New RAMMS Version 1.6

Content

1 General ... 2
  1.1 Restart RAMMS ... 2
  1.2 Additional Preferences ... 2
2 Edit the input file ... 2
  2.1 Draw new polygon shapefiles ... 2
  2.2 Delete points of a polygon when drawing ... 2
  2.3 Change color of polygon shapefiles ... 3
  2.4 Convert polygon shapefile to calculation domain ... 3
  2.5 Import line shapefiles ... 4
  2.6 File tree – remove visualization button ... 4
3 Running a simulation ... 5
  3.1 Improved numerical scheme ... 5
  3.2 Curvature effect implemented ... 6
  3.3 Obstactle / NoFlux feature implemented ... 6
  3.4 Cohesion implemented ... 7
4 Visualization and analysis of the results ... 9
  4.1 Automatic export of ASCII files after a simulation ... 9
  4.2 Generation of ASCII files from multiple output files ... 9
  4.3 Velocity arrows ... 10
  4.4 Summary of moving mass ... 11
  4.5 Time plots and line profile plots ... 12
5 RAMMS::Viewer ... 13
6 References ... 13

1 General

1.1 Restart RAMMS

Restart RAMMS with the menu ‘*Track* ➔ *Restart RAMMS*’.

Note: Don’t use this function after a RAMMS update when asked to restart RAMMS. After an update close RAMMS via ‘Track ➔ *Exit’, Ctrl+Q or the general window close button and start RAMMS again.*

1.2 Additional Preferences

Open the additional preferences with the button (tooltip “Additional Preferences”). The button is located in the vertical toolbar and has formerly been used to open the colorbar properties. It’s also possible to open them with ‘*Help* ➔ *Advanced…* ➔ *Additional Preferences* ➔ *Edit*”. Colorbar properties may still be opened via the menu ‘*Edit* ➔ *Colorbar Properties*’.

2 Edit the input file

2.1 Draw new polygon shapefiles

To draw a new polygon shapefile use the “Draw New Polygon Shapefile” button or the menu ‘*Input* ➔ *Polygon Shapefile…* ➔ *Draw New Polygon Shapefile*’. The menu ‘*Input* ➔ *Polygon Shapefile…’* offers as well an option to load an existing polygon shapefile. This functionality is useful to draw noflux features like dams or houses, additional MuXi areas or the outline of the event.

2.2 Delete points of a polygon when drawing

When drawing a release area, a forest area (::AVALANCHE only), a polygon shapefile or a calculation domain it is possible to delete the last created point. By pressing the mouse wheel (middle mouse button) always the last created point is deleted. *Example: Delete the last three points by pressing the mouse wheel three times.*

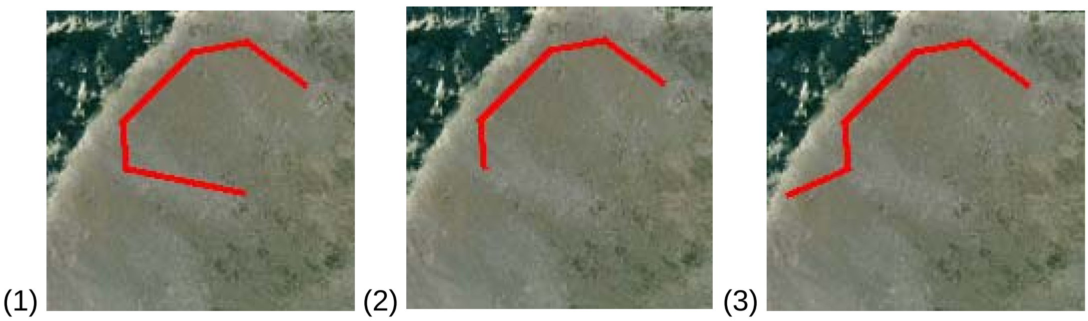


Fig. 1: (1) Draw the release area (in this case counterclockwise), (2) delete the last created point by pressing the mouse wheel and (3) continue drawing the release area.

2.3 Change color of polygon shapefiles

Change the color of imported or drawn shapefiles in the ‘*Additional Preferences*’. Open the ‘*Additional Preferences’* window with the button  in the vertical toolbar or via the menu ‘*Help ➔ Advanced… ➔ Additional Preferences… ➔ Edit*’. The SHAPEFILE_COLOR variable holds values from 0 to 255 (standard value = 100).

```
ARROW_THICK 2
SHAPEFILE_COLOR 50 ( 0 - 255 )
END
```

Fig. 2: Change the color of the shapefile in the ‘*Additional Preferences’*.

