Configuring the simulations

IOconditions.txt (initfiles)
the climatic input
the run name
Restart file

combine_foam.inc (source)
path definition (line 4 and 24)
number of degassing peak (line 27)
starting of the degassing scenario tstart_deg (line 42)
degassing scenario (line 277-281)

cond_p20.dat (initfiles)
the background degassing rate
Trap or not
the run duration (line 22-23)
frequency of printing (line 24)
For steady-state simulations: accelerating the O2 (line 26)
For steady-state simulations: run at least 6 Myr (line 23) and check the age for generating a restart (line 25, close to 6 Myr)

Running the simulations

Restart

1. copy OUTPUT/output.*** into the restart directory
2. in initfiles, change accordingly line 14 in IOconditions.txt

Compiling

1. source directory
2. ./makefile (open it with Xcode if want to change executable name. GEOCLIM by default)
3. check the errors (call yves)
4. run the model: ./GEOCLIM (or new prescribed name by user)

Usual crashes

1. file or directory not found ! Check paths

In any case: read the message from the computer!

Managing the output

having a quick look at output during the simulation (just for checking purposes)

1. GEOCLIM is producing 2 netcdf files: geoclim and geographic (OUTPUT directory)
2. check numerical values:

ncdump -h name.nc : show the name of the variables (h is for header)
ncdump -v time,CO2_atm_level name.inc (v is for variables)

Final output

1. 2 netcdf files: geoclim and geographic (OUTPUT directory)
2. a lot of ascii files which will be used for plotting (OUTPUT directory) with python

graphical output

1. for netcdf: Panoply (can make maps, simple plots, latitudinal transects)
2. python script called geoclim_full.py (python directory): have to define the name of the runs line 13 (WITHOUT dot), you find this in IOconditions.txt file (initfiles) or in the names of the output files (OUTPUT)
3. run the python script: python geoclim_full.py (python directory in a terminal). Will generate pdf output in the python/figs/ directory.