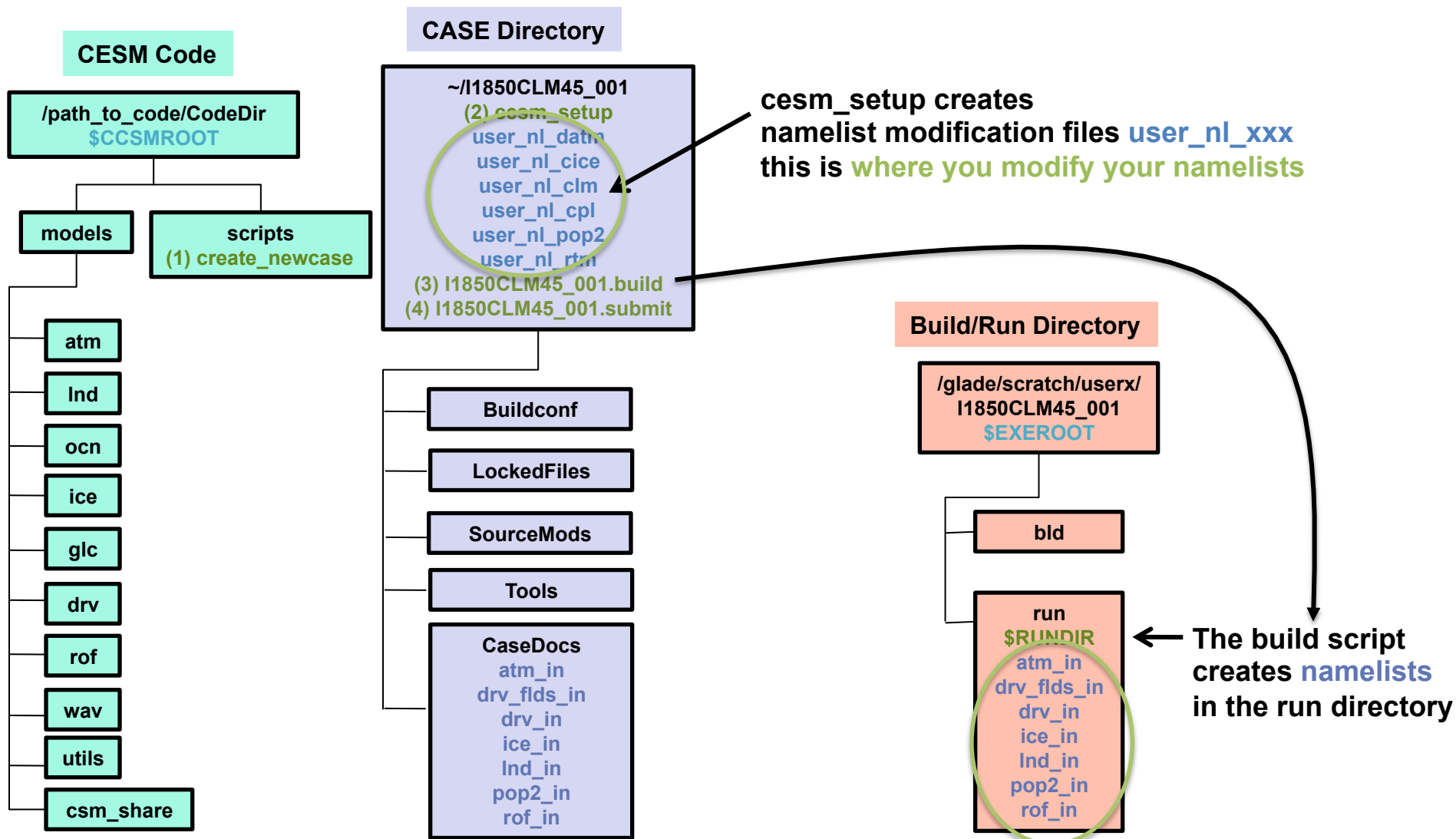


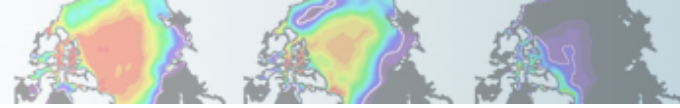
CLM directories & commands



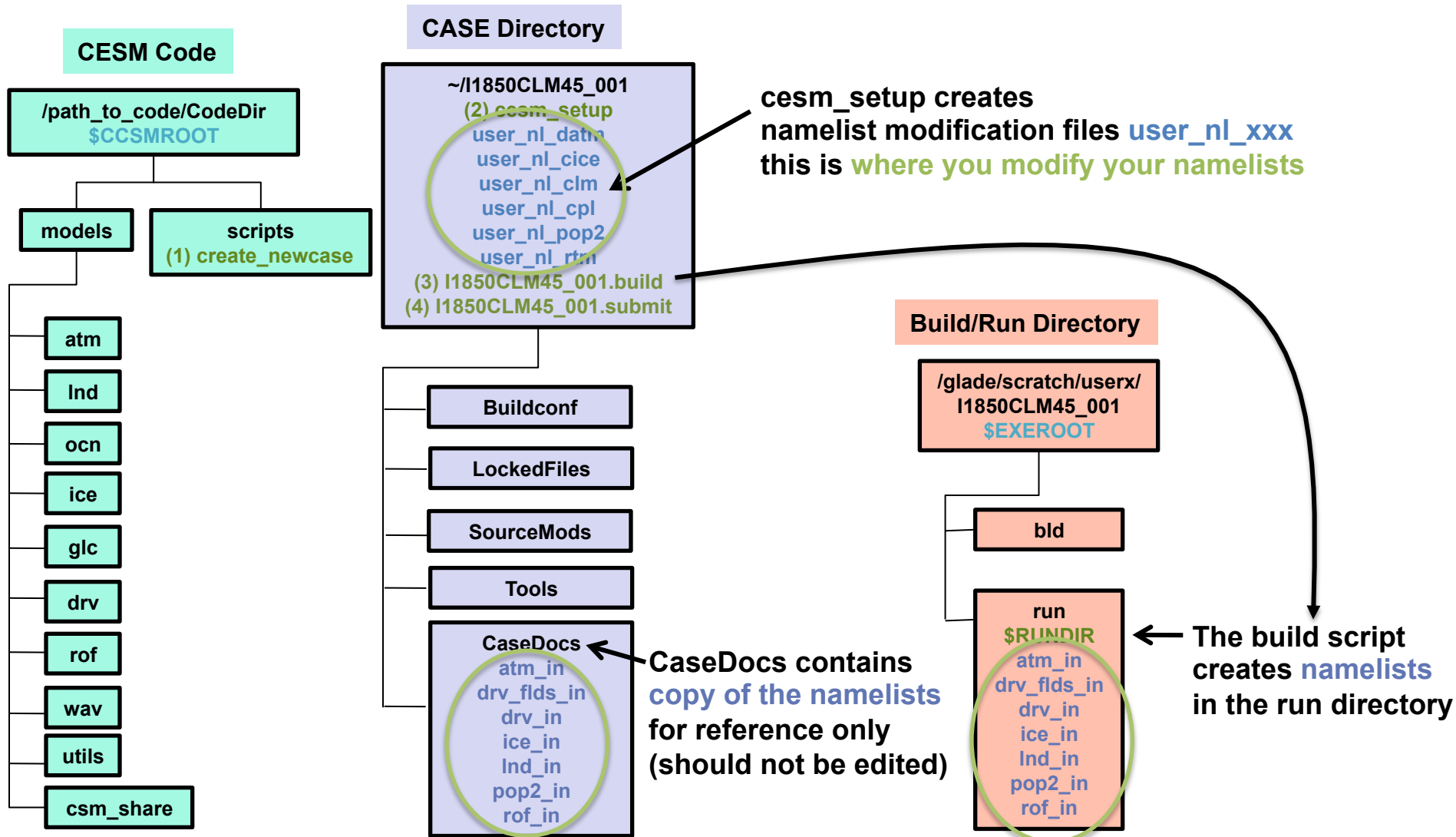


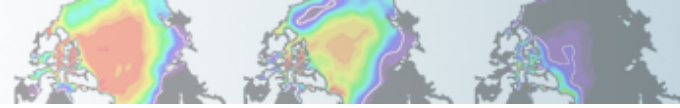
CLM directories & commands





CLM directories & commands





CESM1 Tutorial: Basic Modifications: Namelist variables/tool: **user_nl_<model>**

- Not all changes can be made in env_run.xml.
- **user_nl_<model>** files appear in the case directory after ./cesm_setup is invoked:

user_nl_datm ↔ atmosphere (atm_in)

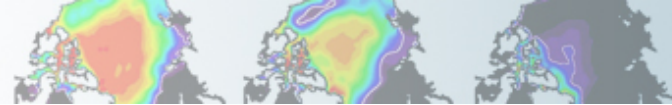
user_nl_clm ↔ land (lnd_in)

user_nl_cice ↔ ice (ice_in)

user_nl_pop2 ↔ ocean (pop2_in)

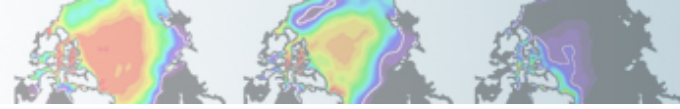
user_nl_cpl ↔ coupler (driver; drv_in)

user_nl_rtm ↔ river transport (rof_in)



Modifying Namelists

- Compsets set up namelists
- **user_nl_clm** modifies `Ind_in` namelist file
Important: Don't modify the namelist file directly. Use `user_nl_clm`.

