CLM5.0 Tutorial 2: Running single point and regional cases

Will Wieder
Erik Kluzek
Sean Swenson
Yaqiong Lu

NCAR is sponsored by the National Science Foundation

Why single-point & regional CLM?

- **Code development**
- **Testing & debugging**
- **Comparison with point observations**

Some publications & references

Bonan et al (2014) Geosci. Model Dev., 7, 2193-2222 * Multi-layer canopy
Fisher et al. (2015) Geosci. Model Dev., 8 3593-3619 * Ecosystem demographics model
Levis et al. (2014) Geosci. Model Dev., 7 613-620 * Agricultural tillage (C cycle!)

Tutorial 2 Objectives:

- **Perfect the four steps for running CLM**
- **Gain familiarity customizing & manipulating CLM datasets**
- **Gain familiarity modifying cases to use different datasets**
- **Introduction to making CLM input data sets**
- **Build complexity incrementally to meet your needs**

Tutorial 2 Exercises:

Options for running single point and regional cases:

1. **Out of the box single point cases**

   Familiarize w/ datasets you can change

2. **Regional scripts to subset existing datasets**

3. **Making datasets and PTCLM**

Exercise 1: Create & run an out-of-the-box single point simulation

Exercise 1: Summary

A. Steps to create a case are:
   1. ./create_newcase
   2. ./case_setup *from working directory
   3. ./case_build
   4. ./case_submit

B. Create an out of the box case for a single point in:
   - Brazil and
   - Mexico City

C. Compare the two, the main difference is the:
   - Domain datasets have grid information
   - Surface datasets have land-type information for each grid point

Exercise 1: Objectives & Questions

1. **Setup a case for Brazil**
2. **Setup a case for Mexico City**
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/lnd_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/lnd_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 CRUNCEP forcing)?**
2. **For theBbrazil 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.streams.txt.CLMCRUNCEP.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 1: Create & run an out-of-the-box single point simulation

What resolutions can be used?

TO DO:

cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/cime/scripts

./manage_case -query grids | more

*** NOTE if this doesn't work you may also need to enter:**

module load python/2.7.7

This will list all the supported resolutions.
The "1x1_<name>" examples are single point runs
1x1_brazil = Novo Progresso Brazil
1x1_mexicocityMEX = urban case w/ meterological forcing

** Next we'll create new cases for the Brazil and Mexico City cases*

Exercise 1: Create & run an out-of-the-box single point simulation

What compsets can be used?

TO DO:

./manage_case -query compsets -setby clm

This will list all the supported compsets.

** Next we'll create new cases for the Brazil and Mexico City cases*

Exercise 1: Create & run an out-of-the-box single point simulation

(1) create a new case

(2) invoke case.setup

(3) build the executable

(4) submit your run to the batch queue

We will progress step by step, starting with step 1

Exercise 1a: Create & run an out-of-the-box single point simulation [Brazil]

(1) create a new case

```
./create_newcase -case ~/Day2Brazil -res 1x1_brazil -compset IMCRUCLM50BGC
```

Stop here

(2) invoke case.setup

(3) build the executable

(4) submit your run to the batch queue

We will progress step by step, starting with step 1

REVIEW: Create a new case

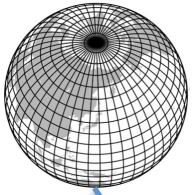
In the scripts directory, create_newcase is the tool that generates a new case.

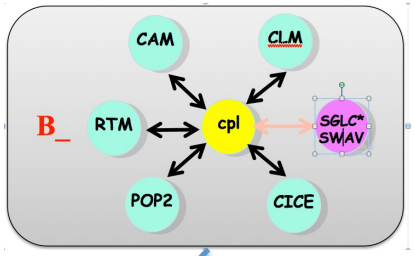
create_newcase requires 3 arguments

What is the casename ?

Which resolution?

Which model configuration?
Which set of components?
Which meteorology?





./create_newcase –case ~/Day2Brazil –res 1x1_brazil –compset IMCRUCLM50BGC

YourCaseName

Single Point, Brazil

CLM5.0, BGC mode, Mosart, etc.
More on this to come…

2000_DATM%CRU_CLM50%BGC_SICE_SOCN_MOSART_SGLC_SWAV

***Note**: A previously required 4th argument, "-mach", is no longer needed when using a supported machine.*
CIME now figures out what machine you are running on. If you are running on an unsupported machine, this argument is required.

Exercise 1a: Create & run an out-of-the-box single point simulation [Brazil]

(1) create a new case

```
./create_newcase -case ~/Day2Brazil -res 1x1_brazil -compset IMCRUCLM50BGC
```

(2) invoke case.setup

Now we'll configure, build & submit the case you just set up.

(3) build the executable

(4) submit your run to the batch queue

Exercise 1a: Create & run an out-of-the-box single point simulation [Brazil]

(1) create a new case

./create_newcase -case ~/Day2Brazil -res 1x1_brazil -compset IMCRUCLM50BGC

Start here

(2) invoke case.setup

move into case directory:

cd ~/Day2Brazil

Type this command line:

./case.setup

(3) build the executable

./case.build

(4) submit your run to the batch queue

./case.submit

Stop here

Exercise 1b: Create & run an out-of-the-box single point simulation [Mexico City]

Now we'll do the same thing for a simulation in Mexico City that uses observed meteorological forcing

Exercise 1b: Create & run an out-of-the-box single point simulation [Mexico City]

(1) create a new case

```
cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/cime/scripts
./create_newcase -case ~/Day2MexDF -res 1x1_mexicocityMEX -compset I1PTCLM50
```

(2) invoke case.setup

```
cd ~/Day2MexDF
./case.setup
```

(3) build the executable

```
./case.build
```

(4) submit your run to the batch queue

```
./case.submit
```

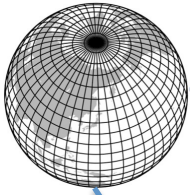
REVIEW: Create a new case

In the scripts directory, create_newcase is the tool that generates a new case.

