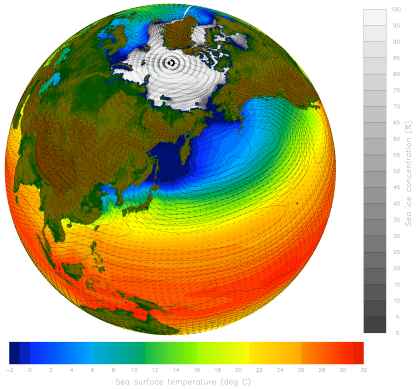
Getting up and running with CESM

Cécile Hannay
Climate and Global Dynamics (CGD), NCAR

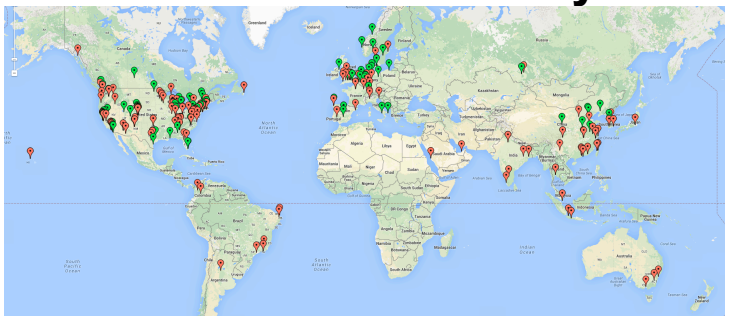
NCAR is sponsored by the National Science Foundation

Why CESM ?

State of the Art Climate Model



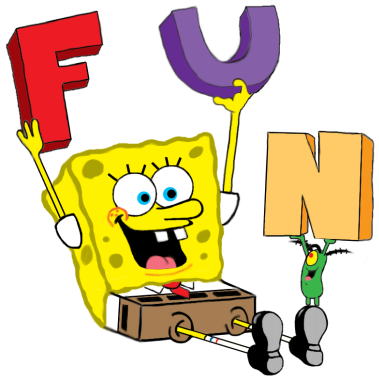
Widely used by the Climate Community



Over 5000 downloads around the world

Well supported

It's fun



Outline

- CESM webpage
- One-Time Setup
- How to set and run an experiment
- Getting More Help

CESM 1.2 Webpage

http://www.cesm.ucar.edu/models/cesm1.2/

Release Notes

Scientific validation

Guidance on model versions

Post processing Tools

Model Documentation

CESM Models

Home » CESM Models » CESM1.2 Series Public Release

CESM1.2 SERIES PUBLIC RELEASE

ABOUT THIS RELEASE SERIES

The CESM1.2 release has numerous new key features among which are the addition of CLM4.5, new science changes to CAM5 running with the CAM-SE dynamical core, and new scripting infrastructure for the generation of component sets, grids and model testing.

CESM1.2 SERIES RELEASE NOTES

Please read the CESM1.2 Series Release Notes which includes What's New - Science, What's New - Software, Answer-Changing Features, Supported Machines, and Known Problems. The new scripting infrastructure is described in detail in the CESM1.2 User's Guide.

SCIENTIFIC VALIDATION

Scientific validation consists of a multi-decadal model run of the given component set at the target resolution, followed by scientific review of the model output diagnostics. All scientifically supported component sets are also accompanied by diagnostic and model output data. Validated CESM1.2 model results and diagnostics will be added to the CESM1.2 website as they become available.

What version of the model should I use?

For a scientifically supported target component set and resolution, please refer to the Scientifically Validated Configurations for that target configuration. For component sets and resolutions that are not scientifically validated in any supported release (e.g. cesm1.0.5 and cesm1.1.1), CSEG strongly urges you to use the latest model release (in this case cesm1.2.0).

DIAGNOSTIC PACKAGES AND NAMING CONVENTIONS

- Post Processing Utilities
- Model File Naming Conventions
- Experiment Case Naming Conventions

MODEL DOCUMENTATION

Background and Sponsors

CESM PROJECT

The Community Earth System Model (CESM) is a fully-coupled, global climate model that provides state-of-the-art computer simulations of the Earth's past, present, and future climate states.

CESM is sponsored by the National Science Foundation (NSF) and the U.S. Department of Energy (DOE). Administration of the CESM is maintained by the Climate and Global Dynamics Division (CGD) at the National Center for Atmospheric Research (NCAR).

How to acquire the code

MODEL SOURCE CODE

Copyright and Terms of Use

All CESM source code is subject to the following Copyright Notice and Disclaimer.

Acquiring the Release Code

The source code for CESM releases is distributed through a public Subversion code repository. This code can be checked out using Subversion client software, such as the command tool svn, or simply view the latest version with a web browser.

A short registration is required to access the repository. After registering, you will receive an email containing a user name and password that is necessary to gain access to the repository.

Acquistion of the code is more fully described in the most recent version of the CESM1.2 User's Guide.

Reporting problems Getting Help

REPORTING A PROBLEM

If you have any problems, please first read the User's Guide including the sections on FAQs and Use Cases. Please also refer to the CESM Bulletin Board, which is in place to facilitate communication within the CESM community. Finally, please also refer to the Release Notes entries that are provided with every release and release update. If questions or problems still exist, then please send an email to cesm-help@cgd.ucar.edu. Support questions will be answered as resources are available.

CESM SUPPORT POLICY

CESM Support Policy - November 2012

CESM 1.2 Webpage

http://www.cesm.ucar.edu/models/cesm1.2/

A short registration is required to access the repository. After registering, you will receive an email containing a user name and password that is necessary to gain access to the repository.

Acquistion of the code is more fully described in the most recent version of the CESM1.2 User's Guide.

REPORTING A PROBLEM

If you have any problems, please first read the User's Guide including the sections on FAQs and Use Cases. Please also refer to the CESM Bulletin Board, which is in place to facilitate communication within the CESM community. Finally, please also refer to the Release Notes entries that are provided with every release and release update. If questions or problems still exist, then please send an email to cesm-help@cgd.ucar.edu. Support questions will be answered as resources are available.

CESM SUPPORT POLICY

CESM Support Policy - November 2012

CESM DATA MANAGEMENT & DISTRIBUTION PLAN

The Community Earth System Model (CESM) Data Management and Data Distribution Plan documents the procedures for the storage and distribution of data associated with the CESM project.