The numbers correspond to the currently selected color table and range from 0 (lower limit) to 255 (upper limit). Regarding the standard color table “Rainbow” (no. 34) colors range from violet (0) to red (255).

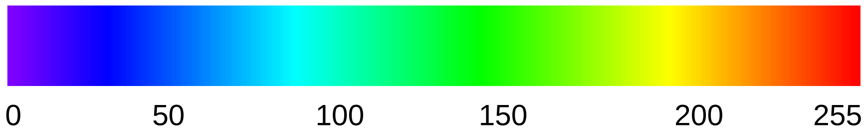


Fig. 3: Shapefile color numbers considering the standard color table "Rainbow".

The color table may be changed as well in the *‘Additional Preferences’* via the variable COLORTABLENR. Color table numbers and names are listed at the end of the additional preferences and may be viewed via the menu ‘*Help ➔ Advanced… ➔ ColorTables… ➔ View Available ColorTables*’.

2.4 Convert polygon shapefile to calculation domain

Convert a shapefile that has been created for example in GIS into a calculation domain with the menu ‘*GIS ➔ Convert Polygon Shapefile… ➔ to RAMMS Domain Shapefile*’. An Open File dialog pops up. Choose the polygon shapefile that should be converted into a calculation domain and name it accordingly. RAMMS converts the polygon shapefile into a calculation domain file, saves the file with the given name in the project folder and loads the calculation domain into the project.

2.5 Import line shapefiles

Import line shapefiles with the *'Import Polygon Shapefile'* button , via the menu '*Input ➔ Polygon Shapefile… ➔ Load Existing Polygon Shapefile'* or via the menu '*GIS ➔ Import Polygon Shapefile'*.

Fig. 4: Small blue dots indicate the points of the imported line shapefile.

2.6 File tree – remove visualization button

At the right side of the file tree two buttons can be found. The *'Refresh Tree'* button  updates the tree view. The *'Remove Visualization'* button  removes the currently displayed visualization. Select the desired entry in the tree view to reactivate the visualization.

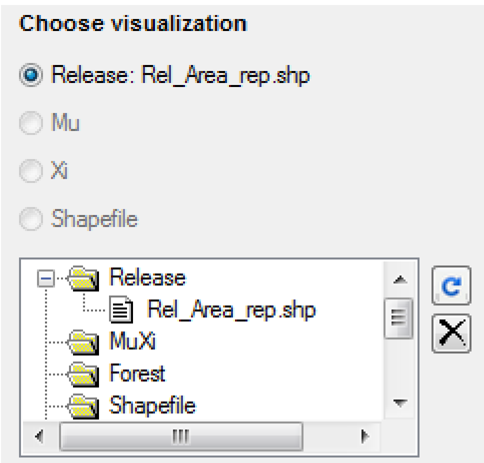


Fig. 5: File tree with the two buttons *'Refresh Tree'* and *'Remove Visualization'*.

3 Running a simulation

3.1 Improved numerical scheme

In all RAMMS versions up to now, an ENO (Essentially Non-Oscillatory) scheme was used to numerically solve the governing differential equations (Christen et al., 2010). However, the numerical solution was implemented on strictly orthogonal grids. This improves computational speed, but introduces numerical instabilities especially in steep terrain. The new version of RAMMS uses the same second order ENO scheme, but now on general quadrilateral grid. This new scheme improves numerical stability, but slows the computational speed somewhat. The introduction of this stable ENO scheme allows us to use lower H_cutoff values minimizing mass loss during calculations. The standard value of H_cutoff is 0.000001 m.