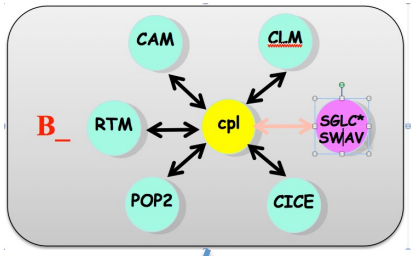
create_newcase requires 3 arguments

What is the casename ?

Which resolution?



Which model configuration?
Which set of components?
Which meteorology?



./create_newcase -case ~/Day2MexDF

Your Case Name

-res 1x1_mexicocityMEX

Single Point, Mexico City

-compset I1PTCLM50

CLM5.0, SP mode, RTM, etc.

2000_DATM%1PT_CLM50%SP_SICE_SOCN_RTM_SGLC_SWAV

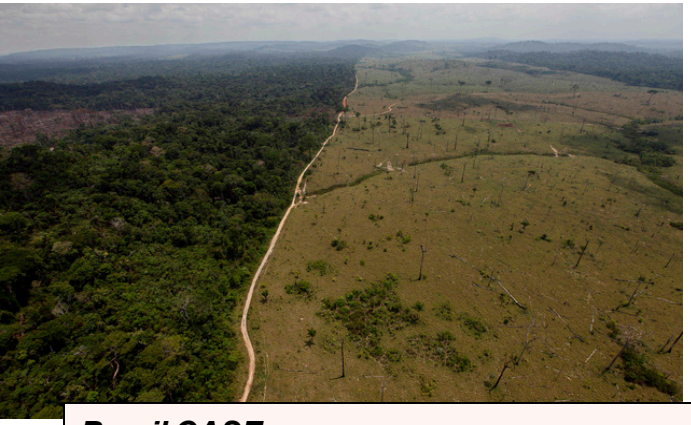
What are the differences between the two cases?

You can try look at each env_run.xml file

diff ~/Day2Brazil/env_run.xml ~/Day2MexDF/env_run.xml

* **NOTE** these differences are summarized on the next slide

What are the differences between the two cases?



SUMMARY
There are more:
BGC vs. SP
Mosart vs. RTM
Start & end dates

Brazil CASE

RUN_STARTDATE = 0001-01-01
STOP_OPTION = ndays
STOP_N = 5
ATM_DOMAIN_FILE = domain.lnd.1x1pt-brazil_navy. 090715.nc
LND_DOMAIN_FILE = domain.lnd.1x1pt-brazil_navy.090715.nc
DATM_MODE = CLMCRUNCEP
ATM_NCPL = 48

Mexico City CASE:

RUN_STARTDATE = 1993-12-01
STOP_OPTION = nsteps
STOP_N = 158
ATM_DOMAIN_FILE = domain.lnd.1x1pt-mexicocityMEX_navy.090715.nc
LND_DOMAIN_FILE = domain.lnd.1x1pt-mexicocityMEX_navy.090715.nc
DATM_MODE = CLM1PT
ATM_NCPL = 24

- **The start date and the time to run for is different.**
- **The domain files are different**
- **The "DATM_MODE" (type of Meteorology) is also different:**
  - **The Brazil case runs for CRUNCEP forcing (@ 30 min time step), while**
  - **The Mexicocity case runs for "CLM1PT" forcing (tower Meteorology, @ 1h)**

Aside: What is DATM_MODE?

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

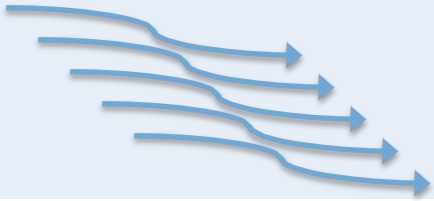
1) CLMGSWP3 *this will most likely be the preferred meteorological data to use w/ CLM5, but the compsets are not yet available

2) CLMCRUNCEP

3) CLM_QIAN

4) CLM1PT

5) **CPLHIST3HrWx**



DATM_MODE provides the specific datasets that are used to provide atmospheric lowest layer: winds, pressure, humidity, solar, long-wave down, temperature, and precipitation.

- **CLMCRUNCEP – Use global NCEP forcing at half-degree resolution from CRU goes from 1900-2010 [GSWP3 similar time period and spatial resolution]**
- **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 are the differences between the two cases? (continued)

You can try look at each lnd_in files (in CaseDocs)

vi ~/Day2Brazil/CaseDocs/lnd_in

And in another window

vi ~/Day2MexDF/CaseDocs/lnd_in

Or, try to diff the two files above

diff ~/Day2Brazil/CaseDocs/lnd_in ~/Day2MexDF/CaseDocs/lnd_in | more

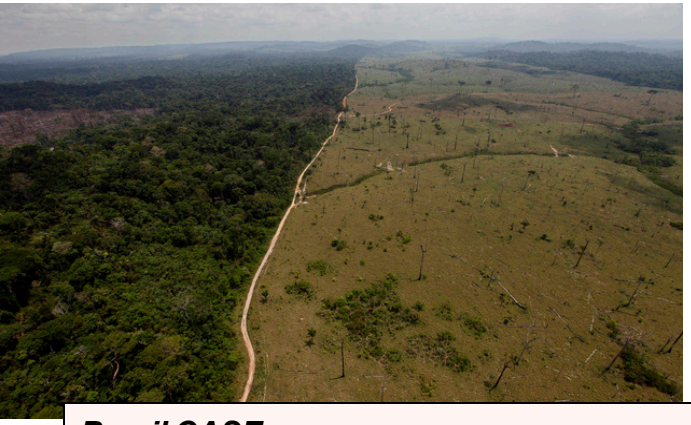
or

diff ~Day2Brazil/Buildconf/clm.input_data_list ~/Day2MexDF/Buildconf/clm.input_data_list

* **NOTE** these differences are summarized on the next slide

What are the differences between the two cases?

