LAHARFLOW: A MODEL OF LAHARS FOR HAZARD ASSESSMENT

Mark Woodhouse[1,2], A.J. Hogg[2], J.C. Phillips[1]

[1] School of Earth Sciences, University of Bristol, Bristol, UK

[2] School of Mathematics, University of Bristol, Bristol, UK

SCHEDULE

- Overview of the LaharFlow model formulation
- Introduction to LaharFlow web-interface
- LaharFlow exercises

Please stop us at any time to ask questions and have discussions.

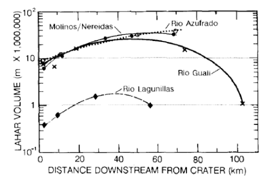
FORMULATION

Lahars are complex sediment-laden flows:

Flows are fundamentally unsteady, with fronts and discontinuous features

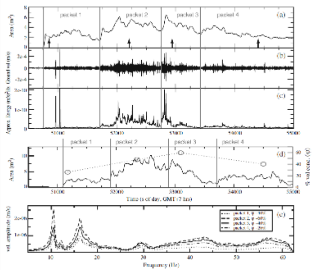
Semeru, Indonesia, 2003

Significant mass is added through erosion which has a critical influence on flow dynamics



(Pierson et al, 1989)

Ice-melt and rainfall triggered lahars have multiple distinctive sources



Semeru lahar pulses from distinctive sources (Doyle et al, 2012)

Particle segregation strongly influences frictional regime and thus flow dynamics

Cayambe, Ecuador, 2017

REQUIREMENTS OF A LAHAR HAZARD MODEL

For a model to be useful for lahar hazard management it must:

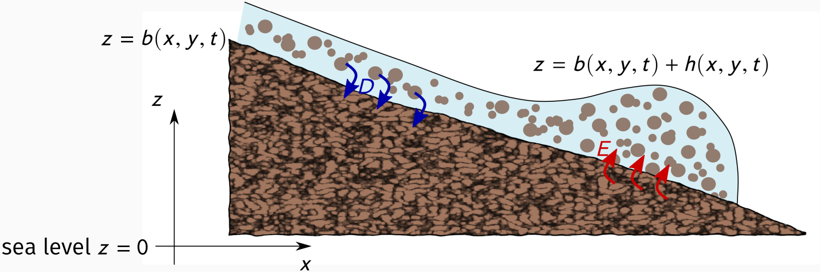
- reproduce observed behaviour over a wide range of conditions;
- need only modest computational resources and freely available forcing data;
- facilitate exploration of inputs;
- not be highly sensitive to parameters that are difficult to measure.

LaharFlow is a dynamic lahar model based on three conservation principles:

- conservation of mass
- conservarion of momentum
- conservation of transported solid material

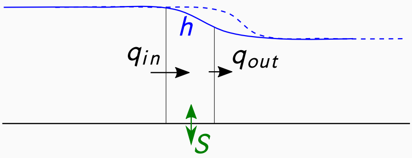
For efficiency, and because we model flows over large distances, we use a shallow-layer approximation. This requires parameterizations of some physical processes:

(i) basal drag
(ii) erosion & deposition



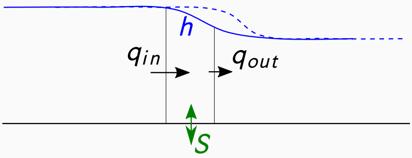
- lahar depth $\boldsymbol{h}$
- depth-averaged velocity $\bar{\boldsymbol{u}} = (\bar{u}, \bar{v})$
- depth-average concentration of solids $\bar{c}$
- erosion entrainment rate E
- deposition rate D
- porosity of erodible bed ϕ

The equations we use are derived from conservation principles on control volumes.



Change in mass = mass flux in - mass flux out + mass added from (or lost to) base boundary

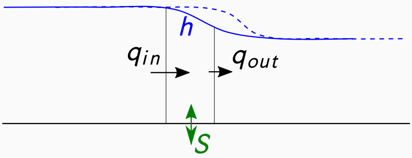
The equations we use are derived from conservation principles on control volumes.



Change in mass = mass flux in - mass flux out + mass added from (or lost to) base boundary

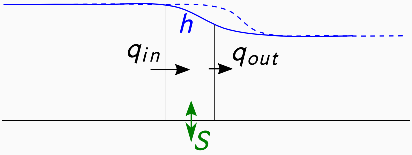
$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}\left(h\bar{u}\right) = S$$

The equations we use are derived from conservation principles on control volumes.