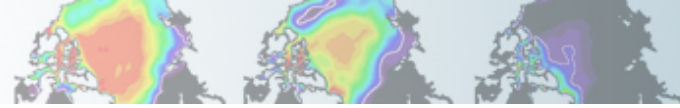


Modifying Namelists

- Compsets set up namelists
- **user_nl_clm** modifies Ind_in namelist file
Important: Don't modify the namelist file directly. Use user_nl_clm.

- Website for CLM namelist variables:
http://www.cesm.ucar.edu/models/cesm1.1/cesm/doc/modelnl/nl_clm.html

** Some namelist variables can also be changed in env_run.xml file



Looking at Namelist Files

Option 1

- cd into your case directory, then CaseDocs
 - (~/I1850CLM45_001/CaseDocs)
- Open **Ind_in** with text editor

Option 2

- cd into your run directory
 - (glade/scratch/I1850CLM45_001/run)
- Open **Ind_in** with text editor

```

&clm_inparm
albice = 0.60,0.40
co2_ppmv = 367.0
co2_type = 'constant'
create_crop_landunit = .false.
dtime = 1800
fatmldfrc = '/glade/p/cesmdata/cseg/inputdata/share/domains/domain.lnd.fv1.9x2.5_gx1v6.090206.nc'
finidat = ' '
fsnowaging = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/snicardata/snicar_drdt_bst_fit_60_c070416.nc'
fsnowoptics = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/snicardata/snicar_optics_5bnd_c090915.nc'
fsurdat = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/surfdata_1.9x2.5_simyr2000_c130927.nc'
maxpatch_glcmech = 0
maxpatch_pft = 17
more_vertlayers = .false.
nsegspc = 20
paramfile = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/paramdata/clm_params.c130821.nc'
urban_hac = 'ON'
urban_traffic = .false.
use_century_decomp = .false.
use_cn = .false.
use_crop = .false.
use_lch4 = .false.
use_nitrif_denitrif = .false.
use_vertsoilc = .false.
/
&ndepdyn_nml
/
&popd_streams
/
&light_streams
/
&clm_hydrology1_inparm
/
&clm_soilhydrology_inparm
/
#!-----
#! lnd_in:: Comment:
#! This namelist was created using the following command-line:
#! /glade/p/cesm/lmwg/CLM2014_tutorial_n02_clm4_5_57/models/lnd/clm/bld/CLM build-namelist -infile /glade/p/work/dll/testcases/CL
M2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clmconf/cesm_namelist -csmdata /glade/p/cesmdata/cseg/inputdata -inputdata /glade/p/
work/dll/testcases/CLM2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clm.input_data_list -ignore_ic_year -namelist &clm_inparm start
_ymd = 00010101 / -use_case 2000_control -res 1.9x2.5 -clm_start_type default -l_ncpl 48 -lnd_frac /glade/p/cesmdata/cseg/inputdata/
share/domains/domain.lnd.fv1.9x2.5_gx1v6.090206.nc -glc_nec 0 -co2_ppmv 367.0 -co2_type constant -config /glade/p/work/dll/testcases/
CLM2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clmconf/config_cache.xml -bgc sp
#! For help on options use: /glade/p/cesm/lmwg/CLM2014_tutorial_n02_clm4_5_57/models/lnd/clm/bld/CLM build-namelist -help
#!-----
~
~

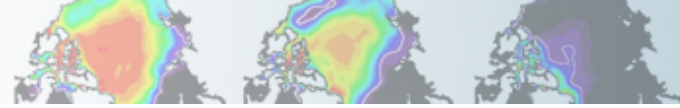
```

```

&clm_inparm
albice = 0.60,0.40
co2_ppmv = 367.0
co2_type = 'constant'
create_crop_landunit = .false.
dtime = 1800
fatmldnfrfc = '/glade/p/cesmdata/cseg/inputdata/share/domains/domain.lnd.fv1.9x2.5_gx1v6.090206.nc'
finidat = ' '
fsnowaging = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/snicardata/snicar_drdt_bst_fit_60_c070416.nc'
fsnowoptics = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/snicardata/snicar_optics_5bnd_c090915.nc'
fsurdatt = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/surfdata_1.9x2.5_simyr2000_c130927.nc'
maxpatch_glcmech = 0
maxpatch_pft = 17
more_vertlayers = .false.
nsegspc = 20
paramfile = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/paramdata/clm_params.c130821.nc'
urban_hac = 'ON'
urban_traffic = .false.
use_century_decomp = .false.
use_cn = .false.
use_crop = .false.
use_lch4 = .false.
use_nitrif_denitrif = .false.
use_vertsoilc = .false.
/
&ndepdyn_nml
/
&popd_streams
/
&light_streams
/
&clm_hydrology1_inparm
/
&clm_soilhydrology_inparm
/
#!-----
#! lnd_in:: Comment:
#! This namelist was created using the following command-line:
#! /glade/p/cesm/lmwg/CLM2014_tutorial_n02_clm4_5_57/models/lnd/clm/bld/CLM build-namelist -infile /glade/p/work/dll/testcases/CL
M2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clmconf/cesm_namelist -csmdata /glade/p/cesmdata/cseg/inputdata -inputdata /glade/p/
work/dll/testcases/CLM2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clm.input_data_list -ignore_ic_year -namelist &clm_inparm start
_ymd = 00010101 / -use_case 2000_control -res 1.9x2.5 -clm_start_type default -l_ncpl 48 -lnd_frac /glade/p/cesmdata/cseg/inputdata/
share/domains/domain.lnd.fv1.9x2.5_gx1v6.090206.nc -glc_nec 0 -co2_ppmv 367.0 -co2_type constant -config /glade/p/work/dll/testcases/
CLM2014Tutorial_20yr2000SP_n02_clm4_5_57/Buildconf/clmconf/config_cache.xml -bgc sp
#! For help on options use: /glade/p/cesm/lmwg/CLM2014_tutorial_n02_clm4_5_57/models/lnd/clm/bld/CLM build-namelist -help
#!-----
~
~

```

Different compsets will change the status of some of these things.



Example Modification: user_nl_clm

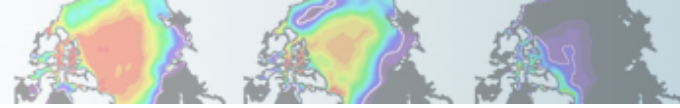
Changing the frequency of model output

hist_mfilt: Number of samples within a file

Default is 1

Setting value to 12 would put 12 records into a single file

* Both hist_mfilt & hist_nhtrfq must be integers



Example Modification: user_nl_clm

Changing the frequency of model output

hist_mfilt: Number of samples within a file

Default is 1

Setting value to 12 would put 12 records into a single file

hist_nhtfrq: Frequency that data are recorded and written to a file

Default: 0 means that output is recorded every month (monthly averages)

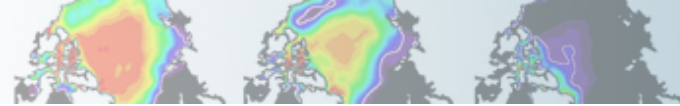
Positive Values: Number of model timesteps (half-hourly) for output record

ex: 48 means output is recorded every day (daily averages)

Negative Values: Absolute value in hours for output record

ex: -1 means output is recorded hourly; -24 means output is recorded daily

* Both hist_mfilt & hist_nhtfrq must be integers



Example Modification: user_nl_clm

Changing the frequency of model output

Daily output with a years worth of daily records in a file:

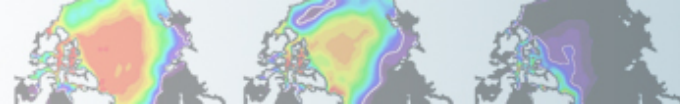
hist_mfilt = 365

hist_nhtfrq = -24

Monthly output with each month written to a separate file (default, as in I2000CLM45_001 case):

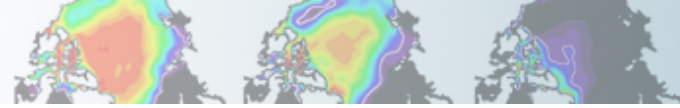
hist_mfilt = 1

hist_nhtfrq = 0



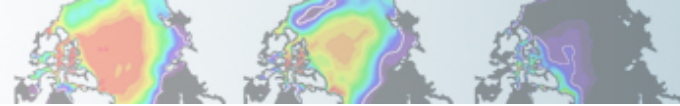
For this tutorial, we changed the default data record setting to daily in the I1850CLM45 compset.

