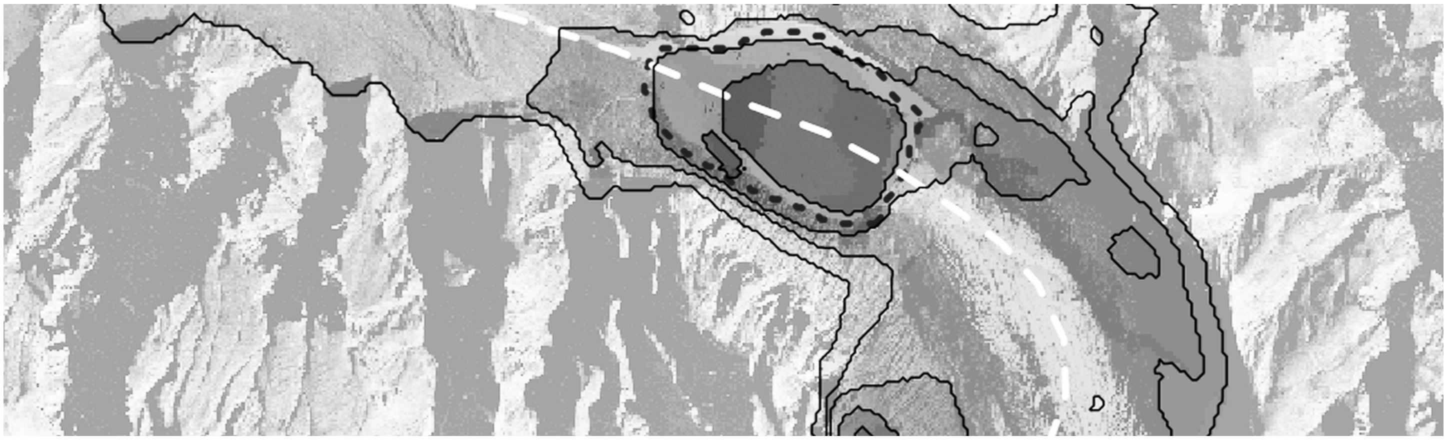
r.avaflow The mass flow simulation tool

r.avaflow 2.4 User manual

by Martin Mergili



r.avaflow 2.4 represents a GIS-supported open source software tool for the simulation of complex, cascading mass flows over arbitrary topography. It empoys the NOC-TVD numerical scheme (Wang et al., 2004) along with a Voellmy-type model, with an enhanced version of the Pudasaini multi-phase flow model (Pudasaini and Mergili, 2019), or with a simple mass point model for the approximation of sliding processes. Complementary functions include entrainment, deposition, stopping, and phase transformations. The starting mass may be defined through raster maps and/or hydrographs. r.avaflow 2.4 includes the possibility to explore multi-core computing environments to run multiple simulations at once as a basis for parameter sensitivity analysis and optimization.

Tutorial videos

r.avaflow 2.4 is most easy to learn from videos.

The following videos are intended to facilitate the first steps with r.avaflow. They were created based on r.avaflow 2.2, but are equally useful for r.avaflow 2.4, even though some input options might have changed. However, note that the installation of r.avaflow now uses the GRASS module *g.extension*, so that no installation script is required any more.

- Lesson 1 | General aspects
- Lesson 2 | Software installation and preparation of data
- Lesson 3 | First simulations with r.avaflow

The Frank Slide data used in the video tutorials were provided as a courtesy by Oldrich Hungr and should only be used for training purposes. Please consult Hungr (2008) for more information on the data and their previous use.

+ Recent changes

r.avaflow 2.4 includes some substantial changes, compared to r.avaflow 2.0-2.3.

The most fundamental changes in r.avaflow 2.4, compared to r.avaflow 2.3, are indicated below. This list is most useful for those users which have already been working with r.avaflow 2.3.

- r.avaflow 2.4 only operates with Python 3 and GRASS 7.8. The use with Python 2.7 and older versions of GRASS GIS has been deprecated.
- The installation of r.avaflow 2.4 uses the built-in GRASS module *g.extension*. No installation script is provided any more.
- Mass point model for sliding: a completely new feature has been introduced, allowing to approximate the motion of slide-type movements with no or limited internal deformation. At each time step, the material at each raster cell is moved through a one-parameter or two-parameter friction model, based on the slope and aspect of the entire sliding mass or a user-defined neighbourhood zone. There is flexibility in transformation from sliding to flowing, and in spatially differentiating between zones of sliding and flowing. This feature is managed and parameterized through the newly introduced options *tslide* and *slidepar*. It is mainly intended for the very initial stage of landslide processes, before disintegration of the mass.
- Entrainment, deposition, and stopping: the models covering the interaction of the flow with the basal surface have been fundamentally revised and extended, allowing for a much more realistic representation of entrainment and deposition, but also stopping. Obsolete approaches have been removed. See options *tstop*, *basal*, and *cvshear* for more details.
- Flow parameters: some changes and rearrangements were made. The option *ambient* was replaced by the newly introduced option *basal*, which serves for the parameterization of the updated entrainment, deposition, and stopping models. The fluid friction number (which is related, but not similar to the Manning number) is now included in the option *friction* whereas the ambient drag coefficient, which has turned out to be hard to handle in practical applications, has been moved to the option *special* with a default value of 0.0.
- The function for the dynamic adaptation of the friction parameters has been modified in order to allow for a more convenient parameterization. If the function is activated (options *controls*), the friction parameters given through the option *friction* are valid for flows with zero kinetic energy (i.e. at rest). The exponential decrease of the friction is governed by a coefficient which is multiplied with the kinetic energy at a given raster cell. Further, the scaling of the friction parameters with the fraction of the respective case has been integrated. However, for most applications, it is strongly recommended to deactivate this function by setting the scaling exponent to zero.
- Progressive collapse: the possibility to consider progressive release though the options *trelease* and *trelstop* has been extended from the first phase to the entire mass. Still, this function should only be used for one single release mass, and cannot be combined with the extrusion-type progressive release.
- Curvature: the effects of topographic curvature can be more rigorously applied than in the previous versions. They can be switched on and off through the second value in the *controls* option: 0 = deactivated, 1 = only applied to decelerationg terms (comparable to the implementation in version 2.3), 2 = applied to all terms (slows down the simulation and may cause numerical issues). The default is 1. Note that deactivating curvature effects can make the simulated flow substantially more mobile.
- Diffusion control: this feature has been deprecated. Even though it has some effect in

reducing computational times, unplausible behaviour has been observed in some specific flow situations.

- Option *hydrocoords*: the aspects of the hydrograph profiles have to be provided in degrees instead of radians.
- The maximum flow velocity displayed in the map plots, which sometimes used to show unrealistically high values, is now computed in a different way, better approximating the real conditions.
- Additional output raster maps of frontal and maximum flow velocities and phase fractions are automatically generated, depending on the model used.
- The accessibility of the map plots displaying the output of the mixture or one-phase model has been improved by adapting the colour schemes.

The major changes in r.avaflow 2.3, compared to r.avaflow 2.2, are indicated below. This list is most useful for those users which have already been working with r.avaflow 2.2.

- Numerical issues with the Voellmy-type mixture model, which were observed in the previous versions, have been fixed. Decelerating forces are now treated separately from the accelerating forces (as it had been implemented earlier for the Pudasaini and Mergili, 2019 multi-phase model), in order to avoid changes of the flow direction due to deceleration effects, having caused the numerical instabilities.
- Progressive collapse: the possibility to consider progressive release though the options *trelease* and *trelstop* has been extended. It is now possible to simulate the progressive collapse of a release mass, starting from its summit, instead of the progressive extrusion of the release mass from its basis. Progressive collapse is activated by providing *trelstop* with negative sign. Note that this function is still experimental and, when used with the multi-phase model, is only applied to the the first phase. It should only be used for one single release mass, and cannot be combined with the extrusion-type progressive release.
- Ambient drag: the default value of the ambient drag coefficient has been changed to 0.0. Experience and feedback from users have shown that setting the ambient drag coefficient (mainly used for air resistance) in a reliable way is a considerable challenge. Therefore, it is now recommended not to consider air resistance in a direct way, but rather to include its effect in the friction parameters (basal friction and turbulent friction in the Voellmy-type model, basal friction in the Pudasaini and Mergili, 2019 model).
- Internal friction and basal friction: if the internal friction is lower than the basal friction, it is set equal to the basal friction. This means that, in simulations with varying basal friction, the internal friction can be globally set to the basal friction by choosing a very low value.
- Fluid friction: Manning's *n* is now used as the fluid friction coefficient, the default value is 0.05. As Mannings *n* is widely used, it is supposed to greatly facilitate the choice of appropriate values. This change was mostly implemented by Sigríður S. Gylfadóttir from the Iceland Meteorological Office.
- Stopping: an additional stopping criterion has been introduced. When setting the fifth value in the *controls* parameters to 7, the flow stops when there is no raster cell at which the dynamic flow pressure is larger than the threshold value for flow pressure given in the *thresholds* parameter.
- Flow pressure: only the dynamic component of the pressure is considered, whereas the static component has been removed.
- Time of reach: the time of reach is now automatically provided as an additional output GRASS raster, ASCII raster, and map plot, indicated by the string *treach*. It shows the time in seconds after which the flow first reaches each raster cell, using the mimimum flow height for display (*thresholds* parameters) as the criterion. Time of reach is also shown in the control points output files. With multiple simulations (flag *m*), the ratio between the time of reach and the

observed time of reach is included in the evaluation. Observed times of reach can be provided through the newly introduced option *reftime*, where comma-separated values indicate the observed travel times to each control point in seconds.