Change in momentum = momentum flux in - momentum flux out + forces (hydrostatic pressure gradient, basal tractions & gravity) + momentum exchange with bed

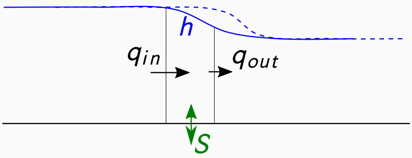
The equations we use are derived from conservation principles on control volumes.



Change in momentum = momentum flux in - momentum flux out + forces (hydrostatic pressure gradient, basal tractions & gravity) + momentum exchange with bed

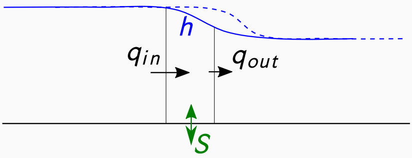
$$\frac{\partial h\bar{u}}{\partial t} + \frac{\partial}{\partial x}\left(h\bar{u}^2\right) = -\frac{\partial}{\partial x}\left(\frac{1}{2}gh^2\right) - \frac{\tau_{xz}(b)}{\rho} - gh\frac{\partial b}{\partial x} + u(b)S$$

The equations we use are derived from conservation principles on control volumes.



Change in solid mass = solid mass flux in - solid mass flux out + solid mass added from (or lost to) base boundary

The equations we use are derived from conservation principles on control volumes.



Change in solid mass = solid mass flux in - solid mass flux out + solid mass added from (or lost to) base boundary

$$\frac{\partial h\bar{c}}{\partial t} + \frac{\partial}{\partial x}\left(h\bar{u}\bar{c}\right) = S_c$$

Mass conservation:

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(h\bar{u}) + \frac{\partial}{\partial y}(h\bar{v}) = \frac{E - D}{1 - \phi}$$

Momentum conservation:

$$\frac{\partial}{\partial t}(h\bar{u}) + \frac{\partial}{\partial x}\left(h\bar{u}^2 + \tfrac{1}{2}gh^2\right) + \frac{\partial}{\partial y}(h\bar{u}\bar{v}) = -\frac{\tau_{xz}(b)}{\rho} - gh\frac{\partial b}{\partial x} + u(b)\frac{E - D}{1 - \phi}$$

$$\frac{\partial}{\partial t}(h\bar{v}) + \frac{\partial}{\partial x}(h\bar{u}\bar{v}) + \frac{\partial}{\partial y}\left(h\bar{v}^2 + \tfrac{1}{2}gh^2\right) = -\frac{\tau_{yz}(b)}{\rho} - gh\frac{\partial b}{\partial y} + v(b)\frac{E - D}{1 - \phi}$$

Solids conservation (assumes a well-mixed flow):

$$\frac{\partial h\bar{c}}{\partial t} + \frac{\partial}{\partial x}(h\bar{u}\bar{c}) + \frac{\partial}{\partial y}(h\bar{v}\bar{c}) = E - D$$

References: Cao et al. (2004), Iverson & Ouyang (2014), Ouyang et al. (2014,2015)

Basal drag is a representation of the flow rheology.

$$\tau_{iz}/\rho = \mathcal{F} \cdot \bar{u}_i / |\bar{\boldsymbol{u}}| \qquad (i = x, y)$$

BASAL DRAG CLOSURE

Basal drag is a representation of the flow rheology.

$$\tau_{iz}/\rho = \mathcal{F} \cdot \bar{u}_i / |\bar{\boldsymbol{u}}| \qquad (i = x, y)$$

Candidate parameterizations for lahars include:
Chézy drag (turbulent fluid):

$$\mathcal{F} = C_d |\bar{\boldsymbol{u}}|^2$$

Basal drag is a representation of the flow rheology.

$$\tau_{iz}/\rho = \mathcal{F} \cdot \bar{u}_i / |\bar{\boldsymbol{u}}| \qquad (i = x, y)$$

Candidate parameterizations for lahars include:
Chézy drag (turbulent fluid):

$$\mathcal{F} = C_d |\bar{\boldsymbol{u}}|^2$$

Coulomb drag (granular flow e.g. Pouliquen, 1999):

$$\mathcal{F} = \mu g h \qquad \text{with} \qquad \mu = \tan\theta_1 + (\tan\theta_2 - \tan\theta_1)\exp\left(-\gamma h^{3/2} / |\bar{\boldsymbol{u}}|\right)$$

Basal drag is a representation of the flow rheology.