Data management and distribution ←

Model Input data →

Timing and load balance →

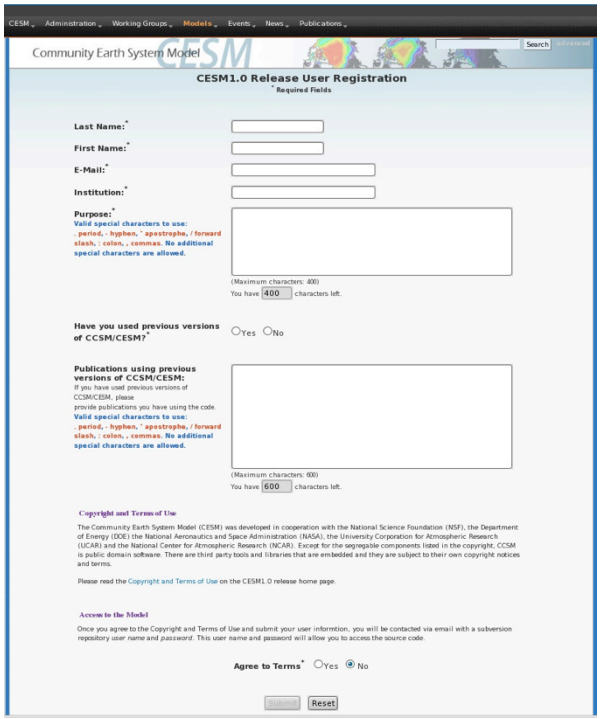
Outline

- CESM webpage
- One-Time Setup
  - Registration
  - Download Source Code
  - Create an Input Data Root Directory
  - Porting
- How to set and run an experiment
- Getting More Help

Registration

- **Go to CESM1.2 home page: http://www.cesm.ucar.edu/models/cesm1.2/**

- **Right hand column has a link to the registration page, click on it**

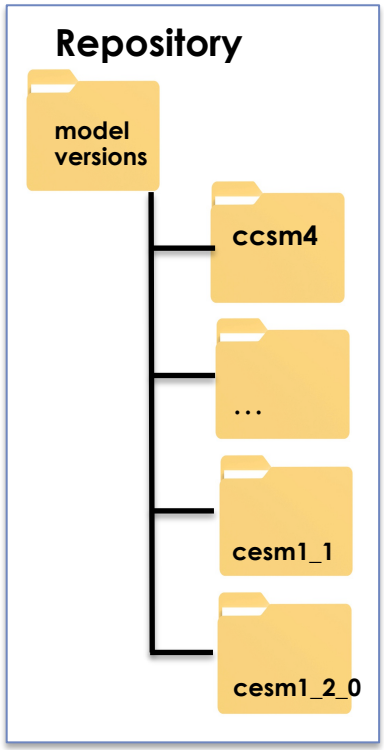


- **Register -- you will be emailed a username and password**

Outline

- CESM webpage
- Software & Hardware Requirements
- One-Time Setup
  - Registration
  - Download Source Code
  - Create an Input Data Root Directory
  - Porting
- How to set and run an experiment
- Getting More Help

Download the Source Code



• **Code and input datasets are in a subversion repository (*)**
https://svn-ccsm-release.cgd.ucar.edu/model_versions

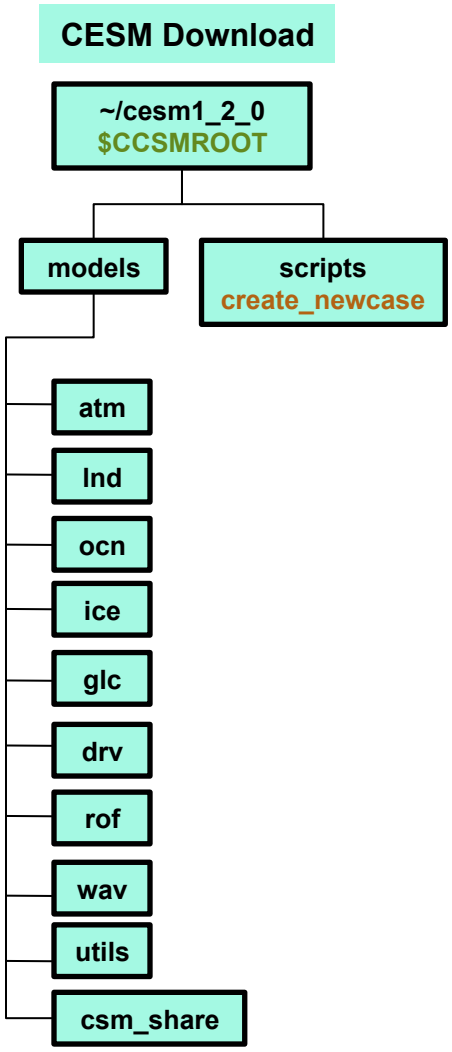
• **List the versions available on the CESM repository**
svn list https://svn-ccsm-release.cgd.ucar.edu/model_versions

• **Check out a working copy from the repository ("Download code")**
svn co https://svn-ccsm-release.cgd.ucar.edu/model_versions/cesm1_2_0

(*) You can get subversion at http://subversion.apache.org/

open source

Overview of Directories (after initial model download)



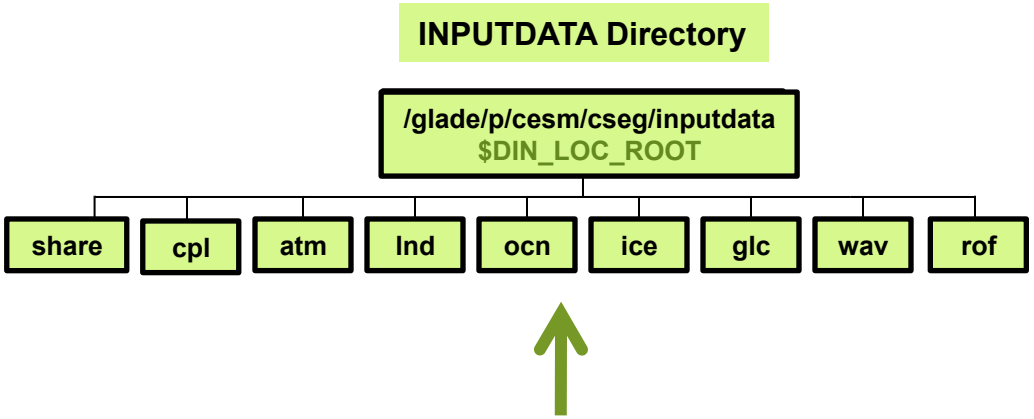
The CESM root consists of 2 directories: models and scripts

Code for all the components, coupler, etc…

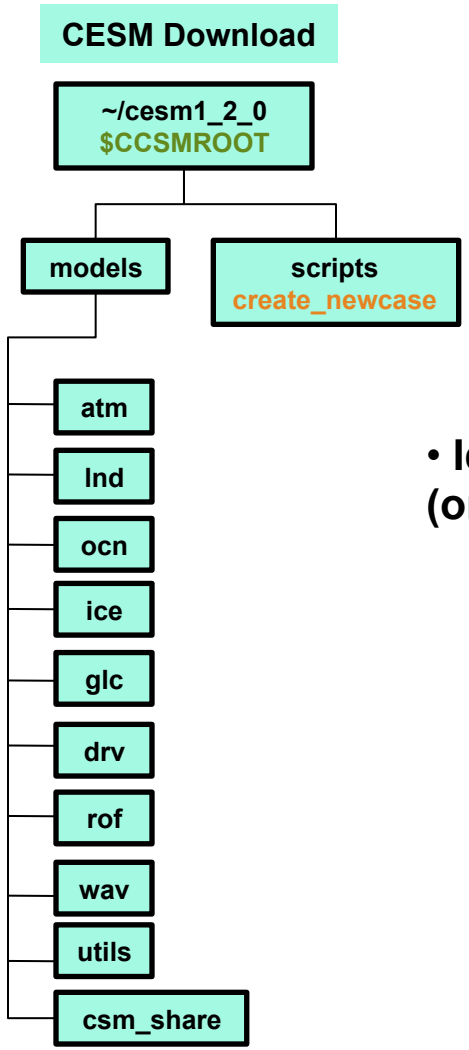
Outline

- CESM webpage
- Software & Hardware Requirements
- One-Time Setup
  - Registration
  - Download Source Code
  - Create an Input Data Root Directory
  - Porting
- How to set and run an experiment
- Getting More Help

