CLM5.0 Tutorial: Single point simulations

Will Wieder
Sean Swenson

NCAR is sponsored by the National Science Foundation

Outline

Lecture/Intro

- Why Single Point
- What is spinup & how do we make it faster?
- Overview for running single point simulations.

Practical

A. Create a global case
B. Extract point data from global datasets
C. Create a case for the single point run
D. Spinup and additional bgc considerations

Other Useful Info

Why single point?

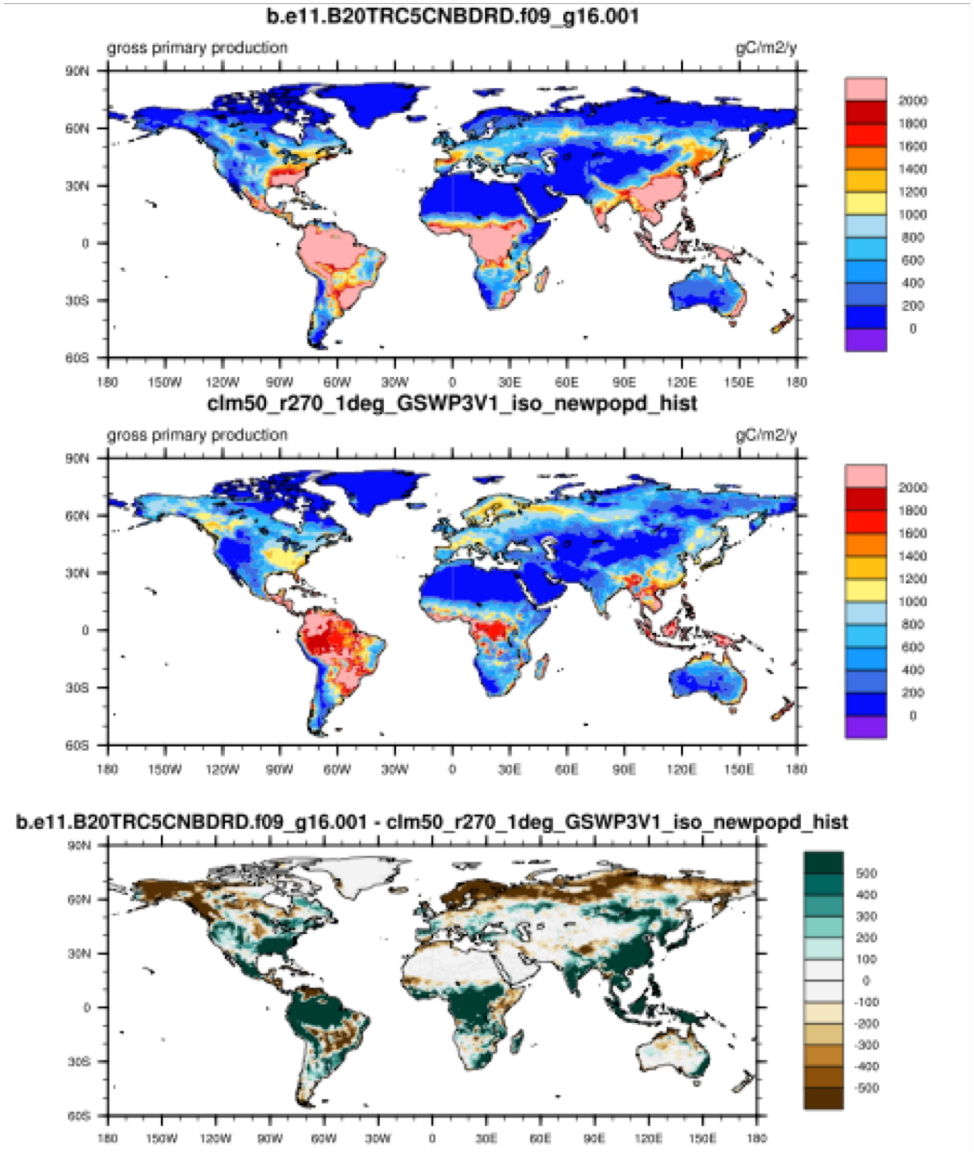
- **Code development**
- **Testing & debugging**
- **Comparison with point observations**

Some publications & references

Swenson et al. (2019) JAMES, 11
Burns et al. (2018) *JAMES*, 10, 617-651.
Cheng et al. (2018) *BG Discuss.*, 1-38.
Schädel, et al. (2018). *ERL*, 13, 105002.
Wieder et al (2017) *JRG-Biogeosci*, 122, 825–845
Bonan et al (2014) GMD, 7, 2193-2222
Fisher et al. (2015) GMD, 8 3593-3619
Levis et al. (2014) GMD, 7 613-620

*Vegetation heat capacity
*Diel cycle of LH fluxes @ tower site
*N uptake in CLM5 & ^{15}N tracers
* Regional permafrost dynamics
* BGC simulations @ LTER site
* Multi-layer canopy
* FATES-beta
* Agricultural tillage (C cycle!)

Spinup?



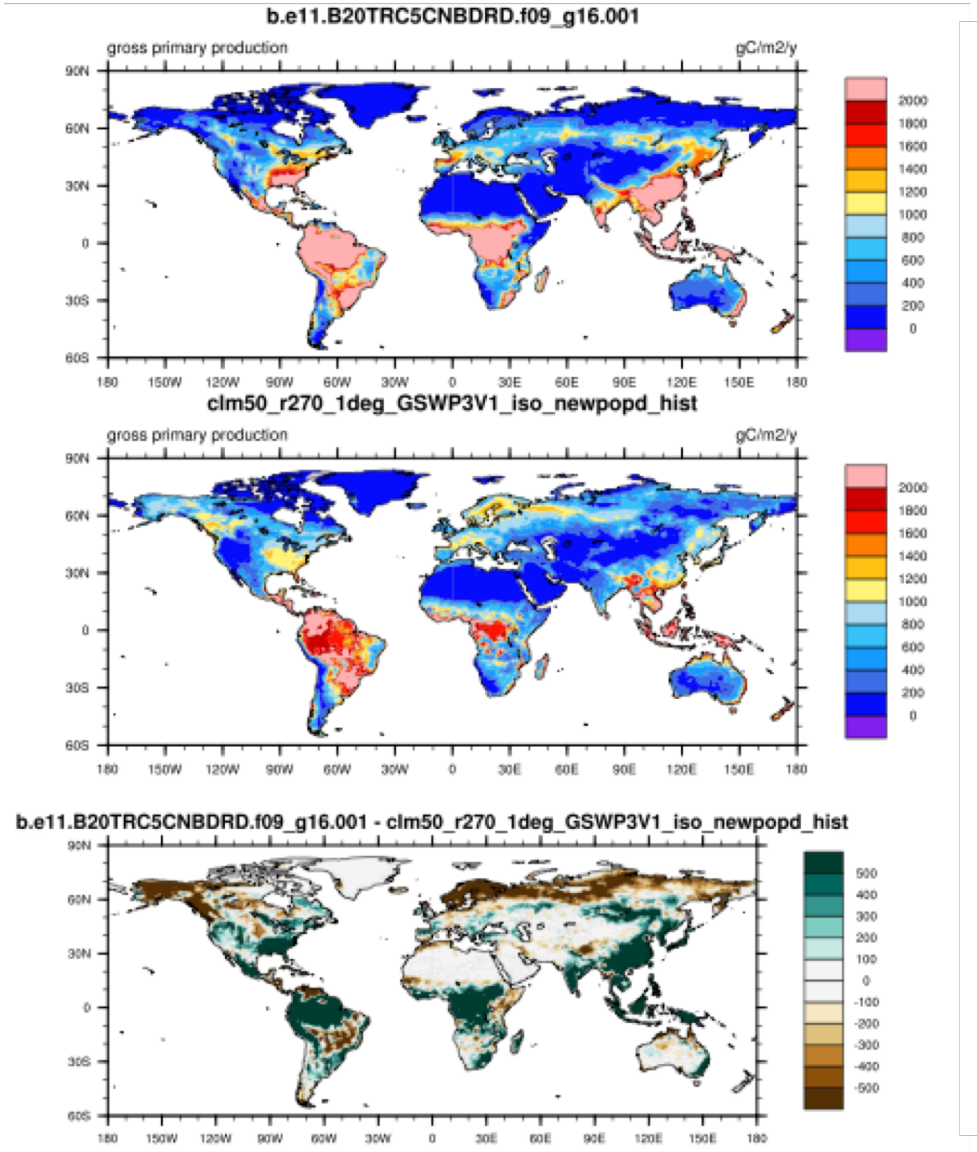
We make make steady state assumptions about 'initial' state of ecosystem properties:

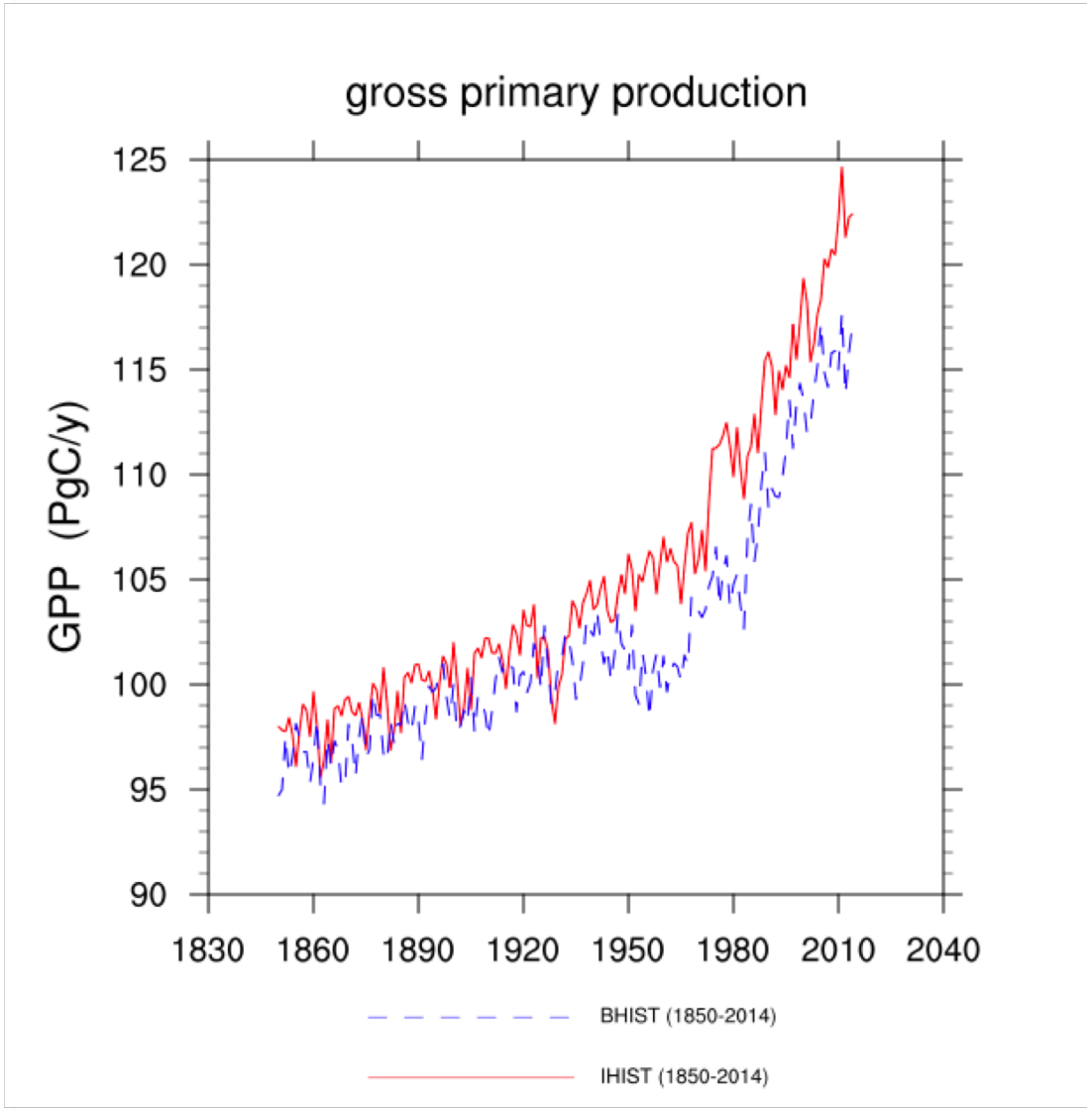
- Temperature
- Water, snow, ice
- Carbon & nitrogen

It is the 'equilibrium' state, given the forcing data e.g. coupled vs. offline GPP (right)

Then, changes in in ecosystem states or fluxes are related to the transient (or forced) responses

Spinup?


b.e11.B20TRC5CNBDRD.f09_g16.001
gross primary production
gC/m2/y
clm50_r270_1deg_GSWP3V1_iso_newpopd_hist
b.e11.B20TRC5CNBDRD.f09_g16.001 - clm50_r270_1deg_GSWP3V1_iso_newpopd_hist


gross primary production
GPP (PgC/y)
BHIST (1850-2014)
IHIST (1850-2014)

How do we accelerate spinup?

Get the 'slow' pools moving!

./xmlchange --file env_run.xml --id CLM_ACCELERATED_SPINUP --val on

echo "spinup_state = 2" >> user_nl_clm [not necessary with above]

http://legendaryliving.life/

- Accelerate mortality of 'dead wood' pools

Net effect = increases wood C turnover &
Reduces size of veg C pools

CNGapMortalityMod.F90

- Accelerate turnover of litter and soil pools
- Accelerate advection and diffusion terms too
- In CLM5, this is calculated as a function of latitude so that spinup is more accelerated in high latitude regions

Net effect = increases soil C turnover & reduces size of soil C pools

SoilBiogeochemDecompCascadeBGCMod.F90
SoilBiogeochemLittVertTranspMod.F90
SoilBiogeochemStateType.F90

How do we go back to normal?

