RAMMS::DEBRISFLOW User Manual

RAMMS

rapid mass movements simulation

A numerical model for debris flows in research and practice

User Manual v1.7.0

Debris Flow

WSL-Institut für Schnee- und Lawinenforschung SLF

WSL Institut pour l‘étude de la neige et des avalanches SLF

WSL Instituto per lo studio della neve e delle valanghe SLF

WSL Institute for Snow and Avalanche Research SLF

SLF

WSL

ETH

Eidgenössische Technische Hochschule Zürich

Swiss Federal Institute of Technology Zurich

Title picture: Debris Flow at Illgraben, WSL test site

Contributors (alphabetical order)

SLF/WSL:
Perry Bartelt
Claudia Bieler
Yves Bühler
Marc Christen
Yolanda Deubelbeiss
Christoph Graf
Brian McArdell
Maren Salz
Maike Schneider

Manuscript update

November 2017

Table of Content

1 Introduction

1.1 Motivation

Mitigation of natural hazards relies increasingly on numerical process models to predict the area inundated by rapid geophysical mass movements. These movements include

- snow avalanches,
- torrent based debris flows and hillslope debris flows,
- mudslides,
- ice avalanches and glacier lake outbreaks
- rockfalls and rock avalanches.

Process models are used by engineers to predict the speed and reach of these hazardous movements in complex terrain. The preparation of hazard maps is a primary application. The models are especially helpful when proposing technical mitigation measures, such as dams and embankments or rockfall protection barriers. The models allow hazard engineers to optimize limited financial resources by studying the influence of different hazard scenarios on defense options.

RAMMS was developed by the RAMMS program team at the WSL Institute for Snow and Avalanche Research SLF. This manual describes the features of the RAMMS program – allowing beginners to get started quickly as well as serving as a reference to expert users.

The RAMMS web page http://ramms.slf.ch provides useful information such as a moderate discussion forum, frequently asked questions (FAQ) or recent software updates. Please visit this web page frequently to stay up to date!

1.2 RAMMS

The RAMMS (**RA**pid **M**ass **M**ovements **S**imulation) software system contains three process modules:

- RAMMS::AVALANCHE
- RAMMS::DEBRISFLOW
- RAMMS::ROCKFALL

The RAMMS::AVALANCHE and RAMMS::DEBRISFLOW modules are designed for flow phenomena containing fast moving particulate debris of snow and rocks. In the avalanche module, the interstitial fluid is air, whereas in the debris flow module the interstitial fluid is mud. The RAMMS::AVALANCHE and RAMMS::DEBRISFLOW models are used to calculate the motion of the movement from initiation to runout in three-dimensional terrain. The models use depth-averaged equations and predict the slope-parallel velocities and flow heights. This information is sufficient for most engineering applications. Information in the slope-perpendicular direction (e.g. mass and velocity distribution) is lost; however, this is seldom of practical interest. Both models require an accurate digital representation of the terrain. Engineers specify initial conditions (location and size of the release mass) and friction parameters, depending on terrain (e.g. roughness, vegetation) and material (e.g. snow, ice or mud content of the debris flow).

The RAMMS::ROCKFALL module is used to study the rigid body motion of falling rocks. The model predicts rock trajectories in general three-dimensional terrain. Rock trajectories are governed by the interaction between the rock and ground. The model contains six primary state variables: three translational speeds and three rotational velocities of the falling rock. From these, kinetic energy, runout distance and jump heights can be derived. Generalized rock shapes are modeled. Rock orientation and rotational speed are included in the rock/ground interaction. The RAMMS::ROCKFALL module is therefore fundamentally different from the RAMMS::AVALANCHE and RAMMS::DEBRISFLOW modules because it is based on hard-contact, rigid-body Lagrangian mechanics, not Eulerian flow mechanics. It also differs from existing rockfall modules because the rock/ground interaction is not governed entirely by simple rebound mechanics, but frictional (dissipative) rock/ground interactions. These govern the onset of rock jumping. The RAMMS::ROCKFALL module predicts all rigid-body motions – rock sliding, rolling, jumping and skipping.

In all RAMMS modules, new constitutive models have been developed and implemented, thanks to calibration and verification at full scale test sites such as St. Léonard/Walenstadt (rockfall, mitigation measures), Vallée de la Sionne (snow avalanches) and Illgraben (debris flow). At present, two new scientific RAMMS modules are under development: RAMMS::AVAL_EXTENDED and RAMMS::DBF_EXTENDED.

1.3 Learning by doing

This manual provides an overview of RAMMS::DEBRISFLOW. Exercises exemplify different steps in setting up and running a RAMMS simulation especially in Chapter 3 *'Setting up a simulation'*. However, to get the most from the manual, we suggest reading it through while simultaneously having the RAMMS program open, learning by doing. We assume RAMMS users to have a basic level of familiarity with windows-based programs, commands and general computer terminology. We do not describe the basics of windows management (such as resizing or minimizing). RAMMS windows, click options and input masks are similar to other windows based programs and can be used, closed, reduced or resized in the same way.

DISCLAIMER

RAMMS is intended to be used as a tool to support experienced users. The interpretation of the simulation results has to be done by a debris flow expert who is familiar with the local as well as with the topographic and geological situation of the investigation area. In no event shall SLF/WSL be liable for any damage or lost profits arising, directly or indirectly, from the use of RAMMS. Swiss law applies. Court of jurisdiction is Davos. If you encounter problems, please contact ramms@slf.ch.

2 Installation and Setup

2.1 System requirements

We recommend the following minimum system requirements for running RAMMS::DEBRISFLOW:

- Operating Systems: Windows 7, 8 or 10 (64-bit)
  32-bit systems (Win XP) are not supported and recommended anymore
- RAM (memory): 4 GB (more recommended)
- CPU: > 1 GHz, 2 cores or more recommended
- Disk space: ca. 220 MB needed for the software

2.2 Installation

Please download the RAMMS::DEBRISFLOW setup file *"ramms_dbf_user_setup_64.zip"* from http://ramms.slf.ch (Downloads section). We recommend to install RAMMS on a **64-bit** Windows system (Windows 7/8/10).

Direct download link: http://ramms.slf.ch/ramms/downloads/ramms_dbf_user_setup_64.zip

Please do the following steps before beginning to install RAMMS:

- Click on the path given above or copy the path to any browser. A window pops up and the automatic download of the file *ramms_dbf_user_setup_64.zip* starts after clicking *Yes*.
- Unzip the file to a temporary location.
- You must have *Administrator privileges* on the target machine. If you do not have such privileges, the installer cannot modify the system configuration of the machine and the installation will fail. Note that you do not need *Administrator privileges* to run RAMMS afterwards.
- Read first, install afterwards! Please read the whole installation process once, before you begin the installation.
- Start the file "*ramms<version>_dbf_user_setup_64.exe*".

Step 1: Welcome

The welcome dialog introduces you to the English setup program and will guide you through the installation process. Click *Next* to continue.

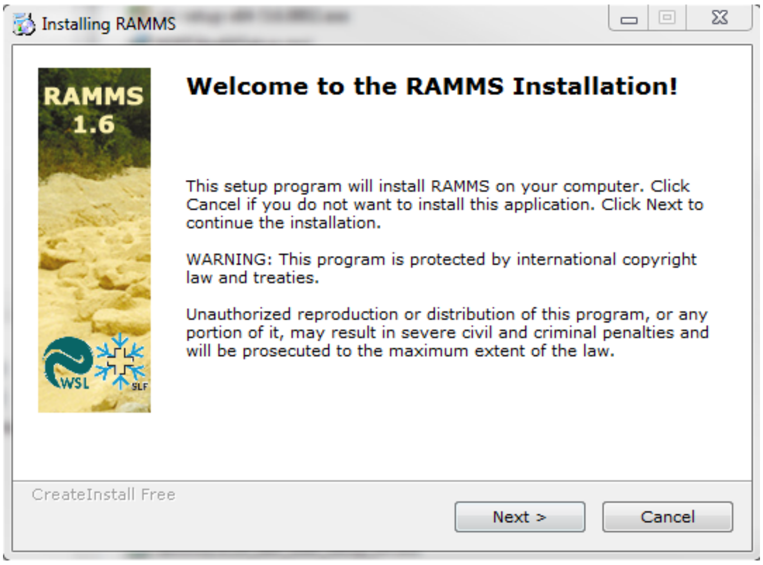


Figure 2.1: Installation - welcome dialog window.

Step 2: Readme

Short introduction to RAMMS. Click *Next* to continue.

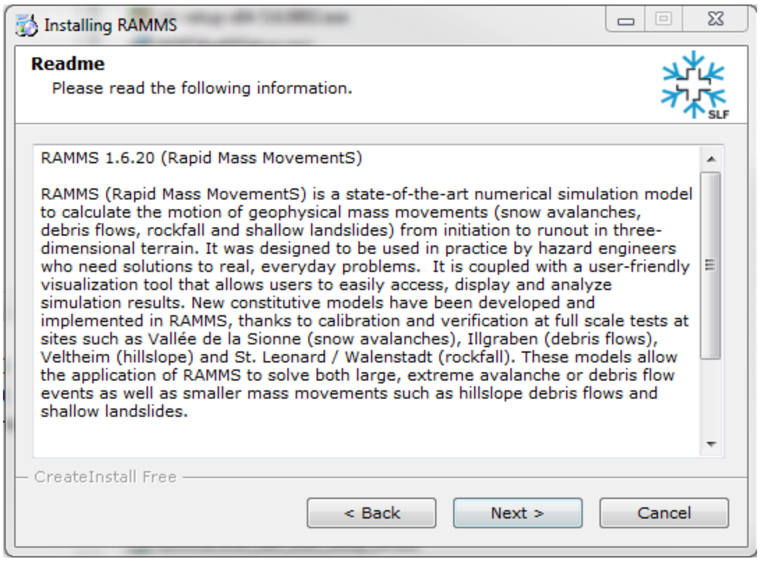


Figure 2.2: Installation - readme dialog window.

Step 3: Accepting the license agreement

Read the license agreement carefully and accept it by activating the check box in the lower left corner. If you do not accept the license agreement, you are not able to proceed with the installation. After accepting the license agreement, click *Next* to continue the installation.

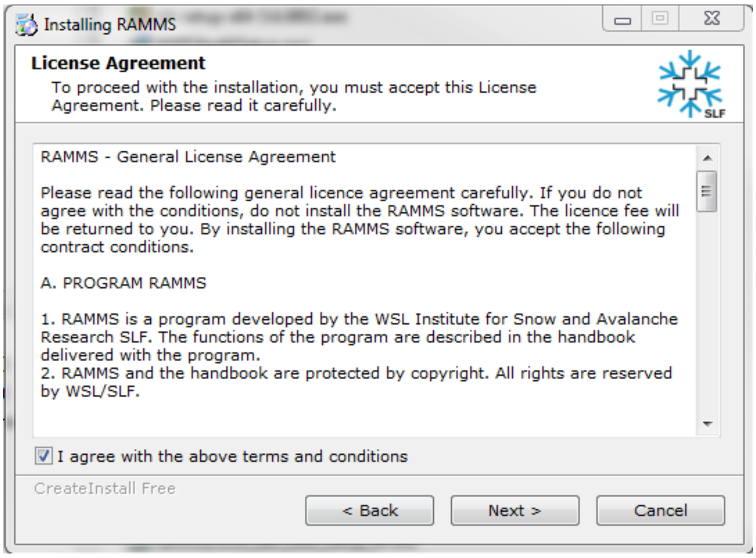


Figure 2.3: Installation - license agreement dialog window.

Step 4: Select destination directory

Choose your destination directory. This dialog shows the amount of space available on your hard disk and required for the installation. Beware: Do NOT use a blank od special character within your installation directory path name (e.g. *C:\program files\RAMMS* is not allowed, use *C:\Programme\RAMMS* or *C:\Programs\RAMMS* instead). Click *Next* to start the installation process.

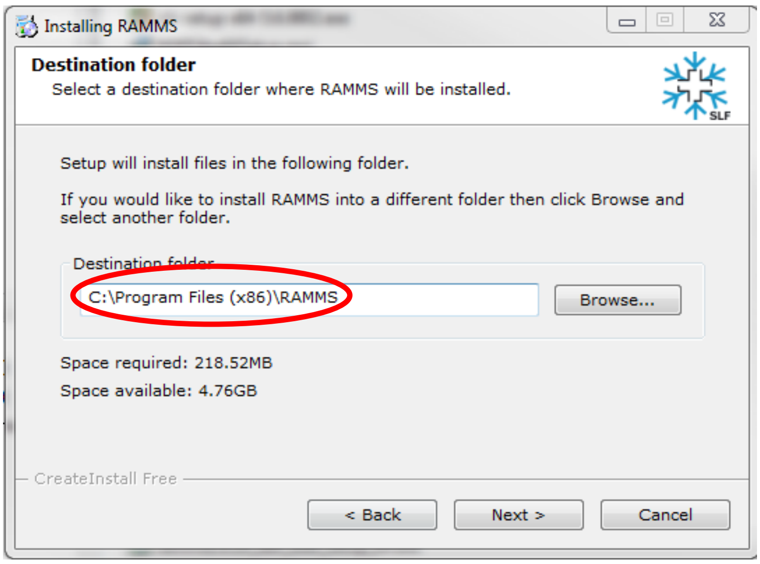


Figure 2.4: Installation - destination directory dialog window.

Step 5: Installing the files

RAMMS is copying the files to the destination location. The window shows the installation progress.

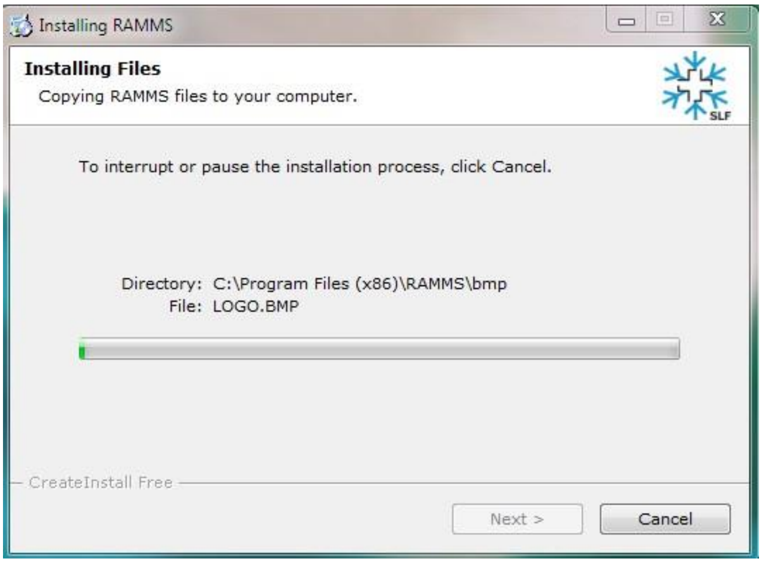


Figure 2.5: Installation - installing files dialog window.

Step 6: Finished installing the files

RAMMS finished copying the files. Click *Next* to finish the installation process.

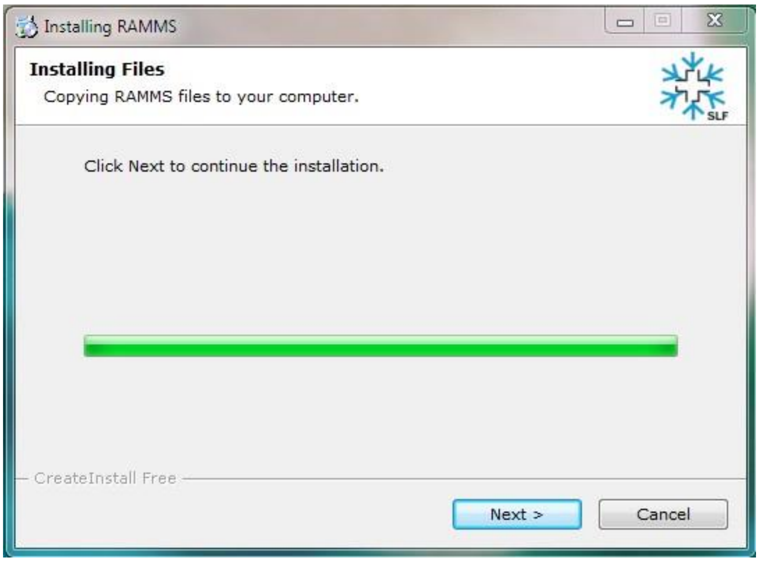


Figure 2.6 : Installation - finished installing files dialog window.

Step 7: RAMMS installation finished!