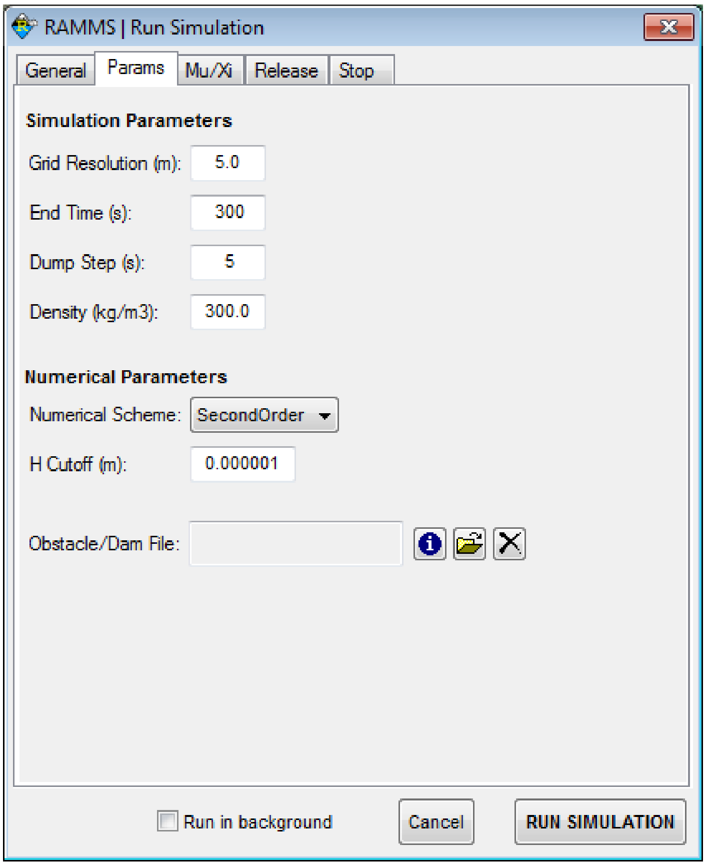


Fig. 6: The Params tab of the Run Simulation window holds the H Cutoff value (standard value = 0.000001 m).

3.2 Curvature effect implemented

In the new version, the normal force N now includes centrifugal forces arising from the terrain curvature. We use the method proposed by Fischer et al. (2012) which was specifically developed for RAMMS. The centrifugal acceleration f is both a function of the avalanche velocity and terrain curvature. The acceleration is calculated according to

$$f = \vec{u}\mathbf{K}\vec{u}^T$$

where $\vec{u}$ is the vector $\vec{u} = (u, v)$, consisting of the avalanche velocity in the $x-$ and $y-$ directions. The matrix $\mathbf{K}$ describes the track curvature in all directions, including the track "twist". The centrifugal force is then

$$F = \rho h f$$

which is added to the normal force N. Typically this increases the friction, causing the avalanche to slow down in tortuous and twisted flow paths. It can change the location of the deposition once the flow leaves the gully.

Curvature may be activated/deactivated via the menu *'Help ➔ Advanced… ➔ Curvature'*.

See http://www.sciencedirect.com/science/article/pii/S0165232X12000183#.

3.3 Obstacle / NoFlux feature implemented

Draw areas where the event should not flow through, e.g. dams or houses. The flow will be deflected. Draw the required objects with the *'Draw New Polygon Shapefile'* function and add the obstacles/noflux features in the Params tab (Obstacle/Dam File:) of the Run Simulation window.

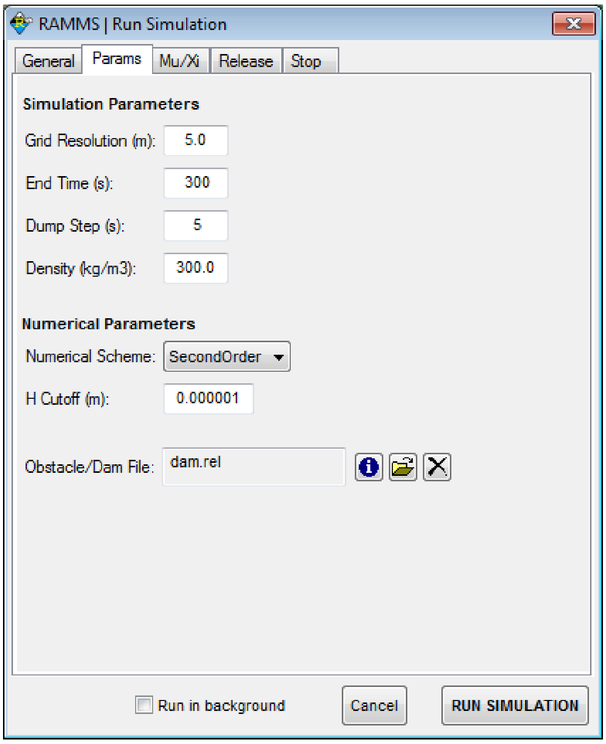


Fig. 7: Add obstacles and noflux features (e.g. dams) in the Params tab.