Get the ‘slow’ pools moving!

```
./xmlchange --file env_run.xml --id CLM_ACCELERATED_SPINUP --val off
echo "spinup_state = 0" >> user_nl_clm
```

- Multiply pool size by ‘acceleration factors’

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**
- **Spinup**

Exercise 4: Create & run single point simulations

A. Create a global case

B. Extract data from global datasets to create domain & surface data (+ RTM directional in regional cases, not covered here).
   - Modify singlept_xr.py

C. Create a new case for the single point run
   - Modify env_mach_pes.xml, env_batch.xml, & env_run.xml
   - Copy and modify user_datm.streams files
   - Modifications for history files
   - Build & submit case

D. Spinup & checking for equilibrium stocks / fluxes
   - SP vs BGC simulations
   - AD mode, postAD & transient runs [BGC]
   - Additional history file modifications.

Start Practical Here

Exercise 4a: Create a global case

(1) create a new case

(2) invoke case.setup

Stop here

---

(3) build the executable

(4) submit your run to the batch queue

Exercise 4a: Create a global case

To Do:

hopefully this looks familiar

Navigate to the scripts directory in the source code directory:

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/cime/scripts
```

(1) create a new case

Type this command line:

```
./create_newcase --case /glade/u/home/$USER/clm_tutorial_cases/IHistCLM50sp_001 --res
f09_g17_gl4 --compset IHistClm50Sp --run-unsupported
```

(2) invoke case.setup

Navigate to your case directory:

```
cd ~/clm_tutorial_cases/IHistCLM50sp_001
```

Type this command line:

```
./case.setup
./preview_namelists
```

preview_namelists will generate a lnd_in file

Look for the path to the surface data set and domain file in your land_in dile

```
cat CaseDocs/lnd_in
```

Exercise 4: Create & run single point simulations

A. Create a global case

B. Extract data from global datasets to create domain & surface data (+ RTM directional in regional cases, not covered here).
   - Modify singlept_xr.py

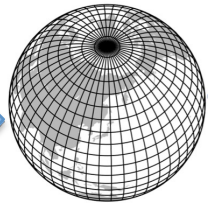
C. Create a new case for the single point run
   - Modify env_mach_pes.xml, env_batch.xml, & env_run.xml
   - Copy and modify user_datm.streams files
   - Modifications for history files
   - Build & submit case

D. Spinup & checking for equilibrium stocks / fluxes
   - SP vs BGC simulations
   - AD mode, postAD & transient runs [BGC]
   - Additional history file modifications.

Exercise 4b: Generate domain and surface datasets

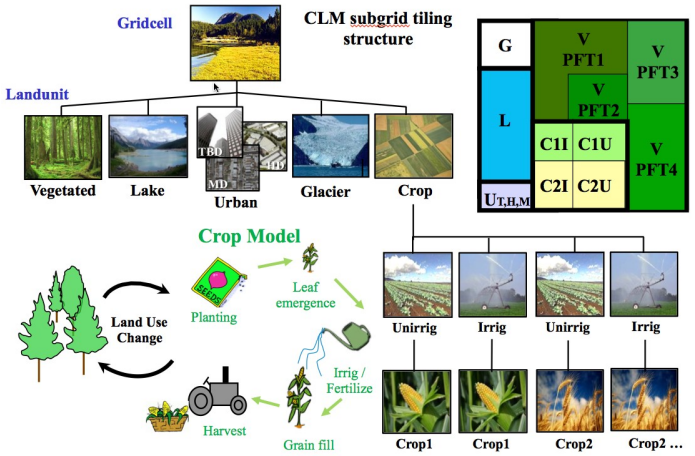
What's in the domain and surface files?

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

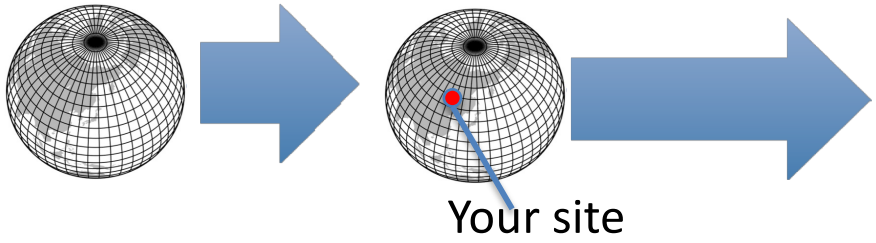
* **NOTE** these can both be modified as appropriate for a particular site / gridcell

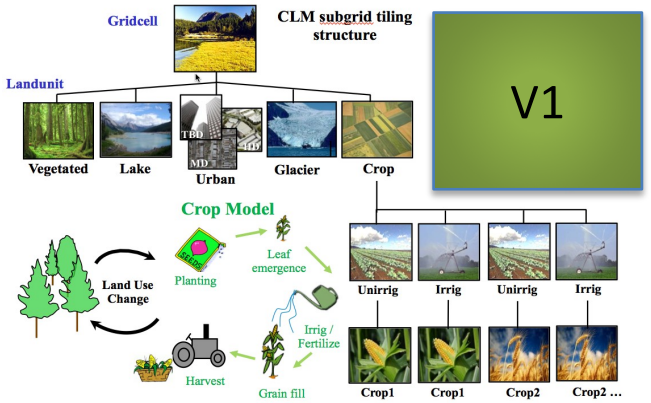
Exercise 4b: Generate domain and surface datasets

The python script we're about to use does the following:

singlept script uses the global information from nearest grid cell to your site

Surface dataset @ resolution Y





Surface Dataset

singlept script it also will pull DATM forcing for your grid cell too!

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

Aside: What is DATM_MODE?

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

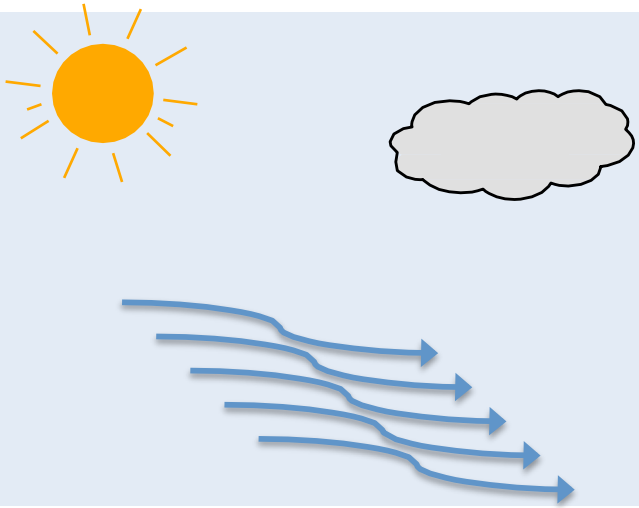
1) CLMGSWP3 *this is the preferred meteorological data to use w/ CLM5,

2) CLMCRUNCEP

3) CLM_QIAN [deprecated]

4) CLM1PT

5) **CPLHIST [this name may have changed]**



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**
- **CPLHIST– Use atmospheric data from a previous CESM simulation**

Exercise 4b: Generate domain and surface datasets

1) Navigate to the tools/contrib directory

cd ~/clm5.0_2019tutorial/tools/contrib

The singlept code will create single point domain, surface data, & datm forcing files by extracting data from global datasets

The script automatically creates directories to store the data in scratch
It's likely worth moving these if you're doing a bunch of simulations down the road…

First, you'll need to load python libraries & run the code

```
module load python/2.7.14
ncar_pylib
./singlept
```

Now you should have new domain, surface, and land use data sets in glade/scratch/$USER/single_point/

Exercise 4b: Generate domain and surface datasets

STOP

2a) We've already done this for your, but for NEW single point simulations, you can modify singlept

STOP

2b) For regional runs, use your favorite text editor to modify subset_surfdata

This code will create regional domain, surface data, & rtm directional files by extracting data from global datasets. ***STOP** **We won't do this today***

Exercise 4b: Generate domain and surface datasets

STOP. This is just background information, you don't need to do anything

There's a bunch of information in the singlept script that's worth reading
Below are the cliff notes of what you'll need to look for and change.

Let's set this up for Harvard Forest.