- Not providing the flag *a* has a more "aggressive" impact than in the previous versions. The output in terms of GRASS raster maps, ASCII rasters, and map plots is reduced to an absolute minimum (flow heights, change of basal topography if relevant, time of reach) in order to save time and memory. Note that no profile graphs can be produced without this flag. However, note that the flag *a* is now automatically enabled for multiple simulations (flag *m*).
- Some further bugs were fixed, such as another issue with progressive release.

The major changes in r.avaflow 2.2, compared to r.avaflow 2.1, are summarized as follows. This list is most useful for those users which have already been working with r.avaflow 2.1.

- An important bugfix has been included, concerning the treatment of hydrographs with Python 3.
- In the input hydrographs, the discharge in cubic metres per seconds now has to be provided instead of the flow height, whereas flow velocity has to be provided in the same way as before. Discharge and flow velocity are always applied to the centre of the hydrograph profile. The profile length is only used for the output hydrographs, but has to be provided also for the input hydrographs.
- After the flexibility in the combination of phases had been strictly constrained in r.avaflow 2.1, r.avaflow 2.4 again allows for the combination of two solid phases and one fluid phase (*phases=s,s,f*). However, this is the only exception from the rule that the sequence always has to be solid - fine solid - fluid when considering more than one phase. Simulations with two solid phases and one fluid phase should still be considered experimental. They can be useful when considering avalanches of rock and ice, since the consideration of granular flows of ice blocks as fine solid is not necessarily appropriate.

The list below summarizes the most important changes introduced in r.avaflow 2.1, compared to r.avaflow 2.0. It is most useful for those users which have already been working with r.avaflow 2.0.

- r.avaflow 2.4 can be run with Python 2.7 or Python 3. Installation with Python 2.7 is only recommended if the setup with all required software packages is already available. The script *grass7.install.sh* is exclusively targeted at the setup for the application of r.avaflow with Python 3 and GRASS GIS 7.8 or higher.
- The combination of phases is more restricted than in r.avaflow 2.0, in order to ensure a more rigorous application of the Pudasaini and Mergili, 2019 model and its extensions. Previously optimized model parameters might have to be revised. From an operational point of view, there is no two-phase model directly available any more, and the first phase in the multi-phase model is always solid, the second phase is always fine solid, and the third phase is always fluid. The multi-phase model is selected by entering *phases=m*. Input of material through the release, entrainment, and hydrograph parameters governs which phases are effectively considered and which are not. Providing input, for example, for the phases 1 and 3, but not for phase 2, effectively results in a two-phase model of solid and fluid material. It is noted that the fine solid phase can be represented by a wide variety of materials, ranging from rather solid ice to rather fluid slurry. Defining *hrelease* together with *rhrelease1* results in an effectively two-phase simulation of solid and fluid (i.e. the phases 1 and 3), where *rhrelease1* represents the fraction of solid release material. One-phase models can be used in the same way as in the version 2.0, with the difference that the Voellmy-type mixture model is selected through *phases=x*.
- Balancing of forces has been improved, so that numerical oscillations on water surfaces are strongly reduced, though not completely eliminated. Still, the new functionality has to be activated through the surface control in the *controls* parameter. Also, the boundary conditions

have been revised, so that water surfaces can now extend all the way to the margin of the area of interest, without draining towards outside. These improvments make r.avaflow potentially suitable for the simulation of tsunamis. As a consequence, an additional set of map plots displaying the evolution and maximum of the heights of impact waves or tsunamis can be generated optionally, by providing the flag *t*.

- There are two new stopping criteria available: with *stopping=4*, the flow stops when its kinetic energy reaches a certain fraction of its maximum kinetic energy during the simulation. This fraction is defined through an additional value in the *ambient* parameter. 0.05 (the default) would mean that the flow stops as soon as its kinetic energy drops below 5 per cent of its maximum kinetic energy. With *stopping=5*, the same principle applies, but considering flow momentum instead of kinetic energy. Note that the additional value in the *ambient* parameter has to be provided in any case - but it is only used if one of the new stopping criteria are applied.
- Also, the flexibility with regard to entrainment has been increased: with *entrainment=2*, the entrainment coefficient is multiplied with the flow momentum instead of the flow kinetic energy at a given raster cell.
- Non-hydrostatic effects are now optionally considered, including enhanced gravity and dispersion. This function is activated by setting the surface control to a value of 2 in the *controls* parameter. Please note that this function is still in its experimental stage and undergoes further testing.
- A new, still experimental function for time scaling can be used for the simulation of very viscous, slow moving one-phase flows. Setting the new *slomo* parameter to a value larger than 1 means that the time is not measured in seconds, but in seconds multiplied with the value provided. *slomo*=86400 would scale the time from seconds to days, resulting in output velocities of metres per day. However, such changes of units are not indicated in the output data.
- The friction parameters can be dynamically adapted to the flow kinetic energy, in order to account for fluidization and lubrication effects. This function is activated by setting the last (additional) value of the *controls* parameter to 1. The necessary constraints are provided through the new option *dynfric*. Also this function is still in its experimental stage.
- Providing the flag *a* in addition to *v* results in the generation of map plots for flow kinetic energy and flow pressure. In older versions of r.avaflow, these plots were created automatically with the flag *v*. However, they are not always needed, so that it appears appropriate to make them optional outputs.

Further, note that some bugs were identified and fixed, in comparison to r.avaflow 2.0. It is therefore not recommended to use r.avaflow 2.0 any more. If you encouter possible bugs in this version, or have ideas for improvement of the software, please do not hesitate to contact martin.mergili@boku.ac.at. Note that r.avaflow is not a commercial software, nor is its development subject of an ongoing funded project. However, it is always attempted to provide adequate support as timely as possible.

1 Requirements

r.avaflow 2.4 runs on LINUX operating systems. It relies on GRASS 7 and R.

r.avaflow 2.4 was developed and tested on Ubuntu 20.04 LTS, but is expected to work in other LINUX environments, too. It relies on GRASS GIS 7.8 and Python 3. Installation of r.avaflow 2.4 requires the grass-dev package. Visualization and evaluation of the model results (flag *v*) makes use of the R Project for Statistical Computing (recommended version: 3.6.3 or higher) and the Python library *pillow*. The following packages of R are required in order to fully explore the functionalities offered by r.avaflow 2.4: *maptools*, *stats*, *sp*, *rgeos*, *rgdal*, *ROCR*, and *raster*. The code builds on Python and C. The script *grass7.install.sh* provided with r.avaflow 2.4 can be used to automatically install all the software needed on fresh installations of Ubuntu 20.04LTS, and probably also on installations of other LINUX environments. However, please note that LINUX environments and the related software are dynamically evolving, so that the script might be outdated quickly, requiring manual modifications in case of failure. Manual interventions might also be necessary if older installations of some of the required software do exist on the system. The script has to be called from the terminal as superuser (requiring administrator rights): *sudo sh grass7.install.sh*

2 Installation

r.avaflow 2.4 can be easily installed through the command *g.extension*.

As soon as all software requirements are fulfilled, the installation of r.avaflow 2.4 is straightforward, using the built-in GRASS module *g.extension*. Assuming that the folder with the r.avaflow source code is stored in the directory */home/user1/*, installation just requires the execution of the following command:

g.extension extension=r.avaflow url=/home/user1/r.avaflow/

Please note that, if you have installed earlier versions of r.avaflow using the script *r.avaflow.install.sh*, you have to manually remove the old files installed e.g. in the folders */usr/lib/grass78/scripts* and */usr/lib/grass78/bin* in order to make sure that the r.avaflow command really calls the newly installed version.

3 Operation

r.avaflow 2.4 is most efficiently operated through shell scripts.

r.avaflow 2.4 is most efficiently operated through shell scripting. Even though beginners often feel uncomfortable first in working with the terminal and with shell scripts, most of them quickly understand that this is a highly comfortable and efficient way to launch simulations and to analyze the results in a systematic way.

First, you have to launch GRASS with the desired Location and Mapset. Please consult the official GRASS GIS website to learn how to do so. You will not use the GUI of GRASS, but work in the terminal. The command line prompt should start with the GRASS version and, in brackets, the location. This means that you are really in GRASS. You can now start r.avaflow 2.4 by typing the arguments directly into the command line:

r.avaflow -e -v prefix=test phases=s elevation=af_elev hrelease1=af_hrelease

In this case, r.avaflow 2.4 would run a simulation with the one-phase model and evaluate and visualize the results, using the elevation map *af_elev* and the solid release height map *af_hrelease* as input. The results are collected in a folder named *test_results*, which will be stored in the current directory. Both maps would have to be available as GRASS rasters in the current mapset.