SUMMARY



Brazil CASE

fatmlndfrc = '/glade/p/cesmdata/cseg/inputdata/share/domains/domain.clm/domain.lnd.1x1pt-brazil_navy.090715.nc'

fsurdat = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/surfdata_1x1_brazil_16pfts_simyr2000_c160127.nc'

Mexico City CASE:

fatmlndfrc = '/glade/p/cesmdata/cseg/inputdata/share/domains/domain.clm/domain.lnd.1x1pt-mexicocityMEX_navy.090715.nc'

fsurdat = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/surfdata_1x1_mexicocityMEX_16pfts_simyr2000_c160127.nc'

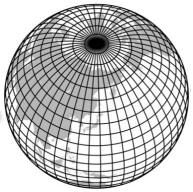
The two files that are different are:

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

Surface dataset – description of surface type, vegetation, soil, etc.

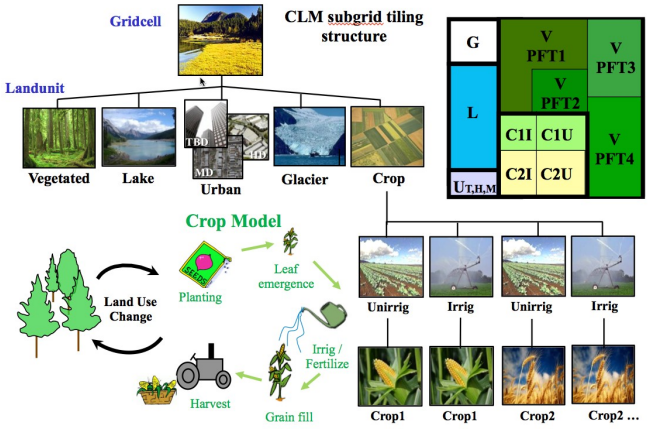
What's in the domain and surface files?

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

* **NOTE** these can both be modified as appropriate for a particular site / gridcell

Exercise 1: Objectives & Questions

1. Setup a case for Brazil
2. Setup a case for Mexico City
3. In your case directory (for either case) in what env*.xml file is the DOMAIN file found? env_run.xml
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? _c160127.nc (Jan/27/2016)
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)? .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 (this case is with CRUNCEP forcing)? datm_atm_in
2. For theBbrazil 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)? 6 (Precip, Solar, Temperature, Pressure, Humidity, and Wind-speed)
3. Looking in the" CaseDocs/datm.streams.txt.CLMCRUNCEP.Solar" file for the Brazil case figure out the creation date for the datm forcing data (from the filepath with a .cYYMMDD format)? c130305 (March/05/2013)
4. What resolution is the forcing dataset for the Brazil case? 0.5 degrees [360x720]
5. In the meteorological forcing streams file for the Mexicocity case – how many active variables are there (as in extra credit question 2 above)? 10 (in addition to those in #2, height, solar split into direct and diffuse, and incoming LW)

Exercise 2: Regional simulations

Using getregional to subset global datasets

Exercise 2: Summary

A. Introduce the directory with tools to make CLM input files.
B. Examine the CLM tool: getregional_datasets.pl
C. Create a regional dataset over Alaska
   - Uses particular domain and surface datasets
D. Create a case that uses the Alaska datasets
E. Demonstrate how to create a simple case using your own datasets.

Exercise 2: Objectives & Questions

1. **Use getregional_datasets.pl in the CLM tools directory to create a domain and surface dataset for a region over Alaska (from global datasets)**
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?**

CLM Tools for creating Datasets

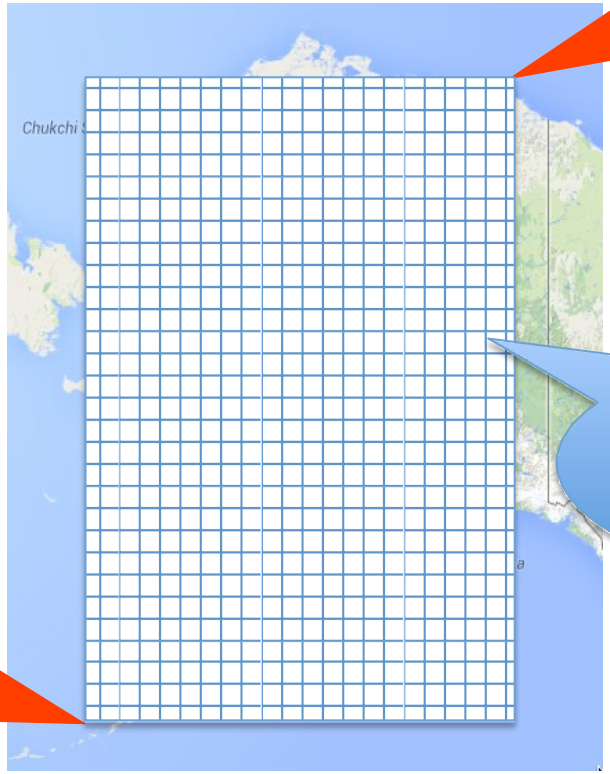
REVIEW:

- **The two datasets that are specific to the resolution are:**
  - **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 "mksurfdata_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.