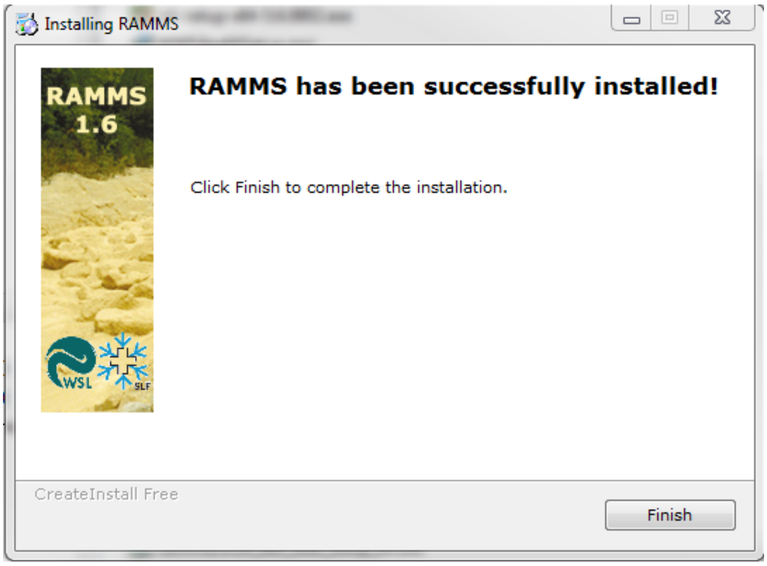


Figure 2.7: Installation - finished installation dialog window.

RAMMS successfully finished the installation. Click *Finish*.

Step 8: Welcome to IDL Visual Studio Merge Modules

To ensure that all important system libraries are installed on your target machine follow the instructions below:

The welcome dialog introduces you to the English setup program and will guide you through the installation process of the IDL Visual Studio Merge Modules. Click *Next* to continue.

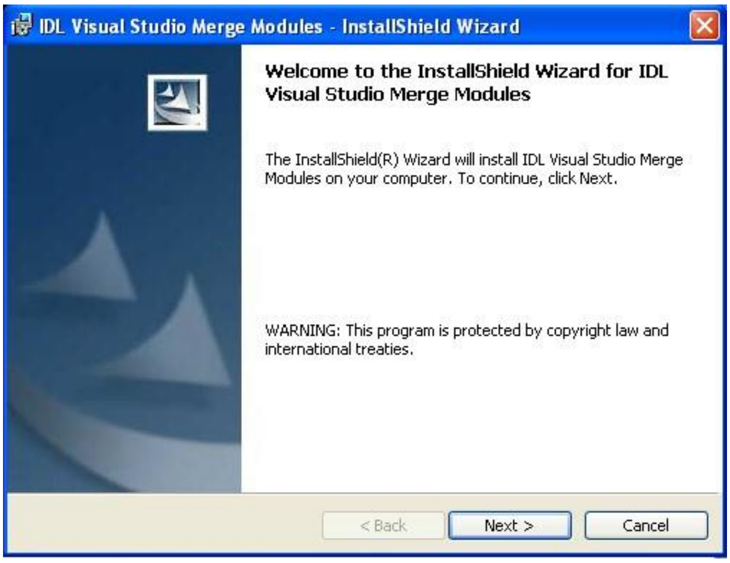


Figure 2.8: IDL Visual Studio Merge Modules - welcome dialog window.

Step 9: Ready to install the program

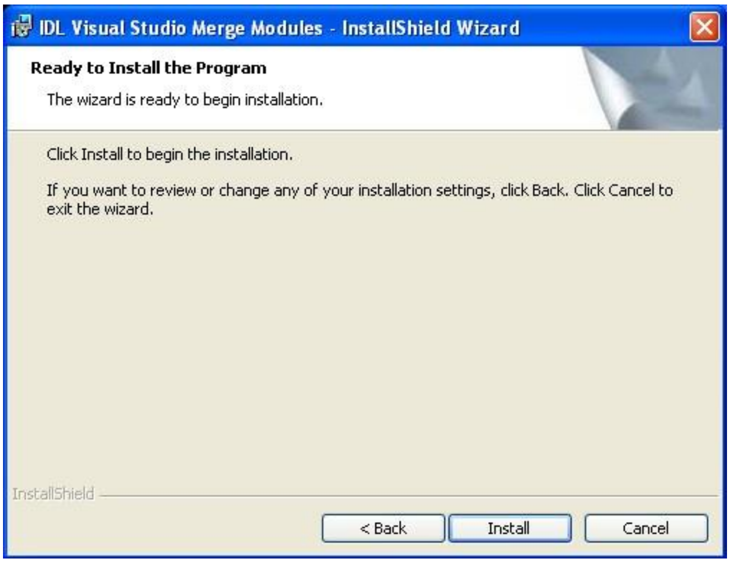


Figure 2.9: IDL Visual Studio Merge Modules - ready to install the program.

Click *Next* to continue.

Step 10: Installing IDL Visual Studio Merge Modules

The wizard is installing the files. Please wait until it is finished.

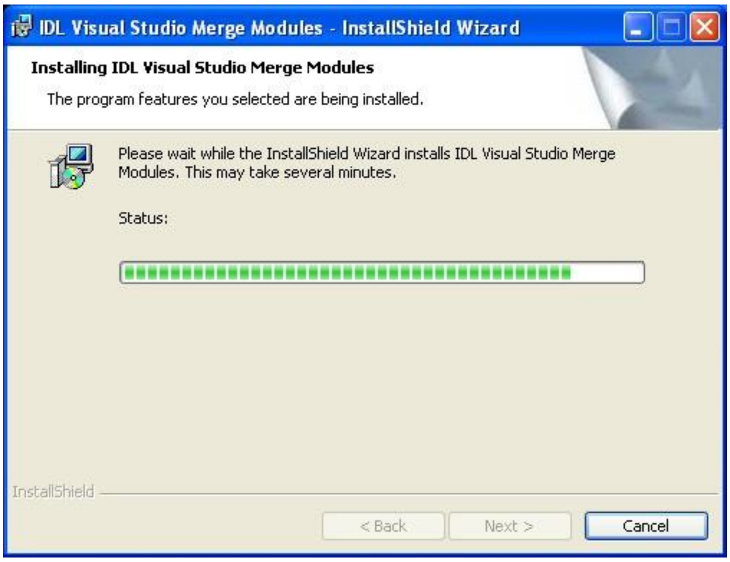


Figure 2.10: IDL Visual Studio Merge Modules - installing...

Step 11: InstallShield Wizard Completed

The wizard completed the installation. Click *Finish*.

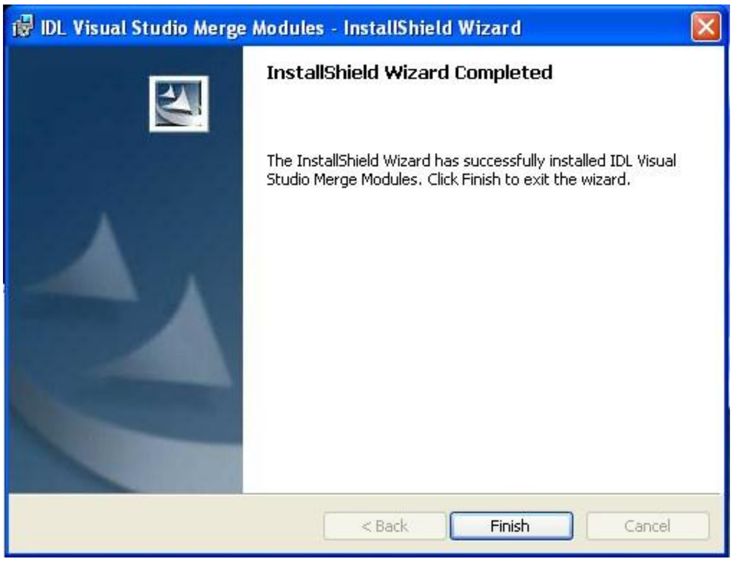


Figure 2.11: Installation - destination directory dialog window.

After having successfully installed RAMMS and the necessary files on your personal computer, you will notice the RAMMS icon on your desktop (for all users):

Figure 2.12: RAMMS icon.

Additionally, a new application folder is created in *Start* → *Programs* (for all users):

- *RAMMS* → *Run RAMMS*
- *RAMMS* → *Uninstall RAMMS*

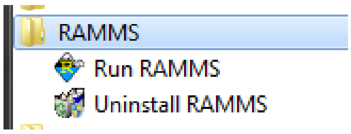


Figure 2.13: RAMMS program group

2.3 Licensing

Access to RAMMS is controlled by a personal use license. Personal use licenses are time limited licenses tied to a single personal computer. This method of licensing requires a machine's unique host ID to be incorporated into a license request file. After the license request file is sent to SLF/WSL, you will receive a license key. Entering the license key on a personal computer enables full RAMMS functionality for the specific personal computer. For more information please visit http://ramms.slf.ch.

2.4 First start

Double-click the RAMMS icon or use ***Start*** → ***Programs*** → ***RAMMS*** → ***Run RAMMS*** to start RAMMS for the first time. Whenever you start RAMMS, the splash screen below will pop up:

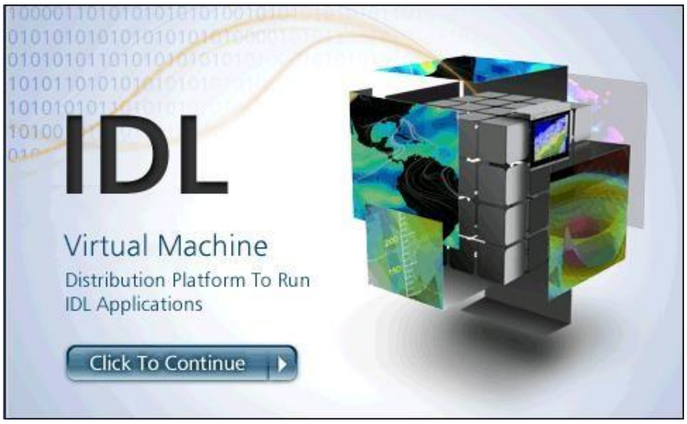


Figure 2.14: RAMMS start window.

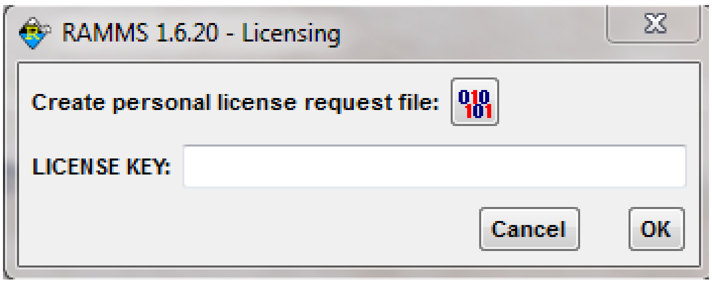


Figure 2.15: RAMMS licensing window

Click on the image. It will disappear and RAMMS will start up. The following dialog window appears (Fig. 2.15 RAMMS Licensing):

2.4.1 Personal license request file

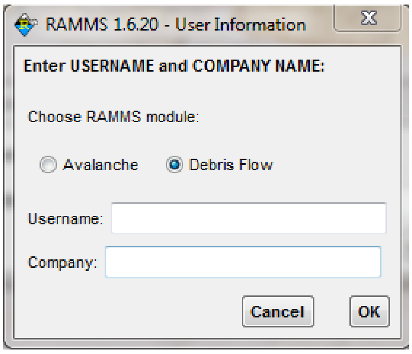


Figure 2.16: Enter user name and company name.

Click the button  to create your personal license request file. In Figure 2.16 enter your full name and the name of your company.

In the next dialog window, choose the destination directory of your personal license request file and save it to your target machine. Your personal license request file should look similar to Figure 2.17.

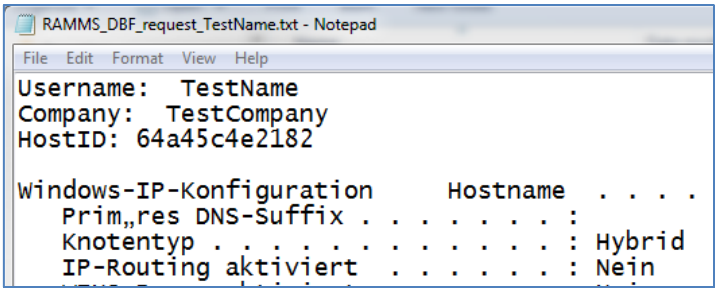


Figure 2.17: Personal license request file RAMMS_DBF_request_TestName.txt

2.4.2 Getting the personal license key

You find an order form on the RAMMS web page (*Order Form* or *Demo Order Form*) at http://ramms.slf.ch. Fill in all your personal information, choose the license period, license type and number of licenses you wish to order, attach your personal license request file(s), accept the license agreement and click *Submit Order*.

An order confirmation email is sent to your email address. We then process your order and send you an invoice. As soon as we received your payment, we will send you your personal license key. Your personal license key is named similar to

DBF_20151013_TestName_RAMMS_TimeLicense.txt.

Open the file in a text editor. It should look similar to Figure 2.18.

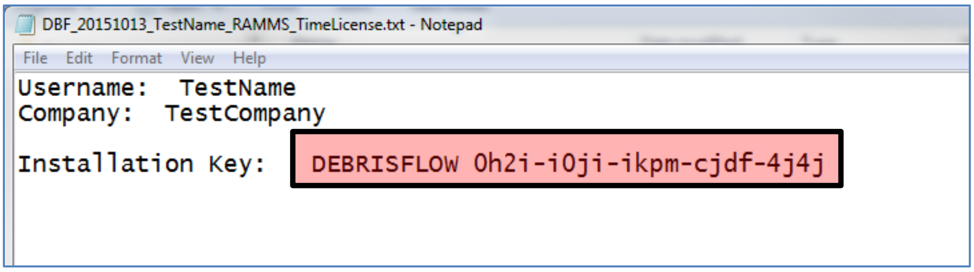


Figure 2.18: Personal license key file RAMMS_license_Muster Test.txt

Now, restart RAMMS (as explained before). The IDL splash screen appears (Figure 2.14) and then the dialog window of Figure 2.15 shows up (RAMMS - Licensing). Copy the license key (in this example: *DEBRISFLOW 0h2i-i0ji-ikpm-cjdf-4j4j*) and paste it at the field *LICENSE KEY* (see Figure 2.15). Notice that there is the prefix DEBRISFLOW. This prefix is part of the license key and has to be inserted as well! If RAMMS accepts your installation key, you successfully finished the installation.

2.5 Update

When you start RAMMS it will automatically check for updates on the internet. This can lead to an error message, if your firewall blocks the executable idlrt.exe (this file starts the IDL-Virtual Machine you need to run RAMMS). Please unblock this file for your firewall. You can also disable the AutoWebUpdate-function by unchecking ***Help*** → ***Advanced...*** → ***AutoWebUpdate***. In the same way you can enable the AutoWebUpdate-function by checking ***Help*** → ***Advanced...*** → ***AutoWebUpdate***.

3 Setting up a simulation

3.1 Preparations

To successfully start a new RAMMS project, a few important preparations are necessary. Topographic input data (DEM in ASCII-, XYZ- or GEOTIFF-format), project boundary coordinates and georeferenced maps or remote sensing images should be prepared in advance (.tif format and .tfw-file, maps and images are not mandatory, but nice to have). Georeferenced datasets have to be in the same **Cartesian coordinate system** (e.g. Swiss CH1903 LV03) as the DEM. Polar coordinate systems in degree (e.g. WGS84 Longitude Latitude) **are not supported**. For more information about specific national coordinate systems please contact the national topographic agency in your country.

3.1.1 Topographic data - Digital Elevation Model (DEM)

The topographic data is the most important input requirement. The simulation results depend strongly on the resolution and accuracy of the topographic input data. Before you start a simulation, make sure all important terrain features are represented in the input DEM. RAMMS is able to process the following topographic data:

- ESRI ASCII grid (Figure 3.1)
- ASCII X, Y, Z regular, single space data (Figure 3.2, irregular data is not supported!)
- GEOTIFF (georeferenced information embedded within a TIFF file)

The first two data types are also available e.g. from www.swisstopo.ch. *ASCII X, Y, Z* data can be converted within RAMMS into an ESRI ASCII grid. Beware, that the XYZ-data must be regular! The header of an ESRI ASCII grid must contain the information shown in Figure 3.1.