$$\tau_{iz}/\rho = \mathcal{F} \cdot \bar{u}_i / |\bar{\boldsymbol{u}}| \qquad (i = x, y)$$

Candidate parameterizations for lahars include:
Chézy drag (turbulent fluid):

$$\mathcal{F} = C_d \, |\bar{\boldsymbol{u}}|^2$$

Coulomb drag (granular flow e.g. Pouliquen, 1999):

$$\mathcal{F} = \mu g h \qquad \text{with} \qquad \mu = \tan\theta_1 + (\tan\theta_2 - \tan\theta_1)\exp\left(-\gamma h^{3/2} / |\bar{\boldsymbol{u}}|\right)$$

Voellmy drag (fluid + grains):

$$\mathcal{F} = C_d \, |\bar{\boldsymbol{u}}|^2 + \mu g h$$

Basal drag is a representation of the flow rheology.

$$\tau_{iz}/\rho = \mathcal{F} \cdot \bar{u}_i / |\bar{\boldsymbol{u}}| \qquad (i = x, y)$$

Candidate parameterizations for lahars include:
Chézy drag (turbulent fluid):

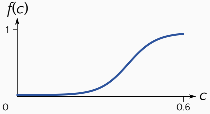
$$\mathcal{F} = C_d \, |\bar{\boldsymbol{u}}|^2$$

Coulomb drag (granular flow e.g. Pouliquen, 1999):

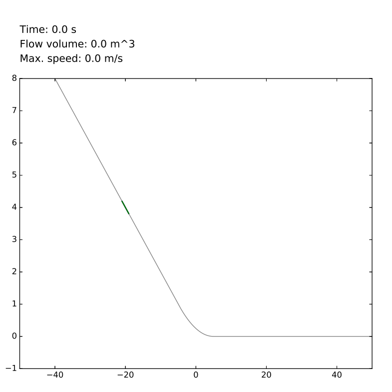
$$\mathcal{F} = \mu g h \qquad \text{with} \qquad \mu = \tan\theta_1 + (\tan\theta_2 - \tan\theta_1) \exp\left(-\gamma h^{3/2} / |\bar{\boldsymbol{u}}|\right)$$

Voellmy drag (fluid + grains):

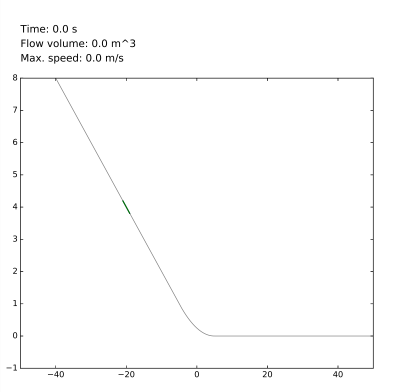
$$\mathcal{F} = C_d \, |\bar{\boldsymbol{u}}|^2 \, (1 - f(c)) + \mu g h f(c)$$



Chézy drag



Pouliquen drag



Erosion in rivers is often modelled using a dimensionless Shields stress

$$\theta = \frac{\tau}{\left(\rho_s - \rho_f\right) gD}$$

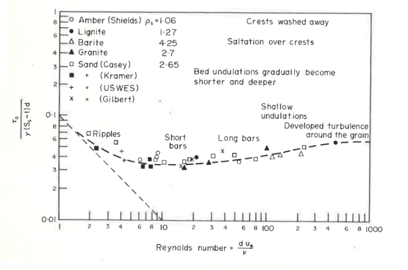
where τ is the magnitude of the shear stress, ρ_s is the density of a grain in the bed and ρ_f is the density of the fluid, g is gravitational acceleration and D is the grain diameter.

Erosion in rivers is often modelled using a dimensionless Shields stress

$$\theta = \frac{\tau}{\left(\rho_s - \rho_f\right) gD}$$

where τ is the magnitude of the shear stress, ρ_s is the density of a grain in the bed and ρ_f is the density of the fluid, g is gravitational acceleration and D is the grain diameter.

Experiments have found that erosion occurs when the Shields stress exceeds a critical value, $\theta > \theta_c$, which is a function of the particle Reynolds number.



If the critical Shields stress is exceeded then erosion occurs and solids are entrained into the flow.

The rate at which solids are entrained has been found to depend on the amount by which the critical Shields stress is exceeded. There have been several proposed parameterizations of the erosion rate, each with a similar form. We use

$$E = m\,(\theta - \theta_c)\,\sqrt{\theta - \theta_c}$$

with the coefficient m (with units of m/s) to be calibrated.