Exercise 2: Sub-setting global datasets

When sub-setting other datasets you select the region to simulate by picking the SW & NE corners of a box to use. The getregional script will then pull out all of the grid points that are within that box (which could be as small as a single point)

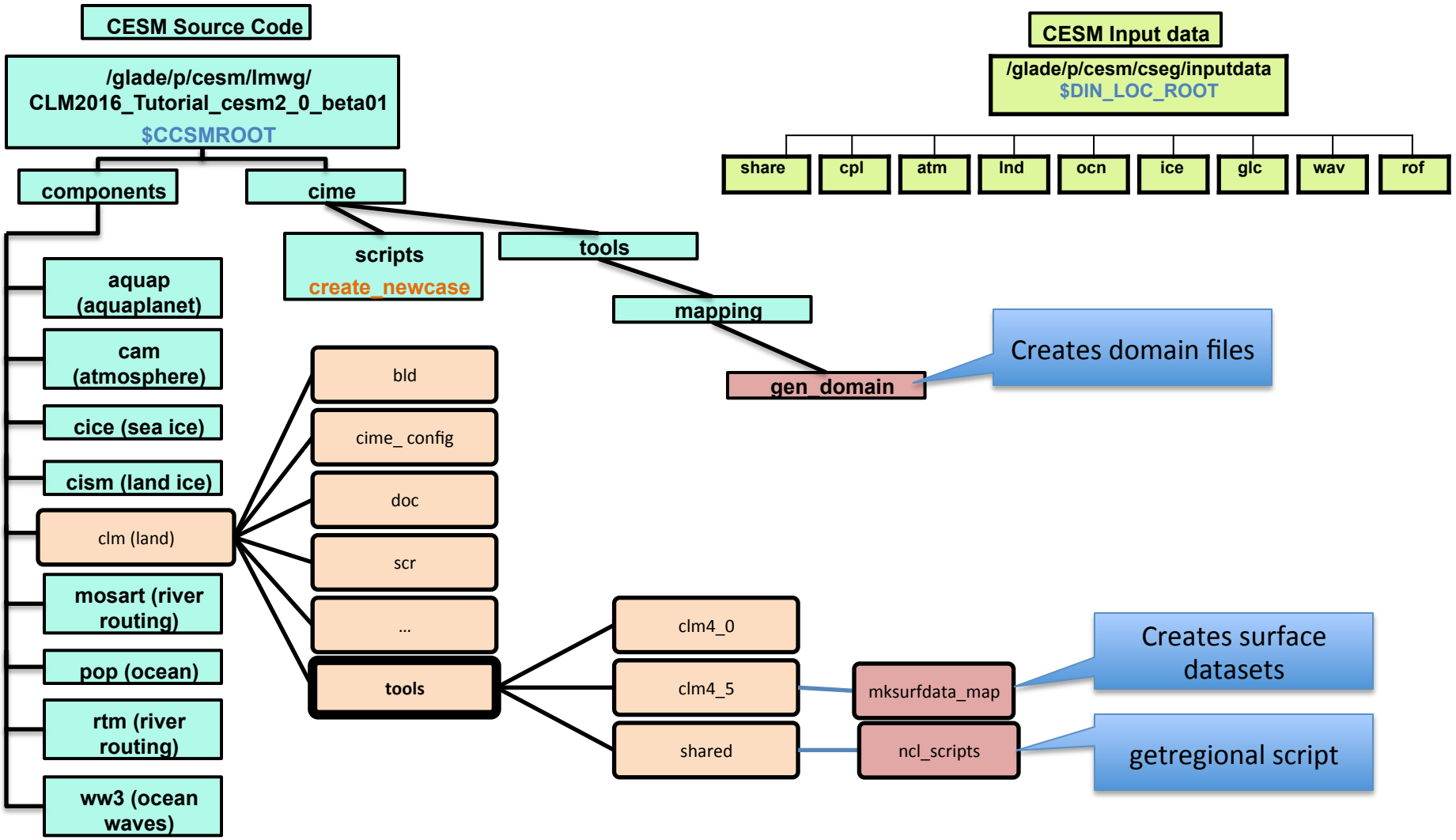


... and pick NE corner

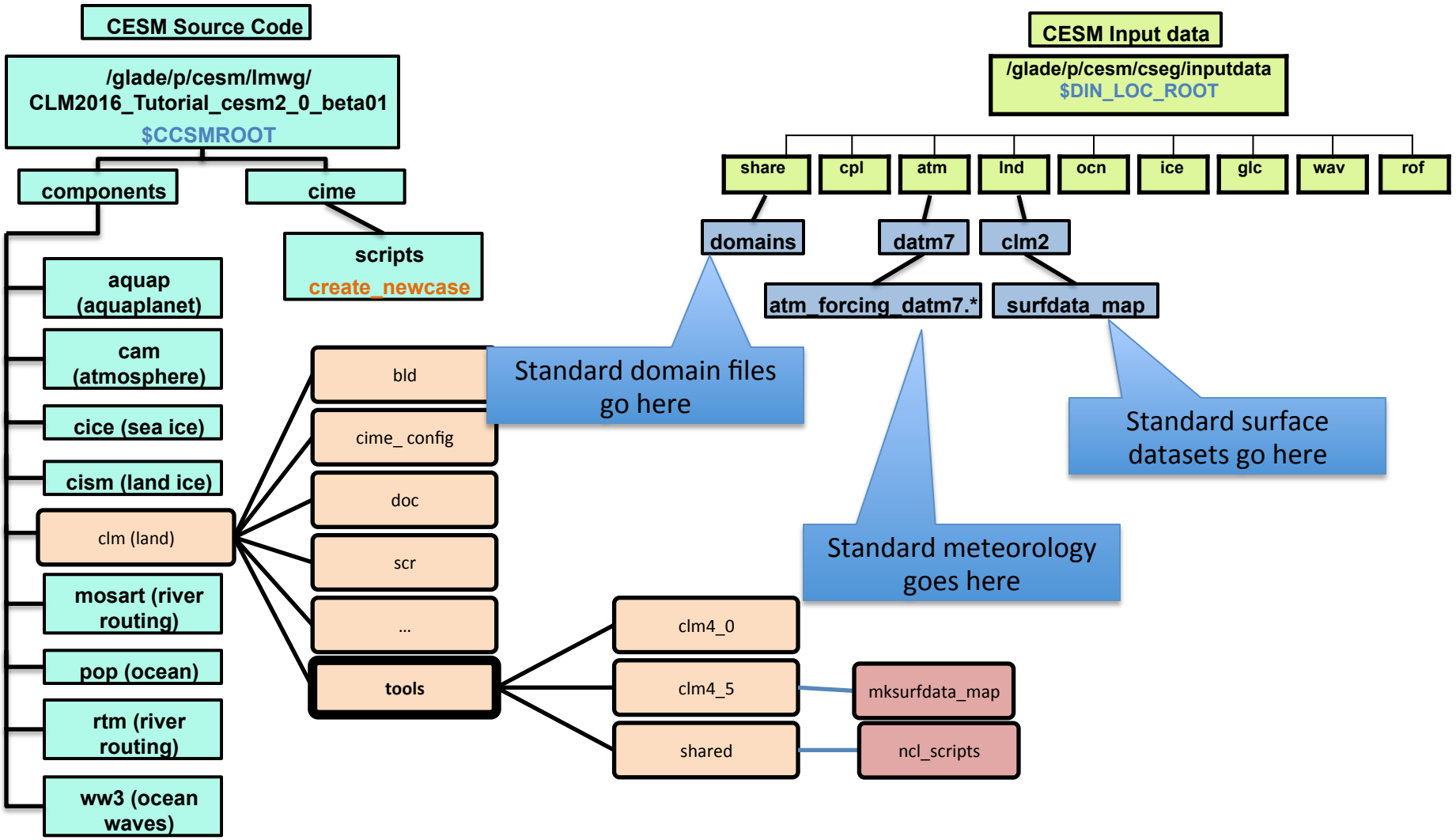
Defines a box of grid points to use as your region to run on

Pick SW corner…

CLM tools directories



CLM tools directories



Exercise 2: 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 getregional_datasets.pl script

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.

TO DO:

```
# go to the source code
cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/

# move to & save the locations the getregional script
cd components/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
```

Exercise 2: Sub-setting global datasets

TO DO (CONTINUTED):

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

 # Run getgetional 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

LOOK AT YOUR NEW DOMAIN AND SURFACE DATA FILES!

- **For more information see the README files in** /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/components/clm/tools/shared/ncl_scripts

Exercise 2: Creating a case based on getregional files

We use the CLM_USRDAT as the resolution

ADDITIONALLY we add 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).

TO DO (FOR EXTRA CREDIT) HOPEFULLY THIS LOOKS FAMILIAR!