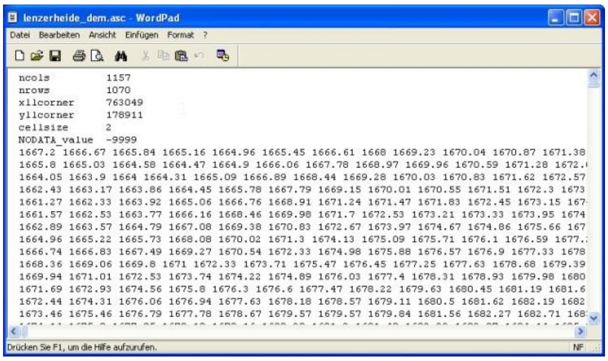


Figure 3.1 : Example ESRI ASCII grid.

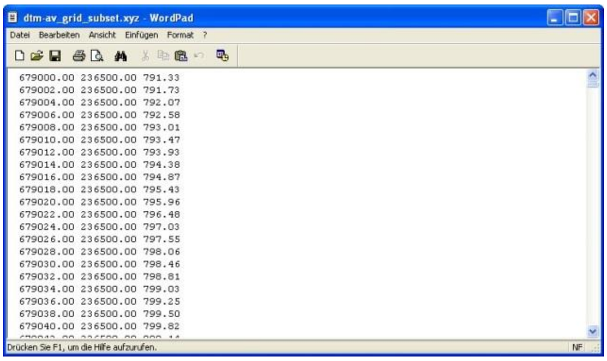


Figure 3.2: Example ASCII X, Y, Z single space data.

Conversion into ESRI ASCII grid

An ESRI ASCII grid can be created in ArcGIS with the function ***ArcToolbox*** → ***Conversion Tools*** → ***From Raster*** → ***Raster to ASCII***. In RAMMS it is possible to import regular ASCII X, Y, Z single space data and convert the data into an ESRI ASCII grid (using ***Track*** → ***New...*** → ***Convert XYZ to ASCII Grid***).

3.1.2 Project and Scenarios

A project is defined for a region of interest. Within a project, one or more scenarios can be specified and analyzed. For every scenario, a calculation can be executed. A project consists therefore of different scenarios (input files) with different input parameters. The basic topographic input data is the same for every scenario. If you want to change the topographic input data (e.g. change the input DEM resolution or the project boundary coordinates) you have to create a new project. Other input parameters (such as release area, calculation domain, calculation grid resolution, end time or time step) can be changed for every scenario.

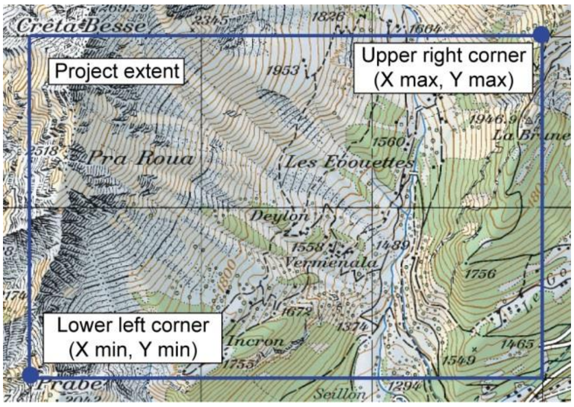


Figure 3.3: The same project extent (area of interest) can be used to calculate different scenarios with different input parameters.

3.1.3 Release information

In RAMMS::DEBRISFLOW there are two options to define the starting conditions (release information) of a simulation:

- Release area (or block release)
- Input hydrograph (or simply hydrograph)

The starting conditions of a simulation can be selected depending on the type of debris flow you want to model. Generally, it is useful to distinguish between *unchannelized* and *channelized* debris flows. Here we use the term *unchannelized* debris flow for hillslope debris flows or shallow landslides, *channelized* debris flows develop in regions where torrents limit the flow paths and the debris material mainly follows the torrent channel.

For small unchannelized debris flows it is useful to use a release area with a given initial depth, which will be released as a block (block release), while for a channelized debris flow it may be more appropriate to use an input hydrograph. The later requires knowledge of the amount of material that might flow past a certain location in the channel. Chapter 5 on page 72 includes several examples for both types of starting conditions.

The definitions of release areas and release depths (block release) have a strong impact on the simulation results. The same applies for simulations with a hydrograph, where the results are sensitive to the discharge hydrograph. Therefore, we recommend to use reference information such as photography, GPS measurements or field maps to draw release areas and to use measured or well-estimated inflow data to define discharge hydrographs. This should be done by people with experience concerning the topographic and historical flow situation of the investigation area.

More details on how to create release areas are given in section 3.5.1, page 34 pp. Details on how to use an input hydrograph are given in section 3.5.2, page 37 pp.

3.1.4 Friction information

RAMMS employs a Voellmy-fluid friction model, which is based on the Voellmy-fluid approach (we refer to Salm et al. 1990 and Salm 1993).

The choice of the friction parameters requires *careful calibration* (section 5.1) of the model by using reference information such as field data, photographs of runout zones, estimations or measurements of flow velocities and flow heights as well as estimations of the material composition. This should be done by an persons with expertise in debris flow characterization.

Physical friction model

The physical model of RAMMS::Debris Flow uses the Voellmy friction law. This model divides the frictional resistance into two parts: a dry-Coulomb type friction (coefficient μ) that scales with the normal stress and a velocity-squared drag or viscous-turbulent friction (coefficient ξ). The frictional resistance S (Pa) is then

$$S = \mu N + \frac{\rho g \boldsymbol{u}^2}{\xi} \quad \text{with} \quad N = \rho h \, \mathrm{g} \cos(\phi) \tag{3.1}$$

where ρ is the density, g the gravitational acceleration, φ the slope angle, h the flow height and **u** the vector **u** = $(u_x,u_y)^T$, consisting of the flow velocity in the x- and y-directions. The normal stress on the running surface, *ρhgcos(φ)*, can be summarized in a single parameter *N*. The Voellmy model accounts for the resistance of the solid phase (μ is sometimes expressed as the tangent of the internal shear angle) and a viscous or turbulent fluid phase (ξ was introduced by Voellmy by using hydrodynamic arguments). The friction coefficients are responsible for the behavior of the flow. μ dominates when the flow is close to stopping, ξ dominates when the flow is running quickly.

Throughout one simulation, the friction coefficients μ and ξ of a calculation domain are constant. However you have the possibility to add up to two polygons within the calculation domain with different μ and ξ friction parameters (see exercise 3.5e *"How to run a calculation" on page 47pp.*)

The Voellmy friction model has found wide application in the simulation of mass movements, especially snow avalanches. For modeling snow avalanches the Voellmy model has been in use in Switzerland for many years and a set of standard parameters is available.

Yield stress

Since Version 1.6.20 the basic Voellmy equation has been modified to include a yield stress (cohesion). Many materials, like mud and snow, do not exhibit a simple linear relation (μ = constant), see Figure 3.4. To model yield stress, we introduce the parameter N_0. With this approach it is possible to model ideal plastic materials. In this case N_0 serves as a yield stress and μ a "hardening" parameter. The new equation for the frictional resistance S is then

$$S = \mu N + \frac{\rho g \boldsymbol{u}^2}{\xi} + (1-\mu)N_0 - (1-\mu)N_0 e^{-\frac{N}{N_0}}$$

where N_0 is the yield stress 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 debris flow to stop earlier, depending on the value of N_0 .

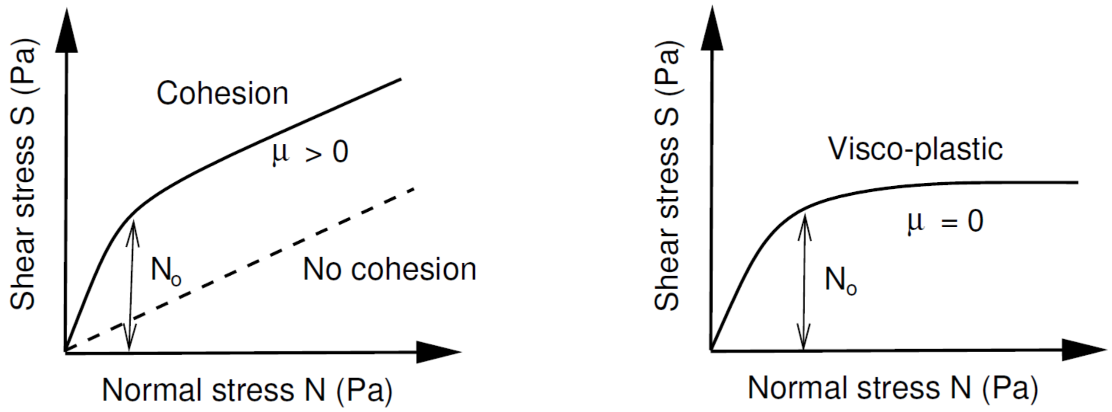


Figure 3.4: Relation between normal and shear stress. Left: Yield stress N_0 serves to increase the shear stress for higher normal pressures. At low normal pressures (small flow heights) the shear stress increases rapidly from $S=0$ to $S=N_0$. The slope of the '*S vs N*' relation remains μ, when the normal pressures are large. Right: If $\mu=0$, we have a visco-plasic behaviour.

Curvature

Since Version 1.6.20, the normal force N 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 flow velocity and terrain curvature. The acceleration is calculated according to

$$f = \boldsymbol{u}\boldsymbol{K}\boldsymbol{u}^T$$

The matrix **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 flow 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 in the *Run Simulation* window (tab *Params*) or via the menu 'Help → Advanced… → Curvature'.

Calibration of the friction parameters μ and ξ

Although the data base for well documented debris flow events is smaller than the one for snow avalanches, we have a good idea of the friction parameters which have to be used. The major difficulty in case of debris flow simulation is the large variety of debris flow compositions, which has a strong influence on the choice of the friction parameters. RAMMS::DEBRISFLOW uses a single-phase model, so we cannot distinguish between fluid and solid phases and the material is modeled as a bulk flow. Therefore, the friction parameters should be varied to match observed flow properties (for calibrating the model) or expected flow properties (if variation is expected as part of a hazard scenario).

It is common that different events in the same torrent show strong differences in composition. This fact makes the calibration of the friction parameters much more difficult and even requires a calibration for different events. Therefore, we strongly recommend careful calibration of the friction parameters by persons with expertise in debris flow characterization.

The calibration procedure of the Voellmy model is explained with an example in section 5.1.

3.1.5 Erosion information

The erosion module in RAMMS predicts the depth of erosion of sediment caused by debris flows. Using this option, it is possible to predict the increase in volume of a debris flow as it travels along a channel. By erosion we mean the net decrease in the elevation of the channel bed as a consequence of the entrainment of sediment from the bed. Here we provide a brief overview of the erosion module and an example of using it. More details can be found in Frank et al. (2015, 2017).

The erosion module is based on a generalization of field observations from the Illgraben Debris Flow observation station in Switzerland, based on repeated terrestrial laser scans by P. Schürch and Alex Densmore, of Durham University. The field observations indicate that the depth of erosion increases with flow strength e.g. with basal shear stress τ (Schürch et al., 2011; Frank et al., 2015) and that the rate of erosion can be quite rapid (Berger et al., 2011). Other observations indicate that small debris flows do not always erode sediment (Berger et al., 2010, Schürch et al., 2011), so the model includes a critical shear stress that permits erosion only when the shear stress in any given cell exceeds the critical shear stress value for the onset of erosion τ_c. The erosion algorithm predicts the maximum potential depth of erosion e_m as a function of the computed basal shear stress in each grid cell:

$$e_m = 0 \text{ for } \tau < \tau_c$$

$$e_m = \frac{dz}{d\tau}(\tau - \tau_c) \text{ for } \tau \geq \tau_c$$

The *potential erosion depth (per kPa) dz/d*τ controls the rate of vertical erosion (in the *z*-direction) as a linear function of channel-bed shear stress.

The *sediment erosion rate* in the field (Illgraben channel) was observed using sensors buried in the channel bed (Berger et al., 2011) to be 0.025 m/s in the downward direction. Sediment is entrained until the erosion depth e_m is reached:

$$\frac{dz}{dt} = 0.025 \text{ for } e_t \leq e_m$$

where e_t is the depth of erosion (relative to the start of the simulation) at time *t* and *z* is the vertical coordinate.

If the shear stress in any given cell is exceeded after erosion takes place, then the maximum depth of erosion (relative to the initial bed elevation at the start of the simulation) is automatically adjusted and additional erosion can take place until the new value of e_m is reached. The elevation of the channel bed does not change during the RAMMS simulation. Users can incorporate changes in topography due to erosion or deposition, e.g. for further use in hazard modeling scenarios or event analyses, by selecting the post-processing options *'Results → Add Deposition to DEM'* or *'Results → Subtract Erosion from DEM'* or both with *'Results → Add Topographic Changes (Erosion/Deposition) to DEM'*.

3.1.6 Calculation parameters

Calculation parameters, such as output name, simulation grid resolution, simulation (end) time, time step etc. can be changed interactively in the RAMMS *Run Simulation* window.

3.2 Preferences

To ease the file handling we recommend setting the preferences prior to start with simulations. The preferences set the path to the working directory and the necessary files such as DEM, maps and orthophotos. If the path to the maps and the imagery files is set correctly in the preferences, RAMMS will automatically open the georeferenced data when you generate a new project.

Use ***Track*** → ***Preferences*** to open the RAMMS preferences window or click the button . For resetting the general preferences use ***Help*** → ***Advanced…*** → ***Reset General Preferences***.

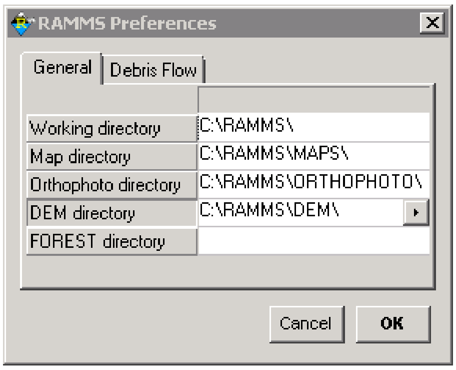


Figure 3.5: General tab of RAMMS preferences.

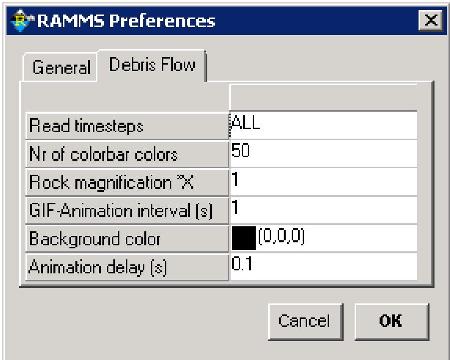


Figure 3.6: Debris Flow tab of RAMMS preferences.

General Tab

Setting	Purpose
Working Directory	Set your working directory. VERY IMPORTANT: DO NOT USE BLANKS in the working directory path!
Map Directory	Set the folder where you place your georeferenced digital maps (consists of a .tif file and a corresponding .tfw file (world-file).
Orthophoto Directory	Set the folder where you place your digital georeferenced orthophotos (aerial picture, consists of a .tif file and a corresponding .tfw file (world-file).
DEM Directory	Set the folder where you place the Digital Elevation Models (format ASCII grid)

Debris flow Tab

Setting	Purpose
Read timesteps	Choose between reading ALL or only 1 timestep. Default is reading ALL timesteps.
Nr of colorbar colors	Set default nr of colorbar colors.
GIF-Animation Interval [s]	Set interval for GIF animation images in seconds.
Background Color	Set background color (greyscale between 0:black and 255:white).
Animation Delay [s]	Set animation delay to decelerate the animation speed.

The following exercise *Working directory* shows how to choose a new working directory. All further settings can be changed in a similar manner. The settings are saved, until they are changed again manually.

Exercise 3.2 : Working directory

Choosing the right working directory is very useful and saves a lot of time searching for files and folders.

VERY IMPORTANT: Do NOT use blanks or special characters in the path names!

- Click  (or use ***Track*** → ***Preferences*** or ***Ctrl+P***) to open the RAMMS preferences window.
- Click into the field *Working directory*. A window pops up where you can choose your new working directory. Click *OK* in both windows. Do this also for other directories if necessary.