```
 plon =287.8                    #lon should be 0 to 360 (no negative values for points in the west!)
 plat = 42.5                   #lat should be -90 to 90
create_domain   = True
create_surfdata = True
create_landuse  = True          # if you want a transient run you should set this flag to true…
create_datm     = False
```

** Note single point runs are much faster if you create new datm files for just a single point (create_datm = True), but this takes some time. Instead we'll point to files that have been generated for you later on.*

The following are designed to simulate sub-grid heterogeneity, which may not be appropriate for a site-level simulation

Here we'll create a surface data set with a broadleaf deciduous temperate pft

```
overwrite_single_pft = True       # makes whole grid 100% single pft
dominant_pft            = 7        # broadleaf deciduous temperature pft
zero_nonveg_pfts     = True       # Sets all non_veg lanunits to 0
uniform_snowpack     = True       # sub-grid elevation controls over snow melt
no_saturation_excess = True      # sets FMAX = 0, below

dir_output='/glade/scratch/'+myname+'/single_point/'
dir_output_datm=dir_output + 'datmdata/'
```

Exercise 4b: Generate domain and surface datasets

STOP! You don't need to the steps below for the purposes of the tutorial.
This information should have been pre-staged for you, but it never hurts to check…
Also, these can be modified if you want to change the names of your output or the sources of global data files.

```
# You can find the path for climate forcing data from your case directory .CaseDocs/datm.streams.txt.CLMGSWP3v1.Precip
dir_input_datm='/glade/p/cgd/tss/CTSM_datm_forcing_data/atm_forcing.datm7.GSWP3.0.5d.v1.c170516/'
dir_output_datm=dir_output + 'datmdata/'
```

Point to the right domain and surface data sets (from lnd_in)

```
fdomain = '/glade/p/cesmdata/cseg/inputdata/share/domains/domain.lnd.fv0.9x1.25_gx1v6.090309.nc'
# Change the resolution and date of the output file
fdomain2 = dir_output + 'domain.lnd.fv0.9x1.25_gx1v7.'+tag+'_c230119.nc'

fsurf = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/surfdata_0.9x1.25_16pfts_Irrig_CMIP6_simyr1850_c170824.nc'
fsurf2   = dir_output + 'surfdata_0.9x1.25_16pfts_Irrig_CMIP6_simyr1850_'+tag+'_c230119.nc'

fluse = '/glade/p/cesmdata/cseg/inputdata/lnd/clm2/surfdata_map/landuse.timeseries_0.9x1.25_hist_16pfts_Irrig_CMIP6_simyr1850-2015_c170824.nc
fluse2   = dir_output + 'landuse.timeseries_0.9x1.25_hist_16pfts_Irrig_CMIP6_simyr1850-2015_'+tag+'_c230119.nc'
```

Exercise 4b: optional

The singlept script allows some control over aspects of the data creation, but some users may want a bit more!

STOP*! his information is provided to get users familiar with these capabilities of the model, but will NOT be used in the examples that follow.*

Local surface datasets and climate forcing may be more important if we want to compare to flux tower observations or other site level measurements. For many FluxNet sites, we have local information on soil properties, pft coverage, and other site information [e.g. flux measurement ranges].

You can add your own site level information to the ***PTCLMDATA*.txt*** *files found in the directory below*

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/tools/PTCLM/PTCLM_sitedata/
```

You can customize ***modify_singlept_site***

-- This script overwrites some fields with site-specific data --

```
cd ~/clm5.0_2019tutorial/tools/contrib/
vi modify_singlept_site
```

Then make the following changes [again for the Harvard Forest example]

```
sitename='US-Ha1'
site_dir= '/glade/u/home/$USER/clm5.0_2019tutorial/tools/PTCLM/PTCLM_sitedata/'
fsurf  = '/glade/scratch/$USER/single_point/surfdata_0.9x1.25_16pfts_Irrig_CMIP6_simyr1850_287.8_42.5_c230119.nc'
dir_output='/glade/scratch/$USER/single_point/'
```

Now you can save your change and run

```
./modify_singlept_site
```

You should see a new surface dataset appended with the sitename provided above in /glade/scratch/$USER/single_point/

Generating climate atmospheric forcing data from flux or met tower data is more complicated. Contact Keith or Sean if you want to do this. We'll show you how to point to particular atmospheric forcings in the next step

Exercise 4: Create & run single point simulations (PTclm)

A. Create a global case

B. Extract data from global datasets to create domain & surface data (+ RTM directional in regional cases, not covered here).
   - Modify singlept_xr.py appropriately

C. Create a new case for the single point run
   - Modify env_mach_pes.xml, env_batch.xml, & env_run.xml
   - Copy and modify user_datm.streams files
   - Modifications for history files
   - Build & submit case

D. Spinup & checking for equilibrium stocks / fluxes
   - SP vs BGC simulations
   - AD mode, postAD & transient runs [BGC]
   - Additional history file modifications.

Exercise 4c: Create a new case for the single point run

(1) create a new case

(2) invoke case.setup

(3) build the executable

(4) submit your run to the batch queue

Exercise 4c: Create a new case for the single point run

hopefully this looks *really* familiar

Navigate to the scripts directory in the source code directory:

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/cime/scripts
```

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here. The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50sp_001 --run-unsupported
```

Wait! What are we actually doing here?

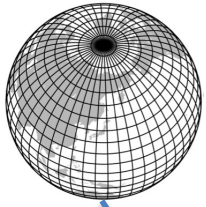
REVIEW: Create a new case

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

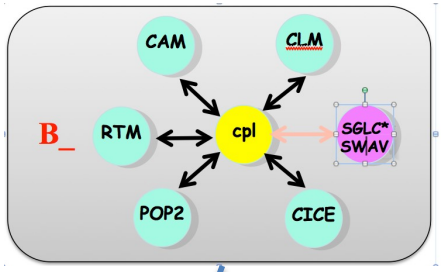
create_newcase requires 3 arguments

What is the casename ?

Which resolution?



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



--case ~/clm_tutorial_cases/Ha1_CLM50sp_001 --res f09_g17 --compset 2000_DATM%GSWP3v1…

Your Case Name

This is a 1 degree global run?!

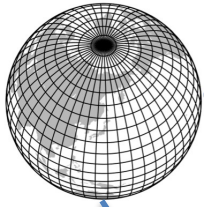
GSWP3 atmosphere + CLM5.0 SP mode, with stub ice, stub river, etc , etc.

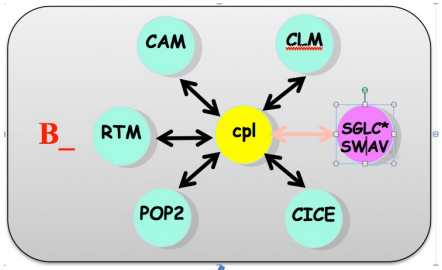
`--compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV`

This does not look like a single point simulation!
But we'll change all that soon…
Also single point runs are cheap to run,
we need to set up the PES configuration accordingly

Which resolution?