Example 3: Modify `user_nl_clm` to get monthly output, 1 file per month in I1850CLM45_001



Run I1850CLM45_001 for 5 years

1. Change user_nl_clm to record monthly output
 - Rebuild the case: I1850CLM45_001.build
2. Change variables in env_run.xml
3. Change wall clock time in I1850CLM45_001.run
4. Rerun the simulation
 - I1850CLM45_001.submit



Review: The 4 commands to run CLM

Set of commands to build and run the model on a supported machine: "yellowstone"

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 ~/I1850CLM45_001 -res f19_g16 -compset I1850CLM45 -mach yellowstone
```

go into the case you just created in the last step

```
cd ~/I1850CLM45_001
```

(2) invoke cesm_setup

```
./cesm_setup
```

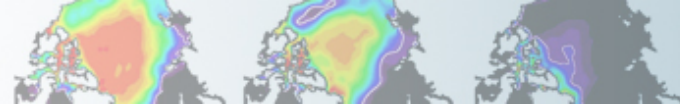
**After modifying namelists,
START HERE**

(3) build the executable

```
./I1850CLM45_001.build
```

(4) submit your run to the batch queue

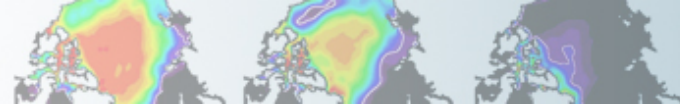
```
./I1850CLM45_001.submit
```



Now **YOU** know how to run the model!

Use these **3 basic modifications** to run a variety of simulations.

1. Component Sets
2. ENV files (env_[command])
3. Namelist files (user_nl_[model])



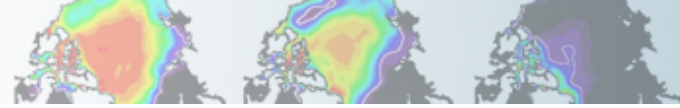
Documenting Your Changes: README files

In your case directory, you will find automatically generated documentation files.

1. **README.science_support:** refers you to the on-line documentation.
2. **README.case file:** detailed information on your compset and resolution, including whether your configuration has science support.

SCIENCE_SUPPORT: NO

***README.case*, we highly recommend YOU document any changes you make to the default scripts. It is YOUR paper trail and opportunity to list modifications.**



For additional information on running & configuring CLM, see CLM User's Guide:

<http://www.cesm.ucar.edu/models/cesm1.2/clm/models/Ind/clm/doc/UsersGuide/book1.html>

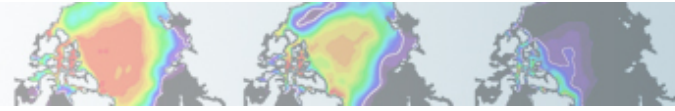


CLM4.5 Tutorial: Running for Single-Point/ Regional Cases



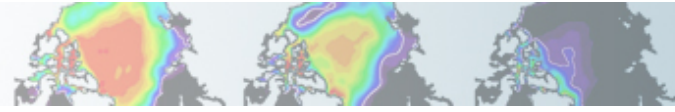
U.S. DEPARTMENT OF
ENERGY

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.



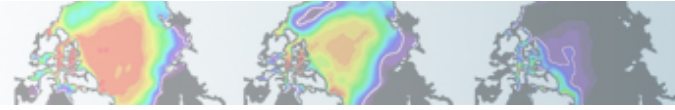
Caveat!

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.

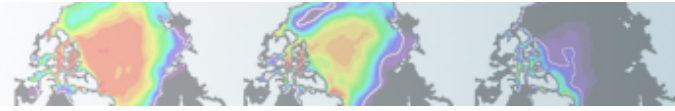
If you really need this version of the model for development – you'll need to ask for access to it after the tutorial.

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`

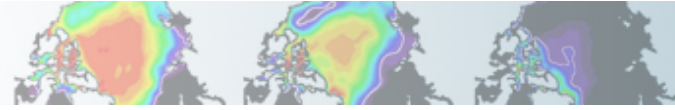
2. In the Day2 subdirectory (and the cases and other examples are in there as well)

3. Three files:

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

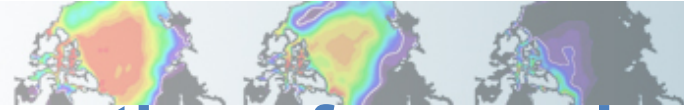


Section I – Running Supported Single Point Resolutions



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.



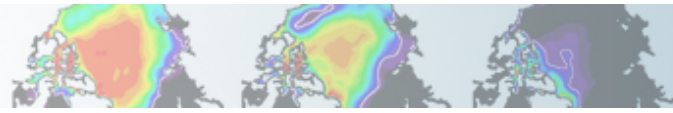
What are the different options for running single point or regional cases?

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

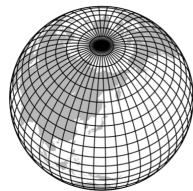
`create_newcase` requires 4 arguments

What is the casename ?



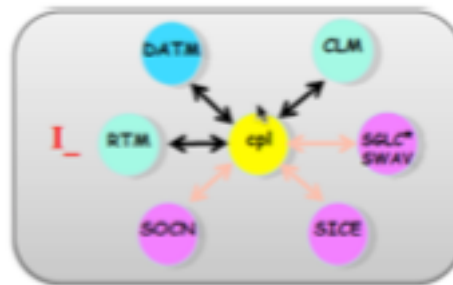
Day2Brazil

Which resolution?
(or location)



1x1_brazil (a single point in brazil)

Which model configuration?
Which set of components?



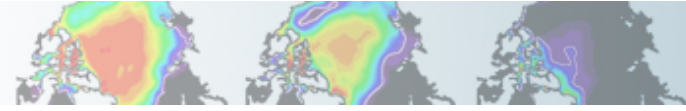
ICLM45
(I = CLM only, 2000)

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
4. Submit to the batch queue and run the simulation

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
```



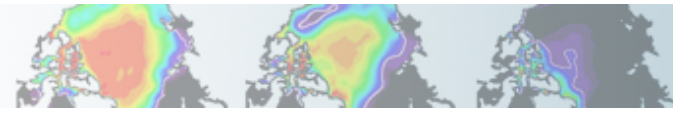
Get a list of the resolutions you can use

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

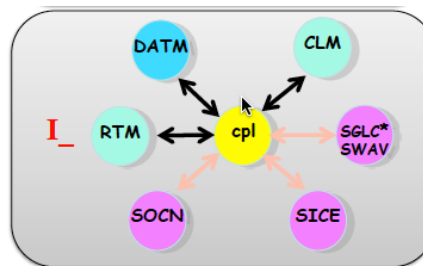
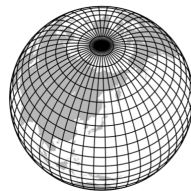
What is the casename ?

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

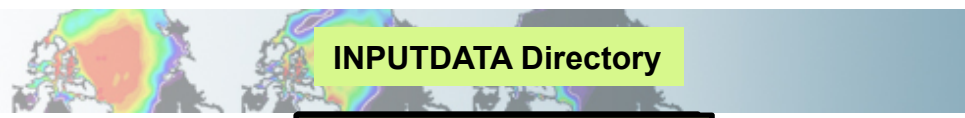
```
./create_newcase -case Day2MexCity -res 1x1_mexicocityMEX -compset I1PTCLM45 -mach yellowstone
```