Overview of Directories (+ inputdata directory)



Inputdata directory $DIN_LOC_ROOT
contains all input data required to run the model



- Ideally directory is shared by a group of users to save disc space (on supported machines - populated inputdata already exists)

Outline

- CESM webpage
- Software & Hardware Requirements
- One-Time Setup
  - Registration
  - Download Source Code
  - Create an Input Data Root Directory
  - Porting
- How to set and run an experiment
- Getting More Help

Porting

- On supported machines - no porting is necessary
- On other machines – porting needs to be done

Porting details are outside the scope of this tutorial

More info about porting:

1. User's Guide

Porting and Validating CESM on a new platform

2. Porting tutorial

http://www.cesm.ucar.edu/events/tutorials/081114/porting-edwards.pdf

Outline

- CESM webpage
- Software & Hardware Requirements
- One-Time Setup
  - Registration
  - Download Source Code
  - Create an Input Data Root Directory
  - Porting
- How to set and run an experiment
- Getting More Help

Congrats !
You are ready to set and run an experiment

Outline

- CESM webpage
- Software & Hardware Requirements
- One-Time Setup
- How to set and run an experiment
- Getting More Help

Work Flow: Super Quick Start

CESM can be run with a set of 4 commands

Set of commands to build and run the model on a supported machine: ”cheyenne”

```
# go into scripts directory into the source code download
cd /path_to_source_code_download/cesm1_2_0/scripts

(1) # (1) create a new case in the directory “cases” in your home directory
./create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne

# go into the case you just created in the last step
cd ~/cases/case01/

(2) # (2) invoke cesm_setup
./cesm_setup

(3) # (3) Build the executable
./case01.build

(4) # (4) submit your run to the batch queue
./case01.submit
```

It is that easy !

Overview of Directories (+ before create_newcase)

INPUTDATA Directory

/glade/p/cesm/cseg/inputdata
$DIN_LOC_ROOT

share | cpl | atm | lnd | ocn | ice | glc | wav | rof

CESM Download

~/cesm1_2_0
$CCSMROOT

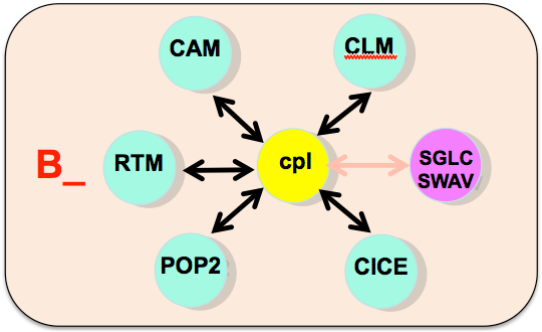
- models
  - atm
  - lnd
  - ocn
  - ice
  - glc
  - drv
  - rof
  - wav
  - utils
  - csm_share
- scripts
  - create_newcase

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

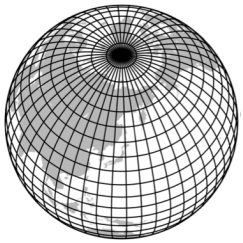
create_newcase requires 4 arguments

What is the casename ?

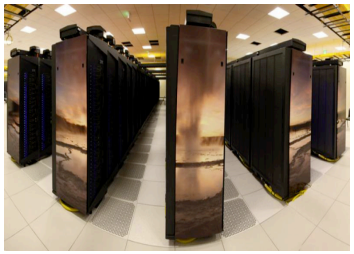
Which model configuration ? Which set of components ?



Which resolution?



Which machine are you running on?



Syntax of create_newcase

create_newcase requires 4 arguments

create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19 -mach cheyenne

Syntax of create_newcase

create_newcase requires 4 arguments

create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19 -mach cheyenne

What is the casename ?

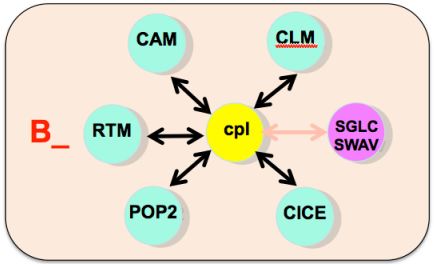
case specify the name and location of the case being created
~/cases/case01

Syntax of create_newcase

create_newcase requires 4 arguments

create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19 -mach cheyenne

Which component set ?



compset specifies the "component set"

Component set specifies component models, forcing scenarios and physics options for those models

Examples:

FC5 = Active atmosphere and Land with prescribed SSTs and sea-ice.

B1850 = All active components (atm, land, ocean, sea-ice)

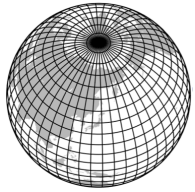
For more information: see the CESM webpage

Syntax of create_newcase

create_newcase requires 4 arguments

create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19 -mach cheyenne

Which resolution?



res specifies the model resolutions (or grid)

Example
f19_f19 (atm/lnd_ocn/ice) => finite volume at about 2 degree resolution

Syntax of create_newcase

create_newcase requires 4 arguments

```
create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne
```

Which machine are you running on?

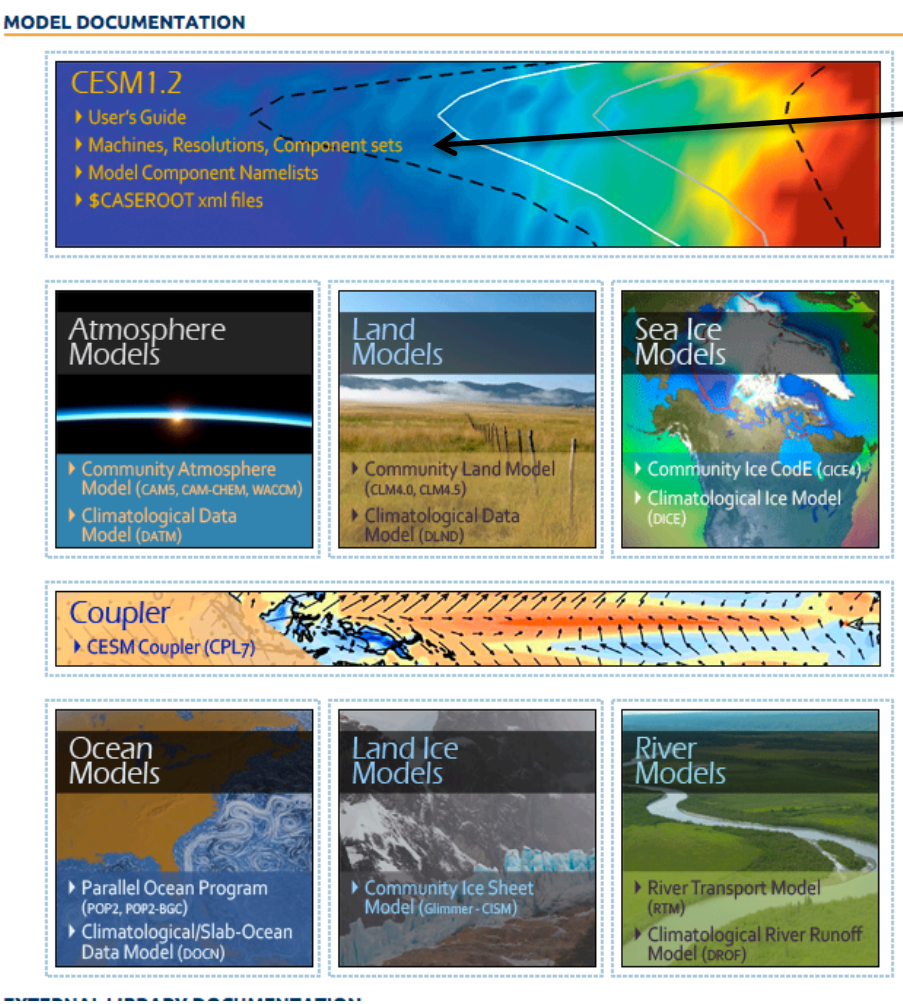
mach specifies the machine that will be used.