(1) create a new case

```
cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/cime/scripts
./create_newcase -case ~/Day2Alaska -res CLM_USRDAT -compset IMCRUCLM50BGC
-user_mods_dir $MYDATADIR
```

(2) invoke case.setup

```
cd ~/Day2Alaska
./case.setup
```

(3) build the executable

```
./case.build
```

(4) submit your run to the batch queue

```
./case.submit
```

Exercise 2: Objectives & Questions

1. **Use getregional_datasets.pl in the CLM tools directory to create a domain and surface dataset for a region over Alaska (from global datasets)**
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 (mksurfdata.pl)?**

Exercise 2: Objectives & Questions

1. Use getregional_datasets.pl in the CLM tools directory to create a domain and surface dataset for a region over Alaska (from global datasets).
2. What is the creation date string (ending _cYYMMDD form) of the files you just created?

- c150114 Jan/14/2015

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?

- ./components/clm/tools/shared/gen_domain
- ./components/clm/tools/mapping/gen_domain_files/

4. In what two CLM tools directories is there a program to create surface datasets (mksurfdata.pl)?

- ./components/clm/tools/clm4_5/mksurfdata_map
- ./components/clm/tools/clm4_0/mksurfdata_map

Exercise 3: PTCLM

Using PTCLMmkdata

Exercise 3: PTCLM

What is PTCLM?

PTCLM is a script that helps you create 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

DEPENDING on the compset (DATM_MODE) you choose.