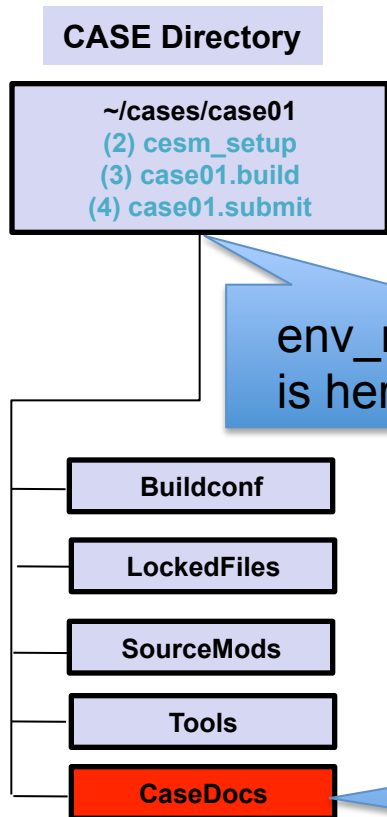
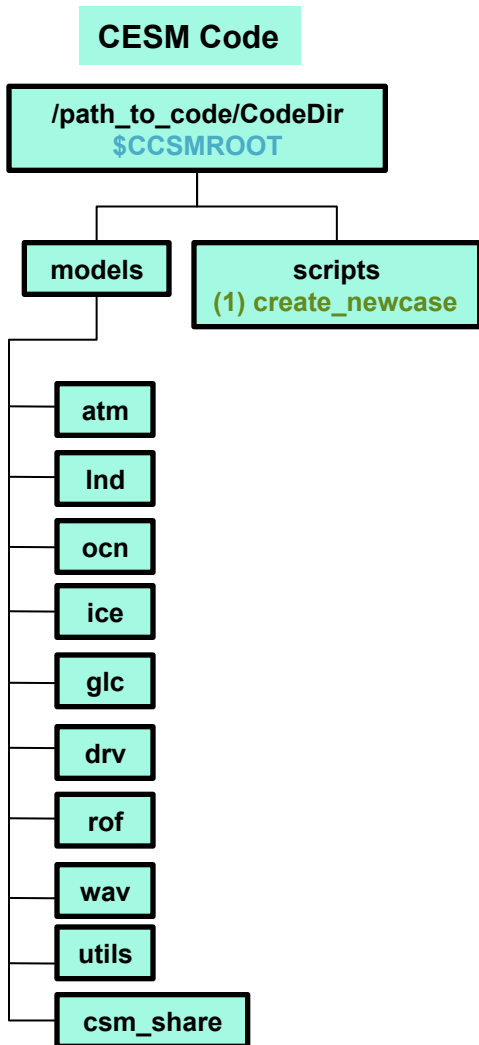
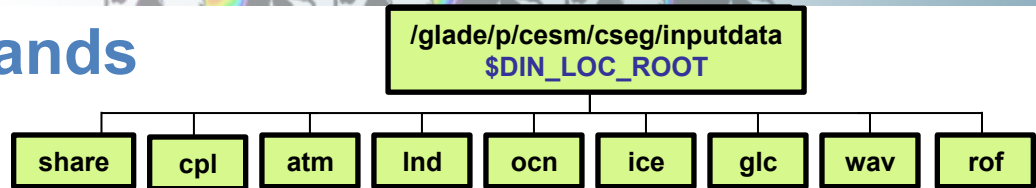
Let's go to Mexicocity!

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

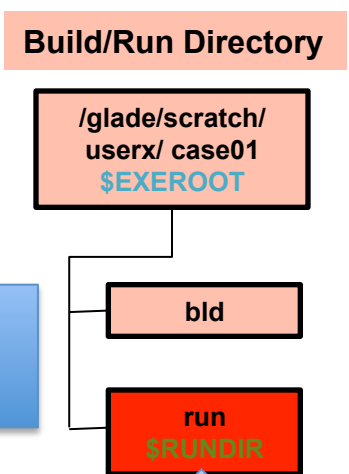
```
# go into scripts directory into the source code download  
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62/scripts  
  
# (1) create a new case in the directory "cases" in your home directory  
./create_newcase -case ~/cases/Day2MexCity -res 1x1_mexicocityMEX -compset I1PTCLM45 -  
mach yellowstone  
  
# 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
```



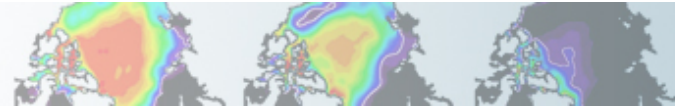
CLM directories & commands



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)

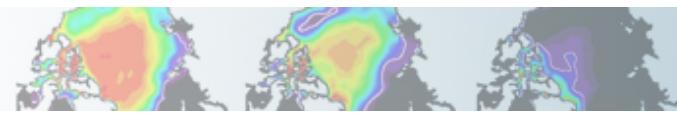
Brazil CASE

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

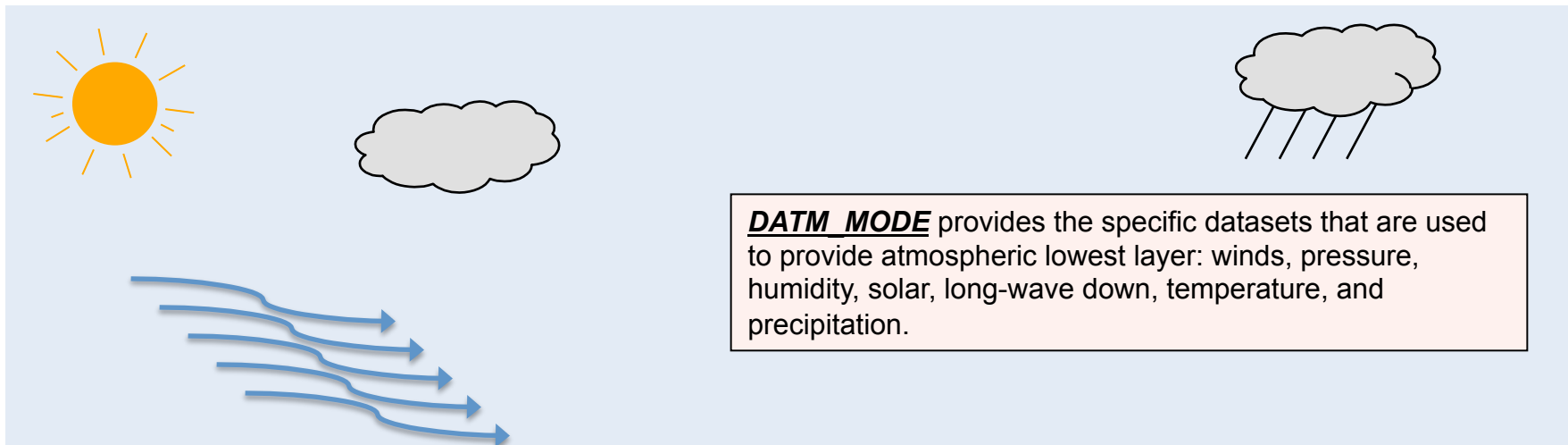
- **The start date and the time to run for is different.**
- **The “DATM_MODE” (type of Meteorology) is also different.**
- **The brazil case runs for Qian forcing**
- **While the Mexicocity case runs for “CLM1PT” forcing (tower Meteorology)**



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/Day2MexicoCity/CaseDocs)

Brazil CASE

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

```
fsurdat = '$DIN_LOC_ROOT/Ind/clm2/surfd_data_map/  
surfd_data_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/surfd_data_map/  
surfd_data_1x1_mexicocityMEX_simyr2000_c130927.nc'
```

The two files that are different are:

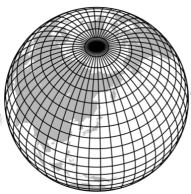
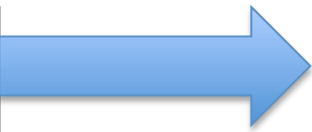
Domain file – Where in the world you are (or for global simulations what specific grid resolution)

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



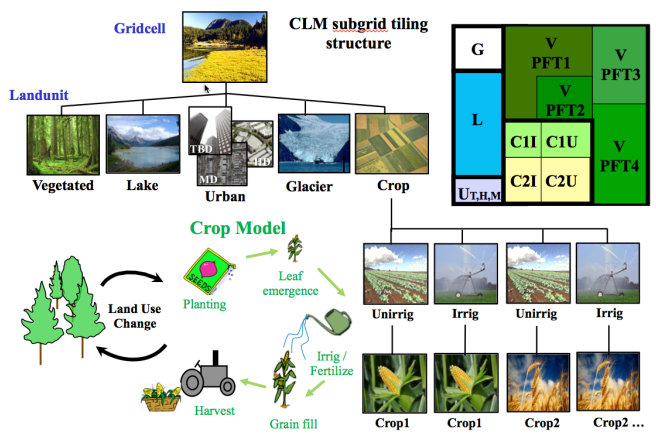
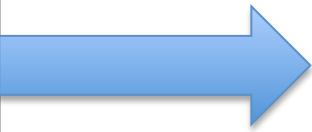
What's in the domain and surface files?

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.

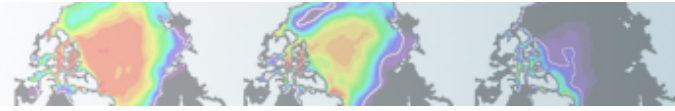


- Percent coverage in the grid cell for each land-unit type.
- Percent coverage of each vegetation type
- Soil type and color
- A whole host of other information that describes the land-cover for each grid cell



Review

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

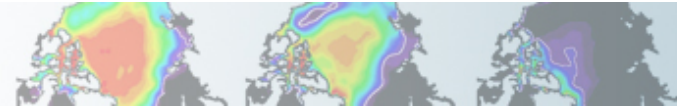


Exercise I

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)?