Valid Values for res, compset, and mach

Command line to list all the valid choices for grids, compsets and machines

./create_newcase -list <type>

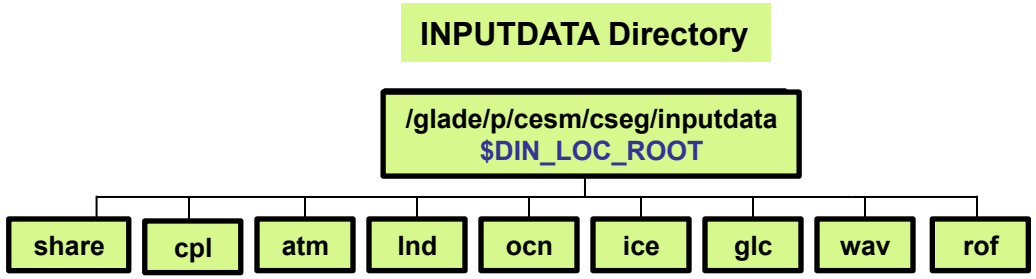
with type can be [compsets, grids, machines]

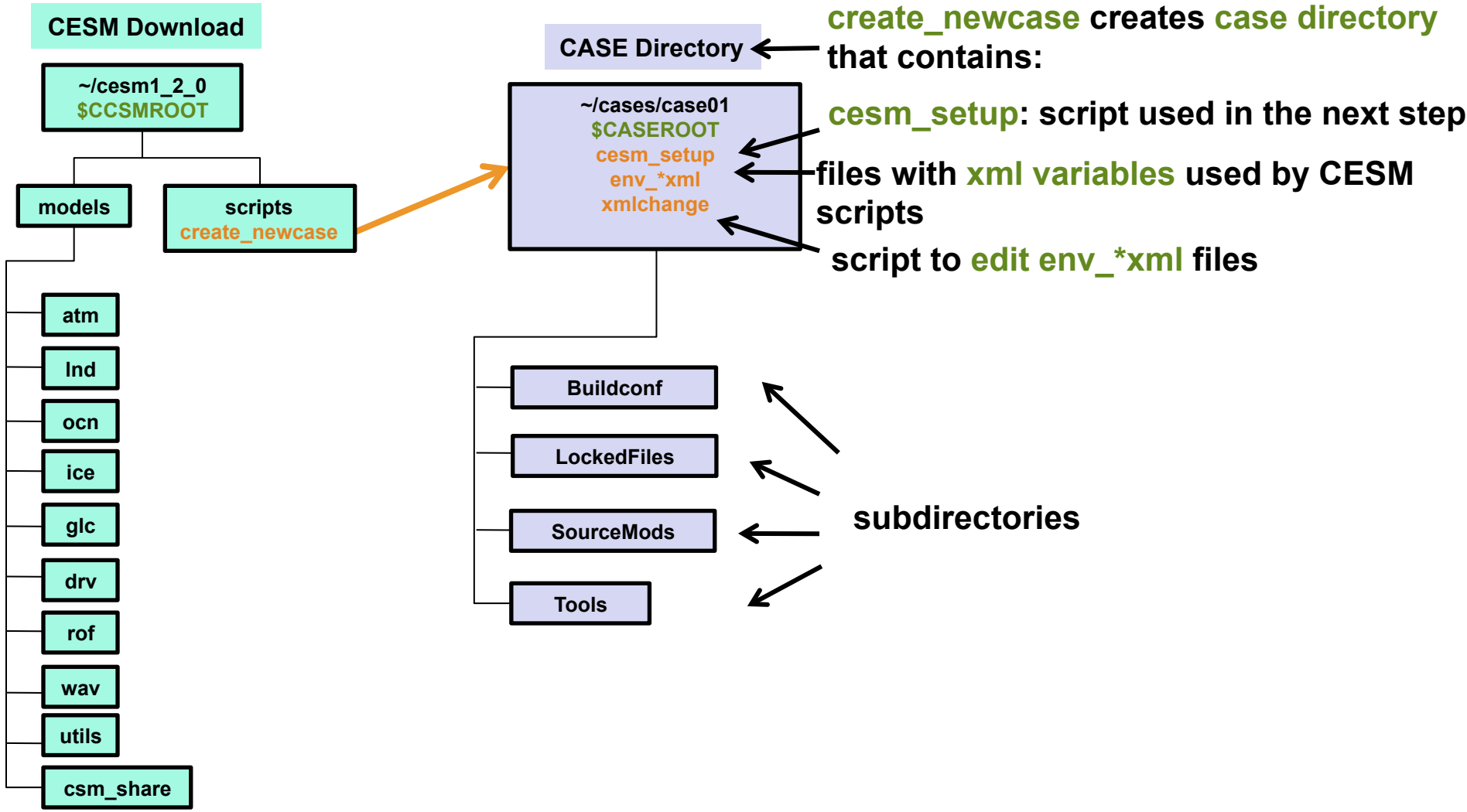


List of valid values is also available from the CESM website

http://www.cesm.ucar.edu/models/cesm1.2/

Overview of Directories (after create_newcase)





About env_*.xml files

- **env_*.xml contains variables used by scripts**
  - **env_case.xml: set by create_newcase and cannot be modified**
  - **env_mach_pes.xml : specifies layout of components**
  - **env_build.xml: specifies build information**
  - **env_run.xml : sets run time information (such as length of run, frequency of restarts, …)**

User interacts with this file most frequently

- **Here's a snippet of the env_run.xml file**

```
<!--"sets the run length in conjunction with STOP_N and STOP_DATE, valid values: none,never,nst
eps,nstep,nseconds,nsecond,nminutes,nminute,nhours,nhour,ndays,nday,nmonths,nmonth,nyears,nyea
r,date,ifdays0,end (char) " -->
<entry id="STOP_OPTION"   value="ndays"  />

<!--"sets the run length in conjunction with STOP_OPTION and STOP_DATE (integer) " -->
<entry id="STOP_N"   value="5"  />
```