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





--res f09_g17

--compset 2000_DATM%GSWP3v1…

This is a 1 degree global run?!

GSWP3 atmosphere + CLM5.0 SP mode, with stub ice, stub river, etc , etc.

```
--compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV
```

This does not look like a single point simulation!
But we'll change all that soon…
we need to set up the PES configuration accordingly

There are a TON of command line changes to modify your simulations in the slides that follow
All of these can be found in the text file below.
You're welcome to copy this text into your terminal window [from the case directory]

```
cd ~/clm_tutorial_cases/Ha1_CLM50sp_001
cp /gpfs/u/home/wwieder/clm_tutorial_cases/Ha1_CLM50sp_001/README.xml_modsSP.txt .
```

Exercise 4c: Create a new case for the single point run

hopefully this looks *really* familiar

Navigate to the scripts directory in the source code directory:

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/cime/scripts
```

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here. The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50sp_001 --run-unsupported
```

Start here

Navigate to your case directory:

```
cd ~/clm_tutorial_cases/Ha1_CLM50sp_001
```

The we can make the following xml changes BEFORE you set up your case

```
./xmlchange --file env_mach_pes.xml --id COST_PES --val 36
./xmlchange --file env_mach_pes.xml --id TOTALPES --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS_PER_INST --val 1
./xmlchange --file env_mach_pes.xml --id ROOTPE --val 0
./xmlchange --file env_batch.xml --id JOB_QUEUE --val share
./xmlchange MPILIB=mpi-serial
./xmlchange --file env_batch.xml --id JOB_WALLCLOCK_TIME --val 1:00:00
```

Exercise 4c: Create a new case for the single point run

hopefully this looks *really* familiar

Navigate to the scripts directory in the source code directory:

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/cime/scripts
```

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here. The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50sp_001 --run-unsupported
```

(2) invoke case.setup

Type this command line:

```
./case.setup
./xmlchange --file env_run.xml --id CLM_FORCE_COLDSTART --val on
./xmlchange --file env_run.xml --id CLM_NML_USE_CASE --val 1850_control
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_START --val 1901
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_END --val 1920
./xmlchange --file env_run.xml --id DATM_PRESAERO --val clim_1850
./xmlchange --file env_run.xml --id CCSM_CO2_PPMV --val 284.7
./xmlchange --file env_run.xml --id STOP_OPTION --val nyears
./xmlchange --file env_run.xml --id STOP_N --val 5
./xmlchange --file env_run.xml --id RUN_REFDATE --val 0001-01-01
./xmlchange --file env_run.xml --id RUN_STARTDATE --val 0001-01-01
```

```
# ----------------------------------------------
#   Optional
# some of the code mods (left) may not be necessary
# if we set up the case using an 1850 compset, e.g.
1850_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV

If you have time:
1.  try to create a new case [Ha1_CLM50sp_002] and
2.  compare your env_run.xml files
# ----------------------------------------------
```

Exercise 4c: Create a new case for the single point run

And a few more change to env_run…

Then we get to point to our new domain files made in 4b

**you can change this to your own directory, or use mine*

```
./xmlchange --file env_run.xml --id ATM_DOMAIN_FILE --val domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
./xmlchange --file env_run.xml --id ATM_DOMAIN_PATH --val /glade/scratch/$USER/single_point
./xmlchange --file env_run.xml --id LND_DOMAIN_FILE --val domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
./xmlchange --file env_run.xml --id LND_DOMAIN_PATH --val /glade/scratch/$USER /single_point
```

And name list changes to:

- *point to our surface data set we made in 4b*
- *interpolation of datm files (e.g. N deposition and aerosols)*

```
echo "fsurdat = '/glade/scratch/wwieder/single_point/surfdata_0.9x1.25_16pfts_CMIP6_simyr1850_287.8_42.5_c170706.nc'" >> user_nl_clm
echo "mapalgo = 'nn','nn','nn','nn','nn'" >> user_nl_datm
```

You'll currently be writing out monthly averaged history files for a single point.

For extra credit you can modify user_nl_clm to write out multiple time steps to a single file!

Exercise 4c: Create a new case for the single point run

datm.streams files point to the atmospheric forcing data including:
Winds, pressure, humidity, solar, long-wave down, temperature, and precipitation.
They also point to the aerosols and topography that is used by the model

You can have a look like this

./preview_namelists

ls CaseDocs/datm.streams.txt*

Since we're pointing to our own data atmosphere for the single point of interest we need to tell the model where these inputs can be found

To do this, you have to point to the atmospheric forcing data for your single point

This is kind of time consuming, so here's the short cut [don't worry details are on the next page if you're keen]

cp /glade/u/home/wwieder/clm_tutorial_cases/Ha1_CLM50sp_001/user_datm.streams.txt.* .

Note, this is also how you'd point to local meteorology from your specific tower site, instead of data from a global reanalysis, as we're using here

Exercise 4c: Create a new case for the single point run

STOP! You don't need to the steps below for the purposes of the tutorial,
But in the future you can follow these step if you have your own data to point to.

** you didn't make the new datm files in 4a, so you you'll have to use my path on this one*

./preview_namelists

#Then copy datm.streams files and rename each with user_* in the title

cp CaseDocs/datm.streams.txt.* .

mv datm.streams.txt.CLMGSWP3v1.Precip user_datm.streams.txt.CLMGSWP3v1.Precip.

***You'll need to do this for each streams file**

#Finally edit the user_datm.streams.txt.CLMGSWP3v1 files so they point to the single point data we generated by indicating the correct: [note, we don't need to change the user_datm.streams.txt.presaero.clim_1850 or *topo.observed files]*

<filePath> /gpfs/fs1/scratch/wwieder/single_point/datmdata_Ha1_gswp3

Domain file domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc

File names *e.g.* clmforc.GSWP3.c2011.0.5x0.5.Prec.**287.8_42.5.**1901-01.nc

Exercise 4c: Create a new case for the single point run

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here.
The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%SP_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --
case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50sp_001 --run-unsupported --project UCGD0004
```

(2) invoke case.setup

Then, navigate to your case directory:

```
cd ~/$USER/clm_tutorial_cases/Ha1_CLM50sp_001
```

Type this command line:

```
./case.setup
```

Start here

(3) build the executable

Type this command line:

```
gcmd -q R4231266 -- ./case.build
```

(4) submit your run to the batch queue

Type this command line:

```
./case.submit
```

Exercise 4: Create & run single point simulations

A. Create a global case

B. Extract data from global datasets to create domain & surface data (+ RTM directional in regional cases, not covered here).
   - Modify singlept_xr.py appropriately

C. Create a new case for the single point run
   - Modify env_mach_pes.xml, env_batch.xml, & env_run.xml
   - Copy and modify user_datm.streams files
   - Modifications for history files
   - Build & submit case

D. Spinup & checking for equilibrium stocks / fluxes
   - SP vs. BGC simulations
   - AD mode, postAD & transient runs [BGC]
   - Additional history file modifications.

Check out the introduction to spinup for the SP runs in your Jupyter Lab space.
Then we'll move on to BGC simulations with single point

First you'll have to update your juypter notbook

