Cascade

Overview:
Brief look at how to run Cascade, and export the output.

Updated 8/16/18 for icecascade_v1_80
Summary of files:

- Required input files for Cascade
  - Grids (*.dat)
    - Tab or space separated (space is best)
    - No header
    - In kilometers
    - [X Y Z Colour ID UID]
  - Topos (*.dat)
    - Tab or space separated (space is best)
    - No header
    - In kilometers
    - [X Y Z ID]
  - Velocity (*.txt)
    - 4 header lines followed by ages, grid and topo files
  - icecascade.in
    - Edited to reflect your model timing, atmospheric and climate parameters, (and more; see below)
  - Run_icecascade.sbatch
    - Doesn't need to be edited (other than to put your email address on it the first time), unless you want to run on a specific node
    - Can change model name so that on terminal it shows
  - Icecascade (executable)

1) Move Setup
2) Exporting the Point Cloud
See Pecube_Setup-Plotting for how to setup and export grid, topo, and lines.

Note: Cascade doesn't allow selection of where in the cross section to analyze, so the area of interest must start at 0, run in the positive direction, to the width of your model (defined in icecascade.in inputs D1 and D2).

3) Model Setup
velocity file: list the ages and corresponding grid for n+1 deformation steps (all def steps+starting); include 10 5 10 1.0 at beginning. (same as pecube)

```
10
5
10
1.0
```

```
age_start      model_grid_step0.dat
.
.
.
0       model_grid_steplast.dat
```

For best practice, allow 0.2-0.5My of initialization time for river channels to form. Do this in the velocity.txt file by adding 0.2-0.5 to age_start, and using the same grids twice. EG:

```
10
5
10
1.0
age_start+0.2  model_grid_step0.dat
age_start      model_grid_step0.dat
age_start      model_grid_step1.dat
.
.
.
0       model_grid_steplast.dat
```

Note: **DO NOT COPY AND PASTE FROM ANY MICROSOFT PROGRAMS INTO THE TEXT EDITOR.** It makes pecube/cascade angry. Pretty much any text editor works, some of use XCode, Smultron, Mac's TextEdit, Atom (Windows).
Note: the first four lines correspond to: (1) the y dimension in km, (2) # nodes in y-direction, (3) interpolation window for averaging over the MOVE grid, and (4) x step-size for interpolation (of topo).

The icecascade.in file
This is where you enter the select thermal/climate parameters, and more. n=#def steps.
*= change per model; ** Orographic precipitation
y is the distance along the cross-section (our normal x)

Model Processes: (fluvial erosion & deposition, hillslope diffusion, landslide & glacial erosion, orographic precipitation)
*** **(input A9):** T for orographic precipitation, F for constant/global precipitation

Model Output: (screen output, model output, etc)
(input B2): How often the model writes into the screen output file (how precise do you want your error tracking to be? *Default is 10000*)
(input B4): how frequent to export topo/landscape [yrs] (if model runs for 5Ma, and you export every 50000yrs, then you get 100 files) (for more complete videos, 50k-*100k* screen output is best)

(input B5): name of this model run, also name of output folder (eg *IceCascade* shown in the figure in section 5)

Time Domain: (dynamic time, model run time)
(input C1): dynamic time; *best practice == T.*
(input C2): 25-125yrs; *best practice == 25*
***(input C3)**: total model time in years. Should be first age in velocity.txt file

Spatial Domain: (Model Size, # nodes, etc) (read Paul's paper on legacy topography and see FAQ for info on model size)
***(input D1)**: number of nodes along edges of model ➔ (model width [km] *desired Cascade resolution [nodes/km]) + 1 (x – perpendicular to section, y – along section) (start at *1km resolution*, and increase once your model confidence is higher)
***(input D2)**: length of model in D1 in km
***(input D4)**: input an initial topography ➔ *T*
***(input D5)**: name of initial topography file *.dat
(input D6): 3

Fluvial erosion: (fluvial/bedrock erosion coefficients & constants)
(input E1): an (inverse) proxy for rock strength; default is *3.5e-4*. Range of permissible values is approximately 3.5e-3:3.5e-5. For stronger rocks, use smaller numbers (eg 8.5e-5 is half an order of magnitude stronger). For weaker rocks, use larger numbers (eg 8.5e-4 is half an order of magnitude weaker)

Hillslope erosion: (hillslope/landslides erosion coefficients & constants) [we don't change anything normally]
(input F1): k_{diff} [km^2/yr]: *2e-6*

Ice: (glaciers, etc)
(inputs G-L)
Tectonics: (rock properties, isostasy, etc)
(input M1): *3*
***(input M1b):** velocity.txt file (which contains ages, move grids, etc)