You can learn much more about the input of r.avaflow in the section below. However, also with this minimum input it can be demonstrated that this way of operation is rather efficient: if you wish to run the simulation again, you can just recover the string by typing the upward key.

But we still go one step further. We can also write the command we have just typed into the terminal in an executable text file, and instead execute this file. Whereas this does not look very straightforward on a first glance, it dramatically increases our flexibility: we can now write several commands into the same script - for example, running r.avaflow for several times with different input. We can go for a beer, to sleep, or even on holiday, and see the results when we have returned, without any intervention required in between.

It is recommended to create a folder for each project we are working on. So, let us create the folder *projects/test* in our home directory. Within this folder, let us create a text file with the name test.avaflow.start.sh. The suffix *sh* stands for shell script, which is the most common type of executable file in LINUX environments. We now copy the text we have executed from the command line into this script, and save our changes. Then, back in the terminal, we have to change to the directory where the script is stored, and execute the script by typing *sh* and its name:

```
cd projects/test/
sh test.avaflow.start.sh
```

You will find the folder with the results of the simulation in the directory *projects/test*. Using shell scripts, you cannot only run several simulations at a time. They also help you to:

- Import tiff or ascii rasters into GRASS (command *r.in.gdal*)
- Create generic landcapes (command *r.mapcalc*)
- Preprocess raster maps (command *r.mapcalc*)
- Compute statistics of deposited volumes (command *r.volume*)

Examples of such start scipts can be found in the training data.

4 Input

r.avaflow 2.4 can be run with a minimum of input, but also with a combination of various parameters.

A large choice of options is available for the definition of the initial conditions and the parameterization of r.avaflow. Not all options are needed for all types of simulations - in fact, simple simulations can be performed with very little input. You can filter the set of options by clicking on one of the keywords below. Klicking on further keywords constrains the number of options. The entire set can be recovered by clicking on *All*. P1 stands for phase 1 (or the mixture), P2 and P3 stand for the phases 2 and 3.

All | Key parameters | Release | Entrainment | Hydrographs | Flow parameters

Operational parameters	Single run	Multiple runs	Mixture model	One-phase model	Multi-phase model

-a Map plots of pressure and kinetic energy

This flag - in combination with the flag *v* - results in the generation of output rasters and map plots of flow pressure and kinetic energy, in addition to the map plots of flow height, change of the basal topography (if relevant), and time of reach. No profile graphs can be produced without this flag. The flag *a* is automatically activated for multiple simulations (flag *m*). Without the flag *m*, it is deactivated by default in order to save time and memory.

-e Model execution *-e*

This flag enables the execution of the simulation model. It is activated by default.

-k Keep result GRASS raster maps

Particularly with a very large number of model runs (flag *m* or time steps *time*), a huge number of GRASS raster maps is produced which are often not needed, but require a considerable amount of disk space. Without the flag *k* all result GRASS raster maps stored in the active mapset are deleted, with the flag they are kept. This flag is deactivated by default.

-m Multiple model runs

Multiple model runs are executed to produce impact indicator index maps, to evaluate parameter performance and sensitivity, and to generate the input for the sensitivity analysis and parameter optimization tool AIMEC (Fischer, 2013). This flag is deactivated by default.

-t Map plots of impact wave or tsunami height

This flag - in combination with the flag *v* - results in the generation of map plots of wave heights in reservoirs. Reserved for the multi-phase model, those plots display solely P3 (fluid), showing the difference between the fluid flow height for each time step and the fluid release height. This function only yields a useful result for simulations involving impact waves or tsunamis. It is deactivated by default.

-v Evaluation and visualization *-v*

Series and animations of maps and profile plots are generated for the flow heights, flow velocities, flow kinetic energies and flow pressures modelled for the different time steps. If reference data are provided (options *impactarea* or *hdeposit*, selected results are evaluated against the reference data provided. This flag is activated by default.

xy prefix Prefix for output files and folders *prefix=afl*

Prefix for the output files and folders. Any type of string can be used, but it is recommended to choose a combination of approx. 3-5 characters to shortly describe the simulation.

7 cores Number of cores to be used *cores=8*

When performing multiple model runs (flag *-m*), the number of processors can be provided. E.g., *cores=8* triggers the use of a maximum of 8 processors (depending on availability), which is the default.

0.5 cellsize Cell size for simulation

Cell size in metres to be used for the model (optional with flag *m*). If the cell size is not given, it is taken from the input elevation raster map (option *elevation*).

7 limiter Numerical limiter *limiter=1*

Four types of numerical limiters can be used, indicated by an integer number:

- 1 = Minmod limiter
- 2 = Superbee limiter
- 3 = Woodward limiter
- 4 = Van Leer limiter

phases Phases to be considered *phases=m*

A maximum of three phases can be defined through shortcuts of one or two characters, separated by commas.

- x = Mixture (Voellmy-type model)
- s = Solid (plastic behaviour, frictional, non-viscous)
- fs = Fine solid (plasticity-dominated viscoplastic behaviour, frictional and viscous)
- f = Fluid (viscosity-dominated viscoplastic, non-frictional, viscous)
- m = Multi-phase (P1: solid, P2: fine solid, P3: fluid)
- s,s,f = Multi-phase (P1: solid, P2: solid, P3: fluid)

The default is the multi-phase model with solid, fine solid, and fluid material (*phases=m*). Note that the multi-phase model can be reduced from three to two phases by providing input only for two phases (see release, entrainment, and hydrograph parameters). Thereby, solid material always has to be assigned to P1, fine solid material to P2, and fluid material to P3. In any case, the density of P1 has to be higher that the density of P2, and the density of P2 has to be higher than the density of P3.

☰ phasetext	Phase labels for map plots	*phasetext=P1,P2,P3*

Optionally, comma-separated strings can be provided, describing the materials associated to each phase. The number of comma-separated strings has to correspond to the number of phases defined through the option *phases*. An example of a multi-phase model would be (*phasetext=Rock,Ice,Water*). By default, the string *Solid* is used for solid material, the string *Fine* is used for fine solid material, and the string *Fluid* is used for fluid material. This parameter has no influence on the simulation itself. Instead, it serves for a more descriptive labelling of the map plots.

☰ aoicoords	Set of coordinates delineating area of interest	

Series of four coordinates (N, S, W, E) delineating the bounding rectangle to applied for the analysis. If this parameter is not given, the bounding rectangle will be determined from the elevation raster map (option *elevation*).

☰ controls	Control parameters	*controls=0,1,0,0,0,0*

Parameter	Description	Range Default
Conversion control	Input and ouptput GIS data commonly measure the thickness of materials at the surface in vertical direction, whereas the flow models applied with r.avaflow use topography-following coordinate systems. Therefore, input release heights (measured in vertical direction) should be converted into release depths (measured perpendicular to the local topography), and the computed flow, entrainment and deposition depths should be reconverted into heights for output. Skipping these steps and considering heights at depths still conserves the columes, but leads to a slightly incorrect	[0-1] 0

	representation of the situation. Applying the conversion, in contrast, may lead to undesired effects as the cell sizes - which also have to be adapted in order to conserve volume - may become very large in steep terrain. • 0 = No conversion of flow heights to flow depths; • 1 = conversion of flow heights to flow depths (multiplication with the cosine of the slope), and reverse.	
Curvature control	Topographic curvature influences the dynamics of mass flows. In r.avaflow, curvature effects can optionally be included in the simulations through the topography-normal component of gravity. • 0 = Curvature is neglected - this leads to a higher mobility of the simulated flow, compared to the other options; • 1 = curvature is considered in the decelerating source terms. • 2 = curvature is considered in all relevant terms. This is the most rigourous application, but it slows down simulations and may cause numerical issues.	[0-2] 1
Surface control	This function is important for the interaction between landslides and reservoirs, i.e. the formation and propagation of impact waves. Explicit balancing of the forces at the shoreline of horizontal water surfaces is important to reduce numerical oscillations (surface control), but can have negative effects on simulations in other circumstances. • 0 = No balancing of forces (recommended for all simulations without landslide-reservoir interactions); • 1 = balancing of forces is applied.	[0-1] 0
Entrainment control	If activated, the flow is allowed to entrain material from its basal surface (see options *basal*, *centr*, and *cshear*). • 0 = No entrainment; • 1 = entrainment coefficient multiplied with flow momentum; • 2 = simplified Pudasaini and Krautblatter (2021) entrainment and deposition model; • 3 = combination of 1 (for entrainment) and 2 (for deposition); • 4 = acceleration-deceleration entrainment and deposition model. It is assumed that accelerating flows can entrain material, whereas decelerating flows deposit material. The entrainment coefficient defined through the options *basal* or *centr* is multiplied with the change of flow velocity at each time step to comute the rate of entrainment or deposition.	[0-4] 0
Stopping control	If activated, a criterion is defined which decides at each time step whether the flow continues or stops and deposits. • 0 = No stopping; • 1 = the flow stops if the flow kinetic energy is equal or lower than the	[0-3] 0

threshold value given in the option *basal*;

- 2 = the flow stops if the flow momentum is equal or lower than the threshold value given in the option *basal*;
- 3 = the flow stops when there is no raster cell at which the dynamic flow pressure is larger than the threshold value for flow pressure given in the option *basal*.