Deposition is modelled as a flux of solids that are settling out from the lahar. There are many parameterizations of the settling velocity, each representing the settling velocity as a function of particle diameter D, fluid viscosity ν and the reduced gravity g':

$$w_s = f(D, \nu, g').$$

The depositional solids flux is then given by $D = \bar{c} w_s f_1(\bar{c})$ where $f_1(\bar{c})$ models hindered settling .

Deposition is modelled as a flux of solids that are settling out from the lahar. There are many parameterizations of the settling velocity, each representing the settling velocity as a function of particle diameter D, fluid viscosity ν and the reduced gravity g':

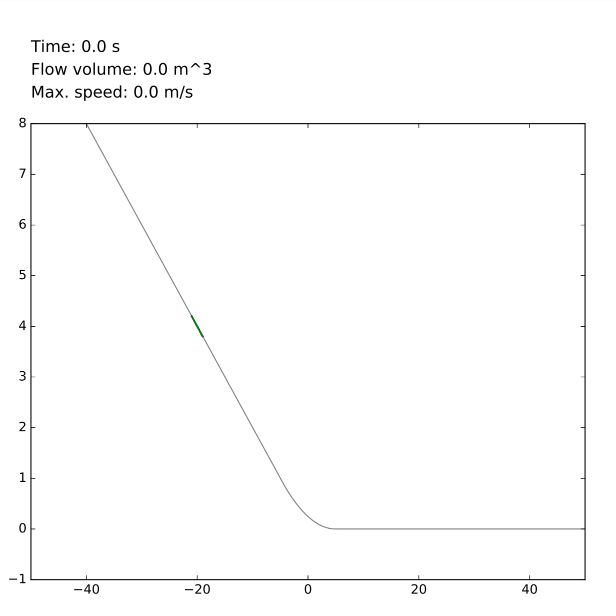
$$w_s = f(D, \nu, g').$$

The depositional solids flux is then given by $D = \bar{c} w_s f_1(\bar{c})$ where $f_1(\bar{c})$ models hindered settling .

Erosion and deposition change the topography. The morphodynamic link between the flow and the topographic evolution is modelled by locally altering the bed elevation:

$$\frac{\partial b}{\partial t} = -\frac{E - D}{1 - \phi}$$

1D simulation – erosion & deposition



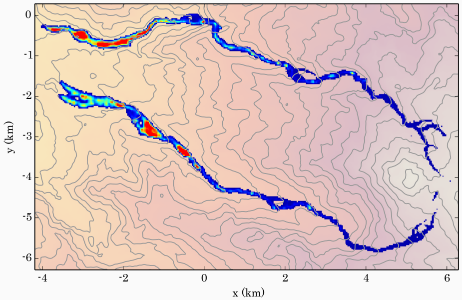
EROSION DEPTH

Erosion can only occur if there is erodible material in the bed. If all of the erodible material is entrained into the flow then no further erosion can occur.

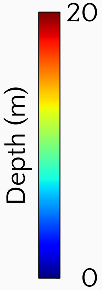
In LaharFlow an erosion depth is specified to model the amount of erodible material in the bed.

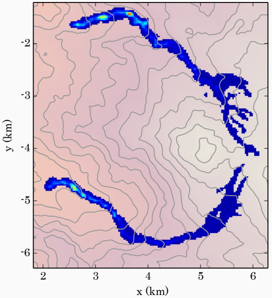
Entrainment of solids due to erosion is a very important process in the lahar dynamics.

The bulking up increases the flow volume, but does so gradually. If we try to model the bulking up by setting a larger initial volume then the flow momentum will not be correct, so dynamic features (flow velocities, lahar path, arrival times) will be incorrect.



Entraining
Volume 17.3 × 10^6 m^3

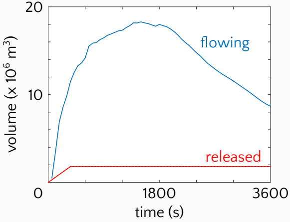


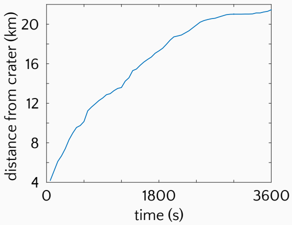


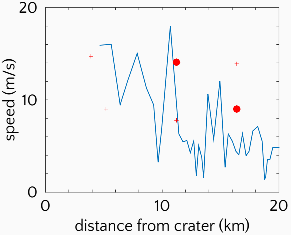
Not entraining
Volume 1.8 × 10^6 m^3

The parameters in LaharFlow have been calibrated for large lahars, primarily using observations from the Ruiz lahars of 1985 using the data of Pierson et al. (1990).