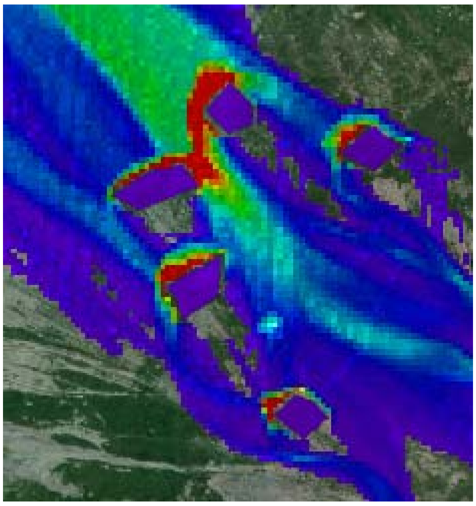


Fig. 8: The event is deflected by the noflux obstacles (violet polygons).

3.4 Cohesion implemented

It is well known that flowing snow and muddy debris are “cohesive” materials. In the first versions of RAMMS, the frictional properties of the flowing material were governed by a simple, but robust, Voellmy-type flow law without cohesion:

$$S = \mu N + \frac{\rho g U^2}{\xi}.$$

RAMMS has several possibilities to select the μ and ξ values (see sections 4.2 and 4.5 in user manual). In the new version the basic Voellmy equation has been modified to include cohension:

$$S = \mu N + (1-\mu)C - (1-\mu)C \exp\left(-N/C\right) + \frac{\rho g U^2}{\xi}$$

where C is the cohesion of the flowing material. Unlike a standard Mohr-Coulomb type relation this formula ensures that $S \rightarrow 0$ when both $N \rightarrow 0$ and $U \rightarrow 0$. It increases the shear stress and therefore causes the avalanche or debris flow to stop earlier, depending on the value of C.

This formula has been established using chute experiments with flowing snow (Platzer et al., 2007) and real scale experiments with debris flows in Illgraben (VS). Snow has different cohesive properties depending on snow temperature. Wet snow avalanches have higher cohesion values; dry snow avalanches have lower cohesion values.

Cohesion can help reduce spurious numerical diffusion in runout zones, providing a clearer delineation of the deposition zone.

Cohesion values (unit Pascal) may be entered in the Mu/Xi tab of the Run Simulation window. Recommended values may be found in the following table.

Tab. 1: Recommended values of cohesion.

::AVALANCHE	dry	0 – 100 Pa
	wet	100 – 300 Pa
::DEBRIS_FLOW	0 – 2000 Pa	

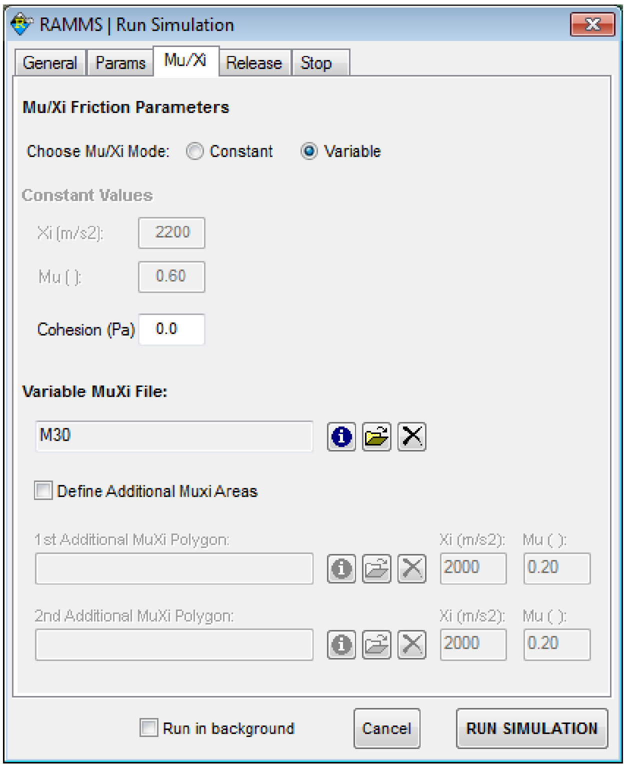


Fig. 9: Run Simulation window with value field for cohesion (Pa).

See http://www.sciencedirect.com/science/article/pii/S0165232X07000808 and http://onlinelibrary.wiley.com/doi/10.1029/2006GL028670/abstract for more details.