```
cp /glade/p/cgd/tss/CTSM_tutorial2019/ctsm_tutorial_jupyter/notebooks/Practical4.ipynb
~/ctsm_tutorial_jupyter/notebooks/.
```

Exercise 4d: Single point BGC_AD

Running in 'accelerated decomposition' (AD) mode

Navigate to the scripts directory in the source code directory:

```
cd /glade/u/home/$USER/clm5.0_2019tutorial/cime/scripts
```

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here. The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%BGC_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50bgc_001 --run-unsupported --project UCGD0004
```

Navigate to your case directory:

```
cd ~/clm_tutorial_cases/Ha1_CLM50bgc_001
```

The we can make the following xml changes BEFORE you set up your case

```
./xmlchange MPILIB=mpi-serial
./xmlchange --file env_mach_pes.xml --id COST_PES --val 36
./xmlchange --file env_mach_pes.xml --id TOTALPES --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS_PER_INST --val 1
./xmlchange --file env_mach_pes.xml --id ROOTPE --val 0
./xmlchange --file env_batch.xml --id JOB_QUEUE --val share
./xmlchange --file env_batch.xml --id JOB_WALLCLOCK_TIME --val 6:00:00
```

#We'll give these a longer time to run

Exercise 4d: Single point BGC_AD

(1) create a new case

(2) invoke case.setup

Type this command line (these are identical to ex 4c)

./case.setup
./xmlchange --file env_run.xml --id CLM_FORCE_COLDSTART --val on
./xmlchange --file env_run.xml --id CLM_NML_USE_CASE --val 1850_control
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_START --val 1901
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_END --val 1920
./xmlchange --file env_run.xml --id DATM_PRESAERO --val clim_1850
./xmlchange --file env_run.xml --id CCSM_CO2_PPMV --val 284.7
./xmlchange --file env_run.xml --id STOP_OPTION --val nyears
./xmlchange --file env_run.xml --id RUN_REFDATE --val 0001-01-01
./xmlchange --file env_run.xml --id RUN_STARTDATE --val 0001-01-01

Additional xml changes for AD mode

./xmlchange --file env_run.xml --id CLM_ACCELERATED_SPINUP --val on

The data you analyze was generated like this…

#./xmlchange --file env_run.xml --id STOP_N --val 400
#./xmlchange --file env_run.xml --id REST_N --val 100

…but for the purposes of this exercise, lets run for something more reasonable

./xmlchange --file env_run.xml --id STOP_N --val 5
./xmlchange --file env_run.xml --id REST_N –val $STOP_N

Exercise 4d: Single point BGC_AD

Then we get to point to our new domain files. <again identical to ex 4c>

**you can change this to your own directory, or use mine*

```
./xmlchange --file env_run.xml --id ATM_DOMAIN_FILE --val domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
./xmlchange --file env_run.xml --id ATM_DOMAIN_PATH --val /glade/scratch/$USER/single_point
./xmlchange --file env_run.xml --id LND_DOMAIN_FILE --val domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
./xmlchange --file env_run.xml --id LND_DOMAIN_PATH --val /glade/scratch/$USER/single_point
```

Then we point to our new surface dataset we just generated in user_nl_clm

#Then copy the lines below to the end of the file

```
echo "fsurdat = '/glade/scratch/wwieder/single_point/surfdata_0.9x1.25_16pfts_CMIP6_simyr1850_287.8_42.5_c170706.nc'" >> user_nl_clm
echo "hist_mfilt = 20" >> user_nl_clm
echo "hist_nhtfrq = -8760" >> user_nl_clm
```

I'm also adding this to user_nl_clm for AD simulations to reduce variables written out to .h0. files.

```
echo "hist_empty_htapes = .true." >> user_nl_clm
echo "hist_fincl1 = 'TOTECOSYSC', 'TOTECOSYSN', 'TOTSOMC', 'TOTSOMN', 'TOTVEGC', 'TOTVEGN', 'TLAI', 'GPP', 'CPOOL', 'NPP', 'TWS',
'H2OSNO'" >> user_nl_clm
```

 # And add this to user_nl_datm

```
echo "mapalgo = 'nn','nn','nn','nn','nn'" >> user_nl_datm
```

Copy the datm.streams files we used in ex. 4c into this directory.

```
 cp ../Ha1_CLM50sp_001/user_datm.streams.txt.* .
```

Exercise 4d: Single point BGC_AD

Now you should be able to build and submit!

[Notice how much longer the build takes now…]

(1) create a new case

For single point runs we need to run with a stub ice and river model, which is easiest to define here.
The rest of the compset doesn't really matter that much, we'll change it later. Type this command

```
./create_newcase --compset 2000_DATM%GSWP3v1_CLM50%BGC_SICE_SOCN_SROF_SGLC_SWAV --res f09_g17 --case /glade/u/home/$USER/clm_tutorial_cases/Ha1_CLM50bgc_001 --run-unsupported --project UCGD0004
```

(2) invoke case.setup

Then, navigate to your case directory:

```
cd ~/$USER/clm_tutorial_cases/Ha1_CLM50bgc_001
```

Type this command line:

```
./case.setup
```

Start here

(3) build the executable

Type this command line:

```
qcmd -q R4231266 -- ./case.build
```

(4) submit your run to the batch queue

Type this command line:

```
./case.submit
```

Use your Jupyter Lab to
Check spinup on AD simulations

Exercise 4d: Single point BGC_postAD

Check to see if the model looks spun up. Then you will Running after 'accelerated decomposition' (postAD) mode

You have a couple options here.

- *For global runs I'd recommend creating a clone of your AD case, prestaging the restart files, and running with AD mode off.*
- *In single pint runs I tend to just do this in the same directory, but change the starting dates so we don't write over our AD restart files. If you do this, just keep track of your files & dates!*

We'll do this just using xml changes and modification to user_nl_clm

```
./xmlchange --file env_run.xml --id CLM_FORCE_COLDSTART --val off
./xmlchange --file env_run.xml --id STOP_N --val 200
./xmlchange --file env_run.xml --id CONTINUE_RUN --val FALSE
./xmlchange --file env_run.xml --id CLM_ACCELERATED_SPINUP --val off
./xmlchange --file env_run.xml --id RUN_REFDATE --val 0401-01-01
./xmlchange --file env_run.xml --id RUN_STARTDATE --val 0401-01-01
```

Then add this to user_nl_clm

```
echo "finidat = '/glade/scratch/wwieder/archive/Ha1_CLM50bgc_001/rest/0401-01-01-
00000/Ha1_CLM50bgc_001.clm2.r.0401-01-01-00000.nc'" >> user_nl_clm
echo "finidat = '/glade/scratch/wwieder/archive/Ha1_CLM50bgc_001/rest/0601-01-
01-00000/Ha1_CLM50bgc_001.clm2.r.0601-01-01-00000.nc'" >> user_nl_clm
```