The parameters in LaharFlow have been calibrated for large lahars, primarily using observations from the Ruiz lahars of 1985 using the data of Pierson et al. (1990).







LAHARFLOW WEB INTERFACE

www.laharflow.bris.ac.uk

ACCESSING LAHARFLOW

LaharFlow is hosted at www.laharflow.bristol.ac.uk

You need to register for an account and agree to the condition of use. Registering requires that you join the laharflow-info email list (for announcements regarding maintenance, upgrades and publications) using your registered email address.

For this demonstration, four temporary user accounts have been made:

Streva1, Streva2, Streva3, Streva4

each with password “Streva”.

LaharFlow is recommendend for use on Chrome and Firefox web browsers. It also works (but has not been tested as thoroughly) on Safari. It has not been tested on IE.

The LaharFlow user interface is the way in which you communicate input parameters and source conditions to the LaharFlow model.

The model is then initiated and calculations are performed remotely on a fast server at the University of Bristol.

Model results are displayed in the user interface as maps of the flow, and data is available to download during a simulation and when the run is complete.

The transfer of images from the server to the user interface can be time consuming on a low bandwidth connection. The automatic updating of plots can be switched off.

DOWNLOAD YOUR DATA!!

Warning: the LaharFlow results are not stored on the server. After a simulation, download and store your data.

The results are lost when you next click ‘Run Model’, but are stored if you log out without starting a new run.

EXERCISE 1

Introduction to the LaharFlow interface

EXERCISE 1: INTRODUCTION TO LAHARFLOW INTERFACE

The LaharFlow interface consists of an upper results panel (initially blank on first log in) and a lower input panel.

The input panel has four tabs; two are concerned with data that is input into the model (‘Domain settings’ and ’Parameters’) and two are for retrieving data from the simulation (‘Plot settings’ and ‘Data export’).

There is also a help box that displays information about the user interface elements, and buttons labelled ‘Run Model’ and ‘Cancel run’.

NOTE: the ‘Run Model’ button is disabled until all inputs are valid.

EXERCISE 1: DOMAIN SETTINGS

The first tab is ‘Domain settings’. This provides a form for setting:

- the location of the simulation;
- the size of the simulation domain;
- the spatial resolution of the simulation;
- the time duration that should be simulated.

Change the location to near Ubinas:

latitude:	-16.38 N
longitude:	-70.84 E

The 'Domain settings' tab is also where sources are specified. Multiple sources can be specified, currently as 'flux sources' where a time series of the volume flow rate and concentration of solids is specified.

Clicking 'Add flux source' opens a form for specifying the source by setting:

- the source location, either as a latitude and longitude, or as distances relative to the centre of the simulation domain;
- the radius of the source (all sources are circles);
- the time series for the volume flux and solids concentration.

Once set, the source can be added to the simulation by clicking 'Add source'. Sources that are added can be reviewed and removed by clicking 'Review sources'. All sources are removed by clicking 'Clear sources.'

Exercise 1: Domain settings – source conditions

Add a source on the slopes of Ubinas volcano.

Set the location to be 4000 m west and 4000 m north of domain centre, and specifiy a radius of 100 m.

Use the following times series:

time (s)	0	100	1000
volume flux (m^3/s)	0	1000	0
concentration	0	0	0

Review the source and view the hydrograph. What will be the total volume released?

Exercise 1: Domain settings – View domain

Once the domain settings form has been completed, the simulation domain and positioning sources can be viewed by clicking ‘View domain’ which produces a map in the results panel.

View the domain and confirm that the source is appropriately positioned.

The ‘Parameters’ tab displays the default, calibrated model parameters, and allows the default values to be altered.

Leave the ‘Parameters’ fixed at default values for this exercise.

To prepare for visualizing the results of the simulation, go to the 'Plot settings' tab. Uncheck 'auto update' so that we do not plot as the results are produced.

The auto-update can usually be left on, and maps are displayed as the results are produced. For shared wifi plotting can take a lot of time, so unchecking 'auto update' can save time. We can still view results while the simulation runs.

Also on this tab, we can select which of four maps are displayed.

Click 'Run Model' to start the simulation.

The inputs specified will be validated and send to the server.

You will see a status report as 'Calculating' and the time that the solver has reached.

EXERCISE 1: LOOK AT SIMULATION RESULTS

As simulation results are produced, we can view maps by navigating to the 'Plot settings'.