The option *tstop* can be used to define a raster map allowing to constrain the time of stopping in a spatially differentiated way.

Friction control	Optionally, the flow parameters internal friction angle, bed friction angle, and fluid friction coefficient can be scaled with the flow kinetic energy, following an exponential relationship, and with the fraction of the respective phase. This type of scaling is based on the assumption that flows with higher kinetic energy rather tend to develop fluidization or lubrication effects than flows with lower kinetic energy, and that the friction associated to a given phase is reduced if the phase only represents a minor fraction of the entire flowing mass. • 0 = No dynamic adaptation of friction parameters; • 1 = dynamic adaptation of friction parameters (ignored for the mixture model). See option *dynfric* for more details.	[0-1] 0

elevation

Name of elevation raster map

Name of the input elevation raster map. Note that, in the release area, the map has to represent the bottom of the flow mass. This parameter is mandatory with flag *e*, its unit is metres. Those raster cells of the elevation raster map where data are available are considered the area of interest, which can optionally be further constrained through the rectangle defined by the option *aoicoords*.

hrelease1

Name of release height of P1 raster map

Name of the input raster map representing the distribution of the release height of P1. The unit is metres. This parameter is ignored if the mixture ore one-phase model is activated through the option *phases*.

hrelease2

Name of release height of P2 raster map

Name of the input raster map representing the distribution of the release height of P2. The unit is metres. This parameter is ignored if the mixture ore one-phase model is activated through the option *phases*.

hrelease3 Name of release height of P3 raster map

Name of the input raster map representing the distribution of the release height of P3. The unit is metres. This parameter is ignored if the mixture ore one-phase model is activated through the option *phases*.

hrelease Name of release height raster map

Name of the input raster map representing the distribution of the total release height. The unit is metres. This parameter is used for the mixture and one-phase models (option *phases*). It can also be employed for the multi-phase model together with the option *rhelease1*.

0.5 rhrelease1 Ratio of P1 release height

Without flag *-m*, dimensionless number in the range 0-1 indicating the fraction of the release height of P1 out of the total release height given by the option *hrelease*. The remaining fraction is assigned to P3, whereas the release height of P2 is set to zero. With flag *-m*, two or three comma-separated dimensionless numbers in the range 0-1 indicating the lower and the upper threshold for randomization or controlled variation (see option *sampling*). This parameter is ignored for the mixture and one-phase models (option *phases*) or when *hrelease* is not given.

0.5 vhrelease Variation of release height

Two or three comma-separated dimensionless numbers indicating the lower and upper thresholds for the randomization or controlled variation (see option *sampling*) of the release height between different model runs. The resulting number is multiplied with the values of the release height given by the option *hrelease*. *vhrelease* is only considered with flag *-m*. If the parameter is not given, the release height is kept constant throughout all model runs.

trelease Name of release start time raster map

Name of input raster map representing the time when the mass at each cell is released, given in seconds after the start of the simulation. This parameter is optional. If it is not given, the mass is released immediately.

trelstop Name of release stop time raster map

Name of input raster map representing the time when the release stops, given in seconds after the start of the simulation. This parameter only comes into action when also *trelease* is provided, giving the start of the release. If *trelstop* is provided, a continuous release is considered and *hrelease*, *hrelease1*, *hrelease2*, and/or *hrelease3* have to be given in vertical metres per second. This process can be described as a continuous extrusion of material, such as it is the case for lava during an effusive volcanic eruption. It also possible to simulate the progressive collapse of a release mass, starting from its summit, instead of the progressive extrusion of the release mass from its basis. Progressive collapse is activated by providing *trelstop* with negative sign. Note that this function is still experimental and, when used with the multi-phase model, is only applied to the the first phase. It should only be used for one single release mass, and cannot be combined with the extrusion-type progressive release.

tslide Name of time of initial sliding raster map

Name of the input raster map indicating for each cell the time in seconds until which the mass point model for sliding, parameterized through the option *slidepar*, should be applied. Cell values of -777 provided with the multi-phase model (*model=m* or *model=s,s,f*) automatically result in the sliding model to be applied to purely solid and fine solid material, and the flow model to be applied to pure fluid or solid-fluid mixtures. Other negative values, zero, or not using this option at all mean that the sliding model is not applied.

If initial sliding is used for the entire area of interest, it is recommended to set the second number in the option *cfl* to something around 0.1-0.5 seconds, in order to avoid unnecessarily short time steps.

tstop Name of stopping time raster map

Name of the input raster map indicating for each cell the time in seconds from which on stopping is enabled. This means that the stopping criterion set through the option *basal* is disabled before that point in time. If this map is not provided, stopping is enabled from five seconds onwards.

Negative raster cell values have a particular meaning: negative values are interpreted as absolute values in seconds, but the minus sign triggers the behaviour that the flow material at a given raster cell stops as soon as the stopping time is reached, irrespective of the fulfilment of the criterion provided in the option *basal*. With the multi-phase model (*model=m* or *model=s,s,f*), however, the fluid P3 material escapes with an equal amount (in terms of volume) of solid P1 material and the P2 material. This function helps to simulate the development of distal debris flows and related processes from depositing solid-dominated flows such as rock avalanches.

vinx Name of release velocity in x direction raster map

Name of the input raster map representing the distribution of the release velocity in x direction in the mixture and one-phase models. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells.

viny Name of release velocity in y direction raster map

Name of the input raster map representing the distribution of the release velocity in y direction in the mixture and one-phase models. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells.

vinx1 Name of release velocity of P1 in x direction raster map

Name of the input raster map representing the distribution of the release velocity of P1 in x direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells.

viny1 Name of release velocity of P1 in y direction raster map

Name of the input raster map representing the distribution of the release velocity of P1 in y direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells.

vinx2 Name of release velocity of P2 in x direction raster map

Name of the input raster map representing the distribution of the release velocity of P2 in x direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells. It is further ignored if the one-phase or mixture model is activated through the option *phases*.

viny2 Name of release velocity of P2 in y direction raster map

Name of the input raster map representing the distribution of the release velocity of P2 in y direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells. It is further ignored if the one-phase or mixture model is activated through the option *phases*.

vinx3 Name of release velocity of P3 in x direction raster map

Name of the input raster map representing the distribution of the release velocity of P3 in x direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells. It is further ignored if the one-phase or mixture model is activated through the option *phases*.

viny3 — Name of release velocity of P3 in y direction raster map

Name of the input raster map representing the distribution of the release velocity of P3 in y direction. The unit is m/s. This parameter is optional and not needed for most applications. If it is not provided, its value is set to zero for all cells. It is further ignored if the one-phase or mixture model is activated through the option *phases*.

hentrmax1 — Name of maximum height of P1 raster map

Name of the input raster map representing the distribution of the maximum height of entrainment of P1. The unit is metres. This parameter is ignored if the mixture or one-phase model is activated through the option *phases*.

hentrmax2 — Name of maximum height of P2 entrainment raster map

Name of the input raster map representing the distribution of the maximum height of entrainment of P2. The unit is metres. This parameter is ignored if the mixture or one-phase model is activated through the option *phases*.

hentrmax3 — Name of maximum height of P3 entrainment raster map

Name of the input raster map representing the distribution of the maximum height of entrainment of P3. The unit is metres. This parameter is ignored for the mixture or one-phase models (option *phases*).

hentrmax — Name of maximum height of entrainment raster map

Name of the input raster map representing the distribution of the maximum height of entrainment. The unit is metres. This parameter is used with the mixture and one-phase models (option *phases*). It can also be employed for the multi-phase model together with the option *rhentrmax1*.

0.5 rhentrmax1 — Ratio of maximum height of P1 entrainment

Without flag *-m*, dimensionless number in the range 0-1 indicating the ratio of maximum height of P1 entrainment out of the maximum total height of entrainment given by the option *hentrmax*. The remaining fraction is assigned to P3, whereas the entrainable height of P2 is set to zero. With flag *-m*, two or three comma-separated dimensionless numbers in the range 0-1 indicating the lower and the upper threshold for randomization or controlled variation (see option *sampling*). This parameter is ignored for the mixture or one-phase models (option *phases*), or when *hentrmax* is not given.

0.5	vhentrmax	Variation of maximum height of entrainment	

Two or three comma-separated dimensionless numbers indicating the lower and upper thresholds for the randomization or controlled variation (see option *sampling*) of the maximum height of entrainment between different model runs. The resulting number is multiplied with the values of the maximum height of entrainment given by the option *hentrmax*. *vhentrmax* is only considered with flag *-m*. If the parameter is not given, the maximum height of entrainment is kept constant throughout all model runs.

