Creating velocity input file for Pecube:

1- Create 2-D move undeformed cross section file (not map view) of all important stratigraphic layers and faults (particularly identifying location of ramps). Make sure that model extends 40-60 km left of the final section and 60-90 km right of the final section (avoid edge effects in thermal model), and that there is a line that represents sea level or topography (referred to below as topo line).

Before modeling with points, make sure you have a good 2-D move sequential restoration and know exactly how much motion is on each fault to replicate the cross section

2- Create point file with points spaced 0.5 km apart (easiest to do in excel) need 4 columns, x,y,z, and ID y is always 0 and ID is a unique identifier for each point.

Save as comma deliminated (separated) values. Specifically export the CSV file as a windows compatible version, this may appear in the file type list as CSV (windows)

3- Importing grid:

In 2DMove, File >Insert to browse the comma delimited file you created in Excel. Select Ascii (\*.\*) from the drop down menu in the bottom right side of the

Insert File window and browse the file.

The Ascii Import Wizard will open. Toggle on Comma in the Column Separators field and choose the appropriate options in the Units field.

Set up your columns as X, Y, Z, and Id respectively. Columns can be changed by right-mouse clicking on the headers. Make sure that other column headers are blank by right clicking on them and selecting the blank field at the top of the list. Then press Load.

4- Create fault from basal decollement on right side of model to where the fault first reaches the surface on the left side of the model. Select ‘move on fault’ under ‘structural modeling’. Click on fault and add it as the active fault. Select ‘fault parallel flow’ (toggle automatically collect HW under ‘active fault’).

5- Select all lines and points above the fault and add to objects to be moved. Move fault some component of the distance (5 or 10 km, a number that can be in fairly equal increments for every fault). (save)

6- To isostatically compensate for the load: Open window for flexural isostacy under ‘operations’. Select values for load density and elastic thickness (mantle density and young’s modulus can stay as default). Edit the topo line (where the fault cuts the surface) so that the fault cut off returns to 0 km (or the topo base). Select deformed topo line as top of load, and import previous topography as base of load. Make sure to select ‘load’ and ‘sub aerial’. Turn on points and select all points and lines, add to ‘other objects’, then press apply. Save when finished. Export deformed, isostatically balanced topo line from 2D move after this step (as ascii file, delimited) and run through topo scrip for new topo line (see step 12 below).

7- To isostatically compensate for erosion: Delete downwarped topo line, add previous topo line (from previous time step, not the new one created after isostacy), select again as ‘load base’ then toggle ‘off load’ and apply. (save)

8- Import new topo (created from script), Remove (cut) ‘eroded’ lines that project above this topo. This is just for aesthetics. save as final image/ step.

9- Move fault if necessary, select and add any new lines, cut lines or new points to ‘objects to be moved’ and then repeat steps 5-9 till final solution.

Note: Shift control ‘P’ toggles points on and off, shift control ‘L’ toggles lines on and off

10- Exporting grid:

when all time steps are done, export points from each times step. Use select all points (under edit) and then go to File > Export. Type the File export name and choose the Ascii format from the drop-down menu. Then press Save. The Ascii Export toolbox will open. Set the XY Unit, Z Unit and Elevation fields and toggle on Delimited in the Data. Type field. Click the Set button next to it. In the Ascii Data Export window you can set the column headers you want to export by right clicking on the column headers bar. For the 5th column, select Id from the menu. Then press ok. (file needs to have x,y,z, color ID and ID)

11- To create velocity files run velocity.py in the same directory that the exported grid is in. You will need to make an input file that lists 1) first line: depth in y dimension in km (10) 2) second line: number of nodes in y dimension (5), and 3 third line, 4th line etc. to end of file: time in Myrs (space) filename. See example in script.

12. check velocity by running plot\_input\_velocities.sh this is a gnuplot script so needs gnuplot installed on system

13- To create topo using makeTopo.py

Export deformed topo line from 2D move make sure x,y,z, and color ID are selected under export window. Run makeTopo.py python script. When you run the script (in the folder where the file is) include necessary components in the command line (see beginning of makeTopo.py for what is needed)

14- Place topolines created for each 2D move time step and velocity files (both .dat) in the input folder for 2D move. You will need to provide the name of each file (there should be a topo file and a velocity file for each increment) in the Pecube.in file.