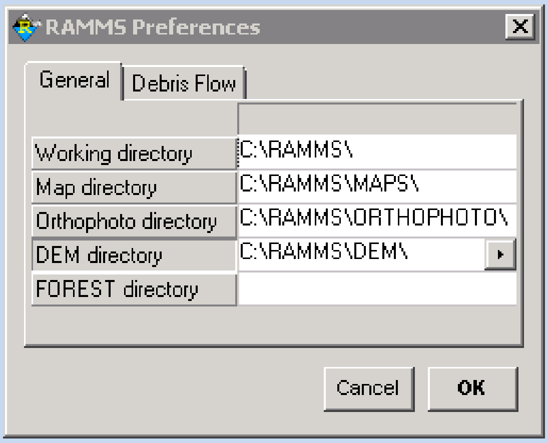


Figure 3.7: RAMMS preferences

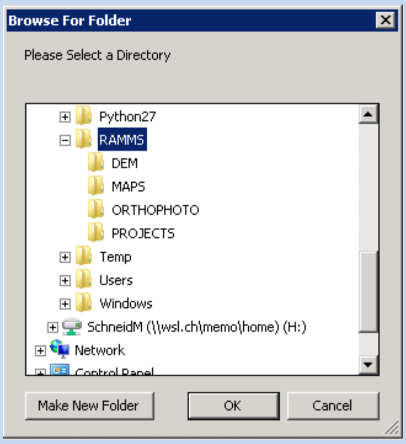


Figure 3.8: Browse for the correct folder.

3.3 Creating a new project

A new project is created with the *RAMMS Project Wizard*, shown in the exercise below. The Wizard consists of four steps:

Exercise 3.3: How to create a new project

- Click  or ***Track*** → ***New...*** → ***Project Wizard*** to open the RAMMS Project Wizard.
- The following window pops up.

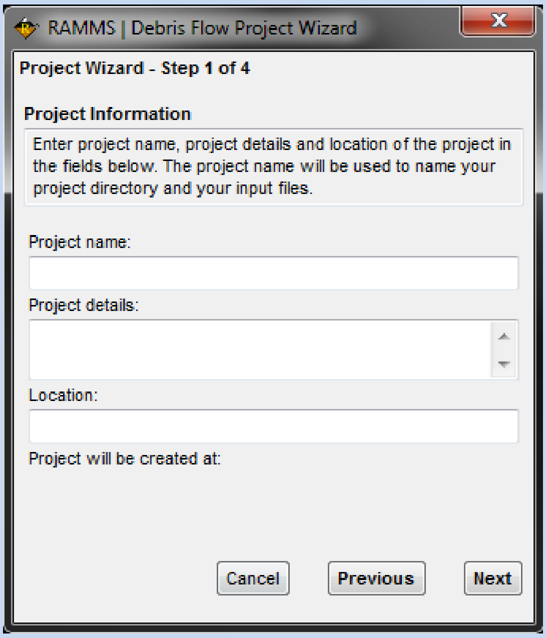


Figure 3.9: RAMMS Project Wizard Step 1 of 4

Continuation of exercise 3.3: How to create a new project

Step 1:

- Enter a project name (1)
- Add project details (2)
- The project location (3) suggested is the current working directory. To change the location click into the *Location* field. A second window appears and you can browse for a different folder (see figure below)
  VERY IMPORTANT: Do NOT use BLANKS or special characters in the project location path!
- Click ***Next*** (4)

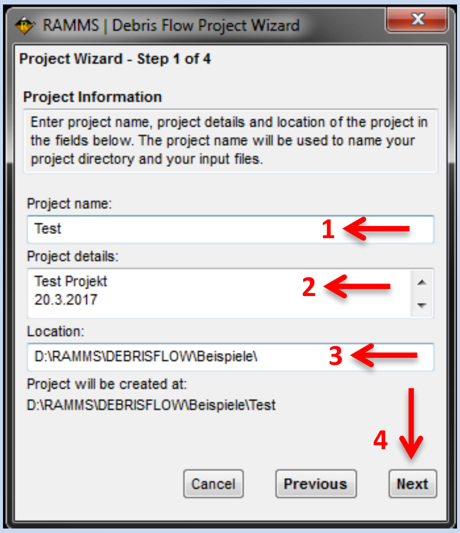


Figure 3.10: Step 1 of the RAMMS Project Wizard Project Information.

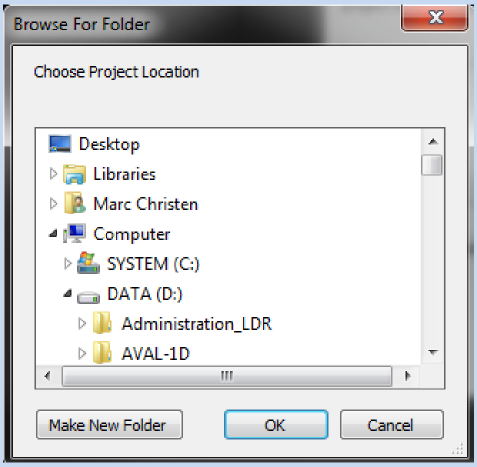


Figure 3.11: Window to browse for a new project location.

Step 2:

- Click into the *Select DEM-file* field to browse for the DEM file. Locate your DEM file in the folder set in the RAMMS preferences.
- The grid resolution of your DEM-file is shown in (2). Change the resolution, if needed (bilinear interpolation).
- Click ***Next***.

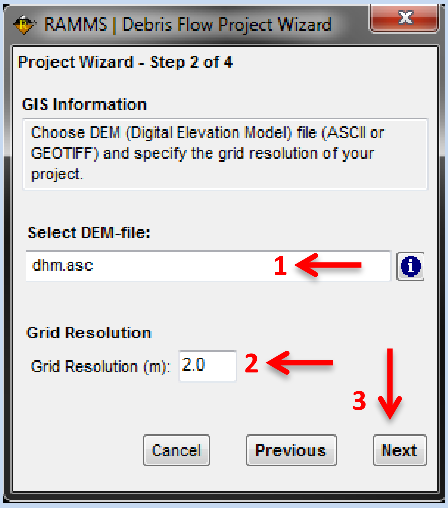


Figure 3.12: Step 2 of the RAMMS Project Wizard: GIS Information.

Continuation of exercise 3.3: How to create a new project

Step 3:

- Enter the X- and Y-coordinates of the lower left and upper right corner of your project area, using the Swiss Coordinate System CH1903 LV03 (or another Cartesian coordinate system), as it is shown below for the Vallée de la Sionne area.
- RAMMS shows the coordinates of your DEM-file (1).
- You can clip the DEM by entering new boundary coordinates or by specifying a polygon shapefile (2).
- Click **Next** (3).

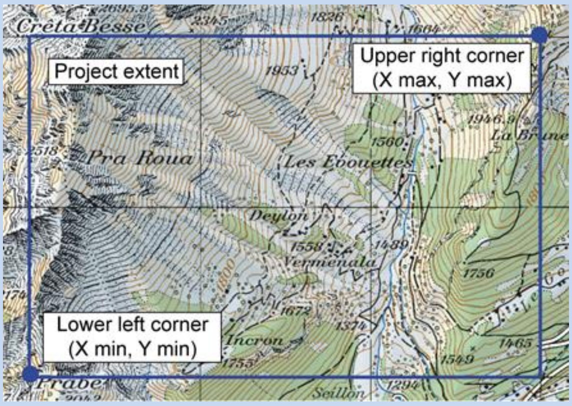


Figure 3.13: Project coordinates: lower left and upper right corner of project area.

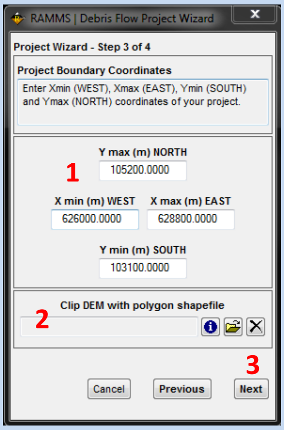


Figure 3.14: Step 3 of the RAMMS Project Wizard: Project Boundary Coordinates.

Step 4:

- Check the project summary.
- To make changes click ***Previous***, to create the project click ***Create Project***.

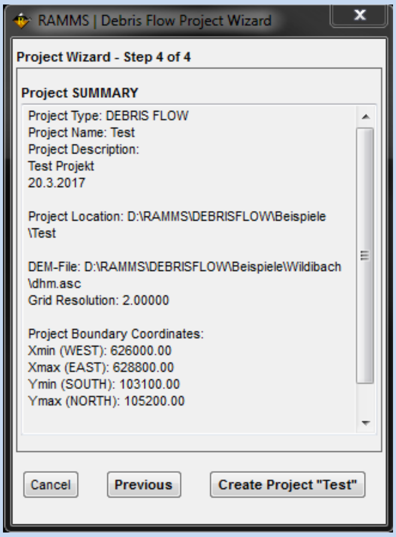


Figure 3.15: Step 4 of the RAMMS Project Wizard: Project Summary.

Project creation:

- The creation process can take a while. Different status bars will pop up and show the progress of the project creation process.

The following files will be created in the project folder.

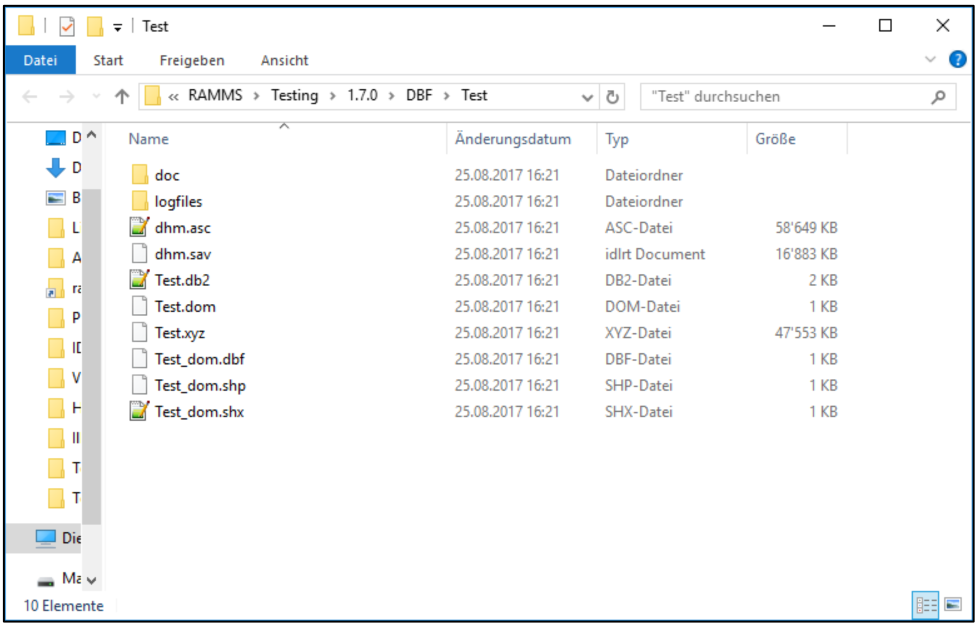


Figure 3.16: Project files

Table 3.1: Listing of files and directories created with a new RAMMS::DEBRISFLOW project.

File / Folder	**Purpose**
doc (folder)	Folder containing input and output log files
logfiles (folder)	Project creation and calculation log files
dhm.asc	ASCII grid with altitude values
dhm.sav	Height information used in RAMMS
_.db2	Input file
_.dom	Calculation domain ASCII file
_.dom.shp	Calculation domain shapefile
_.dom.shx	Calculation domain shapefile
_.dom.dbf	Calculation domain shapefile
_.xyz	Topographic data used in RAMMS

3.4 Working with the RAMMS GUI

Once the project is created, there are several useful tools which can be helpful when working with RAMMS. They are explained in the exercises below.

3.4.1 Visualizing shapefiles and domain-files

There are different ways to visualize your project files (shapefiles and Domain-files). In the exercise below, we will show these possibilities.

Exercise 3.4a : Visualizing shapefiles and Domain-files

a. *Files* tab in the right panel:

- Click on the *Files* tab in the right *DEBRIS FLOW* panel.
- In the file tree below, you will see your available project files (polygon and domain shape-files.

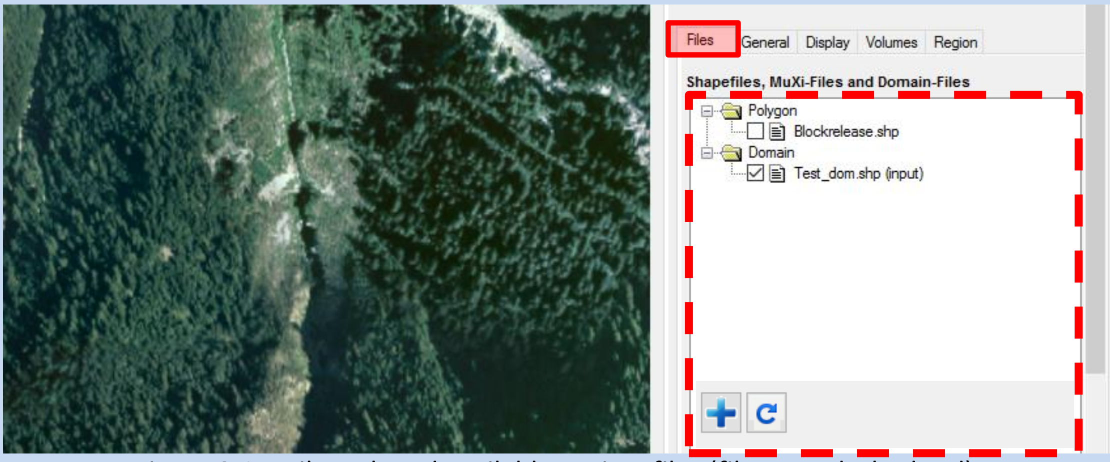


Figure 3.17: Files tab and available project files (file-tree, dashed red). With the **blue + button**, files from external directories can be added to the file-tree. Refresh the tree with the refresh-button.

- Click **the checkbox** (left of the filename) and the file will be shown in your visualization.

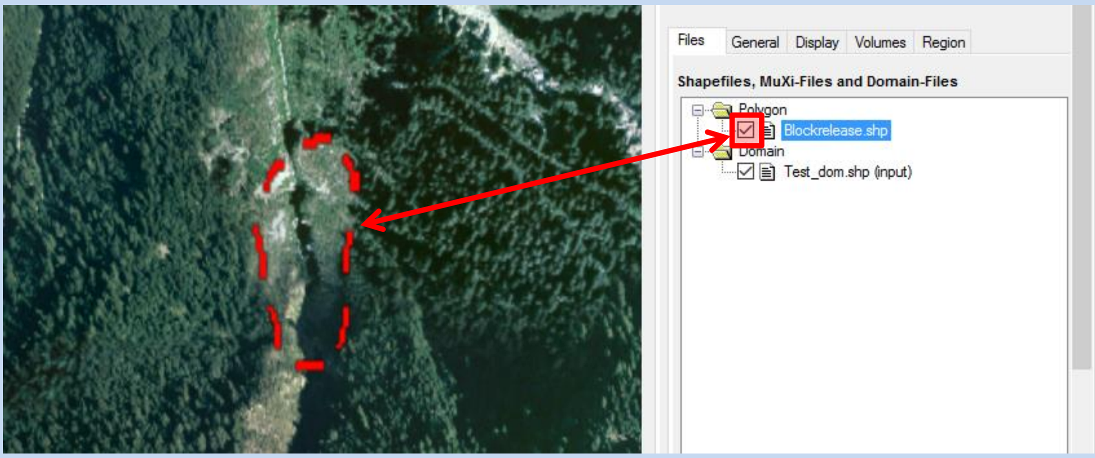


Figure 3.18: Selected file (Blockrelease.shp) on the right is shown in the visualization.

- You can select and visualize as many files as you like!

Shapefile Properties

- Line thickness, color or linestyle can be adjusted for every individual shapefile. Right-click on a **filename** and choose ***Shapefile properties***:

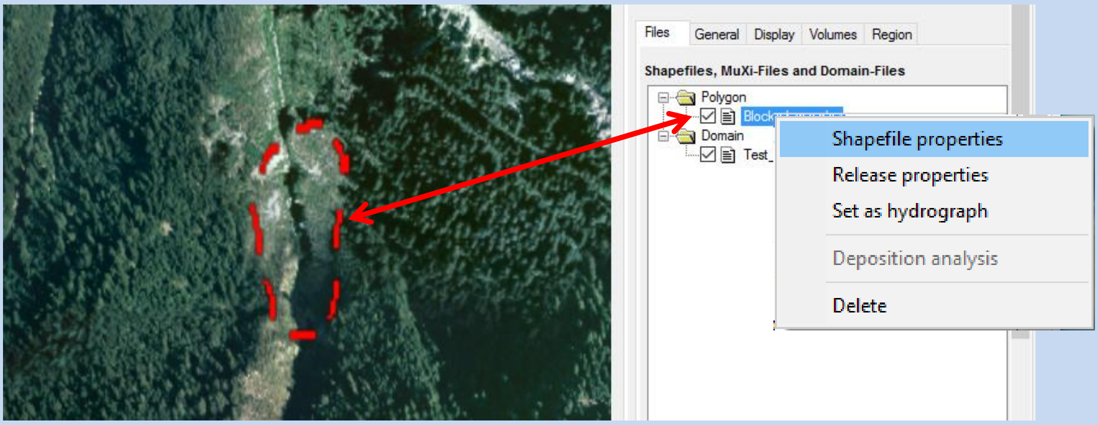


Figure 3.19: Right-click menus ***Shapefile properties***, ***Release properties***, ***Set as hydrograph*** and ***Delete***.