☰	density	Densities of the phases	*density=2700,1800,1000*

Parameter	Remarks	Range Default	Unit
Density of P1	For solid material, the grain density has to be used instead of the bulk density.	[>0,] 2700 for solid 2000 for mixture	kg/m^3
Density of P2	If applicable, the grain density has to be used instead of the bulk density. The density must not be higher than for P1	[>0,] 1800	kg/m^3
Density of P3	The density must not be higher than the density given for P2.	[>0,] 1000	kg/m^3

☰	friction	Friction parameters associated to each phase	*friction=35,20,0,0,0,0,0.05*

Parameter	Description	Range Default	Unit
Internal friction angle of P1	This value is neglected for fluid material. If the dynamic adaptation of friction is activated (see options *controls* and *dynfric*, the value is used to fit the exponential relationship. If the internal friction is lower than the basal friction, it is set equal to the basal friction. Note that, for fine solid material, friction angles >0 degrees may result in numerical	[0,<90] 35	degrees

	issues.		
Basal friction angle of P1	This value is neglected for fluid material. If the dynamic adaptation of friction is activated (see options *controls* and *dynfric*, the value is used to fit the exponential relationship. Note that, for fine solid material, friction angles >0 degrees may result in numerical issues.	[0,<90] 20	degrees
Turbulent friction of P1	The logarithm with base 10 of the turbulent friction has to be entered.	>[0,] 3	s^2/m
Internal friction angle of P2	This value has to be set zo zero for purely viscous material. If the dynamic adaptation of friction is activated (see options *controls* and *dynfric*, it is used to fit the exponential relationship. If the internal friction is lower than the basal friction, it is set equal to the basal friction. Note that, for fine solid material, friction angles >0 degrees may result in numerical issues.	[0,<90] 0	degrees
Basal friction angle of P2	This value has to be set zo zero for purely viscous material. If the dynamic adaptation of friction is activated (see options *controls* and *dynfric*, is used to fit the exponential relationship. Note that, for fine solid material, friction angles >0 degrees may result in numerical issues.	[0,<90] 0	degrees
Internal friction angle of P3	This value is neglected for fluid material.	[0,<90] 0	degrees
Basal friction angle of P3	This value is neglected for fluid material.	[0,<90] 0	degrees
Fluid friction number	The fluid friction number is related, but not identical to Manning's *n*, as the fluid friction number is used with the flow height instead of the hydraulic radius. It only applies to the fluid phase. If the dynamic adaptation of friction is activated (see options *controls* and *dynfric*, this value is used to fit the exponential relationship.	[≥ 0,] 0.05	-

viscosity | Viscosities of the phases | *viscosity=-9999,-9999,-3.0,-9999,-3.0,0.0*

Parameter	Description	Range Default	Unit
Kinematic viscosity of P1	The logarithm with base 10 of the viscosity has to be entered. Very low values have to be used for purely frictional materials. This parameter is neglected for solid material.	[,] -9999	m^2/s

Yield strength of P1	The yield strength of the material has to be entered. This parameter is neglected for solid material. If -9999 is provided for fine solid, the yield strength is computed automatically.	[≥ 0,] -9999	Pa
Kinematic viscosity of P2	The logarithm with base 10 of the viscosity has to be entered. Very low values have to be used for purely frictional materials.	[,] -3	m^2/s
Yield strength of P2	The yield strength of the material has to be entered. If -9999 is provided, the yield strength is computed automatically.	[≥ 0,] -9999	Pa
Kinematic viscosity of P3	The logarithm with base 10 of the viscosity has to be entered.	[,] -3	m^2/s
Yield strength of P3	The yield strength of the material has to be entered.	[≥ 0,] 0	Pa

☰ basal | Parameters governing the interaction of the flow with the basal surface | *basal=-7.0,0.05,0.333,0.0*

Parameter	Description	Range Default	Unit
Entrainment coefficient	Coefficient multiplied with the total kinetic energy of the flow (entrainment=1 in the option *control*) or the total flow momentum (entrainment=2) to derive the entrainment rate of basal material. The logarithm with base 10 of the entrainment coefficient has to be entered, except for 0 which means no entrainment.	[,] -7.0	-
Shear velocity coefficient	Coefficient in the range [>0,-1] describing the rate at which entrained material is included in the flowing mass, or deposited material is removed from the flowing mass. A value of 1 means that all the eroded material is immediately included, and all the deposited material is immediately removed. This parameter is only used with *entrainment=2* or *entrainment=3*, whereas any value can be provided with the other entrainment models.	[0,1] 0.05	-
Basal friction difference	Difference between the basal friction angles of the basal surface and the flow (weighted average of all phases considered in the simulation). Positive values generally result in deposition, whereas negative values generally result in entrainment of material. Also the basal friction difference is only considered with *entrainment=2* or *entrainment=3*.	[,] 0.0	degrees
Maximum water fraction of	Generally, deposition does not differentiate between the different phases, and the material depositing at	[0,1] 0.333	-

deposited material	a given time step will have the same phase fractions as the material remaining in the flow. However, it is useful for certain applications to deposit mainly the solid material, whereas the fluid material remains flowing. If the fluid fraction of the deposited material is larger than the maximum value defined through this parameter (range [0,1], the surplus of the fluid will remain in the flow. Consequently, a value of 0 means that fluid cannot deposit at all, whereas a value of 1 means no differentiation of the material at deposition. This parameter only has to be given for the multi-phase model (*model=m* of *model=s,s,f*). It is used with *entrainment=2*, *entrainment=3*, or *entrainment=4*, whereas it is neglected with *entrainment=1*.		
Stopping criterion	If the values 1 or 2 are provided for stopping (option *control*), the threshold of the total flow kinetic energy (1) or the total flow momentum (2) has to be provided. In both cases, this threshold has to be expressed as fraction of the maximum value reached during the flow (i.e. 0.05 would mean 5 per cent of the maximum). If a value of 3 is provided for stopping, the pressure threshold has to be specified. If stopping has been deactivated, this parameter is not used and any value may be entered.	1,2: [0,1] 3: [0,] 0.0	-

☰ **transformation** Transformation parameters, allowing for example the melting of ice *transformation=0.0,0.0,0.0*

Parameter	Description	Range Default	Unit
Transformation coefficient P1-P2	Coefficient multiplied with the total kinetic energy of the flow to derive the transformation rate between P1 and P2. The absolute value of the logarithm with base 10 of the coefficient has to be entered, except for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P2 to P1.	[,] 0.0	-
Transformation coefficient P1-P3	Coefficient multiplied with the total kinetic energy of the flow to derive the transformation rate between P1 and P3. The absolute value of the logarithm with base 10 of the coefficient has to be entered, except for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P3 to P1.	[,] 0.0	-
Transformation coefficient P2-P3	Coefficient multiplied with the total kinetic energy of the flow to derive the transformation rate between P1 and P2. The absolute value of the logarithm with base 10 of the coefficient has to be entered, except	[,] 0.0	-

for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P3 to P2.

dynfric

Dynamic adaptation of friction parameters *dynfric=0.0,-6.0,0.0*

Parameter	Description	Range Default	Unit
Minimum value of internal and basal friction	Lowest values of internal and basal friction angles that can be derived by the exponential relationship applied for dynamic adaptation. The minimum of the fluid friction is always zero.	[0,] 0.0	degrees
Kinetic energy coefficient	This coefficient is multiplied with the flow kinetic energy to reduce the friction parameters, following an exponential relationship. The logarithm of 10 of the coefficient has to be entered.	[,] -6.0	-
Phase fraction scaling exponent	Exponent for scaling of friction angles with fraction of phase, for example 0.0=no scaling (highly recommended for most cases), 1.0=linear scaling.	[0,] 0	-

slideparam

Parameters for mass point model for initial sliding *slidepar=0.0,0.0,0.0*

Parameter	Description	Range Default	Unit
Search radius	Radius around each cell from which the slope is computed. With 0, the slope is computed from the entire flow mass.	[0,] 0.0	m
Exponent for weighting	Exponent to be applied for down-weighting cell values at a large distance to the central cell. 0.0 means equal weighting of all cells, whereas 1.0 results in a linear down-weighting function, with a weight of 1.0 at the central raster cell and 0.0 at the outer limit of the search radius.	[0.0,] 0.0	-
Coefficient of deformation	The coefficient of deformation governs the degree of internal deformation of the sliding mass. Technically, the absolute value of the provided number denotes the fraction of flow height included in the calculation of the slope. With negative values, the flow height gradients are computed locally from the 3x3 cell enviroment, whereas with positive values, the seach radius and exponent for weighting are applied.	[-1.0,1.0] 0.0	-

special	Parameters further refining the simulation, recommended to be modified only by advanced users	*special=0.0,10,0.12,1,1,1,3,1,0.1, 1,1,1,1,1,0,0,0,1,1,1,10,0,1,1,1*	