Exercise 3: Summary

A. Introduce the tools that make CLM input files.

B. Provide an overview of the process to create input files.

C. Review the steps that PTCLMmkdata uses to create datasets.

D. Create some datasets using PTCLMmkdata.

E. Create a case based on the datasets created.

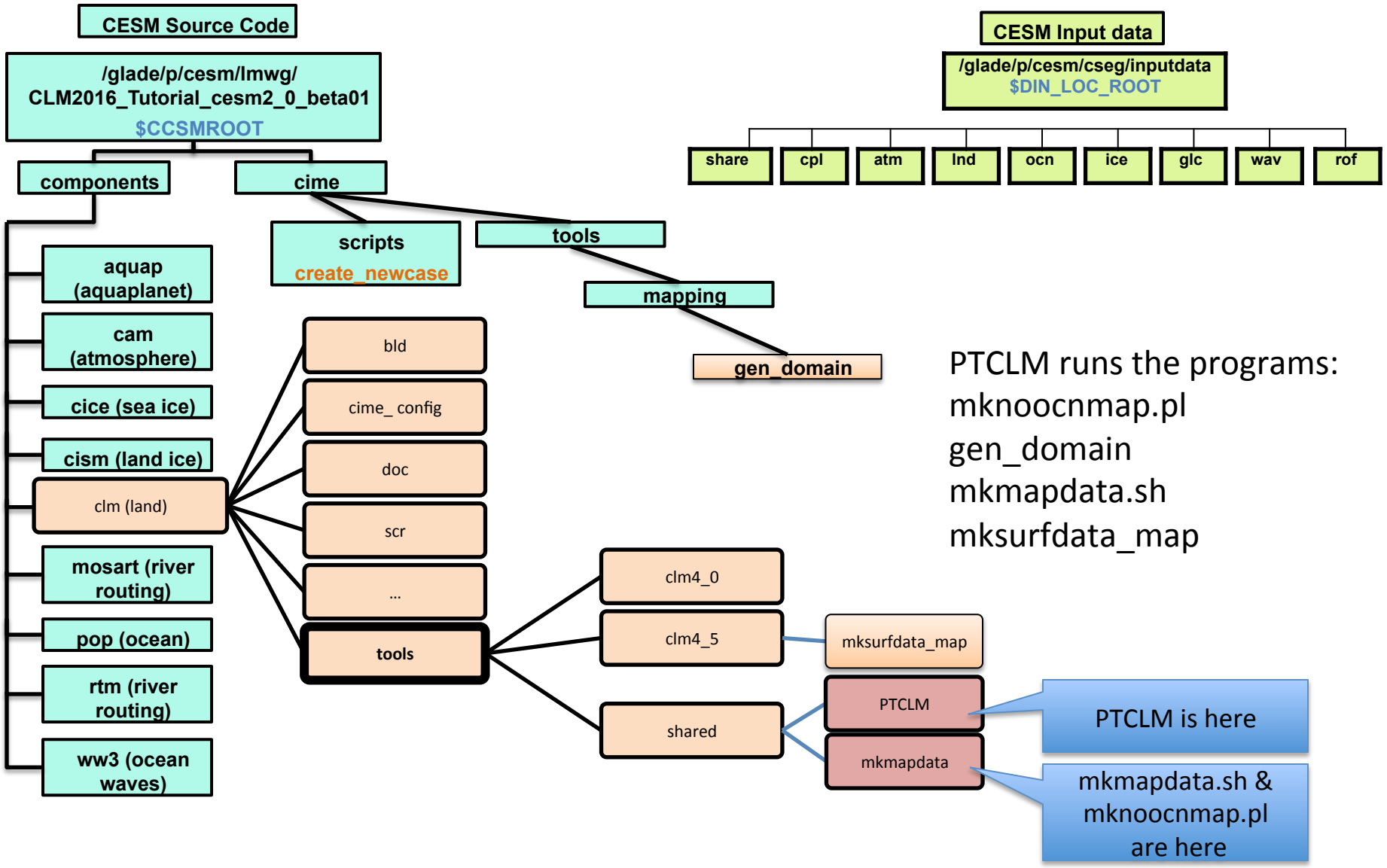
Exercise 3: Objectives & Questions

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 (HINT, look in PTCLM_sitedata)?**
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)?**