CESM will run for 5 days

"id" - variable name

"value" – variable value

- **To modify a variable in an xml file – use xmlchange**

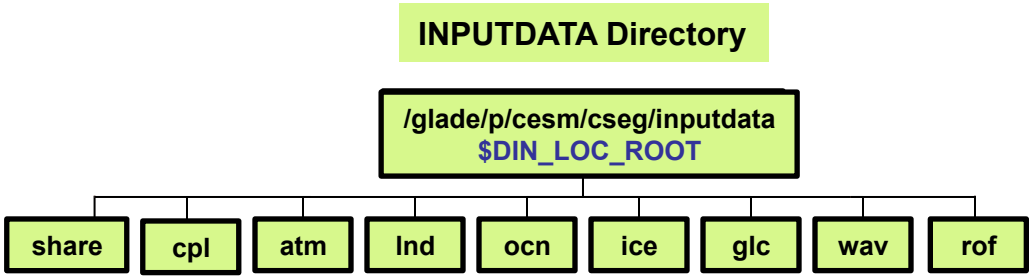
  xmlchange STOP_N=20

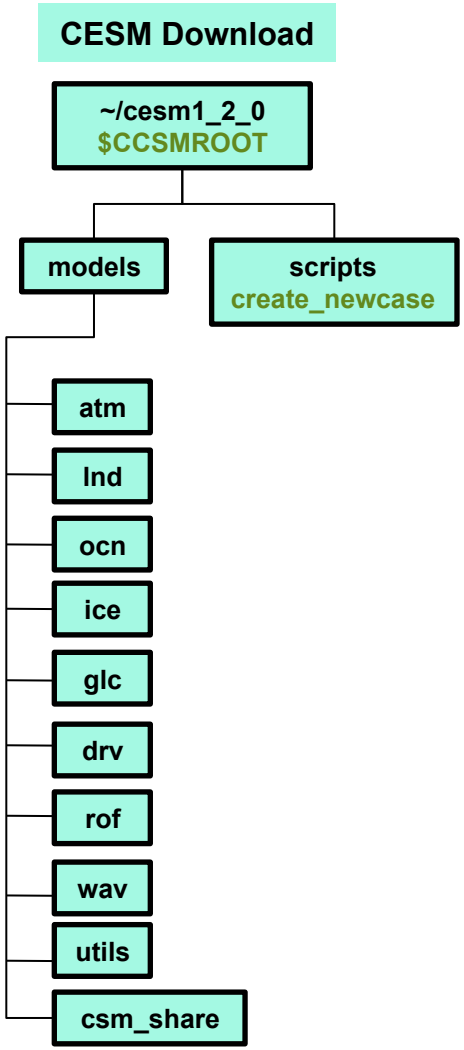
Work Flow: Super Quick Start

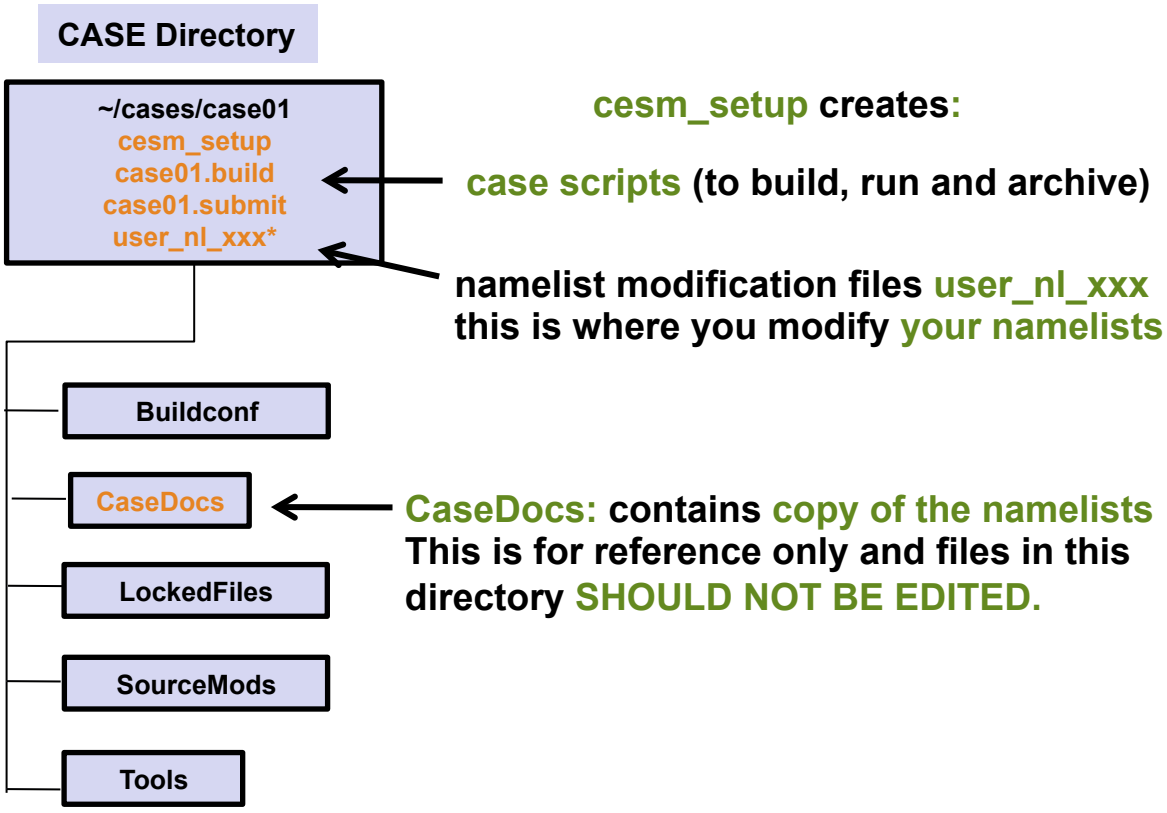
Set of commands to build and run the model on a supported machine: ”cheyenne”

```
# go into scripts directory into the source code download
cd /path_to_source_code_download/cesm1_2_0/scripts

# (1) create a new case in the directory “cases” in your home directory
./create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne

# go into the case you just created in the last step
cd ~/cases/case01/

# (2) invoke cesm_setup
./cesm_setup

# (3) Build the executable
./case01.build

# (4) submit your run to the batch queue
./case01.submit
```

Overview of Directories (after cesm_setup)