Parameter	Description	Remarks	Default
C_{AD}	Ambient drag coefficient	Coefficient to be multiplied with the frontal surface and the velocity of the flow to derive air resistance.	0.0
N_{vm}	Virtual mass number		10
l_{vm}	Parameter related to the virtual mass coefficients	Numerical parameter	0.12
n_{vm}	Parameter related to the virtual mass coefficients	Numerical parameter	1
f_{vm}	Reduction factor for virtual mass coefficients	Increasing this value increases the numerical stability, but weakens the phase interactions	1
K_{Drag}	Mass flux parameter for drag	Parameter determined by the mixture mass flux per unit mixture density	1 m/s
m_{Drag}	Exponent for scaling of the fluid-like drag contributions to flow resistance	Positive number	3

n_{Drag}	Exponent for scaling P_{Drag} with solid fraction	Positive number close to 1	1
U_t	Terminal velocity	Highest possible velocity of an object falling through the flow	0.1 m/s
Re_p	Particle Reynolds number		1
j	Exponent for drag	1=linear, 2=quadratic	1
χ_{P1}	Vertical velocity distribution in P1	0=no shearing, 3=parabolic profile	1
χ_{P2}	Vertical velocity distribution in P2	0=no shearing, 3=parabolic profile	1
χ_{P3}	Vertical velocity distribution in P3	0=no shearing, 3=parabolic profile	1
ξ_{P1}	Vertical distribution of P1	Shape factor: 0=uniform, 3=parabolic	0
ξ_{P2}	Vertical distribution of P2	Shape factor: 0=uniform, 3=parabolic	0
ξ_{P3}	Vertical distribution of P3	Shape factor: 0=uniform, 3=parabolic	0
A_{P2P1}	Exponent for mobility of P2 at interface with P1	1 = linear decrease with increasing fraction of P1	1
A_{P3P1}	Exponent for mobility of	1 = linear decrease with	1

	P3 at interface with P1	increasing fraction of P1	
A_{P3P2}	Exponent for mobility of P3 at interface with P2	1 = linear decrease with increasing fraction of P2	1
r_y	Suitably chosen numerical parameter for regularization		10
J_v	Exponent for scaling of viscosity with fraction of phase	0=no scaling, 1=linear scaling	0
$K_{pmax,P1}$	Maximum value of passive earth pressure coefficient of P1	Low values (around 1) result in more compact flows and can be applied to imitate block sliding. This value is only relevant for solid material.	1
$K_{pmax,P2}$	Maximum value of passive earth pressure coefficient of P2	Low values (around 1) result in more compact flows and can be applied to imitate block sliding.	1

		This value is only relevant for solid material.	
$K_{pmax,P3}$	Maximum value of passive earth pressure coefficient of P3	Low values (around 1) result in more compact flows and can be applied to imitate block sliding. This value is only relevant for solid material.	1
TH	Treatment of input hydrographs	Identifier whether to reset the flow before adding material along the input hydrograph profiles (0); to impose input hydrographs upon the flow along the profiles (1); or to impose the entire discharge upon the centre of each hydrograph profile (2).	2
TD	Treatment of deceleration terms	0 = deceleration (friction, viscosity) terms are not allowed to lead to the change of	0

flow direction; 1 = no control of change of flow direction.

phi1 — Name of P1 internal friction angle raster map

Optional input raster map defining the spatial patterns of the internal friction angle of P1 in degrees. If this parameter is not given, and for all raster cells with value -9999, the global value of the internal friction angle is taken from the friction parameters (see option *friction*). If the internal friction is lower than the basal friction, it is set equal to the basal friction.

phi1 — Name of P1 internal friction angle raster map

tufri — Name of turbulent friction raster map

Optional input raster map defining the spatial patterns of the turbulent friction in s^2/m. If this parameter is not given, and for all raster cells with value -9999, the global value of the turbulent friction is taken from the friction parameters (see option *friction*). This parameter is only relevant for the mixture model (*model=x*.

delta1 — Name of P1 basal friction angle raster map

Optional input raster map defining the spatial patterns of the basal friction angle of P1 in degrees. If this parameter is not given, and for all raster cells with value -9999, the global value of the basal friction angle is taken from the friction parameters (see option *friction*).

delta2 — Name of P2 basal friction angle raster map

Optional input raster map defining the spatial patterns of the basal friction angle of P2 in degrees. If this parameter is not given, and for all raster cells with value -9999, the global value of the basal friction angle is taken from the friction parameters (see option *friction*).

flufri — Name of fluid friction number raster map

Optional input raster map defining the spatial patterns of the fluid friction number, which is related, but not identical to Manning's *n*, as the fluid friction number is used with the flow height instead of the hydraulic radius. This parameter is useful to consider the effects of roughness of the basal surface. If it is not given, and for all raster cells with value -9999, the global value of the fluid friction number is taken from the friction parameters (see option *friction*).

ny1 Name of P1 kinematic viscosity raster map

Optional input raster map defining the spatial patterns of the kinematic viscosity of P1, provided as logarithm (base 10). If this parameter is not given, and for all raster cells with value -9999, the global value of the kinematic viscosity is taken from the viscosity parameters (see option *viscosity*).

ny2 Name of P2 kinematic viscosity raster map

ny3 Name of P3 kinematic viscosity raster map

centr Name of entrainment coefficient raster map

Optional input raster map defining the spatial patterns of the empirical entrainment coefficient, provided as logarithm (base 10). If this parameter is not given, and for all raster cells with value -9999, the global value of the entrainment coefficient is taken from the option *basal*.

cvshear Name of shear velocity coefficient raster map

Optional input raster map defining the spatial patterns of the shear velocity coefficient, needed with the entrainment models 2 and 3 (see option *controls*). If this parameter is not given, and for all raster cells with value -9999, the global value of the shear velocity coefficient is taken from the option *basal*.

deltab Name of basal friction difference raster map

Optional input raster map defining the difference between the basal friction angles of the basal surface and the flow (weighted average of all phases considered in the simulation). Positive values generally result in deposition, whereas negative values generally result in entrainment of material. This parameter is needed with the entrainment models 2 and 3 (see option *controls*). If it is not given, and for all raster cells with value -9999, the global value of the basal friction difference is taken from the option *basal*.

ctrans12 Name of P1-P2 transformation coefficient raster map

Optional input raster map defining the spatial patterns of the empirical transformation coefficient between P1 and P2. The absolute value of the logarithm with base 10 of the coefficient has to be applied, except for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P2 to P1. If this parameter is not given, and for all raster cells with value -9999, the global value of the coefficient is taken from the transformation parameters (see option *transformation*).

ctrans13 Name of P1-P3 transformation coefficient raster map

Optional input raster map defining the spatial patterns of the empirical transformation coefficient between P1 and P3. The absolute value of the logarithm with base 10 of the coefficient has to be applied, except for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P3 to P1. If this parameter is not given, and for all raster cells with value -9999, the global value of the coefficient is taken from the transformation parameters (see option *transformation*).

ctrans23 Name of P2-P3 transformation coefficient raster map

Optional input raster map defining the spatial patterns of the empirical transformation coefficient between P2 and P3. The absolute value of the logarithm with base 10 of the coefficient has to be applied, except for 0 which means no transformation. The value has to be preceded by a minus sign for transformation from P3 to P2. If this parameter is not given, and for all raster cells with value -9999, the global value of the coefficient is taken from the transformation parameters (see option *transformation*).

ambdrag Name of ambient drag coefficient raster map

Optional input raster map defining the spatial patterns of the ambient drag coefficient. If this parameter is not given, and for all raster cells with value -9999, the global value of the ambient drag coefficient is taken from the special parameters (see option *special*).

impactarea Name of observed impact area raster map

Name of the input raster map defining the observed impact area of the flow. Areas with observed impact should be indicated by positive values, areas with no observed impact by 0, no data areas by negative values.

hdeposit Name of observed height of deposit raster map

Name of the input raster map of the height of the observed deposit of the flow. The unit is metres. Areas with no data should be indicated by negative values.

hydrograph Path(es) to input hydrograph text file(s)

Path to input hydrograph text file. This text file has to consist of seven columns:

- Time passed in seconds
- P1 discharge in cubic metres per second
- P1 flow velocity in metres per second
- P2 discharge in cubic metres per second (0 for one-phase or mixture model)
- P2 flow velocity in metres per second (0 for one-phase or mixture model)
- P3 discharge in cubic metres per second (0 for one-phase or mixture model)
- P3 flow velocity in metres per second (0 for one-phase or mixture model)

An unlimited number of hydrographs may be defined additionally to or instead of the helease height parameters (*hrelease*, *rhreleases*, *vhrelease*, *hrelease1*, *hrelease2*, and/or *hrelease3*).

hydrocoords Coordinates, lengths, and directions of hydrograph profile(s)

Each hydrograph profile is characterized by a sequence of four numbers:

- x coordinate of the first hydrograph (m);
- y coordinate of the first hydrograph (m);
- profile length of the first hydrograph (m);
- aspect of the first hydrograph,expressed as the flow direction in degrees, starting from east-west in anti-clockwise direction. -9999 can be entered to align the hydrograph perpendicular to the steepest slope.