4 Visualization and analysis of the results

4.1 Automatic export of ASCII files after a simulation

When RAMMS finishes a simulation and automatically opens the results for visualization four ASCII files are exported:

- <project_name>_MaxHeight.asc: Maximal flow height
- <project_name>_MaxVelocity.asc: Maximal flow velocity
- <project_name>_MaxPressure.asc: Maximal flow pressure
- <project_name>_Deposition.asc: Deposition (at the last dump step)

The four ASCII files are exported to the project folder which may be opened via the menu '*Project ➔ Open Project Folder (Windows Explorer)'.*

4.2 Generation of ASCII files from multiple output files

To export the above listed ASCII files (maximal flow height, maximal flow velocity, maximal flow pressure and deposition) for one or more simulations select the menu '*Track ➔ New… ➔ Export ASCII Files From Multi Simulations (Batch)*'. Choose the simulation output files for which the ASCII files should be generated, multiple selections are possible.

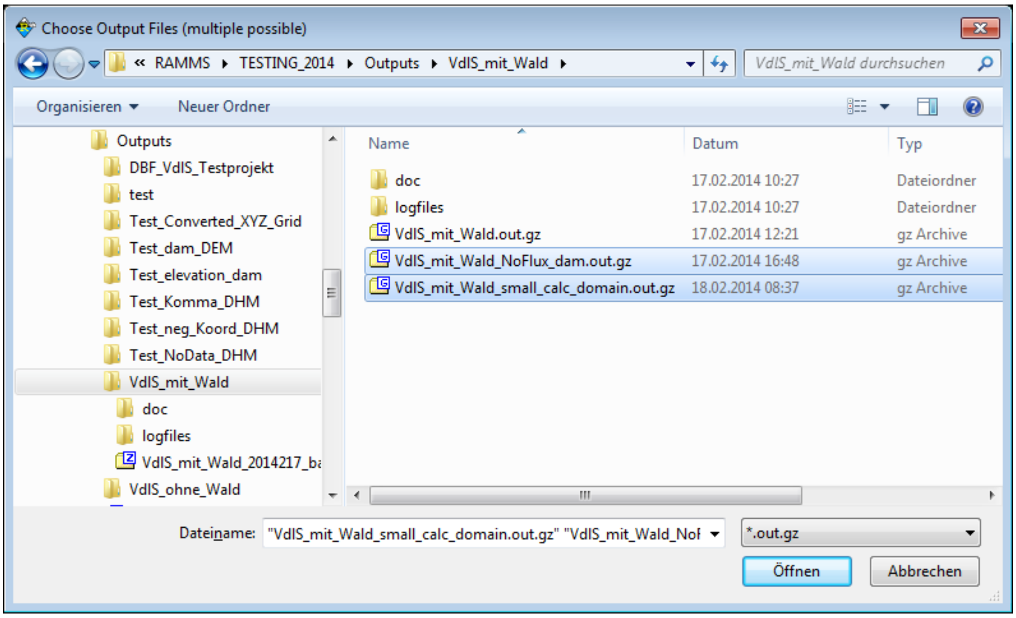


Fig. 10: Selection of simulation output files for which the ASCII files should be generated.

After the simulation output files of a folder have been selected, RAMMS asks if the selection shall be continued in other folders.

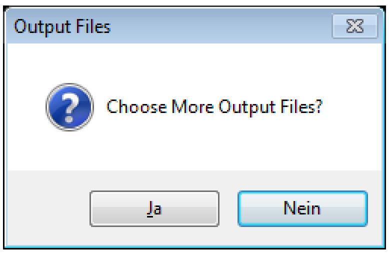


Fig. 11: Press Yes if you want to continue the selection of simulation output files for the generation of the ASCII files.

4.3 Velocity arrows

Display velocity arrows with the *'Show Velocity Arrow'* button or the menu '*Show ➔ Show Velocity Arrow*'. The arrows show the direction (in x- and y-direction) and the velocity intensity (length of the arrow, no absolute velocity values are displayed) of the flow in a cell. The arrows change in size and lengths when zooming into the visualization. Visualization is best with high zoom levels.