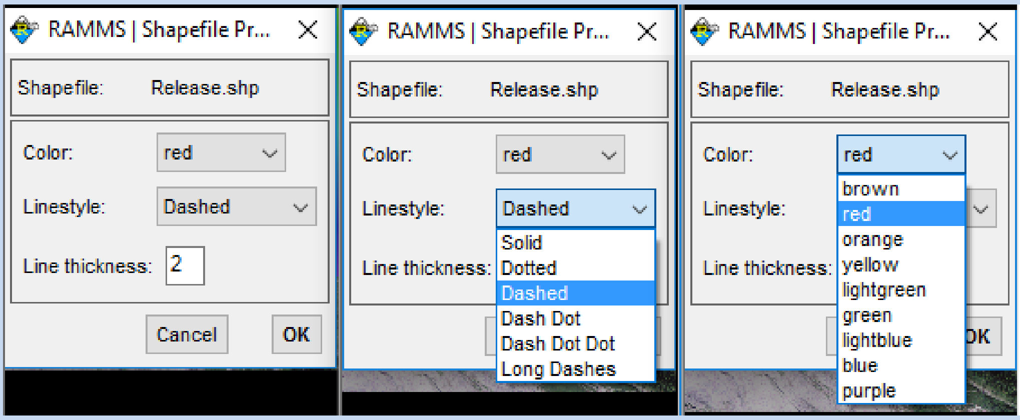


Figure 3.20: Use ***Shapefile properties*** to change line thickness, color or linestyle.

Release Properties

- Please see section 3.5.1 on page 34 on how to specify release area properties.

Set as hydrograph

- With this option you tell RAMMS, that this polygon shapefile is a hydrograph release area. See section 3.5.2 on 37 for more about hydrographs.

Deposition analysis

- This function is only available in output mode. See section 4.2.3 on page 63 on how to do a deposition analysis.

Delete

- Delete a file from disk.

b. Adding files to the project

You can add files to the visualization using one of these options:

- Add data: Use the button or the menu 'GIS – Add data' to add a shapefile. If this shapefile is located outside of your project directory, it will be added to the files-tree.

- Add files from folder: Use the button (Add files from external directory) below the file-tree to add all the files from an external directory to the file-tree. These files are added during this RAMMS session. After you exit and restart RAMMS, you have to add the files again.

- Drag & Drop: see next section.

c. Drag & Drop:

It's possible to Drag & Drop the following files onto the main visualization window:

- Input files (.av2)
- Output files (.out.gz)
- MuXi files (.asc)
- Polygon shapefiles (.shp)
- Domain shapefiles (.shp)

3.4.2 Changing maps and orthophotos (aerial images)

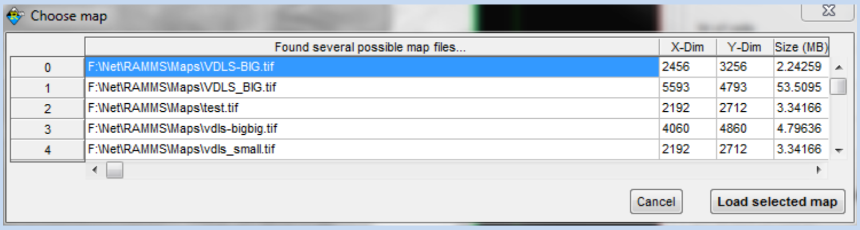
It is possible to change the map or orthophotos of a project anytime. Take into account, that the corresponding .tfw-file (world-file) has to be in the same folder as the actual map (.tif). If this is not the case, the map will not be found!

To check which map and orthophoto is currently loaded in the project, open the project input (or output) log (*Project → Input Log File*). Next to *MAP / ORTHOPHOTO INFO* you will find the location and name of the loaded map and orthophoto, respectively.

Exercise 3.4b : How to add or change maps and orthophotos

d. Add or change a map:

- Go to ***Extras*** → ***Add/Change Map*** or click .
- If more than one map is found, the following window pops up, listing the maps found:


Choose map

	Found several possible map files...	X-Dim	Y-Dim	Size (MB)
0	F:\Net\RAMMS\Maps\VDLS-BIG.tif	2456	3256	2.24259
1	F:\Net\RAMMS\Maps\VDLS_BIG.tif	5593	4793	53.5095
2	F:\Net\RAMMS\Maps\test.tif	2192	2712	3.34166
3	F:\Net\RAMMS\Maps\vdls-bigbig.tif	4060	4860	4.79636
4	F:\Net\RAMMS\Maps\vdls_small.tif	2192	2712	3.34166

Cancel   Load selected map

Figure 3.21: Window to choose map image.

Information on the image dimensions (x-Dim and y-Dim, pixel) and size (in MB) are provided and might be a selection criterion.

- Select the map you wish to add and click ***Load selected map***.

e. Map not found:

- If the question "No map found, continue search?" appears, you either don't have an appropriate map, the map-folder directory is not set (or wrong) or the map is saved in a different folder. In the second case click ***Yes*** and choose the correct folder. A window pops up to browse for the correct map location and file.
- Or click ***No*** to cancel search.

f. Change remote sensing imagery:

- Go to ***Extras*** →***Add/Change Image*** or click .

3.4.3 Moving, resizing, rotating, viewing

Exercise 3.4c: Moving and resizing the model

a. Terrain model has a dimension of 100% or smaller:

- By clicking on the *arrow*  the model can be moved and resized.

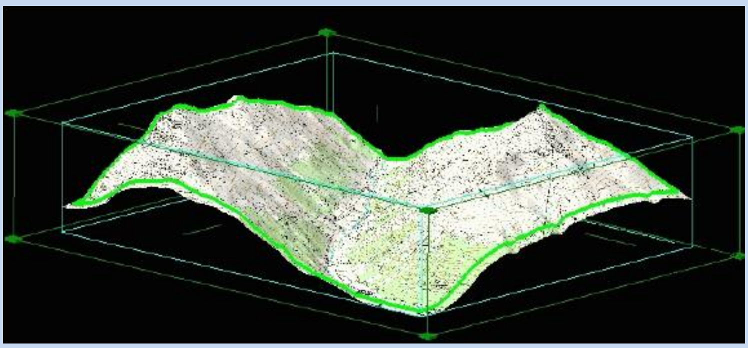


Figure 3.22: *Active* project with lines and corners for resizing.

- To move the model without changing size or aspect ratio, move the cursor to the model and check if the cursor turns to . Then click and hold the left mouse button and drag the model to the desired position.
- To resize the model without changing the aspect ratio, use the mouse wheel to zoom in or out. Alternatively, you can resize the model by changing the percentage value in the horizontal toolbar .

b. Terrain model has a dimension > 100%:

- All steps explained above are still possible.
- In addition to this, the white hand right next to the rotation button becomes active as well. After clicking on this so-called *view pan* button , it is also possible to move the model.

Exercise 3.4d: Rotating the model

After activating the rotation button , the model can be rotated along the rotation axis, by moving the cursor directly on one of the axis until the cursor changes from to . Otherwise a freehand rotation in any direction is possible.

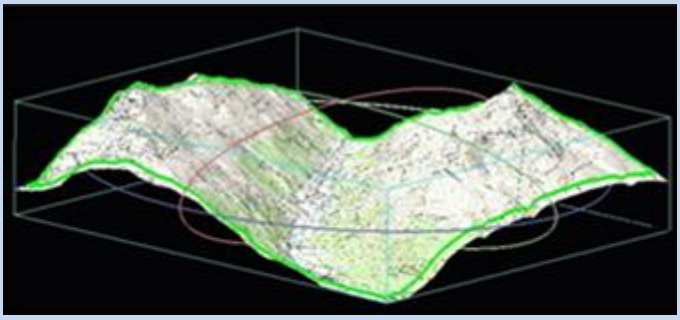


Figure 3.23: *Active* project with rotation axes.

Exercise 3.4e: How to switch between 2D and 3D mode

Click to switch from 3D to 2D view. This button then changes to and by clicking again, you will return to 3D view.

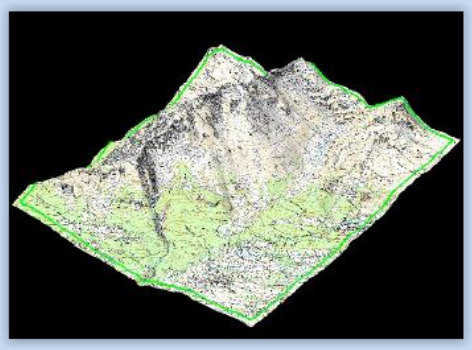


Figure 3.24: 3D view of example model.

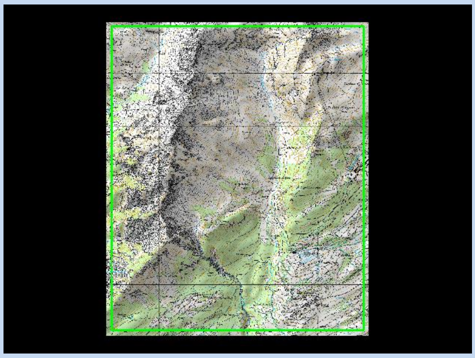


Figure 3.25: 2D view of example model.

In 2D mode you have all possibilities that work for the 3D mode. It works for input files as well as for simulations. **For the following functions of RAMMS it is necessary to switch from 3D to 2D view:**

INPUT:

- Draw new release area
- Release area formation
- Crop release area
- Draw new domain

OUTPUT:

- Draw new line profile

3.4.4 Colorbar

As soon as a parameter is shown in the project, the colorbar appears on the right side of the main window. It can be turned on and off by clicking on .

The colorbar can be moved anywhere in the screen (and can get lost). Use ***Project*** → ***Get Colorbar*** to find a lost colorbar.

Exercise 3.4f: Editing the colorbar

Changing the minimum and maximum values of the colorbar as well as changing the number of colors used is done in the panel *DEBRISFLOW* (right of the map window) in the tab *Display*.

- Simply type a new value into the respective field and hit the return key on the keyboard. The display will be refreshed.
- To view the underlying topography or image, you can change the transparency.
- **ATTENTION**
  Values < x.xxx are not displayed!
  The cutoff depends on the min and max values as well as on the number of colors. Make sure that you have the range of values you want to display!
- Open the editing window by either choosing ***Edit*** → ***Colorbar Properties*** or clicking in the vertical toolbar.
- To change the colorbar properties simply click into the field you want to change, then click *OK*.
- Under ***Edit*** → ***Colorbar White Color*** the text-color of the colorbar can be changed to white. This can be useful when changing the background color of your project to white ***Track*** → ***Preferences*** → ***Debris flow Tab*** → ***Background Color***.

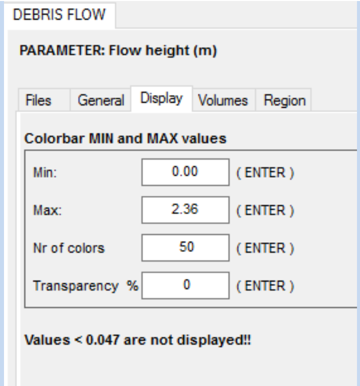


Figure 3.26: The Display tab.

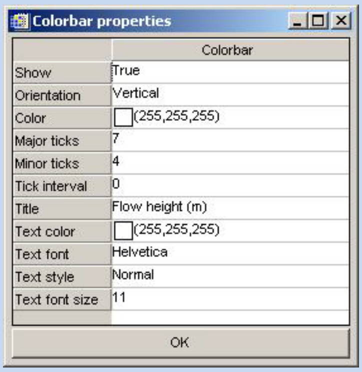


Figure 3.27: The Colorbar Properties window.

3.4.5 How to save input files and program settings

Once a project is created, it is saved under the name and location you entered during step 1 of the RAMMS::DEBRISFLOW Project Wizard (see figure 3.9 on page 26). The created input file has the ending *.db2.

The second situation, in which the input file is saved automatically, is when a calculation is started. The saved input file has the same name as the created output file.

Exercise 3.4g: How to save input files and program settings manually

a. Input file:

- In case you want to save the input file manually before running a calculation, go on ***Track*** → ***Save***. This is helpful when a release area was loaded but you wish to close the project before doing the simulation.
- If you wish to save a copy of your file under a new name, go to ***Track*** → ***Save Copy As*** or click .
- A window pops up to choose an old file which should be overwritten or to type in a new name, then click ***Save.***

b. Program settings

- If you have moved and/or or rotated your project for a better view, you can save this position by going on ***Extras*** → ***Save Active Position***.
- You can now get back to this position anytime by choosing ***Extras*** → ***Reload Position***.

Exercise 3.4h : How to open an input file

- Go to ***Track*** → ***Open*** → ***Input File,*** click or use ***Ctrl+O***.
- A window opens to browse for a debris flow input file (.db2).
- Click ***Open*** after the file name was selected.
- The project will be opened.
- Alternatively, you can drag & drop the input file from your windows explorer onto the RAMMS GUI.

Exercise 3.4i : How to visualize a shapefile

- To visualize a shapefile go to ***GIS*** → ***Add data*** or click .
- A window opens to browse for a shapefile (*.shp).
- Click ***Open*** after the file was selected.
- Alternatively, you can drag & drop the shapefile from your windows explorer onto the RAMMS GUI.

Exercise 3.4j : How to open an output file/debris flow simulation

- Go to ***Track* → *Open...* → *Debris Flow Simulation*,** click or use ***Ctrl+A*.**
- A window opens to browse for a debris flow simulation file (*.out.gz)
- Click ***OK***
- The simulation will be opened.
- Alternatively, you can drag & drop the output file from your windows explorer onto the RAMMS GUI.

3.4.6 About RAMMS

Some information about the RAMMS installation on your computer is found here: ***Help* → *About RAMMS***.

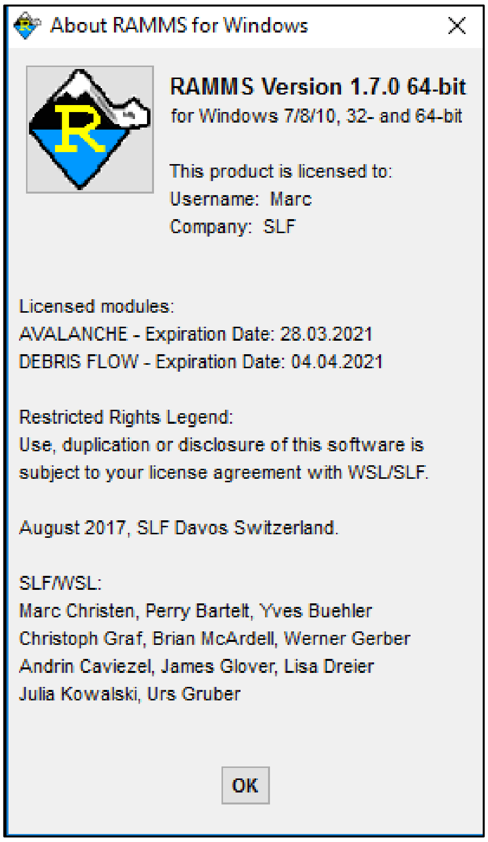


Figure 3.28: About RAMMS::DEBRISFLOW

3.5 Running a simulation

To run a calculation or a specific scenario within a newly created project (creating a project see section 4.3) it is necessary to

- define a release area or an input hydrograph,
- an erosion area (not mandatory),
- a calculation domain and
- friction parameters μ and ξ.

The definition of a smaller calculation domain is especially useful to keep the number of calculation points as small as possible, that is, it is best to test what the flow path of a simulation will be to limit the calculation domain to this extent. The exercises below show you how to create a release area, how to use an input hydrograph and how to create a calculation domain. A simulation also requires a set of friction parameters. Details on the friction model used in RAMMS::Debris Flow are given in section 3.1.4.

3.5.1 Release area

There are different possibilities to include a release area into the project. The following table gives an overview of the possibilities RAMMS offers. For further explanations see the exercises below.