You will see a table of result numbers being produced. In total 100 instantaneous results will be produced.

Click on one of the numbers in the results table and the selected maps will be displayed.

NOTE: on a shared wifi the results can take a few seconds to load, but be patient and they will appear.

We can inspect the maps more closely by either:

- grabbing a plot (click and hold) to zoom in by 500%;
- double clicking the plot to open it at full resolution (one pixel for each cell).

Plots can also be saved by right clicking and saving using your browser.

EXERCISE 2

Small volume lahars at Ubinas – the role of erosion

In this exercise we will simulate small volume lahars at Ubinas, and investigate the effect of erosion.

Cancel the current run if it is not complete.

The current run had an erosion depth of 1 m, meaning that only 1 m of the bed could be eroded. Look at the change in elevation by selecting 'Map of change in elevation' in the 'Plot settings' tab and selecting some instances of time from the results.

On careful inspection, we can see some, but very little, erosion and deposition.

If your browser allows, open the image in a new tab, or save the plot temporarily.

We will keep the same source settings, but now change the erosion depth on the 'Parameters' tab.

We will first set the erosion depth to the maximum allowed value of 10 m.

Re-run the simulation, and look at the new change in elevation.

How has the increase in the erosion depth affected the flow?

EXERCISE 2: SWICHING OFF EROSION.

Now we will switch off erosion by setting a zero erosion depth in the ‘Parameters’ tab.

Run the model again and compare the flow characteristics.

EXERCISE 3

Lahars at Misti – exporting results

We will next simulation lahars at Misti volcano.

Cancel the current run.

The coordinates of Misti are 16.35 W, 71.477 S.

Change the domain settings to select Misti volcano. We will place sources in the southern flank, so using 'View domain', set a domain location that will maximize the domain for the simulation.

Select a source location on the southern flank, and choose some source characteristics.

Keep the peak flow less than $2000\,\mathrm{m}^3/\mathrm{s}$.

Make sure to reset the erosion depth to a suitable value.

Run the model and view some of the maps when they are produced.

EXERCISE 3: EXPORTING RESULTS

In addition to view the results in the LaharFlow interface, we can export results for further examination and processing.

The 'Data export' tab contains the data download options.

There an option to produce a PDF report of the simulation. NOTE: the PDF report is currently not active – we would like to know what you would like to see in the report.

There are options to download a single results files as raw text at a chosen instance of the simulation, to download a Google Earth compatible KML file, or to download a zip file containing all of the raw data produced by the model.

EXERCISE 3: THE RAW DATA FILE

Select 'Plain text file' as the model output type. Here we will download a single results file at one instance of the simulation.

The available results are shown in the table, which will update as the calculation proceeds.

Select one of the available results, and click 'Download model output'.

Your data should download and can be viewed in a text editor. It is a comma-separated values (CSV) file type, with column headers to describe the data.

The data returned are the quantities calculated by the model – the flow depth (h), the volume fluxes per unit area ($h\bar{u}$ and $h\bar{v}$), and the volume of solids in the flow per unit area ($h\bar{c}$). Other quantities of interest can be obtained from these in post processing.

NOTE: in calculating e.g. the speed of the flow, it is necessary to divide $h\bar{u}$ and $h\bar{v}$ by the flow depth. It may not be appropriate to do this in cells where the flow depth is very small.

Select 'KML file' as the model output type. Here we will download a KML file that containing the maximum flow depth that has been found in the simulation.

The data in the KML is updated each time the model results are produced, and the downloaded KML will be the most recently produced results.

If you have Google Earth, the downloaded KML file can be imported directly and viewed.

The KML may also be viewable in gis software. NOTE: we have only tested the import of KML files into Google Earth and QGIS. Some versions of QGIS do not fully support the KML file.

The export of all data is available by selecting 'zipped numerical results'.

NOTE: the zipped data file can be large and may take some time to download is the wifi network is shared.

Unzipping the downloaded file, we find the exported data contains:

- the raw data csv files at each instance of the simulation – 101 data files are produced with names 000000.txt, 000001.txt etc.;
- the kml file of the maximum flow depth, called MaxHeights.kml;
- the maximum flow depth as a csv file, called MaxHeights.txt;
- the maximum flow speed as a csv file, called MaxSpeeds.txt;
- a file containing information on the LaharFlow settings for this simulation, called RunInfo.txt;
- a text file containing data on the flow volume as a function of time, called Volume.txt.

THANK YOU