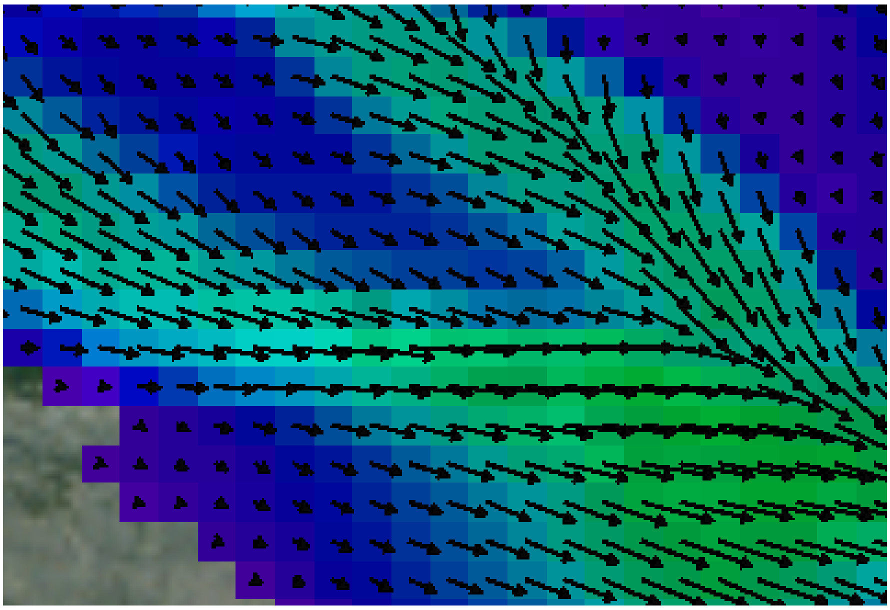


Fig. 12: Velocity arrows show velocity direction and intensity (zoom level 800 %).

Color, head-size, thickness and length can be changed in the 'Additional Preferences' with the keywords

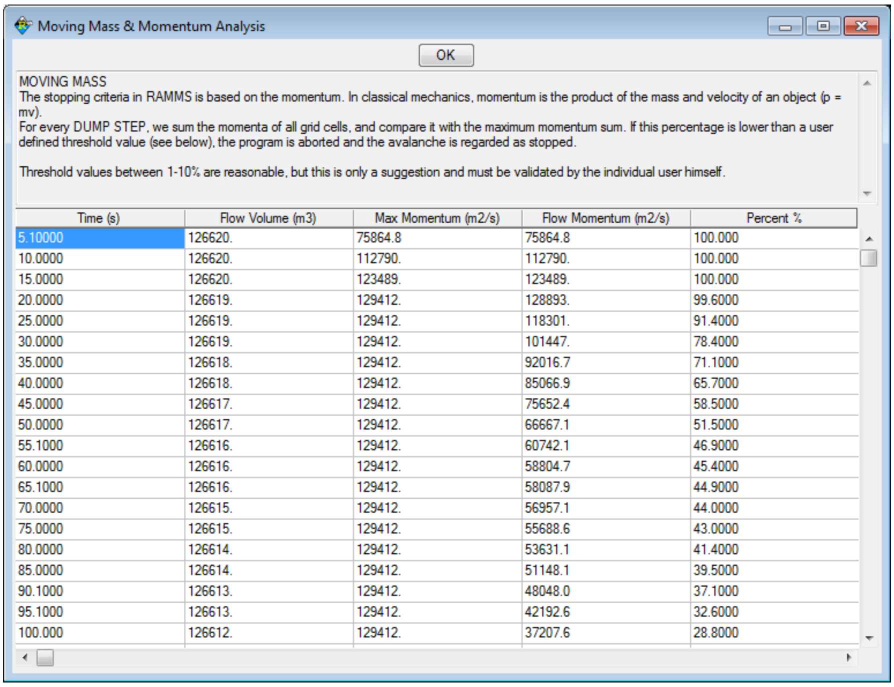
- ARROW_COLOR (black, red, green, yellow, etc.)
- ARROW_HEADSIZE
- ARROW_LENGTHSCALE
- ARROW_THICK

Use ARROW_FACTOR, if you want to reduce the number of arrows. ARROW_FACTOR = 1 displays every single arrow, ARROW_FACTOR = 2 only every second arrow, etc. (default value is 1.0).

Use ARROW_MINVALUE to define the minimum velocity value of the velocity arrows (default value is 0.1 m/s).

4.4 Summary of moving mass

The menu '*Results* ➔ *Summary of Moving Mass'* opens the summary of moving mass. Two windows pop up: a table with flow volume (m^3), maximal momentum (m^2/s), flow momentum (m^2/s) and percent of moving mass (%) per time step(s) as well as a plot with the two graphs moving momentum and flow volume over time.