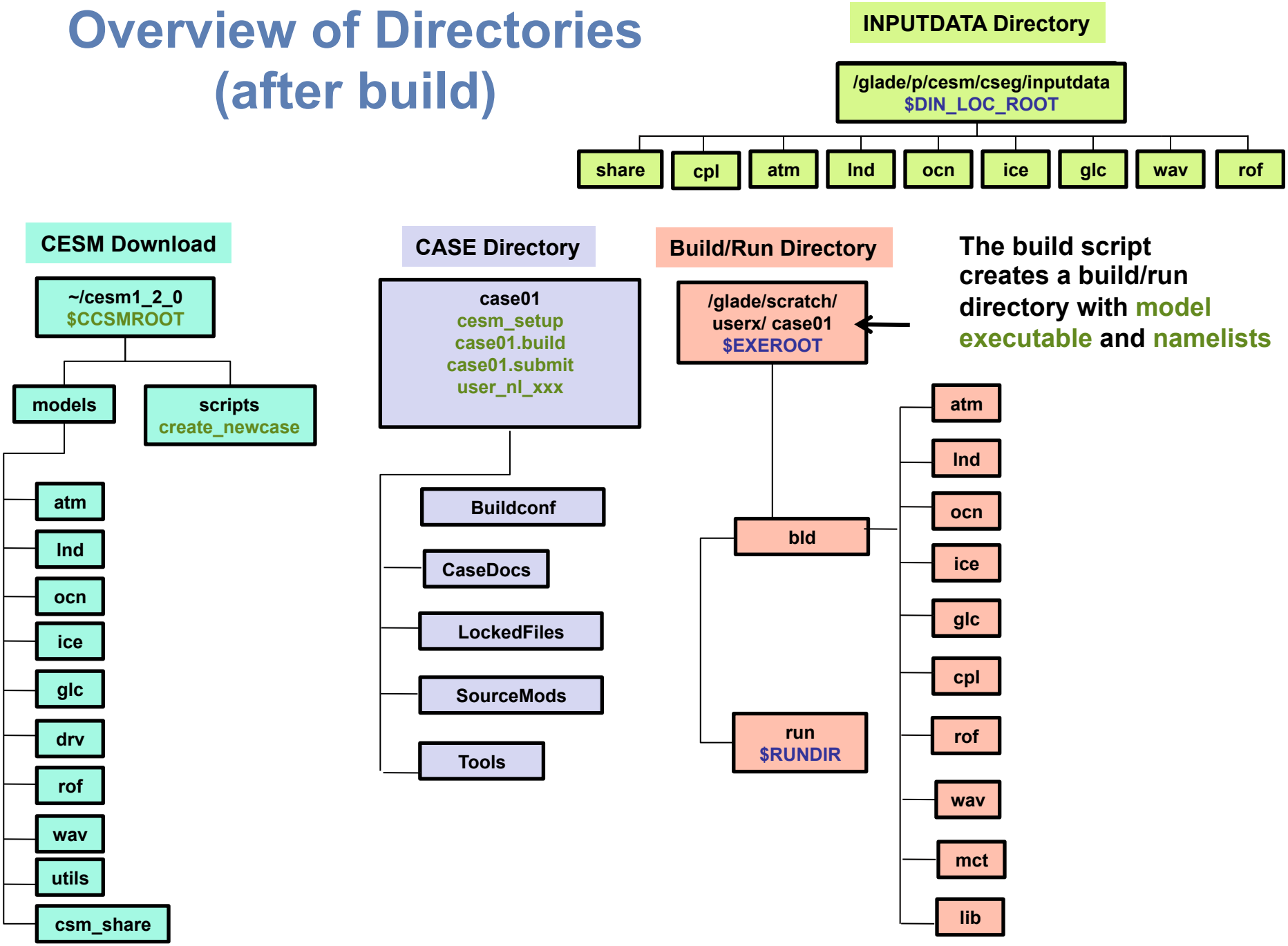


Work Flow: Super Quick Start

Set of commands to build and run the model on a supported machine: ”cheyenne”

```
# go into scripts directory into the source code download
cd /path_to_source_code_download/cesm1_2_0/scripts

# (1) create a new case in the directory “cases” in your home directory
./create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne

# go into the case you just created in the last step
cd ~/cases/case01/

# (2) invoke cesm_setup
./cesm_setup

# (3) Build the executable
./case01.build

# (4) submit your run to the batch queue
./case01.submit
```

Overview of Directories (after build)



Work Flow: Super Quick Start

Set of commands to build and run the model on a supported machine: ”cheyenne”

```
# go into scripts directory into the source code download
cd /path_to_source_code_download/cesm1_2_0/scripts

# (1) create a new case in the directory “cases” in your home directory
./create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne

# go into the case you just created in the last step
cd ~/cases/case01/

# (2) invoke cesm_setup
./cesm_setup

# (3) Build the executable
./case01.build

# (4) submit your run to the batch queue
./case01.submit
```

Running the Model

When you submit your jobs

```
cases/case01> case01.submit

check_case OK
Job <959733> is submitted to queue <regular>
```

Use "qstat –u $username" to check if job is running

```
cases/case01> qstat -u hannay
                                                                  Req'd  Req'd   Elap
Job ID          Username Queue    Jobname    SessID NDS TSK Memory Time  S Time
--------------- -------- -------- ---------- ------ --- --- ------ ----- - -----
1244289.chadmin hannay   regular  case01      10523   5 180    --  01:50 R 00:02
1244299.chadmin hannay   regular  case02        --    5 180    --  01:50 Q   --
```