Climate: (control [orographic] precipitation, climate, etc)
(input N1): keep on order with structures, plotting interval, or 25000
(input N2): precipitation grid size in km (might crash if <1km)
****(input N3):** "Precipitation rate for uniform rainfall model [m/yr]" not actually straight rainfall, affected by MAT and climate parameters, only works with orographic precip on
* ****(input N6):** atmospheric moisture; change a0, leave a1 alone
(input N8): wind speed
(input N9a): wind direction (doesn't differentiate between –y and y)[0-180 permissible] (?)
(input N9b): F to do nothing; T to flip the model 180 on the x-axis (to make orographic precipitation flow from the correct side)

(input N10/N11): wind smoothing scale
****(input N13):** atmospheric lapse rate (only works with atmospheric precip on (?))
(input N14): annual variation in daily temperature C
***(input N18)**: min and max temperature at base of the model

Temperature: (control climate, for all runs)
(input N01): F
***(input N05):** MAT

The run_icecascade.sbatch file

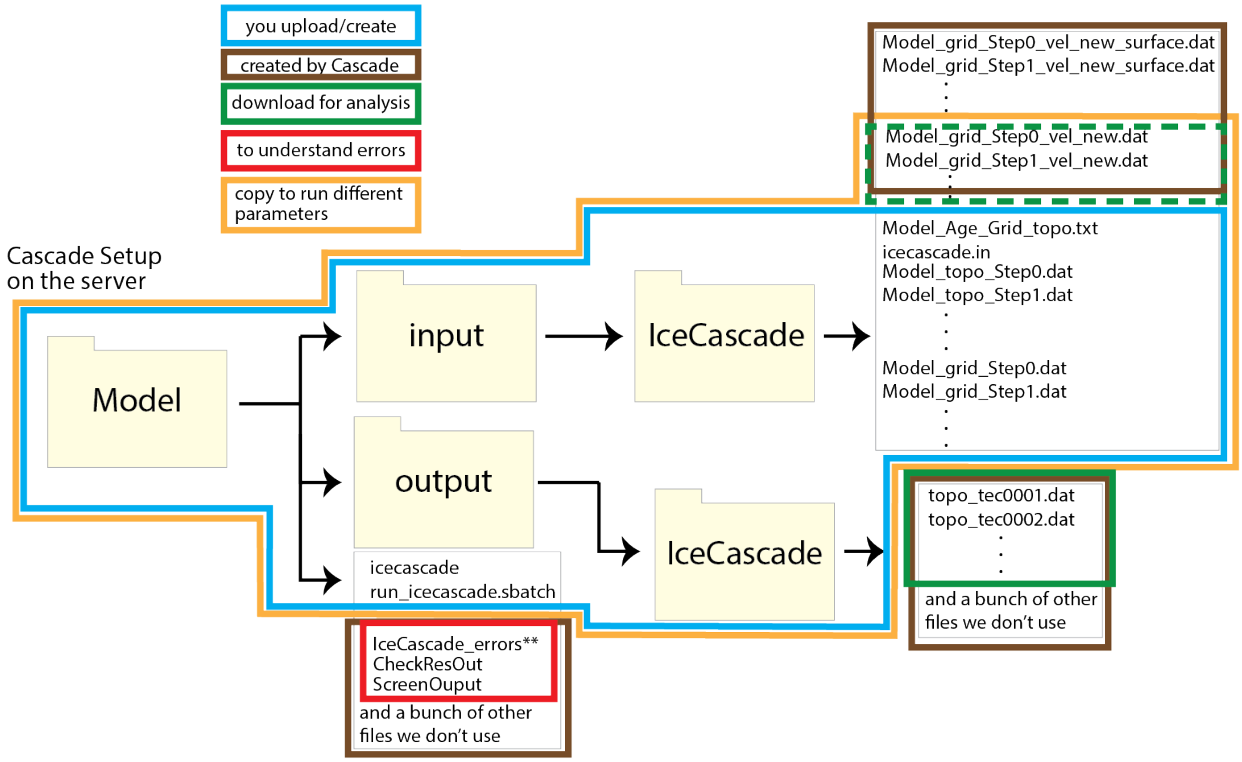
```
#!/bin/bash -l
## Run script for icecascade on esd slurm


## General configuration options
#SBATCH -J MODEL-NAME
#SBATCH -e IceCascade_Errors%j
#SBATCH -o IceCascade_ScreenOutput%j
#SBATCH --mail-user=EMAIL@pitt.edu
#SBATCH --mail-type=ALL


## Machine and CPU configuration
## Number of tasks per job:
#SBATCH -n 1
## Number of nodes:
#SBATCH -N 1
#SBATCH -w u-005-s040

#srun --resv-ports icecascade
srun ./icecascade
```