Time (s)	Flow Volume (m3)	Max Momentum (m2/s)	Flow Momentum (m2/s)	Percent %
5.10000	126620.	75864.8	75864.8	100.000
10.0000	126620.	112790.	112790.	100.000
15.0000	126620.	123489.	123489.	100.000
20.0000	126619.	129412.	128893.	99.6000
25.0000	126619.	129412.	118301.	91.4000
30.0000	126619.	129412.	101447.	78.4000
35.0000	126618.	129412.	92016.7	71.1000
40.0000	126618.	129412.	85066.9	65.7000
45.0000	126617.	129412.	75652.4	58.5000
50.0000	126617.	129412.	66667.1	51.5000
55.1000	126616.	129412.	60742.1	46.9000
60.0000	126616.	129412.	58804.7	45.4000
65.1000	126616.	129412.	58087.9	44.9000
70.0000	126615.	129412.	56957.1	44.0000
75.0000	126615.	129412.	55688.6	43.0000
80.0000	126614.	129412.	53631.1	41.4000
85.0000	126614.	129412.	51148.1	39.5000
90.1000	126613.	129412.	48048.0	37.1000
95.1000	126613.	129412.	42192.6	32.6000
100.000	126612.	129412.	37207.6	28.8000

Fig. 13: Table with mass flow per time step.

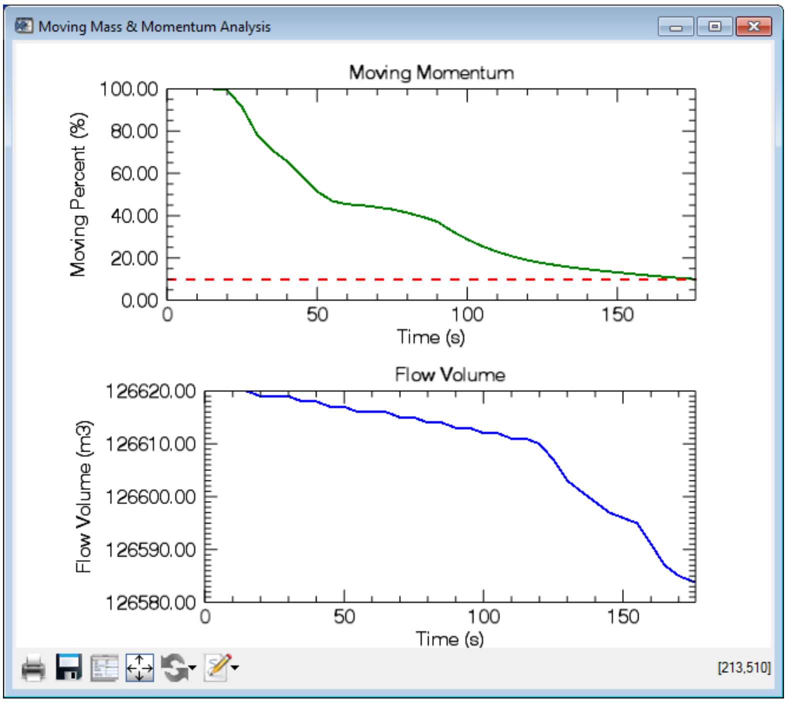


Fig. 14: Graphs with moving momentum and flow volume over time.

4.5 Time plots and line profile plots

The design of time plot and line profile plot windows changed. Title (simulation name) and legend (name of the plotted result) are created automatically for the graph.