Create a new release area (exercise 3.5a)	If there is no release area available for your project, or you wish to create a new one, switch to 2D mode and click (Draw new polygon shapefile)
Open an existing polygon shapefile (exercise 3.5b)	Use the file-tree in the right-hand panel (*Files*) and click the shapefile you want to visualize. Or, use the *'Add data'* button or menu to visualize a shapefile from another source.

Exercise 3.5a : How to create a new release area (polygon shapefile)

- Switch to 2D mode by clicking .
- Activate the project by clicking on the map once.
- Click (*Draw new polygon shapefile*).
- Click into the project where you want to start drawing the outline of the release polygon.
- Continue drawing the release polygon by moving the cursor and clicking the left mouse button.
- To end the release polygon, click the right mouse button. The polygon will be closed automatically.

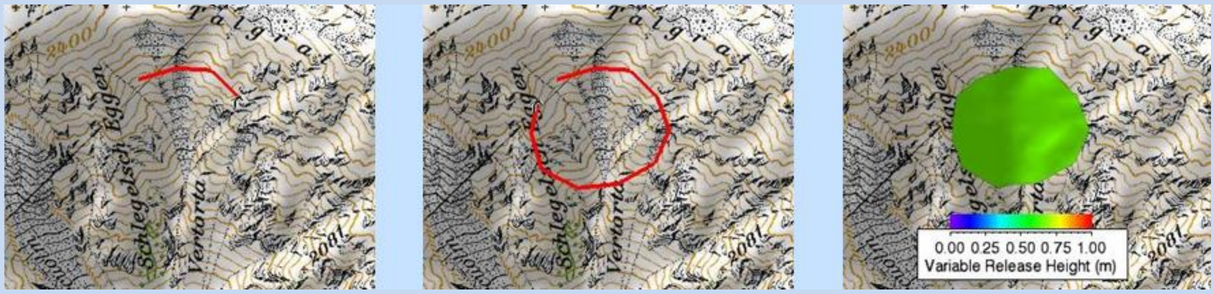


Figure 3.29: Project with emerging release area.

Before the release area is created, you have to answer a few questions:

- **Add more polygon areas?**
  You can either answer with **Yes** and create a second release polygon as explained above or answer with **No** and continue with the next step.
- **Choose a new polygon shapefile name:**
  Enter a new name for the polygon area(s). The ending *.shp* is added automatically.

The polygon shapefile will now be created and opened directly. The colorbar is not yet shown, because you have to define a release depth first (in case of a block release, see next exercise), or you have to define the polygon area as a hydrograph area (see exercise 3.5c, no colorbar needed).

Exercise 3.5b : How to open an existing release area (polygon shapefile)

- Use the file-tree in the right-hand panel (Files) and click the shapefile you want to visualize (see chapter 3.4.1.).
- Alternatively, you can use the Add data button or menu *'GIS → Add data'* to add a shapefile from another source than your project directory to your visualization.

Once a polygon area is created or opened, you can specify the release depth (in case of a block release). Switch to **2D mode**, choose **Input → Release area... → Details/Edit release area** , click the button or right-click the polygon shapefile in the Files-Tab and choose ***Release properties***, and choose the release area polygon by selecting it with the left mouse button. The appearing window yields information about release area, mean slope angle, mean altitude and estimated release volume. And, most importantly, the release depth can be entered, see exercise below.

Additional release information is found in the Debris Flow panel, tab Volumes (Figure 3.30 and Figure 3.31) below.

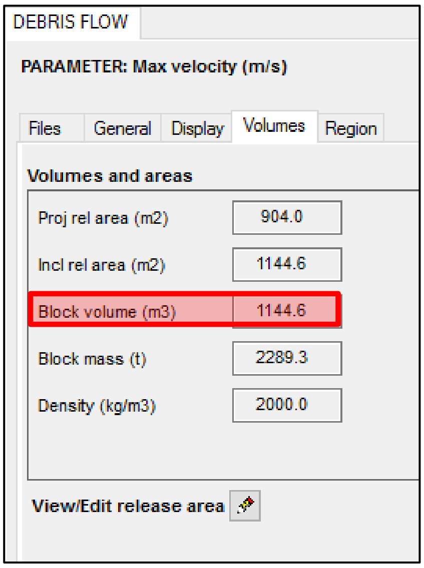


Figure 3.30 : Block release area and volume information.

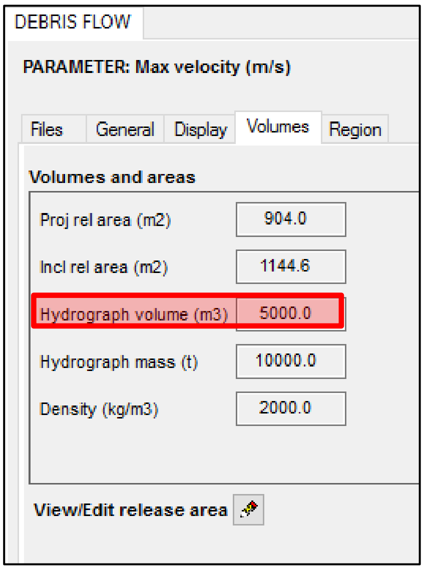


Figure 3.31: Hydrograph area and volume information.

Exercise 3.5b : Specify release depth and view release information (block release)

- Switch to 2D mode by clicking .
- Activate the project by clicking on the topography once.
- Click on the **View/Edit release area** button (in the horizontal toolbar or in the volumes tab in the panel), choose **Input → Release area... → Details/Edit release area** or right-click the shapefile in the *Files* tab and select **Release properties.**
- Then click on the release area you want to get information on. A red polygon is drawn around the selected release area. The following window appears:

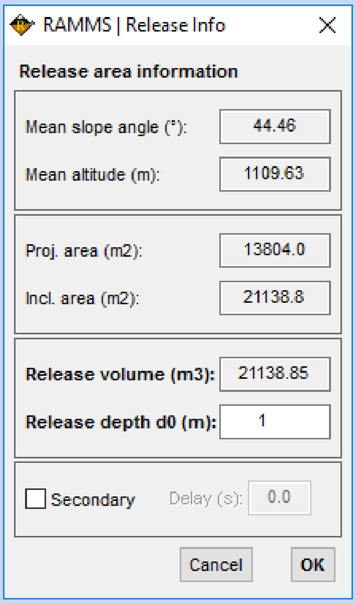


Figure 3.32 Release area information window.

- To change the release depth enter a new value (the resulting release volume is directly adjusted). Click **OK** if you want to keep the changes, **Cancel** otherwise.

3.5.2 Input hydrograph

To simulate channelized debris flows it is advantageous to use an input hydrograph. However, this requires knowledge of the amount of material (discharge) that flows past at a given location. For a RAMMS simulation with a hydrograph you have two options. Either you know the discharge at different times at a given location, e.g. estimated by measured flow heights and corresponding channel cross sections, or you use the estimated total volume in combination with empirical relationships between total volume and maximum discharge (e.g. Rickenmann et al. 1999).

Field measurement of debris flow properties is beyond the scope of this handbook, however if debris flow activity in a catchment is frequent, it may be possible to obtain additional information for calibrating RAMMS by installing a monitoring station. Many debris flows, when the monitoring data are simplified, have a relatively simple triangular wave-like shape (Figure 3.33). Assuming a well-estimated total volume (e.g. field data), maximum debris flow discharge (*Q*) and corresponding time (*t1*), as well as the end time (*t2*) can be calculated. We call this a three-point hydrograph calculation.

In nature, discharge hydrographs are generally better described by four points (Figure 3.34 and Fig-

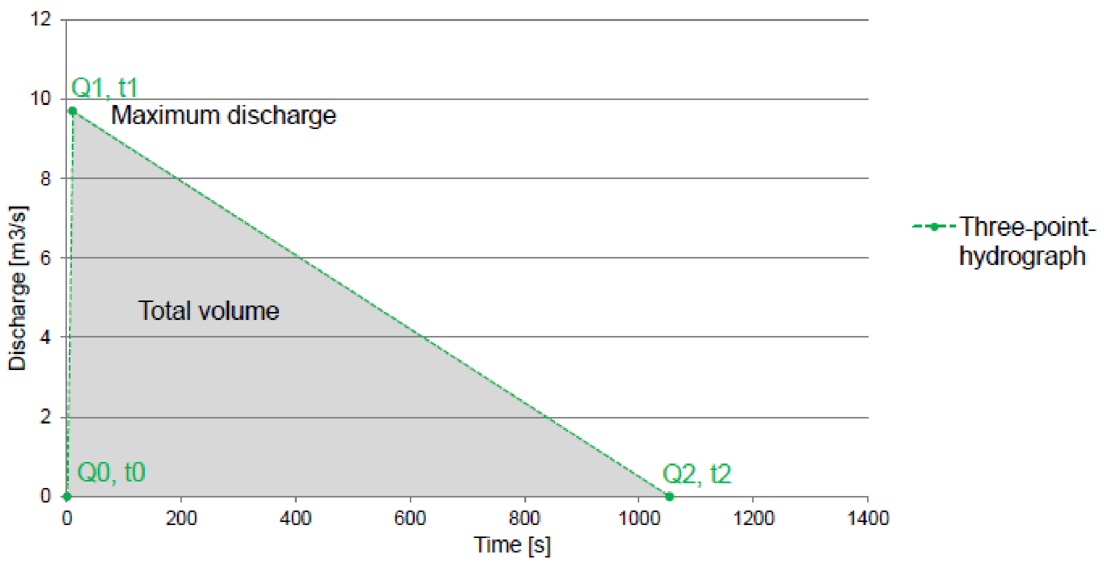


Figure 3.33: Three-point hydrograph with total volume of debris flow.

ure 3.35). However, a three-point hydrograph results in slightly larger flow heights and flow velocities after the maximum discharge. If the detailed discharge hydrograph is not known it is useful to choose a three-point hydrograph. In critical applications we encourage users to evaluate the sensitivity of the model results to the shape of the input hydrograph, e.g. when preparing different scenarios as a basis for hazard maps.

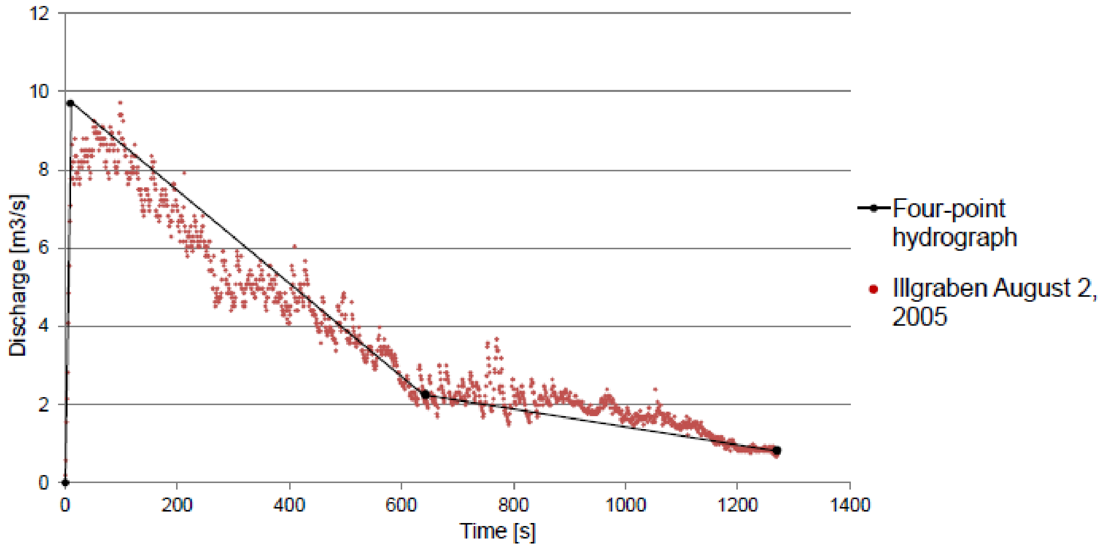


Figure 3.34: Four-point hydrograph for discharge values of an event, August 2, 2005, at Illgraben,Valais, Switzerland.

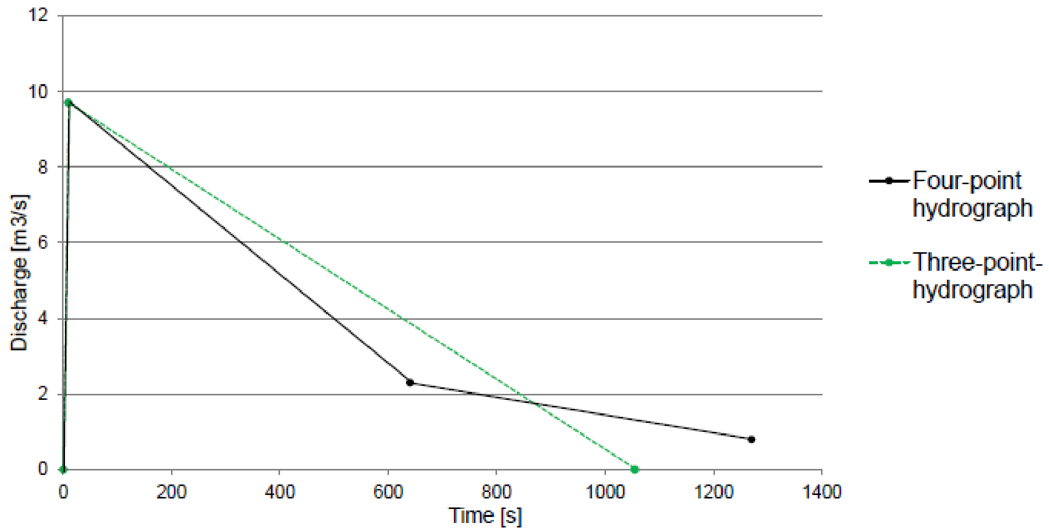


Figure 3.35: Comparison of a three-point with a four-point hydrograph for the given discharge data with the same total volume.

Input data for a discharge hydrograph

In RAMMS::Debris Flow one has the option to enter up to 10 points to define an input hydrograph. It requires discharge *Q*, time *t* and velocity *v* at different times. A three-point hydrograph for the event shown in Figs. 4.24 and 4.25 might look as following:

Table 3.2: Data for discharge hydrograph.

	Time *t* [s]	Discharge *Q* [m^3/s]	Velocity *v* [m/s]
Point 0	0	0	6
Point 1	2.2	8.8	6
Point 2	1500	0	3

The other option is to define total volume V_{tot}, corresponding maximum discharge Q_{max} and time

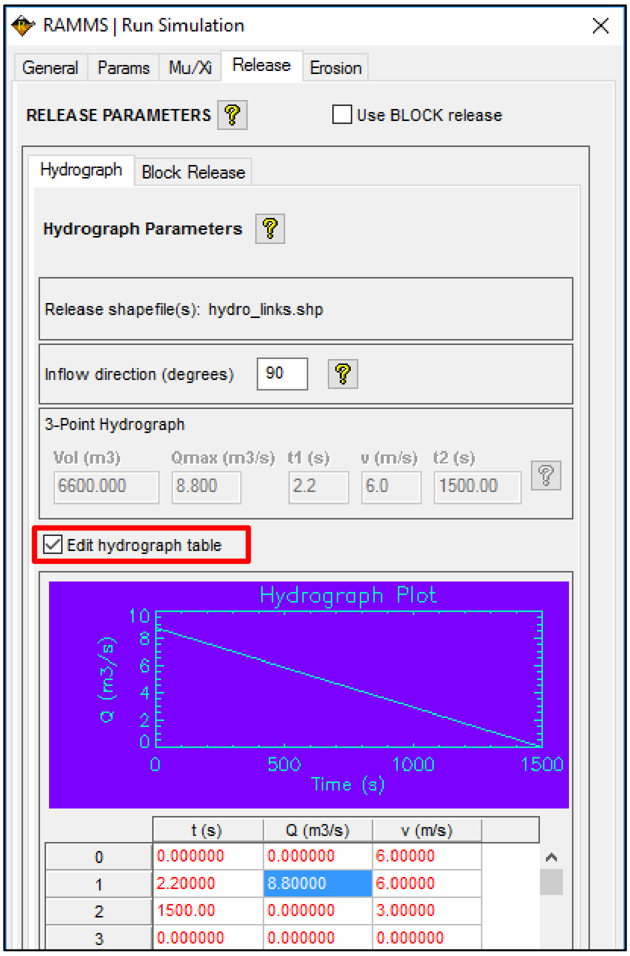


Figure 3.36: Edit hydrograph table. The checkbox (red rectangle) must be set to be able to edit the hydrograph table.

when the maximum discharge occurs t_{max}. The time of maximum discharge is typically observed just upslope of the leading edge of the flow, probably on the order of a few seconds to perhaps a few 10's of seconds after the arrival of a debris flow at any given location. This allows the calculation of the time at which the total volume passed the hydrograph location (t_{end}). Discharge is linearly interpolated between known Q-values.