Note: if you want to run your model on a less busy node, check which nodes are busy using `squeue`*.   (See section 6 for more details on how to implement Command Prompt commands.) Include this in the .sbatch file in file: the highlighted area:*

#SBATCH –w u-005-s038 (to run on #38)
#SBATCH –w u-005-s039 (to run on #39)
#SBATCH –w u-005-s040 (to run on #40)

4) Setting up SFTP connections using Fetch (or another ftp program; WinSCP for windows)
Follow the instructions in Pecube_Setup-Plotting

5) Upload your model according to the folder structure:



6) Running your model

a) Using Terminal (Mac) or Command Prompt (Windows; or Putty) use
`ssh –l username –X –p 6307 134.2.5.40`
to logon to the server (replace `username` with yours). Then type in your password. It won't be visible.

b) Use `cd /esd/esd01/data/username/` to switch to your folder, and then navigate to your specific model (eg `cd /model_runs/Model`).

c) Then, type in `sbatch run_icecascade.sbatch` (for cascade). It should confirm the job and give you a number. Type in `chmod 777 *` to give yourself permission to run the job. Sometimes you have to run sbatch and chmode twice to give full permission.

d) Check to make sure it's still running with `squeue` OR `squeue –u username`. If it computes the velocity files (found in the input file), you're most likely good (this takes 1-4 hrs).

Terminal commands

`ssh –l username –X –p 6307 134.2.5.40`	log on to the server
`squeue`	find out what's running
`squeue –u username`	what are you running
`cd ..`	go up a directory
`cd /esd/esd01/data/username/`	change directory to yours
`sbatch run_icecascade.sbatch`	run a cascade job
`chmod 777 *`	give permission to run a job
`scancel job#`	cancel a job
`ls`	display contents of folder
`scp –r source/folder destination/folder`	copy a folder (source) and its contents to destination
`exit`	log off

Note: if you want to run your model on a less busy node, check which nodes are busy using `squeue`*. Include this in the run_pecube.sbatch file:*

#SBATCH –w u-005-s038 (to run on #38)
#SBATCH –w u-005-s039 (to run on #39)
#SBATCH –w u-005-s040 (to run on #40)

7a) Analyzing your Cascade Model output

Cascade exports in topotec format; designed to be plotted with tecplot (an expensive program). You can extract the files using MATLAB since they are in ASCII format, and plot/analyze from there). Use Cascade_plot.m or Cascade_plot_mult(…etc).m to extract and plot general elevation and precipitation over the model space. For Ksn extraction, you'll need TopoToolbox by Schwanghart and Scherler (there is a an app you can add to matlab, or you can download it as a .m and put it in the home folder).

8) FAQ:

What are the "required" parameters for setup?

C3 (model start time)
D1 (model size in km)
D2 (model size in # nodes)
D5 (name of initial topo file)
M1b (name of velocity txt file)
For your first run, do a run with dynamic time off **C1** and time step **C2** 100-200, orographic **A9** off; check to make sure you have uplift, surface velocities, the precipitation is approx what you want, etc.

How big should my model space be? How long should my model run for?

In theory, you need a ~5-10km buffer on each end, the desired model view space at t=now, and (if you're not using dynamic remeshing) the amount of shortening.
However, as we learn more about the effects of legacy topography, it seems you should start (if possible) from a state where the surface is mostly flat (eg while the entire model view is foreland basin. In the initialization time, you may want to create a "false" topography with a slope of 0.1-0.5° (or from 0km to say, 500 or 800m over the model space) to allow CASCADE to create river channels. These will get overprinted by the true landscape (see Paul's paper).
The model space should ideally cover all topography you care about: so, if at t=now you want to see 100km of landscape, and you have 50km of shortening, the model space needs to be *at least* 150km. The model space *definitely* needs to include the "important" locations of uplift: so, a basal ramp, if you have one, the location of imposed uplift, etc.

Note: CASCADE only models from 0 to however big you say the model is. If your deformation front isn't at (or close to 0), you may need to shift the locations.
Andes run time/model space (VBuford, model McQ02N9v1-10): 50km x 200km, node spacing ~1km, ~16My run time

What should I change if I change the velocity?

C3 (model start time)
M1b (name of velocity txt file)

How do I fine tune the model?

Increase node spacing **D1** up to ~0.5km; make sure dynamic time **C1** is T and C2=25; turn on adaptive remeshing **D3**; change precipitation node size (caution, may crash) **N2**; increase frequency of model output **B4**.

What controls erosion?