This sequence is repeated for the second hydrograph and so on. All entries are separated by commas. If more pairs of coordinates than input hydrographs (option *hydrograph*) are given, the remaining pairs of coordinates define the locations of output hydrographs. If less pairs of coordinates than hydrographs are given, the model run(s) will crash.

thresholds Threshold parameters. *thresholds=0.1,10000,10000,0.001*

Parameter	Remarks	Default value	Unit
Minimum flow height for display	Flow heights below this value will not be displayed in the map layouts. The value specified has no influence on the simulation itself.	0.1	m
Minimum flow kinetic energy for display	Flow kinetic energies below this value will not be displayed in the map layouts. The value specified has no influence on the simulation itself.	10000	J
Minimum flow pressure for display	Flow pressures below this value will not be displayed in the map layouts. The value specified has no influence on the simulation itself.	10000	Pa
Minimum flow height for simulation	Only flow heights above this value will be considered in the simulation. Equal or lower values are set to 0 in order not to compromise numerical stability.	0.001	m

7 sampling Sampling strategy for multiple model runs *sampling=100*

This parameter is only applicable along with the flag *m*. It defines the type of sampling of the options *vhrelease*, *rhreleases*, *vhentrmax*, *rhentrmaxs*, *density*, *friction*, *viscosity*, *ambient*, *transformation*, and *special*: integer value larger than 0 = random sampling (the default; the value given denotes the number of model runs, i.e. the sample size), 0 = controlled sampling, integer value smaller than 0 = one-at-a-time sampling (the value given, if multiplied by -1, denotes the number of model runs, i.e. the sample size, for each parameter). With controlled or one-at-a-time sampling, the number of model runs is applied to generate a uniform distribution between the minimum and the maximum of each varied parameter. This means that the actual number of model runs is the product of the sample sizes associated to all the varied parameters.

☰ time Time interval for output and end time *time=10,300*

Two comma-separated floating point numbers. The first number indicates the real-time interval in seconds at which output information is displayed and written to files. The second numer indicates the real time in seconds after which the simulations stops.

☰ slomo | Factor for time scaling (slow motion) | *slomo=1*

Time scaling can be used for the simulation of very viscous, slow moving one-phase flows of fine solid or fluid material. Setting *slomo* to a value larger than 1 means that the time is not measured in seconds, but in seconds multiplied with the value provided. *slomo*=86400 would scale the time from seconds to days, resulting in output velocities of metres per day. However, please note that such changes of units are not indicated in the output data. The time scaling function is still experimental and needs more testing.

☰ cfl | Control of time step length | *cfl=0.4,0.001*

Two comma-separated floating point numbers governing time step length (optional):

- CFL criterion: a number which has to be lower than or equal to 0.5. The default value is 0.25.
- Time step length in seconds used if the CFL criterion is not applicable (e.g. start of the simulation). The default is 0.001 seconds. If initial sliding is used for the entire area of interest (see option *tslide*), it is recommended to set this parameter to something around 0.1-0.5 seconds, in order to avoid unnecessarily short time steps.

Note that higher values of the CFL criterion and the maximum time step length help to decrease the computational time, but lead to an increased risk of numerical failure. The default values serve well for most simulations, so that this parameter should only be provided in case of numerical issues.

☰ profile | Coordinates of profile vertices

Main flow path, given by the coordinates of an unlimited number of points along the flow path, separated by commas in the following sequence: x of first point, y of first point, x of second point and so on (all in metres). Note that the sequence of points has to strictly follow the course of the path, starting at the top and ending at the bottom, without juming forth and back.

☰ ctrlpoints | Coordinates of control points

An unlimited number of control points for which the main model output parameters are written to a text file. The control points are defined by comma-separated coordinates: x of first point, y of first point, x of second point and so on (all in metres).

reftime — Reference time of reach for each control point

Optional parameter containing comma-separated values indicating the observed travel times (times of reach) to each control point in seconds. The number of values has to correspond to the number of control points. This parameter is only used with the flag *m* and the option *ctrlpoints*. The ratio between the simulated time of reach and the observed time of reach is included in the evaluation.

0.5 phexagg — Factor for exaggeration of flow height — *phexagg=1.0*

Factor for exaggeration of flow height in profile plots (flag *v*). A value of 1 (the default) means no exaggeration. Exaggeration of flow height may be useful in case of a large ratio between flow length and flow height. However, the interpretation of profiles with exaggerated flow height requires specific care.

orthophoto — Background orthophoto for map plots

Optional path to an orthophoto of the investigation area, only useful along with flag *v*. If provided, this orthophoto will be used as background for the map display, otherwise a hillshade automatically generated from the elevation raster map will be used as background.

5 Results

r.avaflow 2.4 automatically creates maps, diagrams and animations, and offers interfaces for export.

The names of all output raster maps, folders and files start with the prefix defined by the option *prefix*. r.avaflow 2.4 produces a set of output GRASS raster maps stored in the active mapset as well as a set of asc, gif, png and txt files stored in subfolders of the folder *[prefix]_results*:

- Input files for model evaluation, parameter sensitivity analysis and parameter optimization with the software *AIMEC* are stored in the subfolder *[prefix]_aimec*.
- Exported ascii raster maps for use with other software packages are stored in the subfolder *[prefix]_ascii*.
- Text files are stored in the subfolder *[prefix]_files*.
- Output hydrographs, maps and derived animations, profile plots and derived animations and ROC Plots are stored in the subfolder *[prefix]_plots*.

The subfolders *[prefix]_hydrographs*, *[prefix]_maps* and *[prefix]_profiles*, including their content, are produced only if the flag *v* (visualization of model results) is specified. In addition, the subfolder *[prefix]_hydrograph* depends on the specification of the options *hydrograph* and *hydrocoords*. The folder *[prefix]_aimec* is only produced with the flag *m*, whereas the folder *[prefix]_profiles* is only produced without the flag *m*.

If the flag *v* was not specified when executing the model, the visualization can be performed afterwards by running the command. Note that, in this case, also the flags *a* and *t* may be provided at this point:

r.avaflow -a -v -t prefix=[prefix]

This step is only possible if the output raster maps still exist in the active GRASS mapset (i.e. if the flag *k* was activated when running the simulation), and the directory *[prefix]_results* has remained unchanged since the original computation. However, advanced users may manually modify the content of the text file *[prefix]_documentation.txt*.

GRASS raster maps	Result raster maps stored in the active GRASS Mapset.	*Active GRASS Mapset*

r.avaflow 2.4 produces the following output raster maps which are, however, stored permanently only with the flag *k*:

- *[prefix]_hflow[timestep]*: flow height for the time step *[timestep]* in metres, one map is produced for each time step;
- *[prefix]_vflowx[timestep]*: flow velocity in x direction for the time step *[timestep]* in m/s, one map is produced for each time step (only with flag *a*);
- *[prefix]_vflowy[timestep]*: flow velocity in y direction for the time step *[timestep]* in m/s, one map is produced for each time step (only with flag *a*);
- *[prefix]_vflow[timestep]*: flow velocity for the time step *[timestep]* in m/s, one map is produced for each time step (only with flag *a*);
- *[prefix]_tflow[timestep]*: flow kinetic energy for the time step *[timestep]* in J, one map is produced for each time step (only with flag *a*);
- *[prefix]_pflow[timestep]*: flow pressure for the time step *[timestep]* in Pa, one map is produced for each time step (only with flag *a*);
- *[prefix]_basechange[timestep]*: entrained or deposited height (change of basal surface) at the time step *[timestep]* in m, one map is produced for each time step - deposition is indicated by positive values, entrainment by negative values.
- *[prefix]_tsun[timestep]*: impact wave or tsunami height of P3 for the time step *[timestep]* in m. One map is produced for each time step, but only for the multi-phase model with flag *t*.
- *[prefix]_vfront*: frontal flow velocity in m/s (weigthed average of all phases).
- *[prefix]_r1front*: volumetric fraction of P1 at the flow front.
- *[prefix]_r3front*: volumetric fraction of P3 at the flow front.
- *[prefix]_r1max*: volumetric fraction of P1 at the time of maximum momentum throughout the entire simulation.
- *[prefix]_r3max*: volumetric fraction of P3 at the time of maximum momentum throughout the entire simulation.
- *[prefix]_vhmax*: flow velocity in m/s (weigthed average of all phases) at the time of maximum momentum throughout the entire simulation.
- *[prefix]_treach*: time of reach. It shows the time in seconds after which the flow first reaches each raster cell, using the mimimum flow height for display (*thresholds* parameters) as the

criterion. Time of reach is also shown in the control points output files.

The number of the phase each map refers to is inserted before *[timestep]*. Maps without that number refer to the entire flow. Maxima of flow heights, flow kinetic energies, and flow pressures over the entire simulation are provided with the suffix *_max*. The final values of the flow heights and entrained/deposited heights are provided with the suffix *_fin*. *[prefix]_elev_mod* denotes the elevation in metres at the end of the simulated event (initial elevation corrected for entrainment and deposition.

With the flag *m*, mainly three output maps are produced, representing the impact indicator indices for maximum flow height (*[prefix]_iii_hflow.png*), for maximum flow kinetic energy (*[prefix]_iii_tflow.png*) and for maximum flow pressure (*[prefix]_iii_pflow.png*). The impact indicator index represents the number of simulation runs where the maximum value of the considered parameter computed for a raster cell is equal or larger than the corresponding impact threshold (option *imparam*) divided by the total number of successful simulation runs.