In both cases, the inflow direction has to be defined, which is described as the angle in counterclockwise direction from the x-coordinate of the topographic data (Figure 3.37).

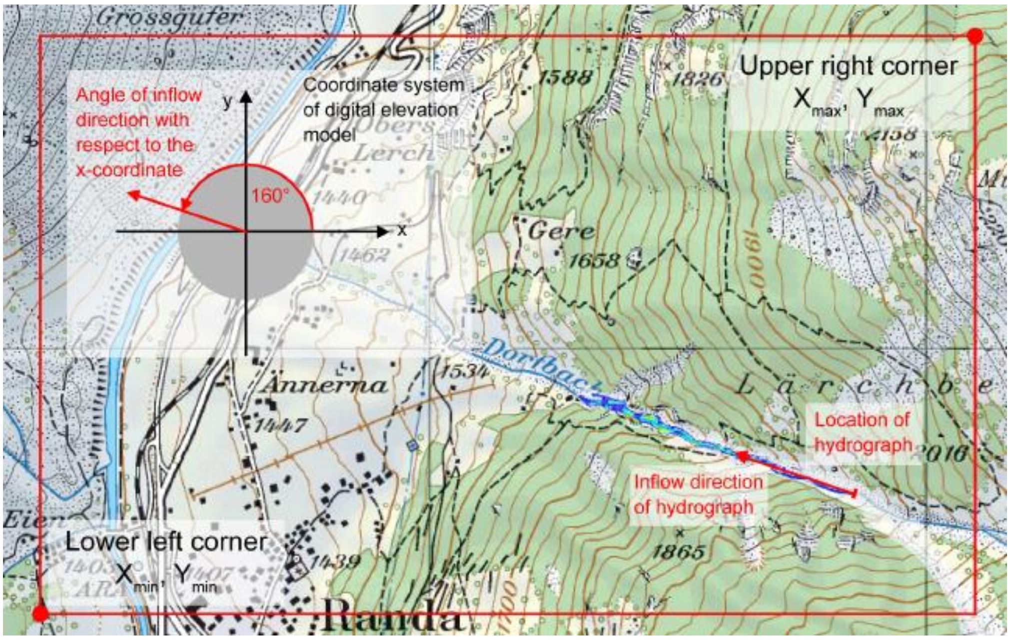


Figure 3.37: Definition of inflow direction of an input hydrograph.

Exercise 3.5c: How to use the input hydrograph

Hydrograph location and calculation domain

- Draw a polygon area which extends over the potential debris flow width with the **'Draw new polygon shapefile'** tool (see section 3.5.1) where you want to place the input hydrograph (Figure 3.50). Be sure your release area covers several grid cells. The input will initially flow downhill perpendicular to the line. Name your release area accordingly.
- Right-click your release area shapefile and choose *Set as hydrograph*.

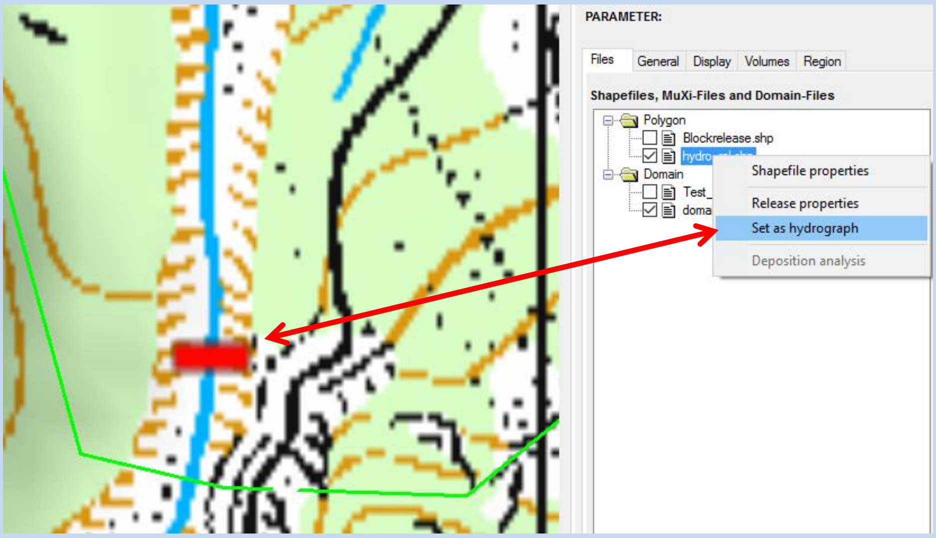


Figure 3.38: Beam shaped polygon area (red rectangle) for the input hydrograph inside the calculation domain (green line).

- Draw a **calculation domain** (see section 3.5.4), such that the area of the input hydrograph is located inside the domain

Enter data of discharge hydrograph

- Once you start to **RUN a calculation** (see section 3.5.4) you can enter the discharge hydrograph information in the corresponding tab **Hydrograph**. For details on how to enter the hydrograph data see beginning of this section 3.5.2.

3.5.3 Erosion

To include one or several erosion areas in the simulation, do the following:

- Draw polygon shapefiles for varying erosion areas. Use *Input* → *Polygon shapefile* → *Draw New Polygon Shapefile* to do this.
- Open the *Run Simulation* window, change to the *Erosion Tab* and start to enter the erosion characteristics.

1. Select shapefile

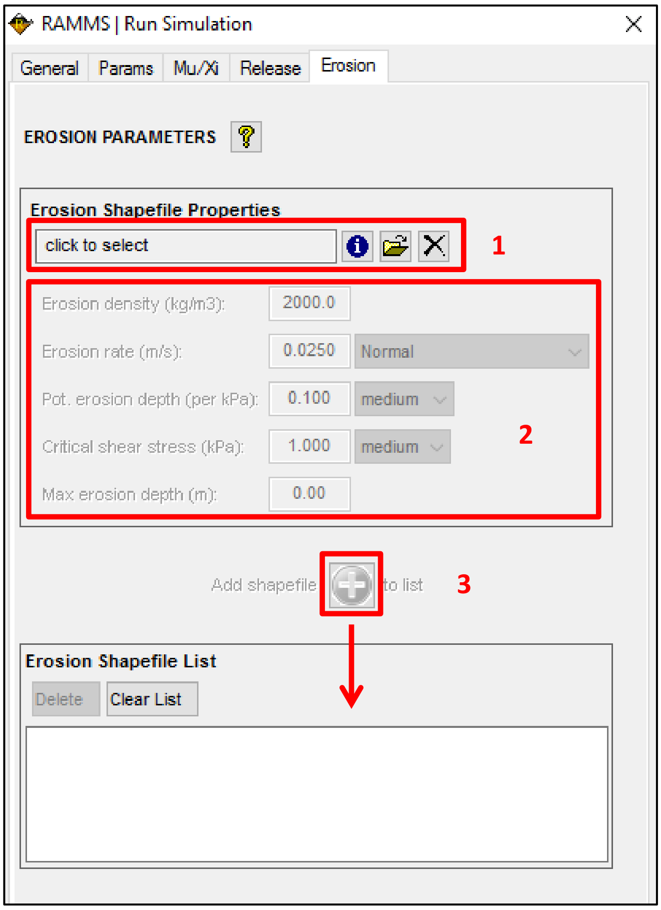


Figure 3.39: Erosion Tab

2. Choose/select parameters, see Table 3.3 below for details.
3. Click the **+ button** to add your selection to the *Erosion Shapefile List*.

...do the same for every shapefile you want to use.

Erosion Parameters

Table 3.3: Parameters for erosion areas.

Erosion density: (kg/m3)	We suggest to use the default density (or the density that you selected in the Params tab) unless you have more detailed information.
Erosion rate: (m/s, default 0.025)	The rate at which debris flows entrain material from the sediment bed. There are very few values available from the field (see Frank et al.), so we propose using a constant value based on the work published by Berger et al., 2011 (0.025 m/s), for the Illgraben torrent (erosion rates measured using sensors buried in the channel bed). This is a rate that the model will use for entrainment until the predicted erosion depth is reached. Increasing this value will cause sediment to be entrained at a faster rate, potentially resulting in relatively large debris flow snouts.
Pot. erosion depth: (per kPa, default 0.1)	This parameter controls the maximum potential erosion depth (variable e_m in the paper by Frank et al.) as a function of the maximum shear stress reached in each cell. This value actually is the slope of the line which relates e_m to the maximum shear stress calculated for each cell. By changing the value of this parameter to e.g. 0.2 (menu choice *deep*) you will double the maximum potential erosion depth.
Critical shear stress: (kPa, default 1.0)	This is the value at which erosion can start occurring. It might be reasonable to expect that a channel bed consisting of interlocked boulders would have a larger critical shear stress, while a channel bed of saturated sandy gravel would have a lower value. Our value is just a curve fit to the data reported by Schürch et al. (2011).
Max erosion depth: (m)	Here we can set a maximum thickness of the layer of erodible sediment if such data are available (e.g. in some cases we may know that there is only a limited amount of sediment available).

3.5.4 Calculation Domain

To reduce calculation time you can specify a smaller calculation domain to reduce the number of computational cells. By analyzing a calculation with a coarse grid (large cell size), e.g. with a cell size of 5 or 10 m, you get an idea where the flow path is situated and you can limit the calculation domain to the area of interest.

Switch to *2D mode* and choose ***Input → Calculation Domain... → Draw New Domain*** or click . Now you can draw a polygon containing the area of interest similar to drawing a new polygon area (see exercise "Create release area" on page 34). We strongly recommend using smaller calculation domains especially if you calculate with small cell sizes (e.g. < 5m).

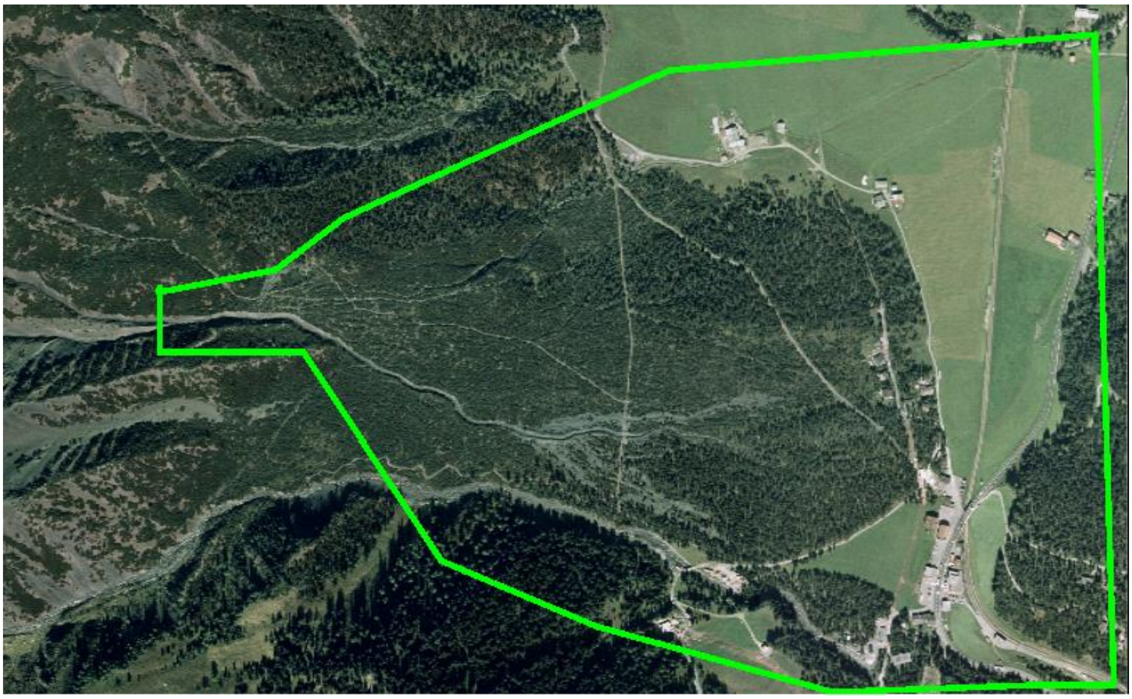


Figure 3.40: Calculation domain in green encloses the area of interest and reduces calculation time incomparison with the default rectangular domain which is automatically generated.

Exercise 3.5d : Finding an optimized calculation domain

- Open your input file.
- Draw a rough calculation domain as explained above, see Figure 3.41 below.

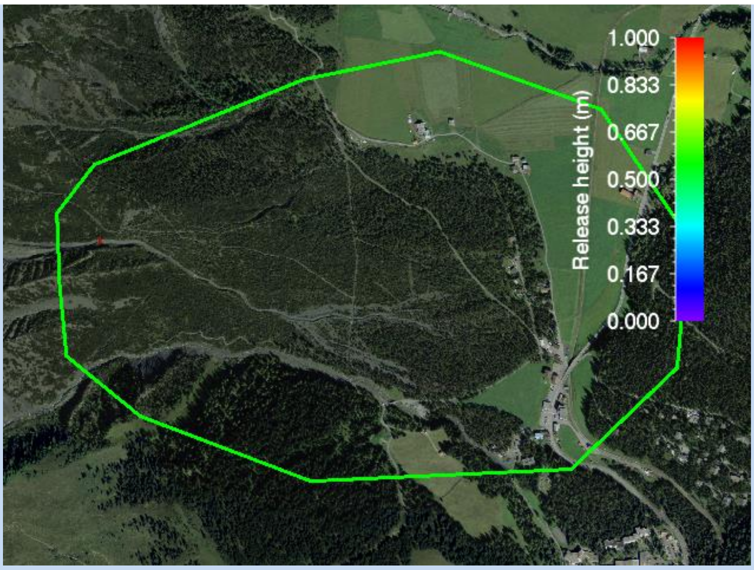


Figure 3.41: Input file with a rough calculation domain

- Do a rough calculation with a simulation resolution of 4m (instead of 2m, e.g.).
- Wait for the simulation to finish. The simulation result will be displayed.
- Click the *Max Flow Height* button .

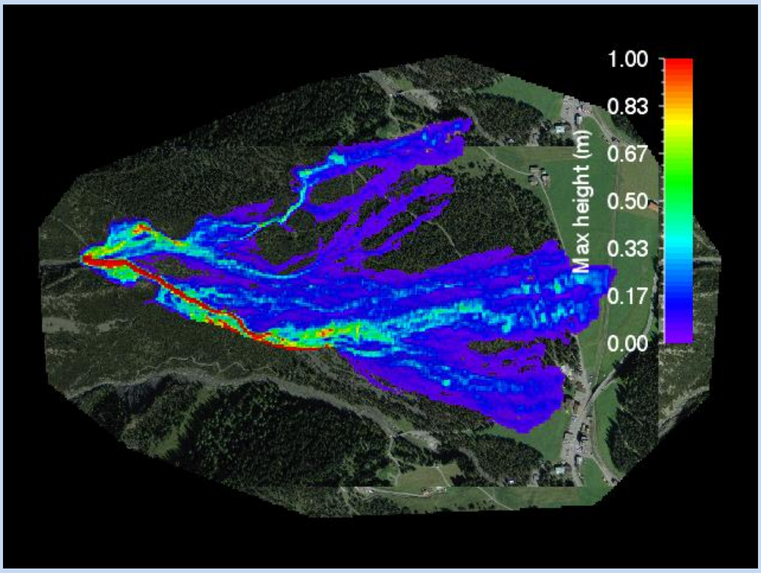


Figure 3.42: Max Flow Height of a 4m simulation

- Click *GIS* → *Export…* → *Create Envelope Shapefile*
- Question: *Use buffer?*
- Click **Yes**. We want to use the envelope shapefile as a calculation domain, and therefore we want to buffer it a little bit (click **No** if you want to have the exact envelope).
- Choose a filename for the envelope shapefile (a name is proposed).
- The created envelope shapefile is shown in the visualization as a dashed red line.