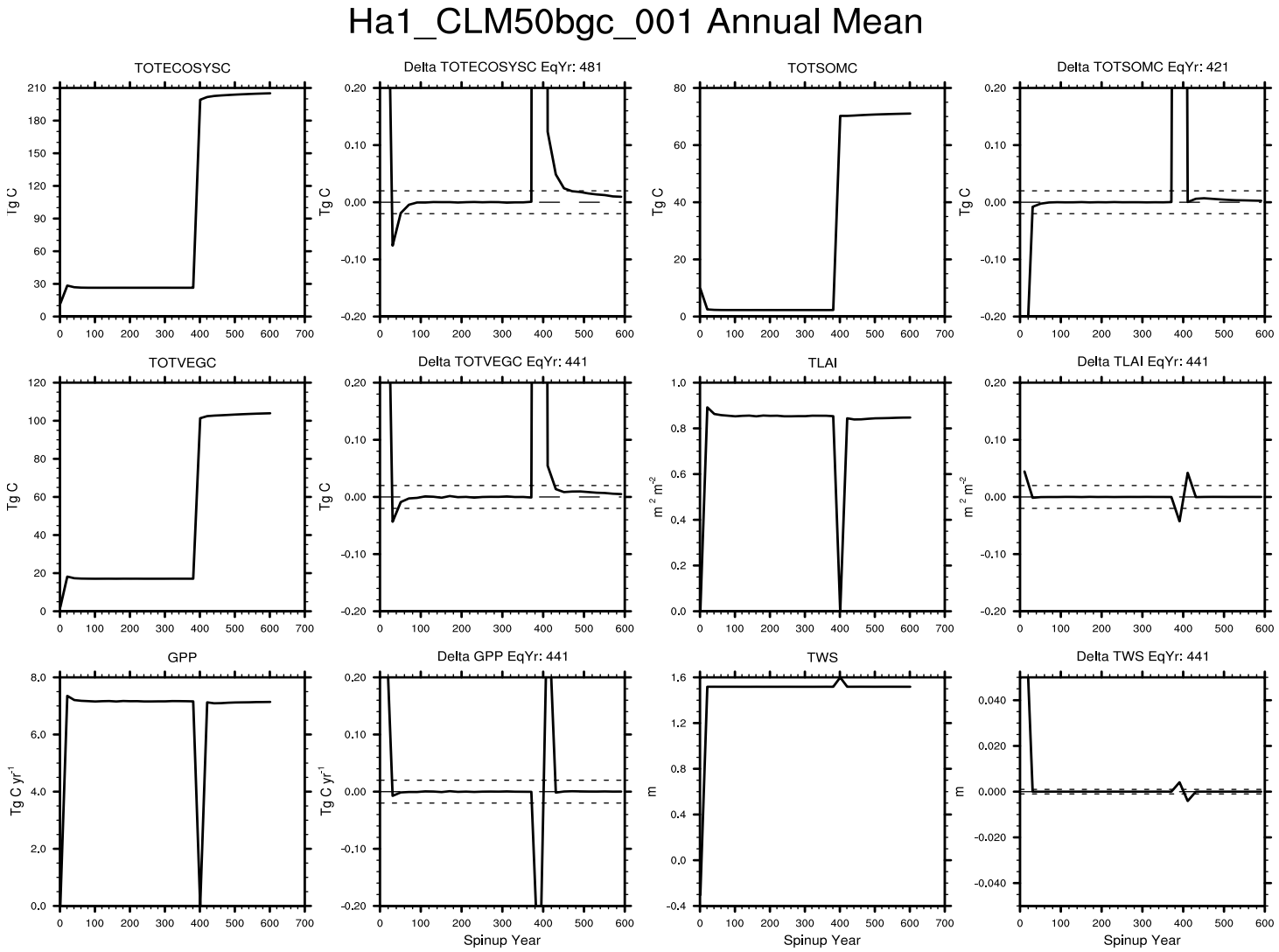
And submit!

```
./case.submit
```

Use your Jupyter Lab to
Check spinup on postAD simulations

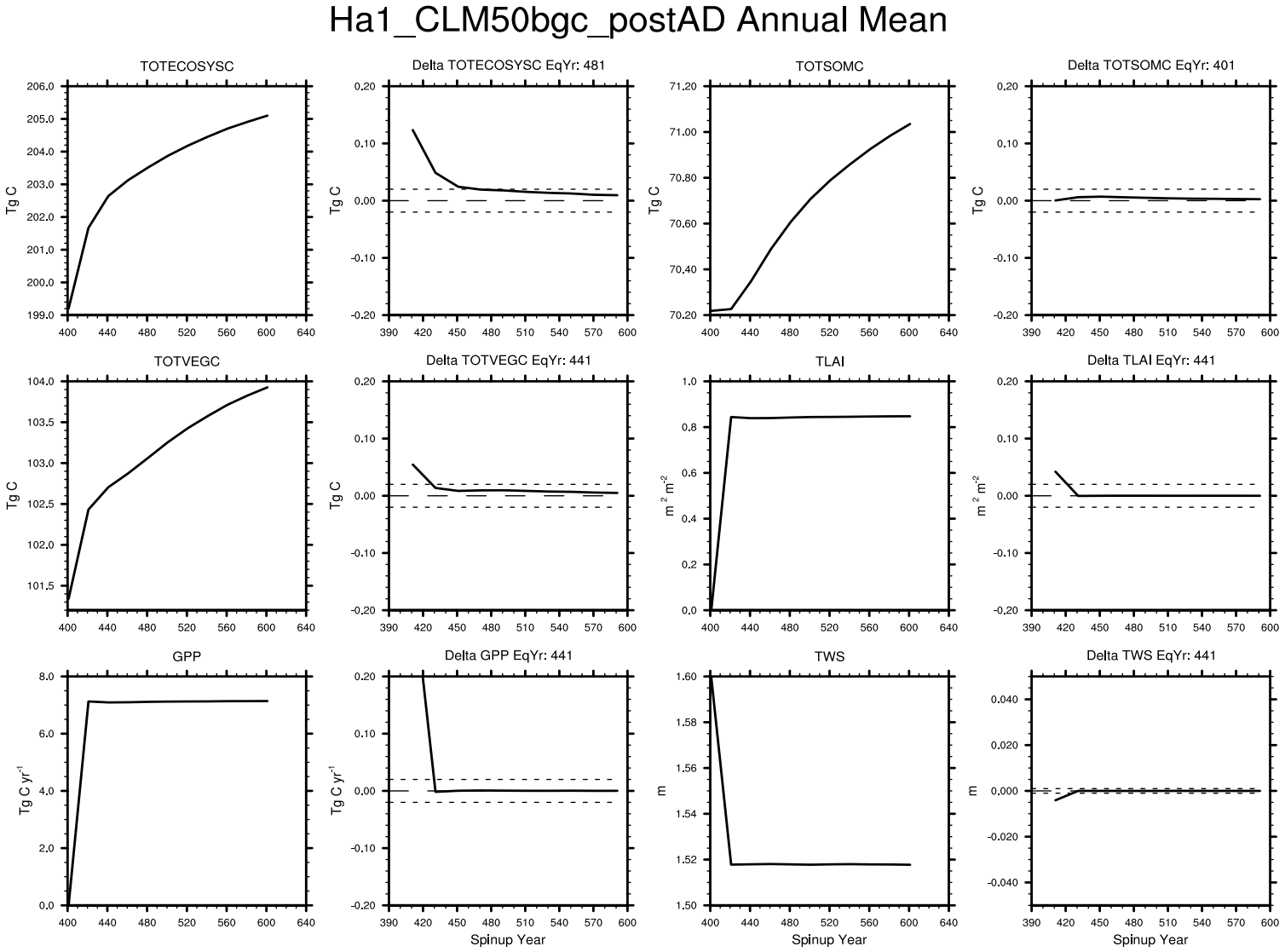
There's also a tool called SpinupStability.ncl that can be found here: /clm5.0_2019tutorial/tools/contrib. Below are results from the AD & postAD runs

What is causing the jump in stocks around year 400?