Your job is running

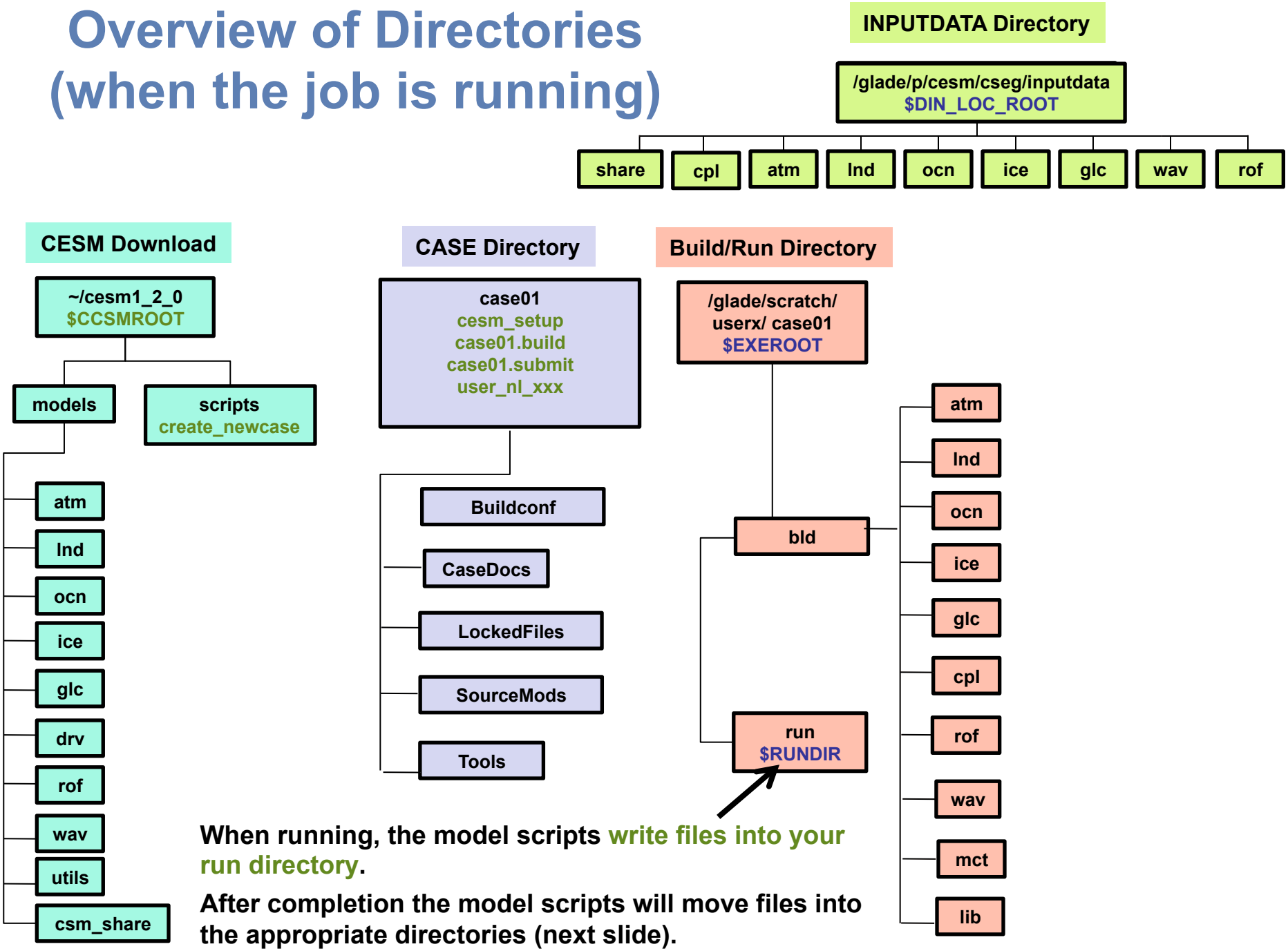
Your job is waiting in the queue

Use "qdel Job ID" to kill a job

```
                                                                  Req'd  Req'd   Elap
Job ID          Username Queue    Jobname    SessID NDS TSK Memory Time  S Time
--------------- -------- -------- ---------- ------ --- --- ------ ----- - -----
1244306.chadmin hannay   regular  case01      47644   5 180    --  01:50 R 00:00

cases/case01> qdel 1244306
```

Kill Job #1244306

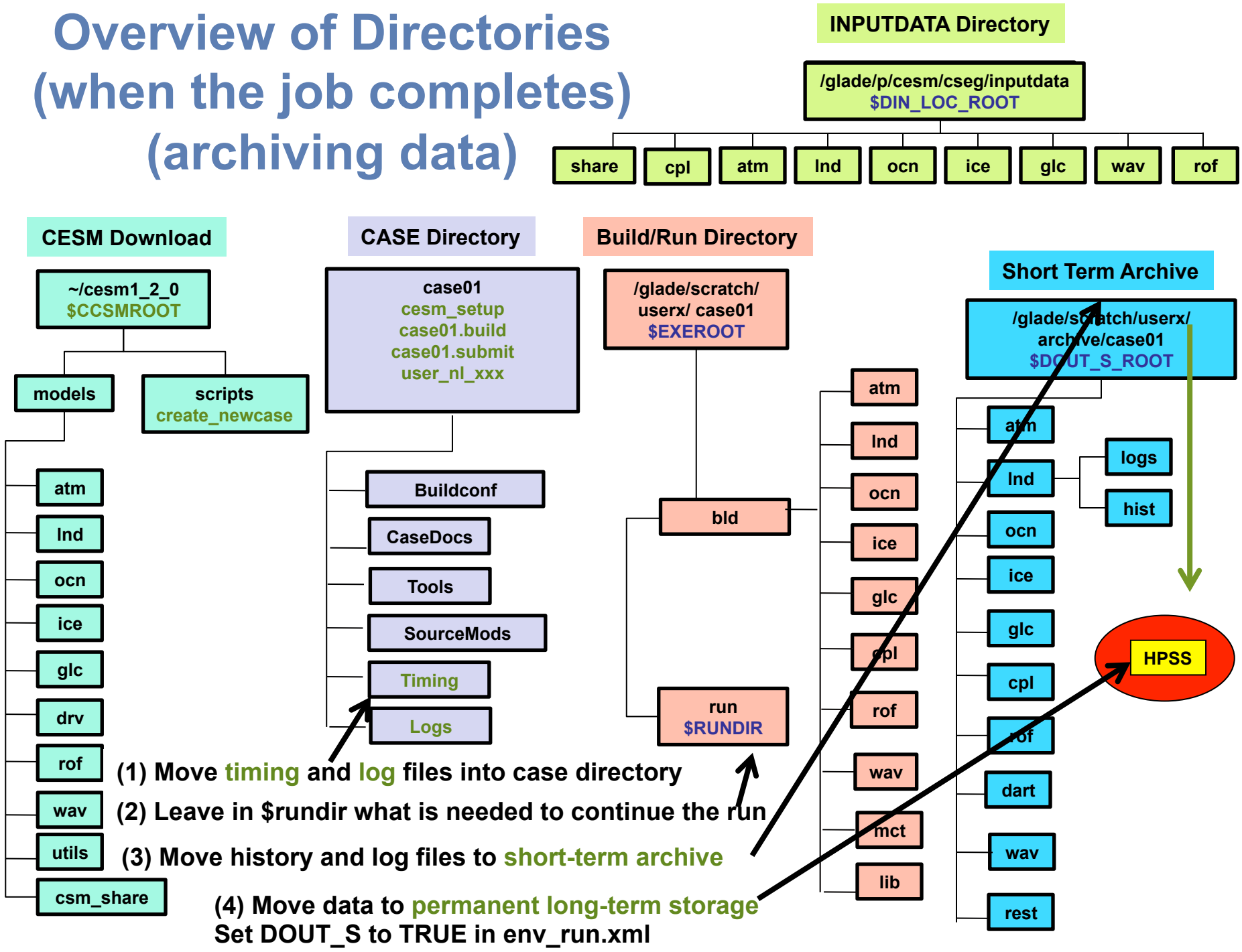
Overview of Directories (when the job is running)



When running, the model scripts write files into your run directory.

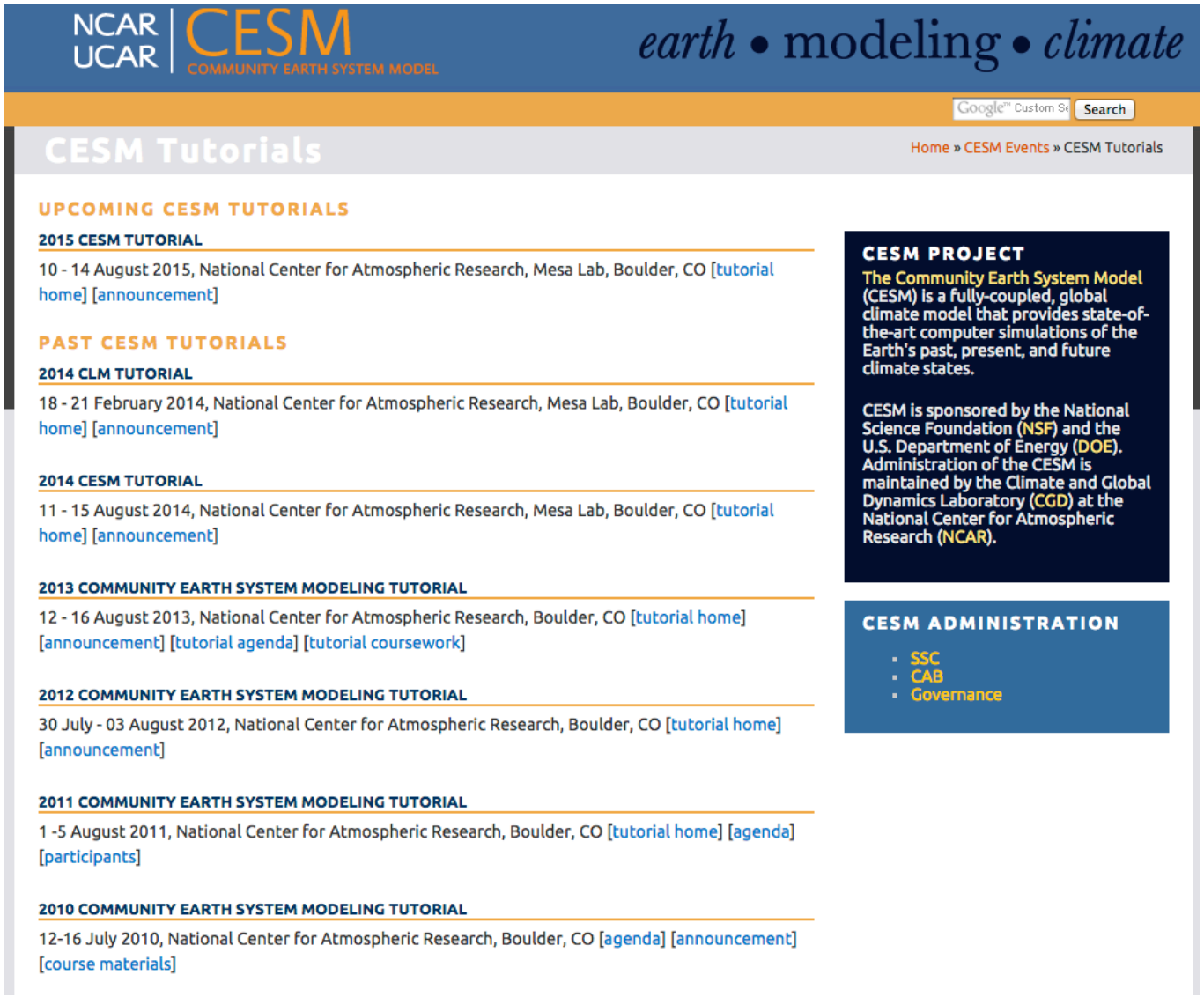
After completion the model scripts will move files into the appropriate directories (next slide).