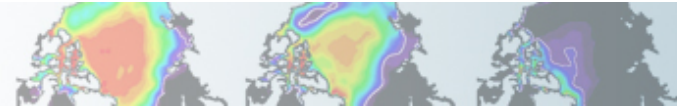


Exercise I

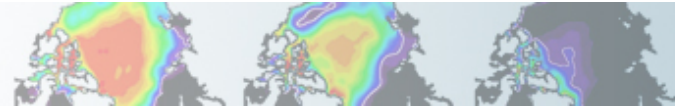
1. Setup a case for brazil
2. Setup a case for Mexicocity
3. In your case directory (for either case) in what env*.xml file is the domain file found?
 - **env_run.xml** file
4. Looking in your case directory for the brazil case in the CaseDocs/Ind_in file figure out the creation date string (_cYYMMDD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

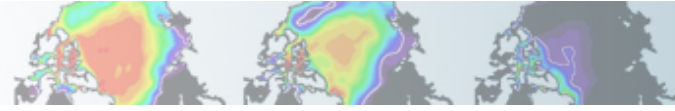


Section II – Using getregional to subset global datasets



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

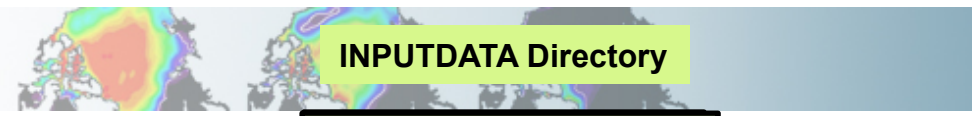
To create the domain file you use “gen_domain”.

To create the surface dataset you use “mksurfddata_map”.

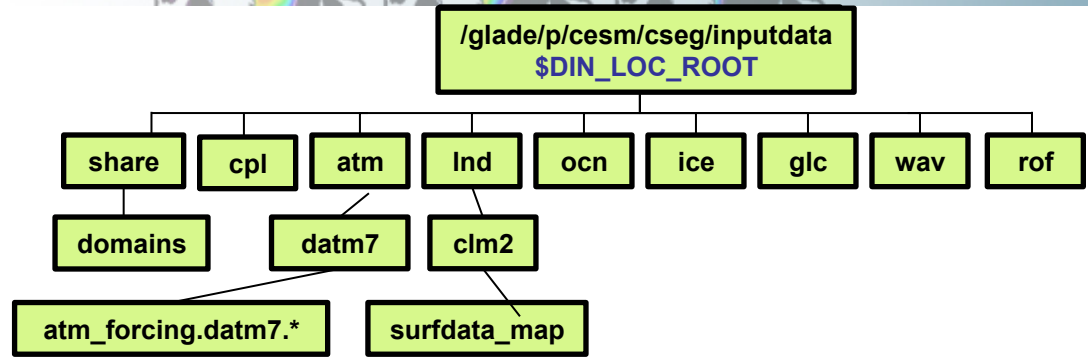
Here we will use the script “getregional_datasets.pl” to create these two files by pulling out a sub-region from an existing 2-degree resolution datasets.