Erosion is controlled by: **velocity** (so time for erosion to happen); **uplift** (ramps/loading & imposed); **precipitation** (discussed above); **fluvial and hillslope erosion** (so, E1-5, F1)

What does "error sill" mean?

This is a super generic error that basically means CASCADE couldn't find a river channel or catchment. This can be due to: (i) no initialization time, (ii) too strong of rocks, (iii) too fast velocity, (iv) not enough precipitation, (v) random error/bad luck/something we haven't figured out yet.

What parameters should I vary?

Constant Precipitation (A9=F): constant precipitation amount is affected by parameters **N6** (atmospheric moisture a0) and **N18** (min and max sea level temperatures) (and to a small degree **E1**). It is not affected by changing N3 (uniform precip rate), N8 (wind speed), N13 (atmospheric lapse rate) or NN05 (mean annual temperature.

The figure(s) below shows the effects of various parameters on the output precipitation. These were run using "default" values (with the exception of N3. N3 default =3.0, oops, but it doesn't affect the results). The following two figures show how N6 and N18 can be varied to affect precipitation; it primarily seems that N6 should be for fine-tuning the amount, while N18 can control larger swings.

Effect of Cascade Parameters on Precipitation
For "Global" Precipitation (A9=F)

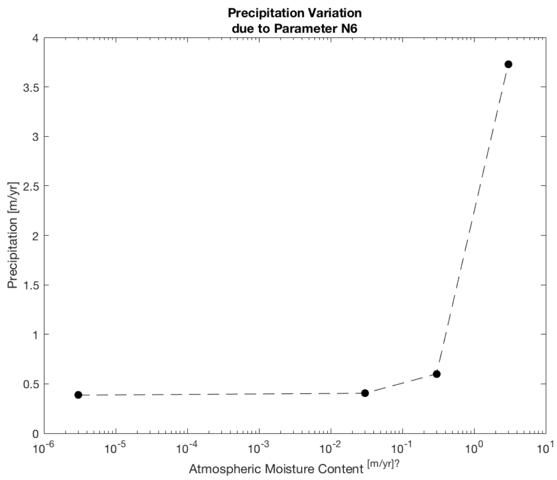
N3- Uniform Precip Rate			N8-Wind Speed			N13-Atm. lapse	
1.0	4.0	10	1.00	1.25	1.50	0.0065	5.3
0.6006	0.6006	0.6006	0.6006	0.6006	0.6006	0.6006	0.6006

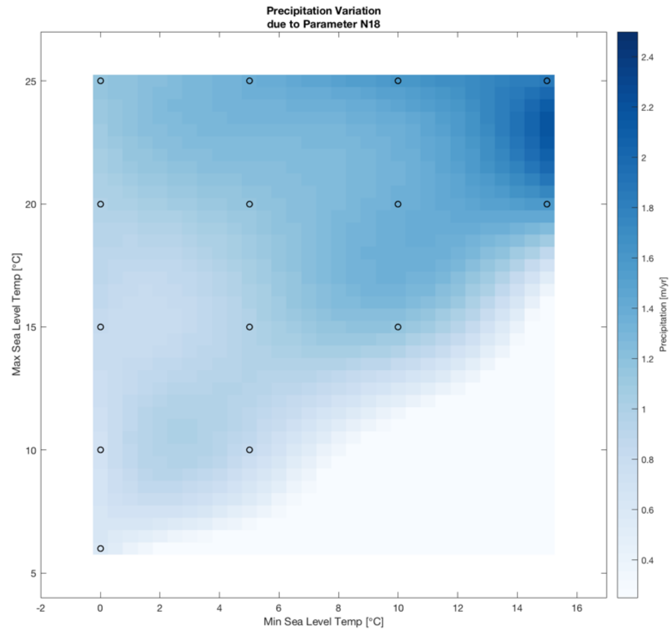
NN05- Mean Annual Temperature				
7.9	10	15	20	25
0.6006	0.6006	0.6006	0.6006	0.6006

default
has effect

N6-Atmospheric Moisture			
3E-6	.03	.3	3
0.3858	0.4047	0.6006	3.7297

N18-Sea Level Temps					
		Min			
		0	5	10	15
max	6	0.6006			
	10	0.6924	0.8209		
	15	0.8244	0.9741	1.1472	
	20	0.9782	1.1519	1.3524	1.5828
	25	1.1567	1.3578	1.5891	1.8544





Orographic Precipitation (A9=T): In theory, affected by: **N3** (uniform precip rate), **N6** (atmospheric moisture), **N8** (wind speed), **N13** (atmospheric lapse rate), **N18** (Min and max sea level temperatures), **NN05** (Mean Annual Temperature) (Maybe also N14, N9, N10, N11, N12)