. Below are results from the postAD runs

- Thresholds and “equilibrium” can be modified for you needs!
- If you know anything about Harvard Forest, do these simulations seem reasonable?



Exercise 4d: Single point BGC_historical

Check to see if the model looks spun up.

If thing look OK we can run historical simulations

This is more for reference than anything

You have a couple options here.

- *For this example we'll create a new case for the historical simulation, but is isn't necessary*
- *Examples can be found at the bottom of this file /glade/u/home/wwieder/clm_tutorial_cases/Ha1_CLM50bgc_001/README.xml_modsBGC.txt*

I'm going to move through these steps pretty quick, as you've done them before

```
./create_newcase --compset HIST_DATM%GSWP3v1_CLM50%BGC_SICE_SOCN_SROF_SGLC_SWAV --res
f09_g17 --case ~/clm_tutorial_cases/Ha1_CLM50bgc_002 --run-unsupported --project UCGD0004
```

Compare the env_run.xml files from your 001 and 002 cases

The full listing of xml changes you'll need follow, but can also be found /gpfs/u/home/wwieder/clm_tutorial_cases/Ha1_CLM50bgc_001/README.xml_modsBGC.txt

Exercise 4d: Single point BGC_historical

Since we created a new case we'll have to go back ands format pes layout and point the right domain files

```
./xmlchange --file env_mach_pes.xml --id COST_PES --val 36
./xmlchange --file env_mach_pes.xml --id TOTALPES --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS --val 1
./xmlchange --file env_mach_pes.xml --id NTASKS_PER_INST --val 1
./xmlchange --file env_mach_pes.xml --id ROOTPE --val 0
./xmlchange --file env_batch.xml --id JOB_QUEUE --val share
./xmlchange --file env_batch.xml --id JOB_WALLCLOCK_TIME --val 6:00:00
./xmlchange MPILIB=mpi-serial
./xmlchange --file env_run.xml --id LND_DOMAIN_PATH --val /glade/scratch/$USER/single_point
./xmlchange --file env_run.xml --id LND_DOMAIN_FILE –val
domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
./xmlchange --file env_run.xml --id ATM_DOMAIN_PATH --val /glade/scratch/$USER/single_point
./xmlchange --file env_run.xml --id ATM_DOMAIN_FILE --val
domain.lnd.fv0.9x1.25_gx1v7_287.8_42.5.151020.nc
```

Exercise 4d: Single point BGC_historical

Then changes for a 20th century transient (some of these may not be necessary)

```
./xmlchange CCSM_BGC=CO2A
./xmlchange DATM_PRESAERO=trans_1850-2000
./xmlchange DATM_CO2_TSERIES=20tr
./xmlchange CLM_NML_USE_CASE=20thC_transient
./xmlchange CLM_CO2_TYPE=diagnostic


# -- changes to run from 1850-1901 --
./xmlchange DATM_CLMNCEP_YR_ALIGN=1850
./xmlchange RUN_TYPE=startup
./xmlchange CONTINUE_RUN=FALSE
./xmlchange DATM_CLMNCEP_YR_ALIGN=1850
./xmlchange CLM_FORCE_COLDSTART=off
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_START --val 1901
./xmlchange --file env_run.xml --id DATM_CLMNCEP_YR_END --val 1920
./xmlchange --file env_run.xml --id STOP_OPTION --val nyears
./xmlchange --file env_run.xml --id STOP_N --val 51
./xmlchange --file env_run.xml --id RUN_REFDATE --val 1850-01-01
./xmlchange --file env_run.xml --id RUN_STARTDATE --val 1850-01-01
./xmlchange --file env_run.xml --id CLM_ACCELERATED_SPINUP --val off
./xmlchange --file env_run.xml --id CLM_FORCE_COLDSTART --val off
```

Copy user_datm.streams files from your previous case

```
cp ../Ha1_CLM50sp_001/user_datm.streams.txt.* .
```

Exercise 4d: Single point BGC_historical

Finally make the name list changes & submit

```
echo "mapalgo = 'nn','nn','nn','nn','nn'" >> user_nl_datm
echo "finidat = '/glade/scratch/wwieder/archive/Ha1_CLM50bgc_001/rest/0601-01-01-
00000/Ha1_CLM50bgc_001.clm2.r.0601-01-01-00000.nc'" >> user_nl_clm
echo "fsurdat =
'/glade/scratch/$USER/single_point/surfdata_0.9x1.25_16pfts_CMIP6_simyr1850_287.8_42.5_c170
706.nc'" >> user_nl_clm
```

We're going to turn off transient land use too, because our single point time series has multiple pfts!

```
echo "do_transient_crops= .false." >> user_nl_clm
echo "do_transient_pfts = .false." >> user_nl_clm
echo "do_harvest = .false." >> user_nl_clm
echo "use_init_interp = .true." >> user_nl_clm

qcmd -q R4231266 -- ./case.build
./case.submit
```

Exercise 4d: Single point BGC_historical

We won't have time for this in the practical, but the following is for your reference:

```
#------------------------------------
```

 # after you get to 1901 you get to make the following changes

 # this could be done for longer with single point, code below just copied from global simulations

/tools/contrib/run_clm_historical

```
#------------------------------------

./xmlchange STOP_OPTION=nyears
./xmlchange STOP_N=22
./xmlchange DATM_CLMNCEP_YR_ALIGN=1901
./xmlchange DATM_CLMNCEP_YR_START=1901
./xmlchange DATM_CLMNCEP_YR_END=2014
./xmlchange CONTINUE_RUN=TRUE
./xmlchange RESUBMIT=3
```

remove the .bin files from your scratch directory

then copy or move user_datm.streams files that have the full transient time series

```
cp /gpfs/u/home/wwieder/clm_tutorial_cases/Ha1_CLM50bgc_002/user_datm* .
```

Exercise 4d: Single point BGC_historical

We won't have time for this in the practical, but the following is for your reference:

#-----------------------------------

 # after you get to 1989 make the xml changes below

 # this could be a good time to change the format of history file output

#-----------------------------------

```
./xmlchange RUN_TYPE=branch
./xmlchange RUN_REFCASE={$CASENAME}
./xmlchange RUN_REFDATE=1989-01-01
./xmlchange STOP_OPTION=nyears
./xmlchange STOP_N=26
./xmlchange CONTINUE_RUN=FALSE
./xmlchange RESUBMIT=0
```

Finally look at the output!
We don’t have any examples in Jupyter Lab, but h0 files are here

/gpfs/fs1/scratch/wwieder/archive/Ha1_CLM50bgc_002/lnd/hist

For a quick look I used SpinupStability.ncl that can be found here: /clm5.0_2019tutorial/tools/contrib. Below are results from the historical runs

Given changes to C stocks over the 20th century, how much does does the equilibrium matter?