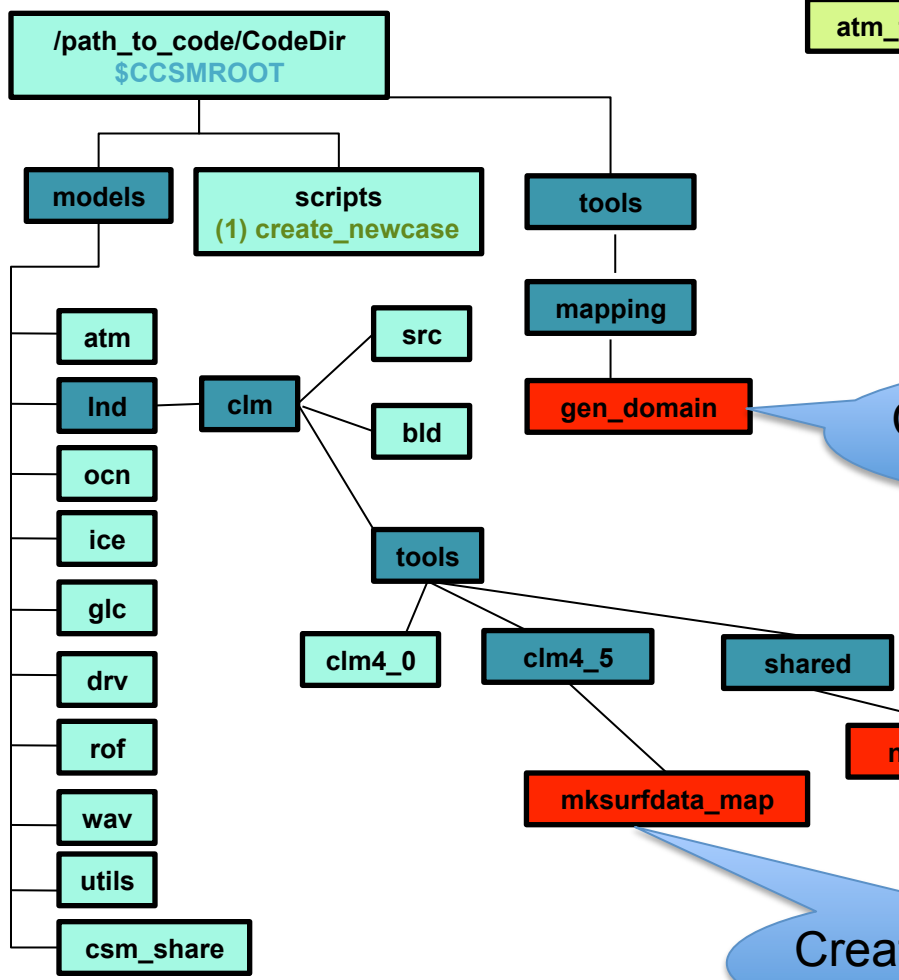
CLM tools directories



INPUTDATA Directory



CESM Code



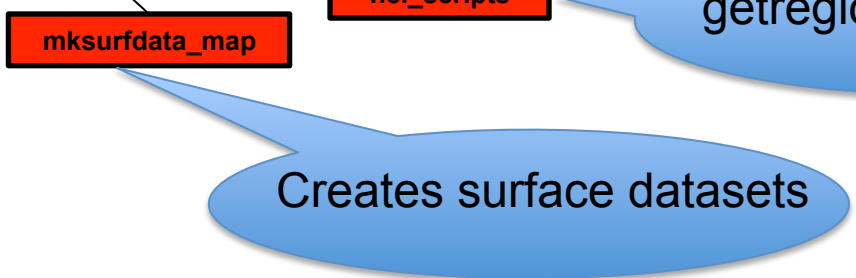
My Own DATA Directory

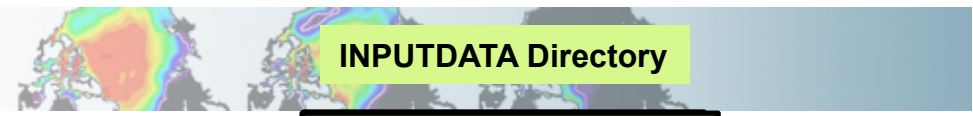
Creates domain files

getregional script is here

Creates surface datasets

gen_domain

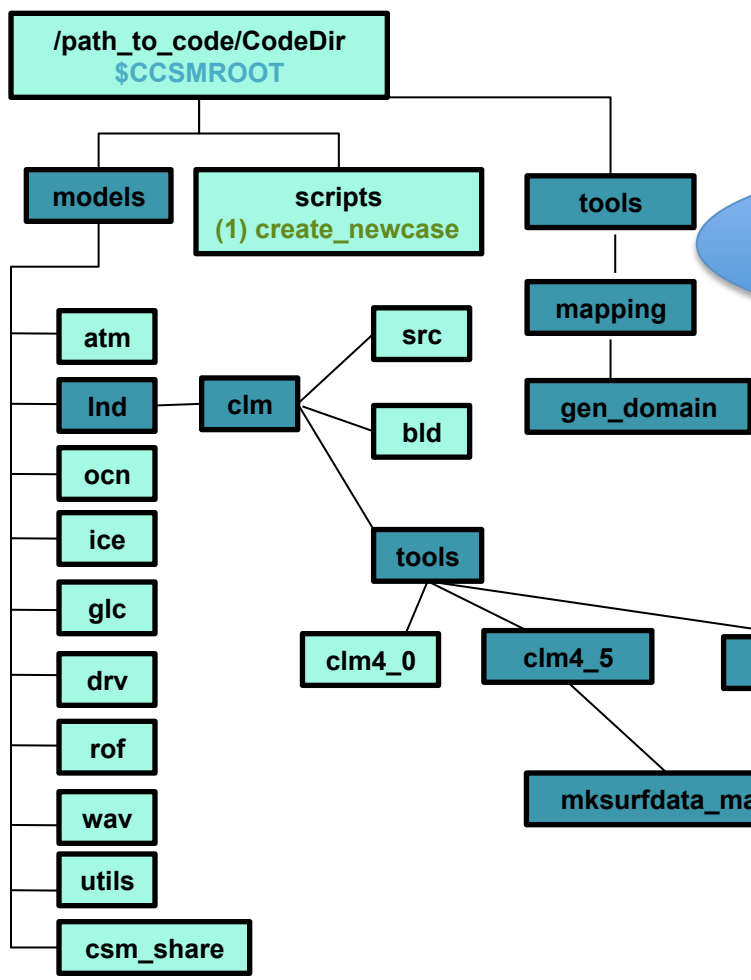




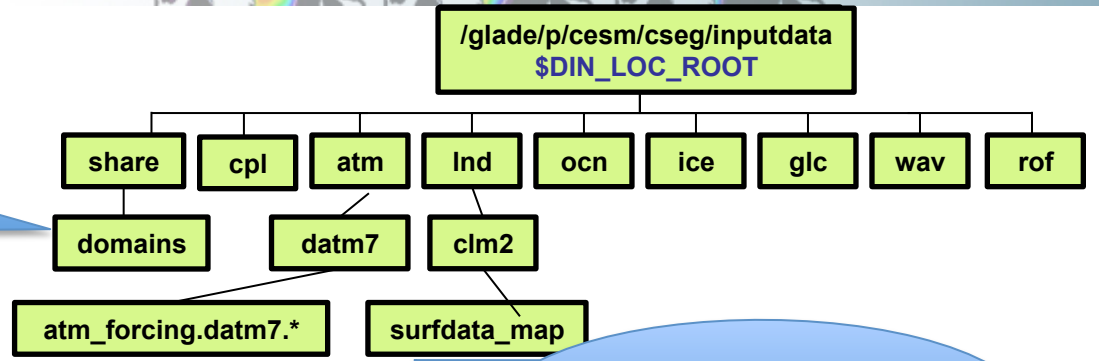
CLM tools directories

Standard domain files go here

CESM Code



INPUTDATA Directory



Standard Meteorology goes here

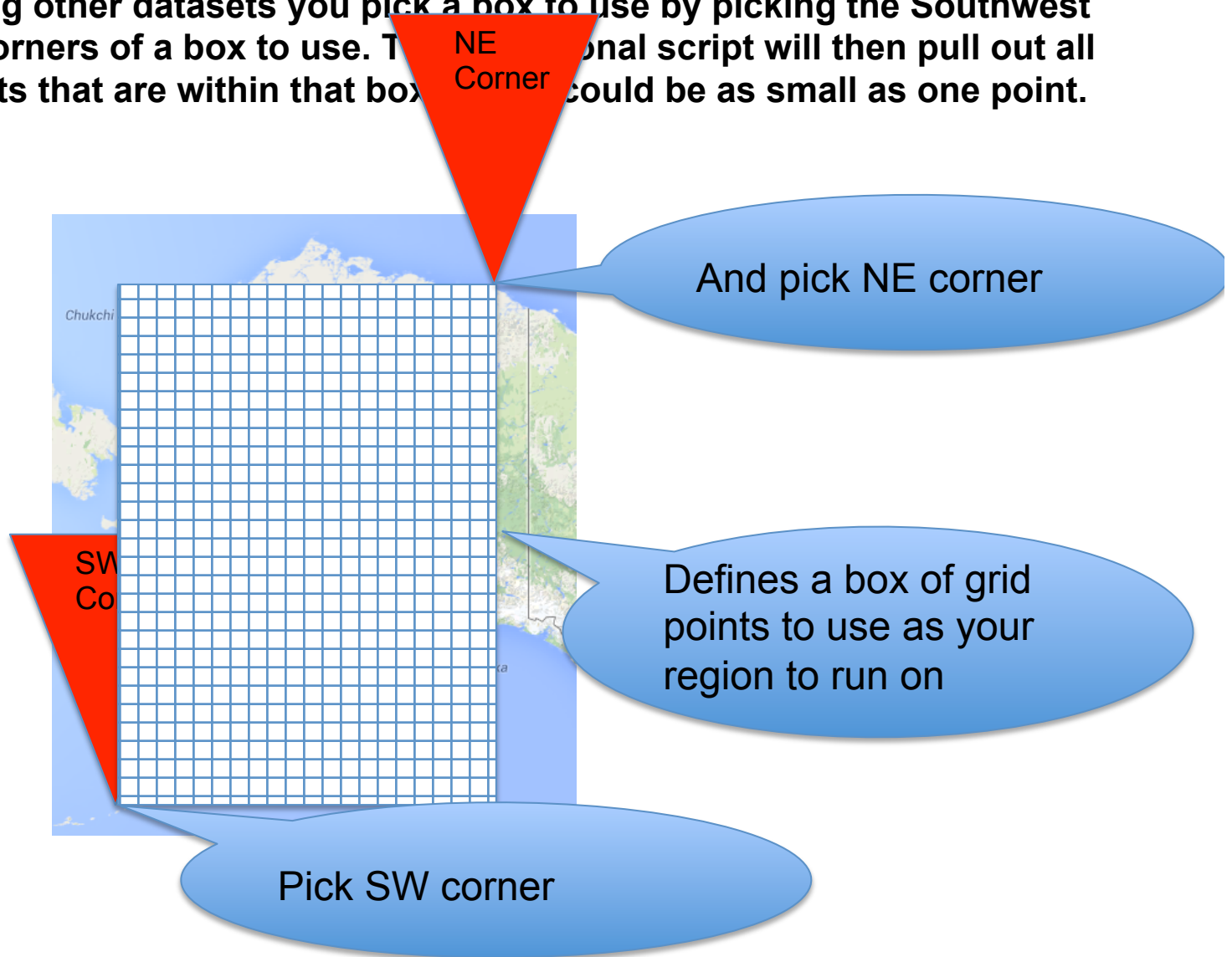
Standard surface datasets go here

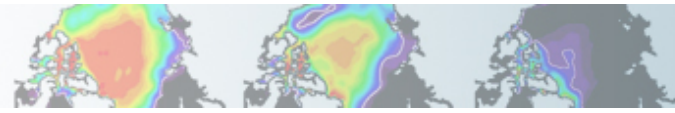
My Own DATA Directory

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.