CLM tools directories

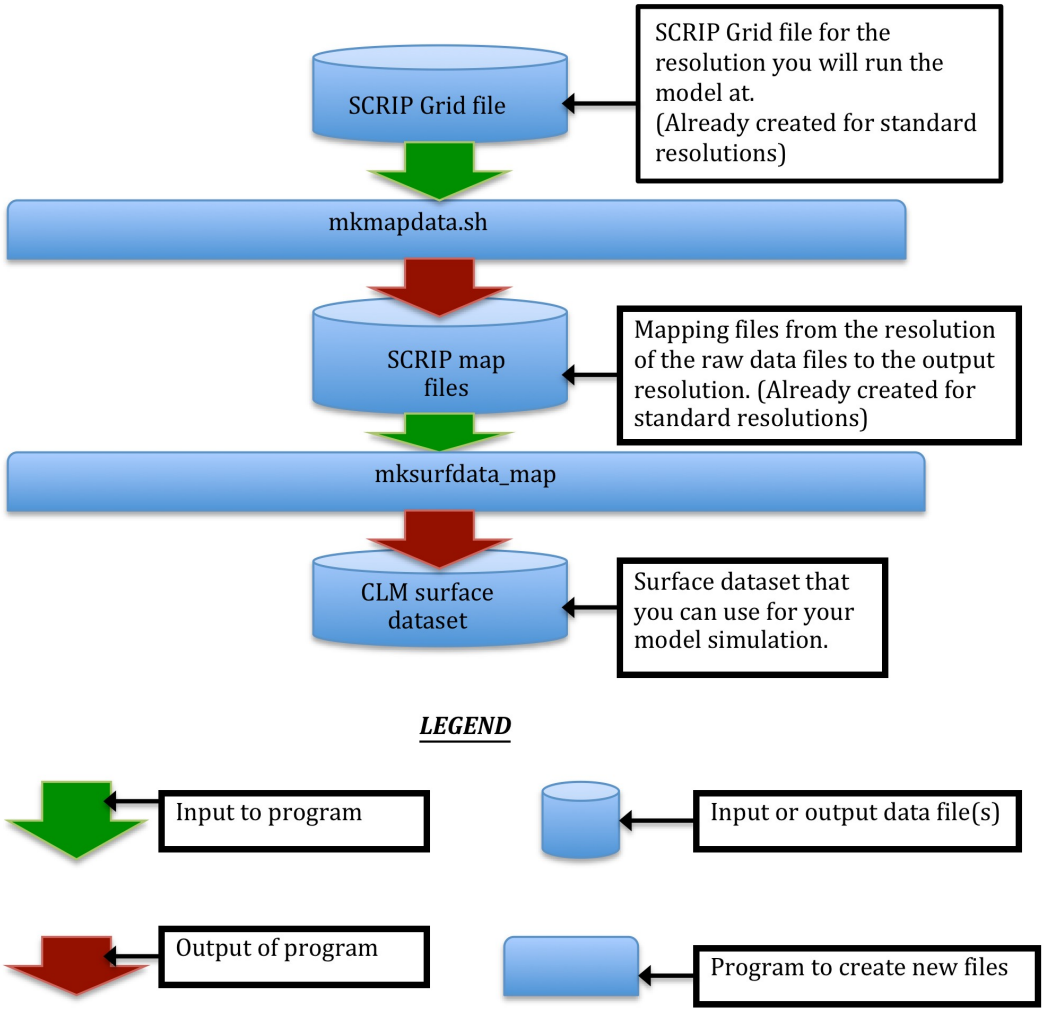


PTCLM runs the programs:
mknoocnmap.pl
gen_domain
mkmapdata.sh
mksurfdata_map

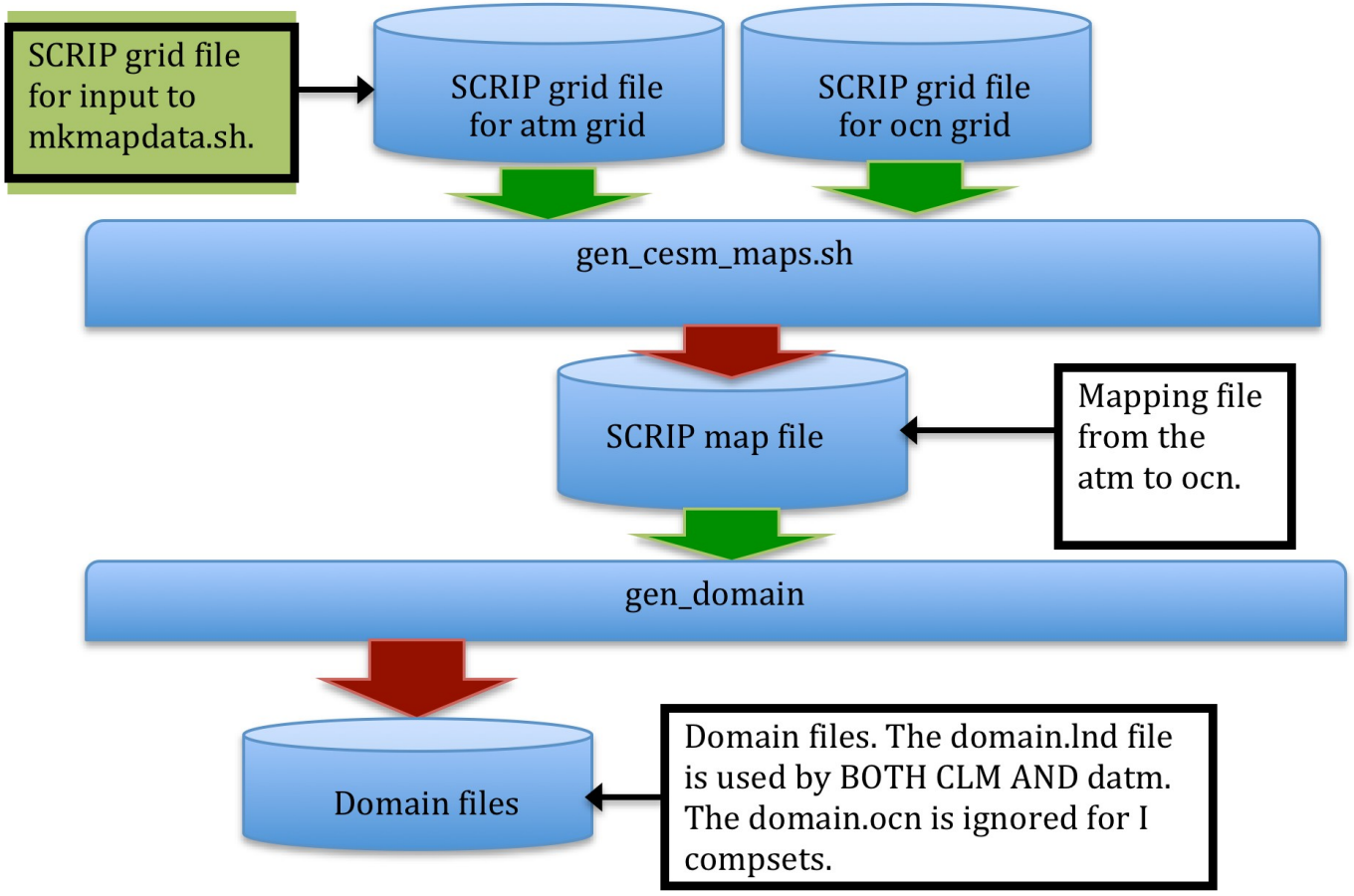
PTCLM is here

mkmapdata.sh &
mknoocnmap.pl
are here

Exercise 3: data flow for creating dataset



Exercise 3: data flow for creating domain files

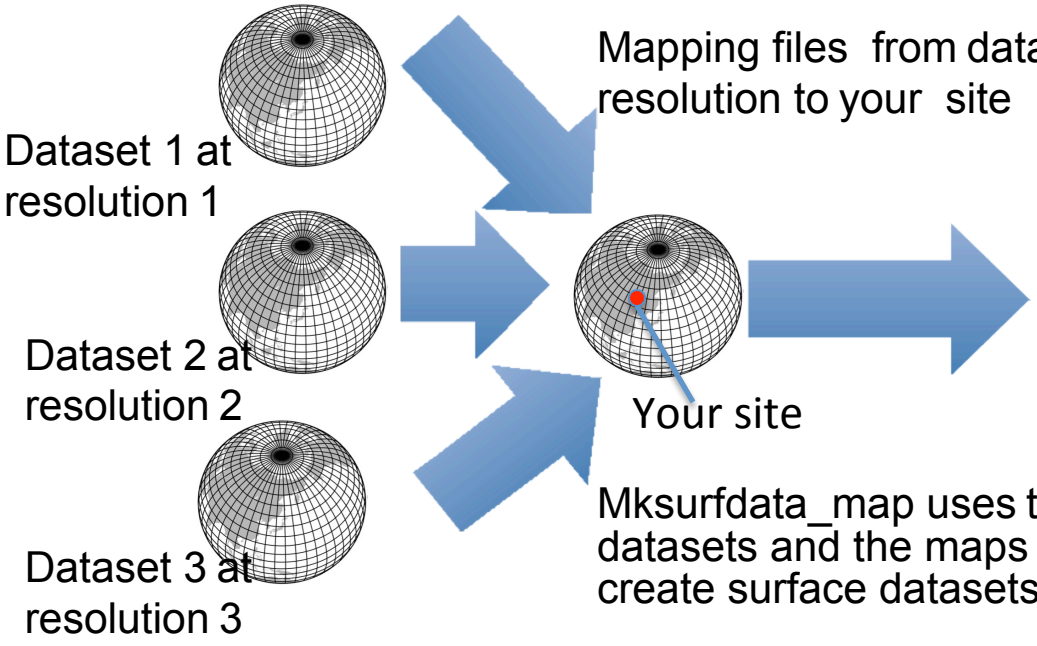


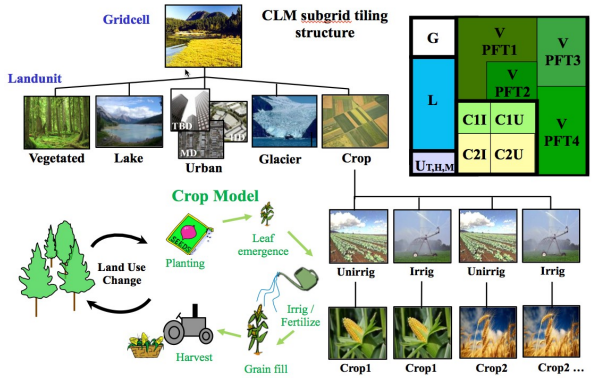
Exercise 3: Steps that PTCLMmkdata does for you to create datasets

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 mksurfdata_map uses.
4. Runs mksurfdata.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).

Exercise 3: PTCLM 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

***NOTE these can always be check & changed**

Exercise 3: Running PTCLM!

PTCLMmkdata 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.***

TO DO:

 # go to the source code
cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/

 # setup the names that we'll use
setenv CESM_ROOT `pwd`
setenv CSMDATA /glade/p/cesmdata/cseg/inputdata
setenv MYSITE 1x1pt_US-UMB ***#Univ. of Michigan Biological Station, go BLUE!***

 # Setup your data directory (NOTE: NORMALLY THIS WOULD NOT BE IN SCRATCH!!)
setenv MYDATADIR /glade/scratch/$USER/mydatadir
mkdir -p $MYDATADIR/$MYSITE

 # copy the mapping files over so they won't have to be created again
 # NOTE: creating these maps takes a LONG time. See the README files in the PTCLM directory if you need to do this for your site.
cp -p $CSMDATA/lnd/clm2/PTCLMmydatafiles.c160208/$MYSITE/map*160208.nc $MYDATADIR/$MYSITE/