Overview of Directories (when the job completes) (archiving data)



More Information/Getting Help

Online tutorial: http://www.cesm.ucar.edu/events/tutorials/

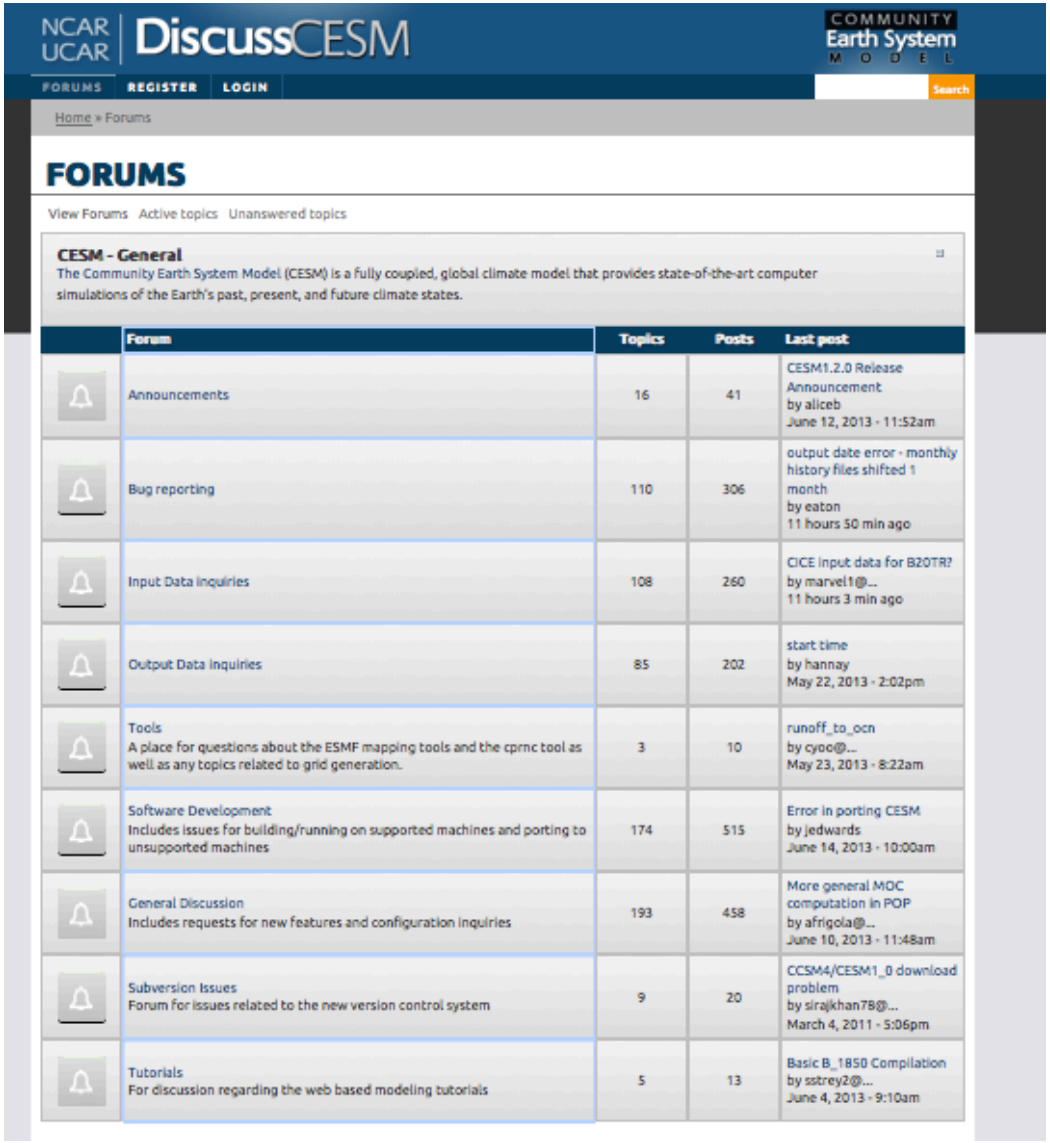


More Information/Getting Help

Model User Guides: **http://www.cesm.ucar.edu/models/cesm1.2/**

More Information/Getting Help

CESM Bulletin Board: http://bb.cgd.ucar.edu/



Super important message

CESM Work Flow Cheat Sheet for the Colloquium

Set of commands to build and run the model on cheyenne during this tutorial
This is what you will use when you do an CESM experiment this week
Please "bookmark" this slide

```
# go into scripts directory into the source code download
cd /glade/p/cgd/asp2017/CESM/cesm1_2_2_1/scripts

# (1) create a new case in the directory "cases" in your home directory
./create_newcase -case ~/cases/case01 -compset FC5 -res f19_f19  -mach cheyenne

# go into the case you just created in the last step
cd ~/cases/case01/

# (2) invoke cesm_setup
./cesm_setup

# (3) Build the executable on a compute node
 go_to_compute_node
./case01.build
exit

# (4) submit your run to the batch queue
./case01.submit
```

CAUTION

Your homework for today => Please do this!

- **One-time steps (do this FIRST)**

Make the directory ~/cases: mkdir ~/cases

Add to your .tcshrc the line: alias go_to_compute_node 'qsub -l select=1:ncpus=36:mpiprocs=1 -l inception=login -l walltime=01:00:00 -l -q regular -A UASP0001 '

- **Submit a 5-day run**

Use the instructions from "CESM Work Flow for the Colloquium"

- **Check it worked**

Call me or Rich to check with you everything worked fine