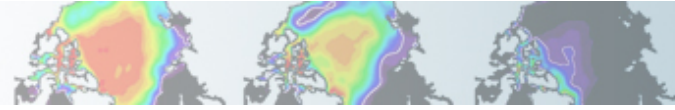
```
# go into the source code download
cd /glade/p/cesm/lmwg/CLM2014_tutorial_n03_clm4_5_62

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

# 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_* .

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

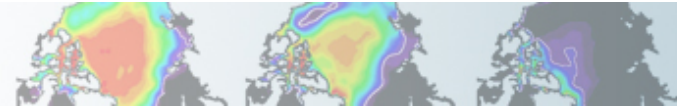


Exercise II

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?

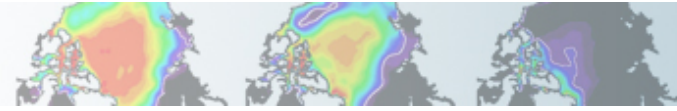


Exercise II

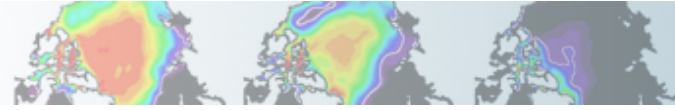
1. Use `getregional_datasets.pl` in the CLM tools directory to create a domain and surface dataset from the 0.9x1.25 global datasets for a region over Alaska
2. What is the creation date string (ending `_cYYMMDD` form) of the files you just created?
 - **c131122 Nov/22/2013**
3. What are the name of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

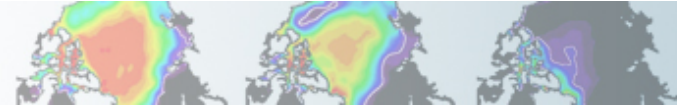


Section III – Using PTCLMmkdata



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 be put into a case.

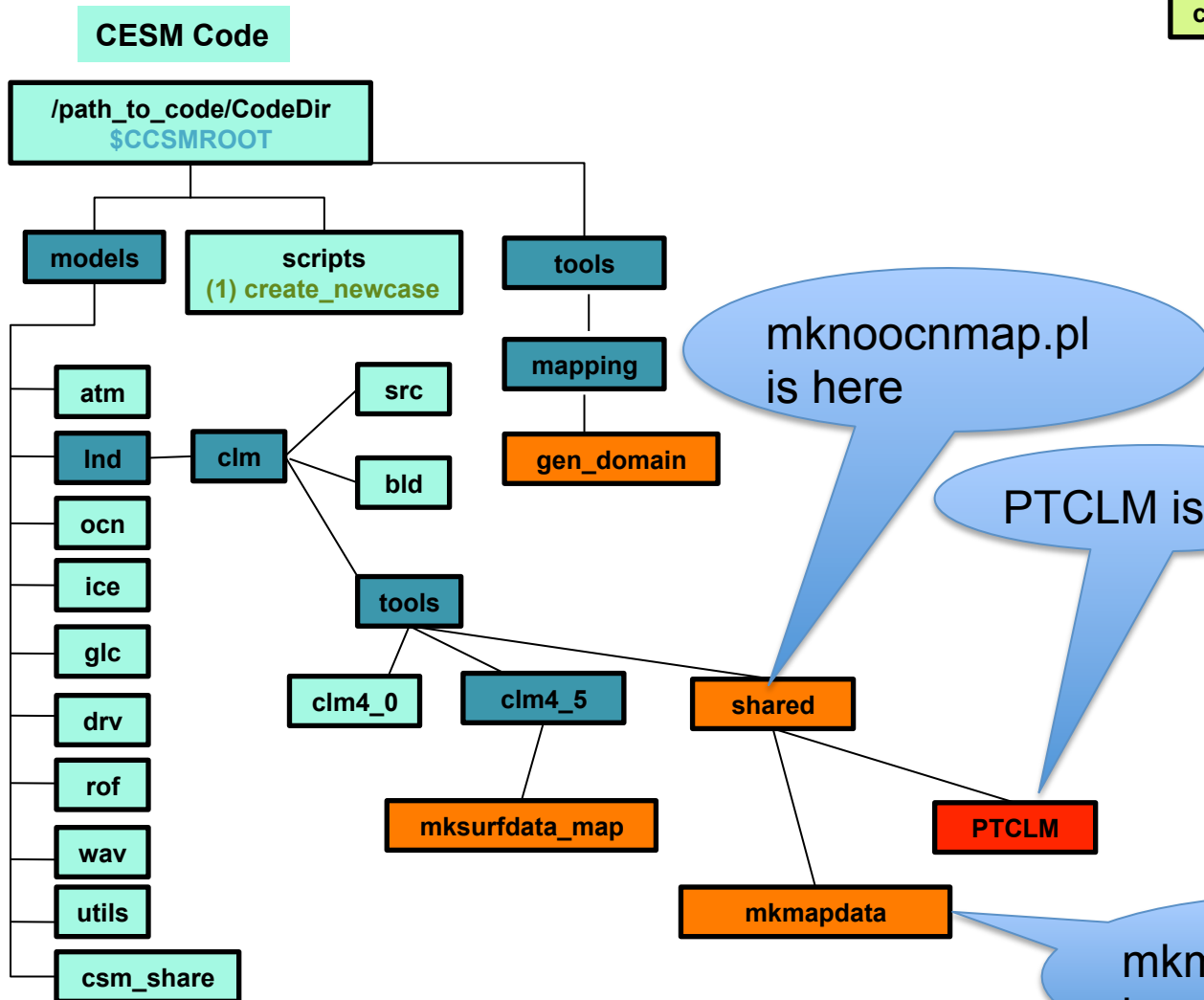
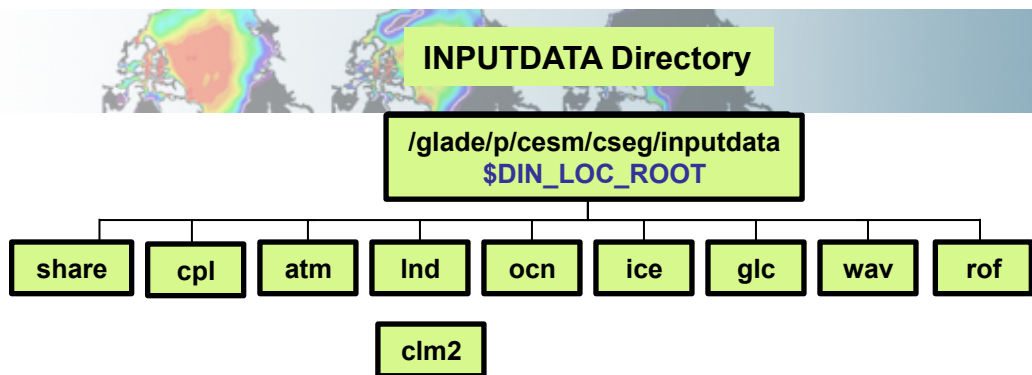
The meteorology that you force the sites with can either be from:

Tower site

Standard global forcing



PTCLM directory



My Own DATA Directory

mknoocnmap.pl is here

PTCLM is here

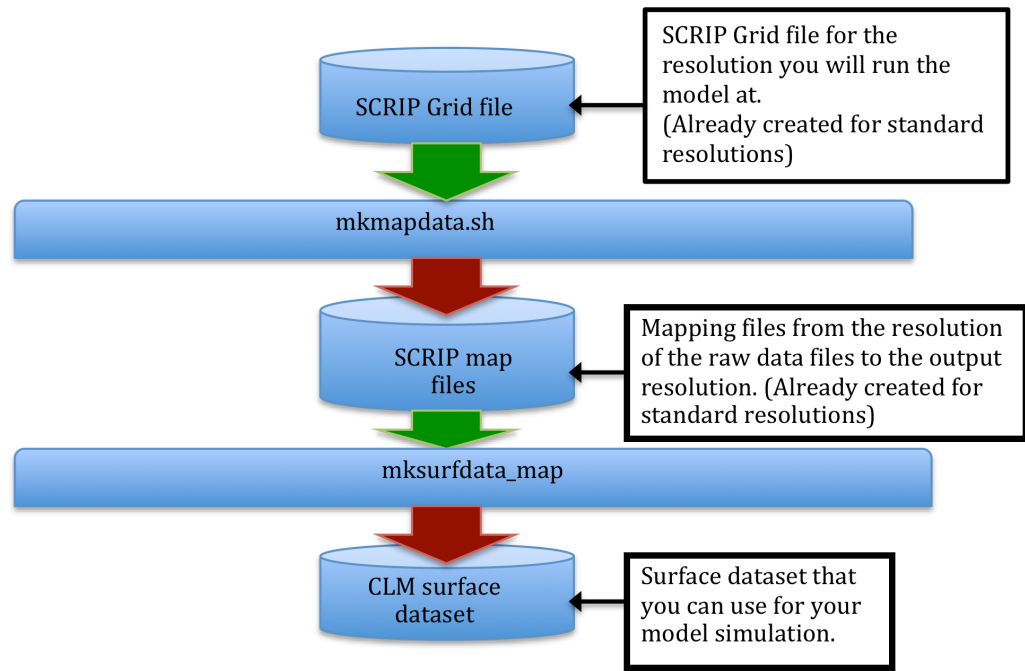
mkmapdata.sh is here

PTCLM runs the programs:

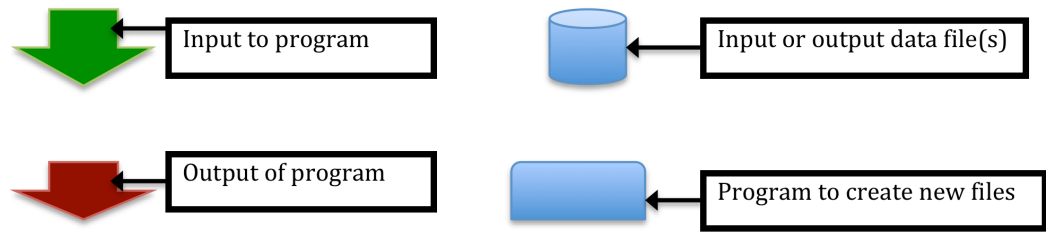
- mknoocnmap.pl
- gen_domain
- mkmapdata.sh
- mk surfdata_map

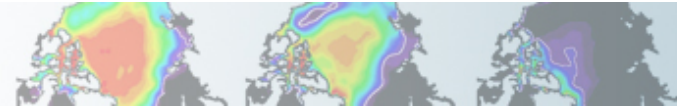


What is the data flow for creating datasets?

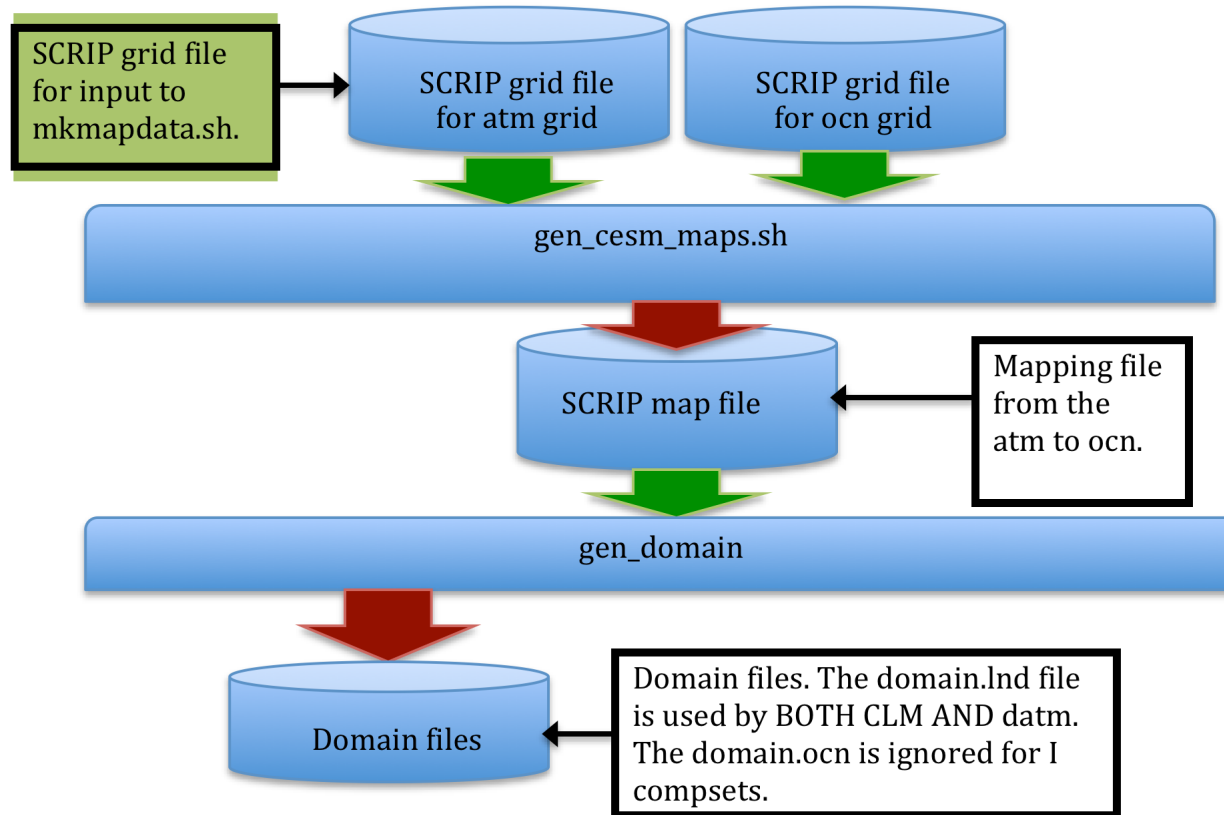


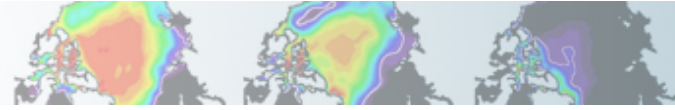
LEGEND





What is the data flow for creating the domain file?





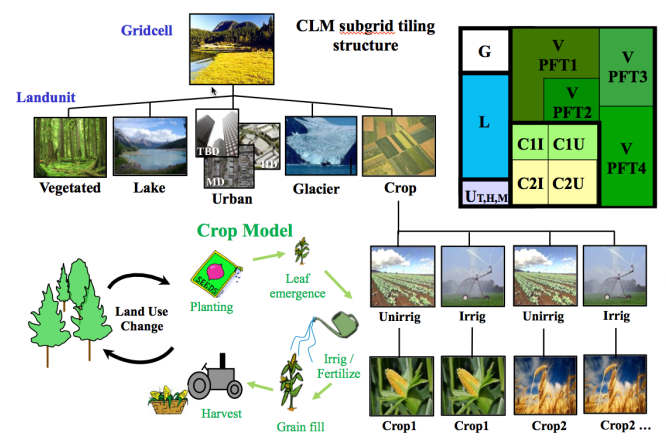