Exercise 3: Running PTCLM!

TO DO (continued):

 # 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/components/clm/tools/shared/PTCLM/PTCLMmkdata --map_gdate 160208 -s US-UMB -d $CSMDATA --mydatadir $MYDATADIR

NOW PTCLM has made your surface dataset and domain files for us University of Michigan Biological Station

Exercise 3: Creating a case based on PTCLM

We use the CLM_USRDAT as the resolution

ADDITIONALLY we add 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).

TO DO (FOR EXTRA CREDIT) HOPEFULLY THIS LOOKS FAMILIAR!

(1) create a new case

```
cd /glade/p/cesm/lmwg/CLM2016_Tutorial_cesm2_0_beta01/cime/scripts

./create_newcase -case ~/Day2UMB -res CLM_USRDAT -compset I1PTCLM50 -
user_mods_dir $MYDATADIR/$MYSITE/
```

(2) invoke case.setup

```
cd ~/Day2UMB

./case.setup
```

(3) build the executable

```
./case.build
```

(4) submit your run to the batch queue

```
./case.submit
```

Exercise 3: Objectives & Questions

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 3: Objectives & Questions

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?** 42
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, --mksurfdata_opts
5. **What options to PTCLMmkdata will change how your case is setup (but not change datasets)?**
   --cycle_forcing, --donot_use_tower_yrs, --clmnmlusecase, and –phys (changes both)

FINAL REMINDER, it's good practice to document 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

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