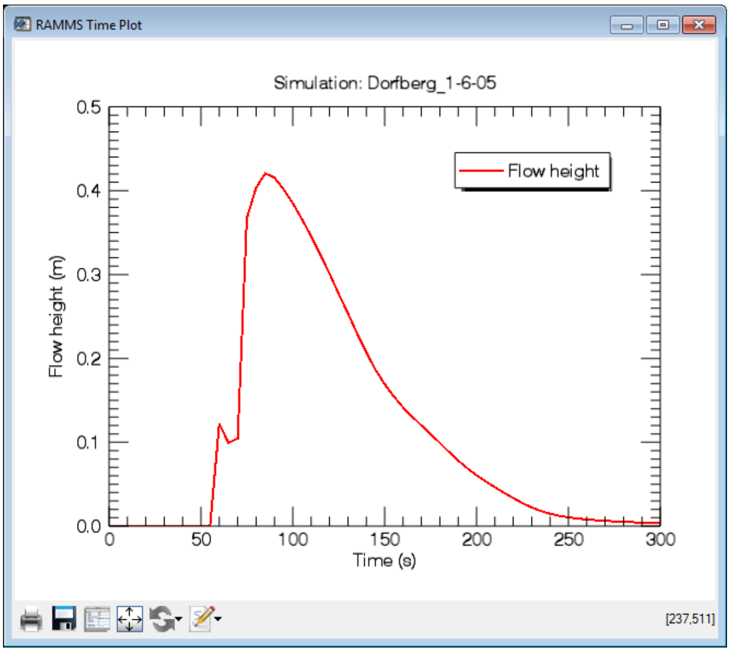


Fig. 15: Time plot window.

With the buttons at the left bottom of the window it’s possible to print, save and edit the graph:

- Print the graph with the “Print…” button .
- Save the graph in different formats with the “Save as…” button .
- Open the the properties window of the plot with the “Properties…” button  and modify the property values if desired.
- The “Reset axis range…” button  resets the axis range according to the currently displayed result values.
- Edit the graph with the options offered by the “Edit” button .
- Add text, forms or a legend to the graph with the “Insert” button .

In addition to the result values (gray) the line profile plot displays the terrain elevation along the line profile (green curve) and the result values added to the terrain (red curve).

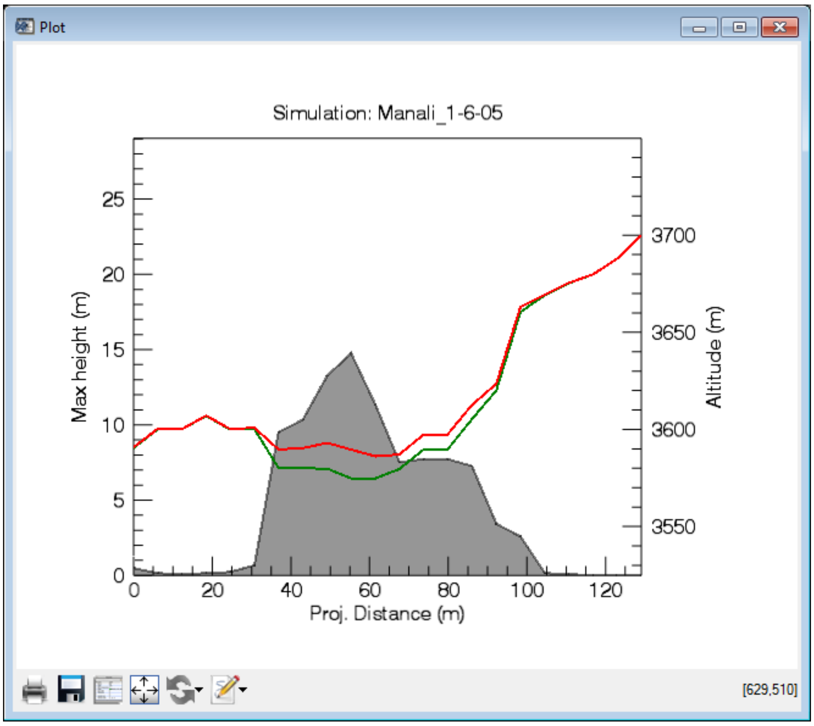


Fig. 16: Line profile plot window.

5 RAMMS::Viewer

New RAMMS::Viewer available now! Did your RAMMS license expire? Do you need to look at some old RAMMS results? This is now possible with the new RAMMS::Viewer.

Download the Viewer from our website:
http://ramms.slf.ch/ramms/index.php?option=com_content&view=article&id=53&Itemid=70

You are able to look at results, export images, GIF animations, ASCII files and shapefiles. You are not able to run new simulations. Please order a new license, if you need to do new simulations for a project.

6 References

Fischer, J.T., Kowalski, J. and Pudasaini, S.P., 2012. Topographic curvature effects in applied avalanche modeling. Cold Regions Science and Technology, 74-75: 21-30.

Platzer, K., Bartelt, P. and Kern, M., 2007. Measurements of dense snow avalanche basal shear to normal stress ratios (S/N). Geophysical Research Letters, 34(7).

Platzer, K.; Bartelt, P.; Jaedicke, C., 2007: Basal shear and normal stresses of dry and wet snow avalanches after a slope deviation. Cold Reg. Sci. Technol. 49: 11-25.