In addition to the output maps, some preprocessed input maps are produced (with the flag *m* they may amount to a large number, depending on the number of model runs). Most output maps and preprocessed input maps are also stored as ascii rasters in the folder *[prefix]_ascii*.

AIMEC input	Files needed for post-processing with the tool AIMEC.	*[prefix]_aimec*

With the flag *m*, this folder contains the automatically generated input folders and files for the evaluation, parameter sensitivity analysis and optimization tool AIMEC. In order to enable applying this tool to the results produced by r.avaflow 2.4, the content and structure of this folder should not be manipulated manually.

Ascii rasters	Ascii rasters of result files which can be exported for use in other software packages.	*[prefix]_ascii*

Most of the output and preprocessed input raster maps stored in the active GRASS mapset are also exported as ascii rasters in order to be accessible with other software packages. The naming scheme follows the one used for the GRASS raster maps.

Text files	Text files describing the simulation and its results.	*[prefix]_files*

The following text files summarize the main parameters of the simulation or are needed for the construction of the maps and plots in the folders *[prefix]_hydrographs*, *[prefix]_maps* and *[prefix]_profiles*:

- *[prefix]_averages.txt*: portion of area of interest with an observed deposit and average of impact indicator index (with flag *m* only).
- *[prefix]_ctrlpoints.txt*: IDs, coordinates and selected result raster values (*hmax* = maximum flow height and the flow heights of all time steps, and *hentrmax* = maximum height of entrainment and the heights of entrainment of all time steps without flag *m*; *iii* = impact

indicator index according to flow height with flag *m*) of all control points specified by the option *ctrlpoints*.

- *[prefix]_directions[phase].txt*: Velocities in x and y direction. This file is needed for the construction of arrows indicating flow direction and velocity in the maps stored in the folder *[prefix]_maps* (not applicable with flag *m*).
- *[prefix]_documentation.txt*: summary of the flags and parameters specified when starting the simulation as well as some further parameters needed for constructing the maps and plots in the folders *[prefix]_hydrographs*, *[prefix]_maps* and *[prefix]_profiles* when running r.avaflow 2.4 with the flag *v* only.
- *[prefix]_hydinfo[hydrograph].txt*, where [hydrograph] stands for the number of the hydrograph: output hydrograph data for each time step (not applicable with flag *m*). *T* = time passed; *H* = flow height; *V* = flow velocity; *E* = height of entrainment/deposition; *Q* = discharge. Numbers at the end of the headers indicate the phase the column refers to.
- *[prefix]_hydprofiles.txt*: x and y coordinates of the centre (*xC*, *yC*) and terminal points (*x1*, *y1*, *x2*, *y2*) of all hydrographs. Hydrograph IDs starting with *I* indicate input hydrograps, hydrograph IDs starting with *O* indicate output hydrographs. This file is not produced with flag *m*. The hydrograph profiles along with the IDs are shown in the maps stored in the folder *[prefix]_maps*.
- *[prefix]_nout1.txt*: two-line file, in the first line displaying the number of time steps, in the second line the model success (1=success; 2=failure, usually for numerical reasons). With the flag *m*, one file is produced for each model run, the number of the model run is added to the file name as suffix.
- *[prefix]_param.txt*: Raw input parameter file created by r.avaflow.py (only without flag *m*).
- *[prefix]_params.txt*: This file is produced along with the flag *m* only. It summarizes the input parameters for all model runs.
- *[prefix]_profile.txt*: Profile data illustrating elevation, flow height, depth of entrainment and deposition, flow velocity, flow kinetic energy and flow pressure at all time steps along the pre-defined profile (option *profile*). This file is needed for the construction of the profiles stored in the folder *[prefix]_profiles* (not applicable with flag *m*).
- *[prefix]_summary.txt*: summary file of the simulation, each line documenting the main parameters of each time step. With the flag *m*, one summary file is produced for each model run, the number of the model run is added as suffix. The content of the summary file(s) is largely identical to the screen output during routing of the flow. With *model=1* the meanings of the column headers are: *nout*: number of time step; *nsum*: number of internal time step of the routing algorithm (these time steps are very short in order to ensure numerical stability); *cfl*: indicator for numerical stability (value should be smaller than 0.5); *tdim*: duration of one internal time step (1/100 s); *tsum*: time passed since start of the flow (s); *dmax*: maximum flow depth at time step (m); *vmax*: maximum flow velocity at time step (m/s); *volume*: flow volume at time step (cubic metres in the summary file; 1000s of cubic metres on the screen output); ekin: kinetic energy summed up over the entire flow (J in the summary file; MJ on the screen output). Numbers at the end of the headers indicate the phase the column refers to.
- *[prefix]_time.txt*: computational time (excluding visualization) in seconds.
- *[prefix]_evaluation.txt*: This file includes some model outcomes, in particular the evaluation scores derived from the comparison of the model results with the observed impact area (option *impactarea*) or the observed height of deposit (option *hdeposit*). Thereby, the modelled impact area is defined by the area where the maximum flow height is equal or larger than the threshold defined by the option *imparam*. The modelled deposit is defined by the area where the flow height at the end of the simulation is equal or larger than the threshold defined by the option *imparam*.

Plots and animations	Graphic representation of the main simulation results.	*[prefix]_plots*

- Hydrograph plots are produced for all input and output hydrographs (flow height and discharge) defined by the option *hydrocoords*, using the data stored in the files *[prefix]_hydinfo[hydrograph].txt* and *[prefix]_hydprofiles.txt*. The hydrograph profiles along with the IDs are shown in the maps stored in the folder *[prefix]_maps*. This file is not produced with flag *m*.
- Without the flag *m*, colour layouts of the simulated flow height (*[prefix]_hflow[timestep].png*), flow kinetic energy (*[prefix]_tflow[timestep].png*, only with flag *a*), flow pressure (*[prefix]_pflow[timestep].png*, only with flag *a*), height of entrainment/deposition, if applicable (*[prefix]_basechange[timestep].png*), and impact wave or tsunami height (*[prefix]_tsun[timestep].png*, , only with flag *t*) are drawn for each time step *[timestep]*. They are stored in the sub-folder *[prefix]_maps_timesteps*. The release area and/or release hydrographs (option *hydrocoords*) are displayed as well as the flow path (option *profile*), the observed impact area (option *impactarea*), the observed deposit (option *hdeposit*) and the locations and IDs of the control points (option *ctrlpoints*), if specified. If the option *orthophoto* is not provided, a hillshade derived from the elevation raster map is shown as background. Otherwise, the orthophoto is shown as background.
- Colour layouts of the maximum values (suffix *_max*) of flow height, flow kinetic energy and flow pressure (both only with flag *a*), and impact wave or tsunami height (only with flag *t*) over the entire simulation, and a map of the final height of entrainment and deposition, if applicable (suffix *_fin*), as well as a map of the time of reach are drawn at the top level of the folder along with animated gifs illustrating the evolution of the flow parameters during the simulation. The suffix *c* in the file names of the gifs refers to compressed versions. With the flag *m*, only four maps are drawn, illustrating the impact indicator indices for maximum flow height (*[prefix]_iii_hflow.png*), for maximum flow kinetic energy (*[prefix]_iii_tflow.png*), and for maximum flow pressure (*[prefix]_iii_pflow.png*) as well as the deposition indicator index for maximum flow height (*[prefix]_dii_hflow.png*).
- Without the flag *m*, series of vertical profiles following the flow path (option *flowpath*) are constructed, illustrating the flow height (put on top of the terrain) along with bar plots representing flow velocity (*[prefix]_vflow[timestep].png*), flow kinetic energy (*[prefix]_tflow[timestep].png*) and flow pressure (*[prefix]_pflow[timestep].png*). They are stored in the sub-folder *[prefix]_profiles_timesteps*. Animated gifs visualizing the complete sequences of profiles are stored at the top level of the folder. The suffix *c* in the file names of the gifs refers to compressed versions.
- In the case of multiple model runs (flag *m* and option *cores*) and the availability of a map of the observed deposit (option *hdeposit*), 2 ROC Plots relating impact indicator index map to the observed deposition area are created: one plot includes the entire study area (*[prefix]_roc.png*), the second one uses a normalized area of true negative predictions (*[prefix]_rocn.png*). The area under the curve AUCROC is displayed as the key indicator for the prediction quality.

Two additional text files, *aucroc.txt* and *aucrocn.txt*, are produced along with the ROC Plots. These files are stored directly in the working directory and display the prefix of the computation along with the corresponding value of AUCROC: in *aucroc.txt*, the value refers to the entire area of interest, while in *aucrocn.txt* the true negative area is normalized. These two files are not overwritten during the next execution of r.avaflow. Instead, a new line is added each time r.avaflow is run with the flag *v*. This facilitates the analysis of the AUCROC values in case of multiple executions of r.avaflow 2.4.

Please cite this site and its content as: Mergili, M., 2014-2020. r.avaflow - The mass flow simulation tool. r.avaflow 2.4 User manual. https://www.avaflow.org/manual.php

Responsible for this web site: Martin Mergili, Graz, Austria.
Contact: martin.mergili@uni-graz.at. Latest update on 16 May 2022.