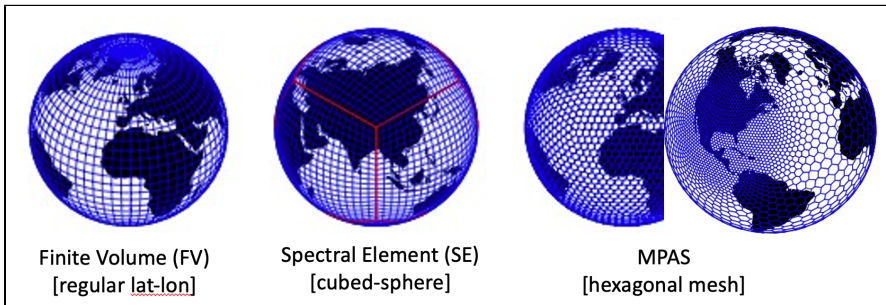


# MUSICA-V0 Prerequisites

## Background and Prerequisites

If you are not familiar with CESM and running CAM-chem, please go through the following material:

- Read the CESM Tutorial coursework and do the exercises: <http://www.cesm.ucar.edu/events/tutorials/2019/>
- Become familiar with CAM-chem by looking through the wiki page: <https://wiki.ucar.edu/display/camchem/Home>
- Read about the Spectral Element dynamical core: [https://ncar.github.io/CAM/doc/build/html/cam6\\_scientific\\_guide/dynamics.html?highlight=spectral#spectral-element-dynamical-core](https://ncar.github.io/CAM/doc/build/html/cam6_scientific_guide/dynamics.html?highlight=spectral#spectral-element-dynamical-core)
- See Peter Lauritzen's lecture on SE at the end of his CESM 2019 Tutorial lecture on Dynamics: <http://www.cesm.ucar.edu/events/tutorials/2019/files/Lecture2-lauritzen.pdf>



## Download MUSICA-V0 source code

MUSICA-V0 is a configuration of CESM available in CESM2.2. If you will be running MUSICA-V0 on your own computer, please check the hardware, OS and software requirements in the [CESM2 documentation](#).

Whether you are working on the NCAR supercomputer or another computer, first get your own copy of the source code (also see [CESM2 download instructions](#)):

```
> git clone https://github.com/ESCOMP/CESM.git {my_cesm_sandbox}
> cd {my_cesm_sandbox}
> git fetch origin
> git checkout cesm2.2-rcbranch
> ./manage_externals/checkout_externals
```

"my\_cesm\_sandbox" should be a directory in your home or work directory (not scratch, see [explanation of glade file spaces on cheyenne](#)), and could include the CESM version, etc., in the name. The steps above will ensure your code includes any bug corrections made since the initial release. This directory is referred to as \$CESM\_ROOT.

To test your code, create a newcase with standard resolution of CAM-chem:

```
> $CESM_ROOT/cime/scripts/create_newcase --case <your_path+$CASENAME> --res f09_f09_mg17 --compset FCnudged --run-unsupported
```

Replace "<your\_path+\$CASENAME>" with an existing path and new name for the case (which will be a new directory) without the brackets. Again, "your\_path" should be in home or work, not scratch. On cheyenne, by default the model is run from, and output written to, scratch. Now go to the new case directory that was created, and "setup" and "build" your executable:

```
> cd <your_path+$CASENAME>
> ./case.setup
> qcmd -- ./case.build
```

There is a mistake in the default settings for the met field nudging for the f09 resolution CAM-chem compset, so correct the number of model\_times\_per\_day namelist variable before running. Add to the file 'user\_nl\_cam':

```
model_times_per_day = 48
```

If there are other namelist variable values in ./CaseDocs/atm\_in you would like to modify, you can put them in 'user\_nl\_cam'.

Settings for the simulation start date and simulation length are in `env_run.xml`. The default is to start 2010-01-01 and run for 5 days. You can change `env_run` to have `STOP_OPTION = nmonths` and `STOP_N = 1`, to run 1 month and get the default monthly mean output file (written to `/glade/scratch/$USER/$CASENAME/run`). If you only run for 5 days you will not get a complete monthly average file (the partial average is written to `$CASENAME.cam.rh0.*`). You can add a second output file to save daily average, or 6-hourly output, of a few variables to check your results after 5 days. See the [CAM-chem wiki page](#) for more an explanation of how to do that. Check the `fincl1` list in your `atm_in` file to see possible variables to include in a `fincl2` list.

Now run the model:

```
./case.submit
```

Find more explanations and examples on the [CAM-chem wiki](#) and the [CESM instructions](#).