What are the steps that PTCLMmkdata does for you to create datasets?

1. Runs `mknocnmap.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?

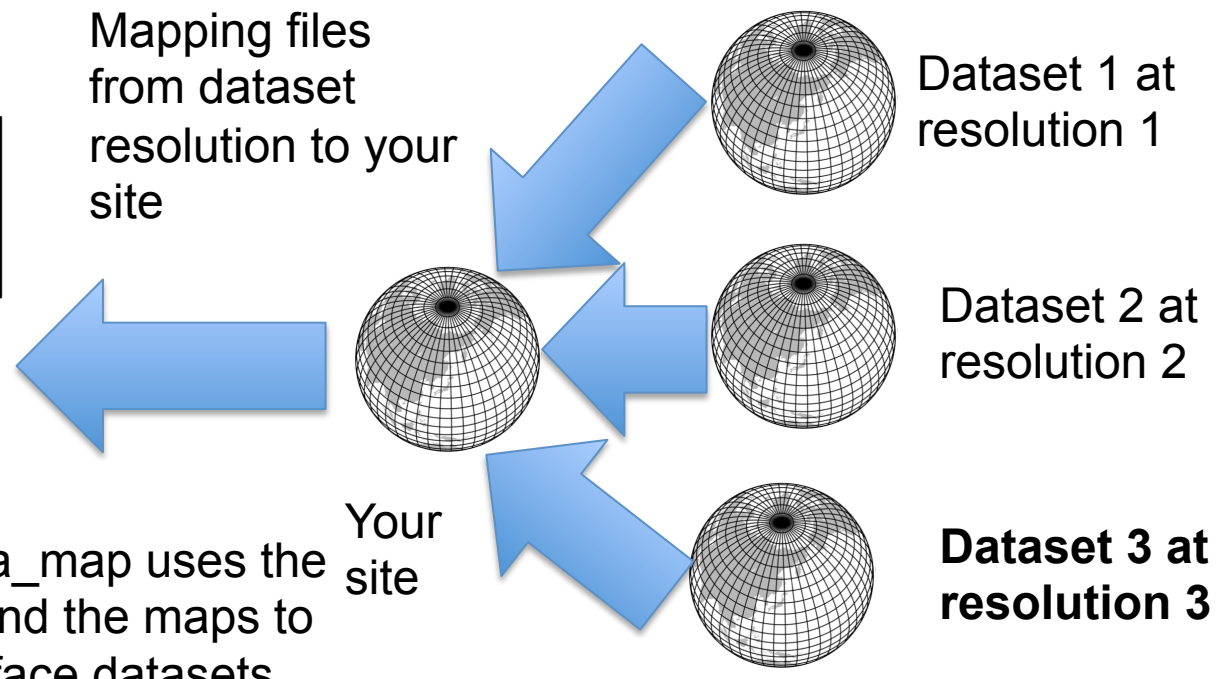
To create surface datasets you read in various “raw” datasets (for PFT, lake, glacier, etc.) at different resolutions and regrid them to your site. The mapping files regrid from the “raw” dataset resolution to your output site.



Surface Dataset

Mksurfdata_map uses the datasets and the maps to create surface datasets

Mapping files from dataset resolution to your site

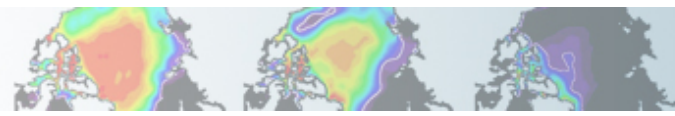


Dataset 1 at resolution 1

Dataset 2 at resolution 2

Dataset 3 at resolution 3

Your site



OK so how do I run PTCLM now?

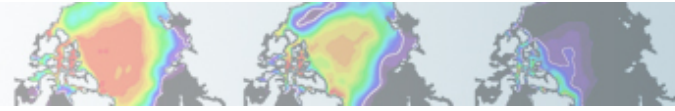
PTCLMmkdata as we said before runs the other file creation scripts to create datasets for you. This is easier than learning each of the previous programs. **However, if something goes wrong you may have to look into the programs that PTCLMmkdata calls to figure out what is up.**

```
# 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

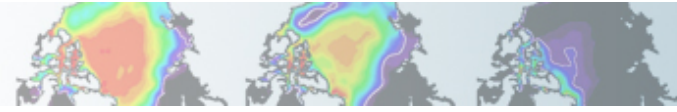
#copy the mapping files over so they won't have to be created again
cp -p $CSMDATA/Ind/clm2/PTCLMmydatafiles/$MYSITE/map_*_140121.nc $MYDATADIR/$MYSITE/

# 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

```
# (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 ICLM45BGC  
  
# 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
```

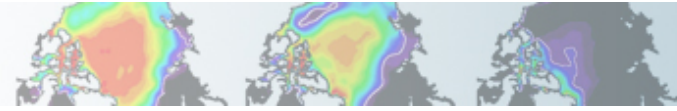


Exercise III

- 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)?**

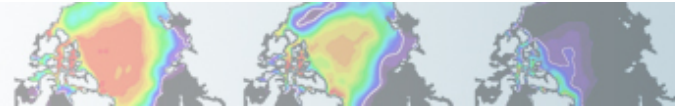


Exercise III

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.