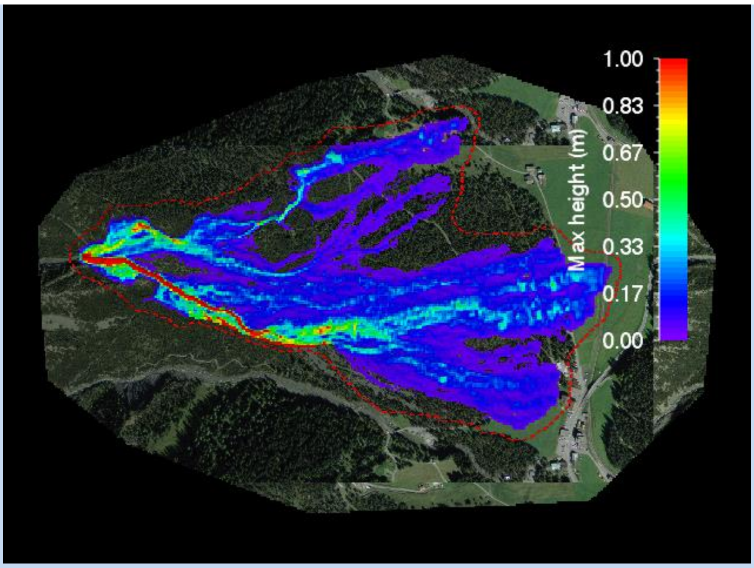


Figure 3.43: Envelope shapefile (dashed red line) of Max Flow Height extent

- Switch back to the input file
- Use *Input* → *Calculation Domain…* → *Load Existing Domain* to load your envelope shapefile as a new calculation domain, see Figure 3.44 below.

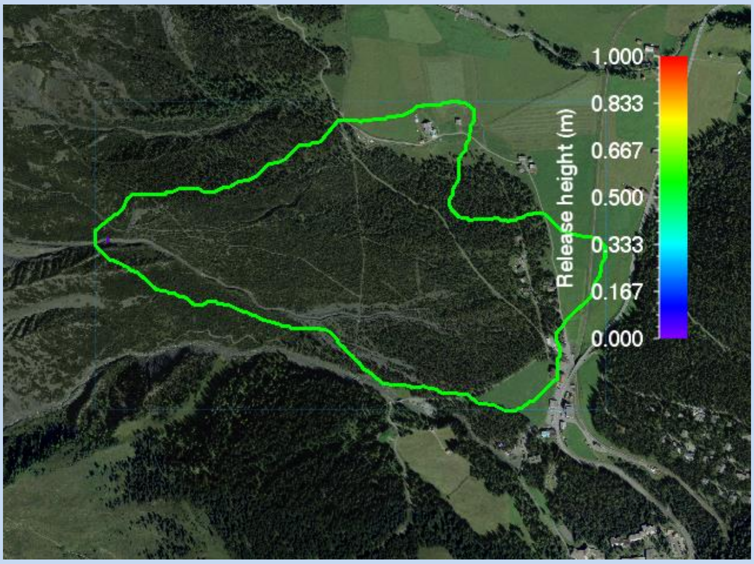


Figure 3.44: Input file with optimized calculation domain (envelope shapefile)

- Now redo your simulation with a simulation resolution of 2m.

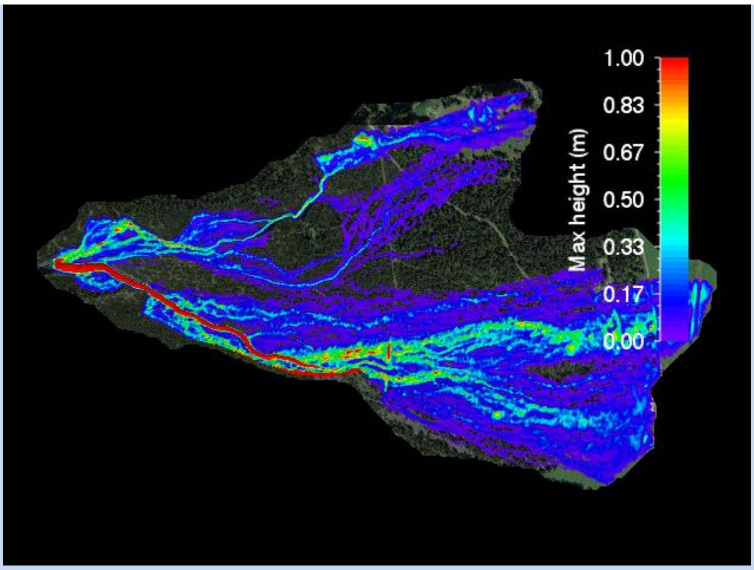


Figure 3.45: Max Flow Height result of a 2m simulation

- In this example, the much smaller new calculation domain saves **50% !!!!!** of the computational time.

3.5.5 How to run a calculation

To run a calculation you have to open a created project (section 3.3), load a release area (section 3.5.1), and draw a calculation domain (section 3.5.4). Below you find an example for running a calculation with constant release height and constant friction parameters μ and ξ.

Exercise 3.5e: How to run a calculation

- To run a simulation choose ***Run*** → ***Run Calculation*** or click 
- The **RAMMS | Run Simulation** window opens. Before clicking ***Run Simulation***, you should check the input parameters.

General Tab:

(1) **Output Name:** Choose a good output filename, add parameter information to the filename to recognize the output file.

(2) **Project name:**

(3) **Details:** Add valuable project information to this field.

(4) **Additional information:** Calculation domain file and digital elevation model (DEM).

(5) **Stop Parameter:** The stopping criteria in RAMMS is based on the momentum, see section 4.2.5 on page 65.

(6) **Remarks:** “Escape” and “Ctrl+R” can be used to cancel resp. start a simulation.

(7) Check box **Run in background**: Option to run simulations in background mode. The RAMMS interface remains active and allows the user to start e.g. new simulations.

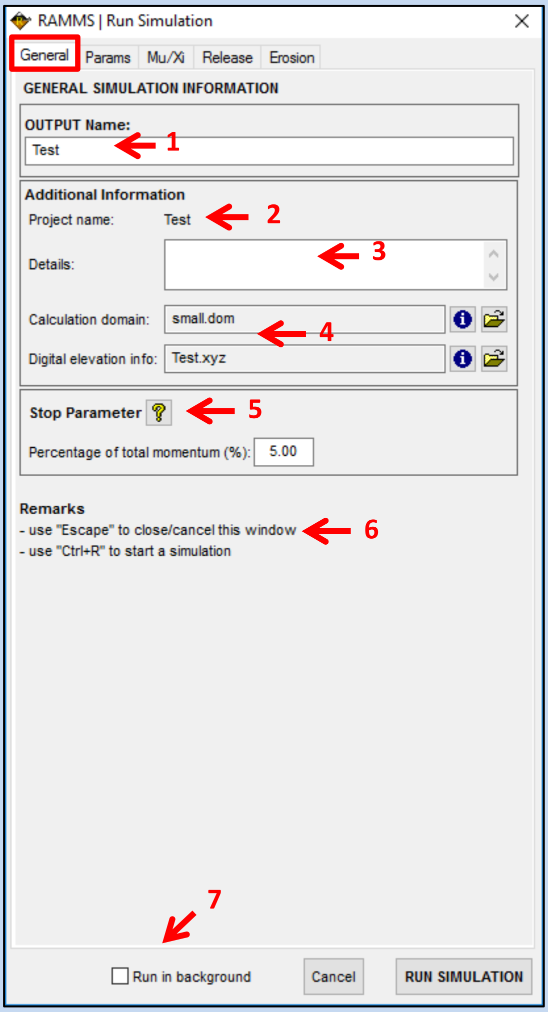


Figure 3.46 General Information

Continuation of exercise 3.5e: How to run a simulation

Params Tab

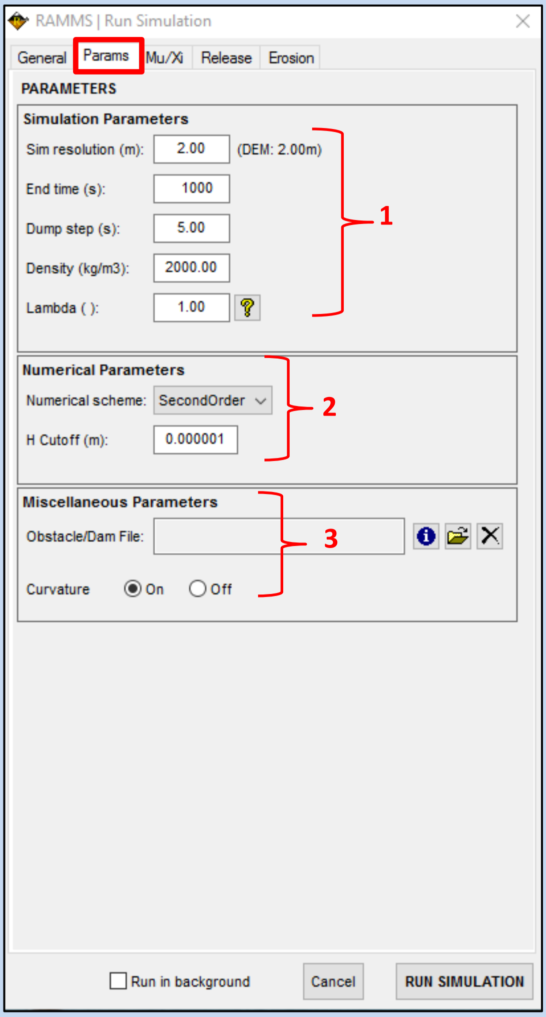


Figure 3.47: Parameter Tab

(1) Simulation Parameters

- Sim resolution: Change, if necessary. The resolution should always be chosen so that important features of the terrain are represented in the terrain model. High resolution grids will extend your calculation time.
- End time: Choose simulation end time.
- Dump-step: The dump-step interval defines the resolution of the animation of your simulation but has no effect on the simulation results.
- Density: Keep the default value for density if no further information on the debris flow material is available (2000 kg/m^3).
- Lambda: Keep the default value for the earth-pressure coefficient Lambda (1.0). The parameter Lambda modifies the longitudinal pressure gradients driving the flow.

The default Lambda value of 1.0 disables the effect of Lambda. The use of Lambda other than 1.0 is only possible in 1st order numerical solver and will influence the simulation results. Results with a Lambda other than 1.0 have to be checked carefully and are not recommended.

(2) Numerical Parameters

- Numerical Scheme: Change numerical solver, 1st or 2nd order scheme. We recommend using 2nd order, because it provides more accurate solutions than 1st order. However if you encounter stability problems it may be useful to run your calculation using the 1st order numerical scheme
- Keep the default value for the Null-height H cutoff (0.000001 m). Unrealistic shallow flow heights of the simulation are eliminated to minimize numerical errors.

(3) Miscellaneous Parameters

- Obstacle/Dam File: Draw polygons of areas, where no flow should pass (houses, deflecting dams, obstacles). The flow is deflected.
- Curvature: Switch *Curvature* on or off.

Continuation of exercise 3.5e: How to run a simulation

MuXi Tab

(1) Enter the friction parameters **μ and** ξ. Start first scenario simulation with default values (μ = 0.2, ξ = 200 m/s^2). More information on frictional resistance is given in sections 3.1.4 and 5.1.

(2) Check box **Define Additional MuXi Areas:** if you want to add polygon regions with different μ and ξ values. e.g. to represent significant changes in the channel, heavily forested areas or perhaps smooth roads. Load the corresponding polygon shapefiles and define μ and ξ for up to two additional regions. Be very careful in using this feature!

(3) **Yield Stress:** Define a polygon area where you want to use an additional yield stress (cohesion). Specify a yield stress in the *Yield stress (Pa)* field. See section 3.1.4 on page 15 for more details about yield stress.

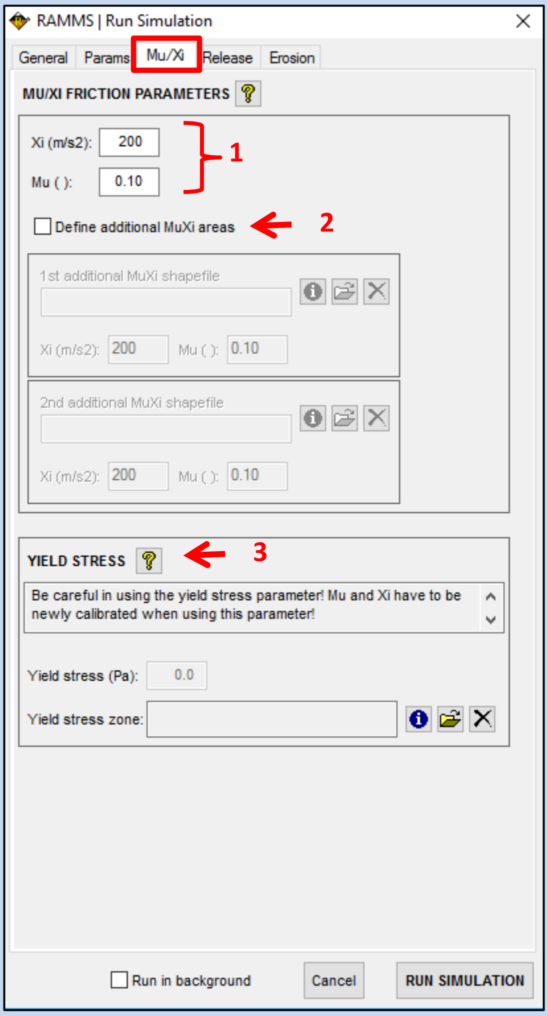


Figure 3.48: MuXi Tab

Continuation of exercise 3.5e: How to run a simulation – Hydrograph/Block release

Release - Hydrograph Tab

The default RELEASE mechanism for debris flows is a **HYDROGRAPH**!

(1) **Use BLOCK release** (Check box): As an alternative to a simulation with a hydrograph.

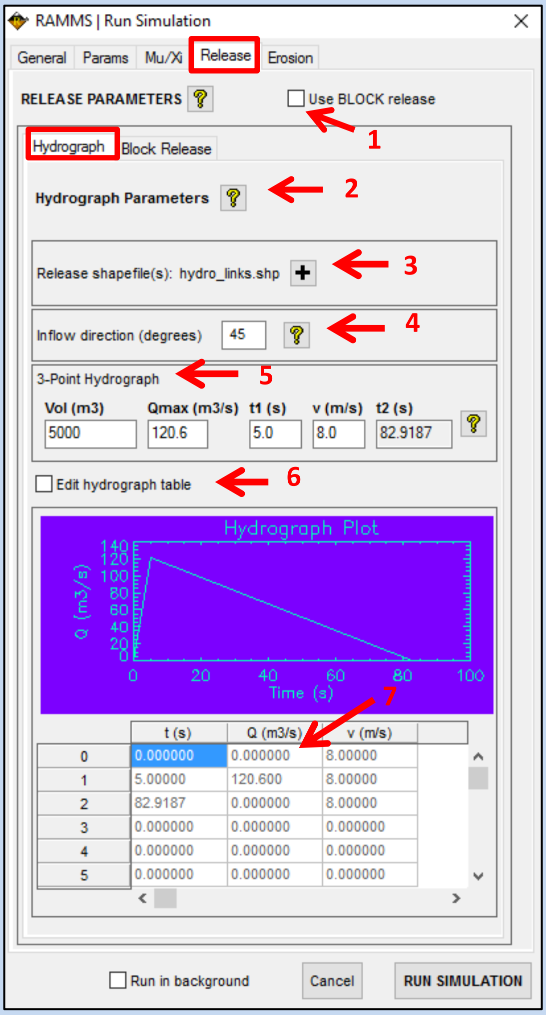
(2) **Hydrograph Help**

(3) **Hydrograph/Release Shapefile(s)**: Release shapefile(s). If more than one shapefile is specified, a + button is shown. A tooltip will show the additional shapefile names when moving the mouse over the button.

(4) **Inflow direction (degrees)**: Define the inflow direction of the debris flow at the hydrograph input location. Click the info button for a detailed description of inflow direction.

(5) **3-Point Hydrograph Calculation**: Define a discharge hydrograph by entering total volume *V* (maximum discharge Q_{max} is calculated automatically according to Rickenmann, 1999), corresponding time at which the maximum occurs (t_{max}) and velocity *v*. End time (t_{end}) of the input hydrograph is automatically calculated.

(6) **Edit hydrograph table** (Check box): Activates the table (7) and is an alternative to a 3-Point Hydrograph Calculation (5).



	t (s)	Q (m3/s)	v (m/s)
0	0.000000	0.000000	8.00000
1	5.00000	120.600	8.00000
2	82.9187	0.000000	8.00000
3	0.000000	0.000000	0.000000
4	0.000000	0.000000	0.000000
5	0.000000	0.000000	0.000000

Figure 3.49: Hydrograph Tab

Release - Block Release Tab

(1) **Use BLOCK release** (Check box): Click this checkbox to do a block release simulation.
(2) **Block Release Help**
(3) **Subtract release height from DEM** (Check box): Subtracts the release height from the DEM and uses a new DEM for the calculation.
(4) **Shapefile and Volume Information**: Overview of all used release areas with volume, depths and delays, as well as the total release volume.

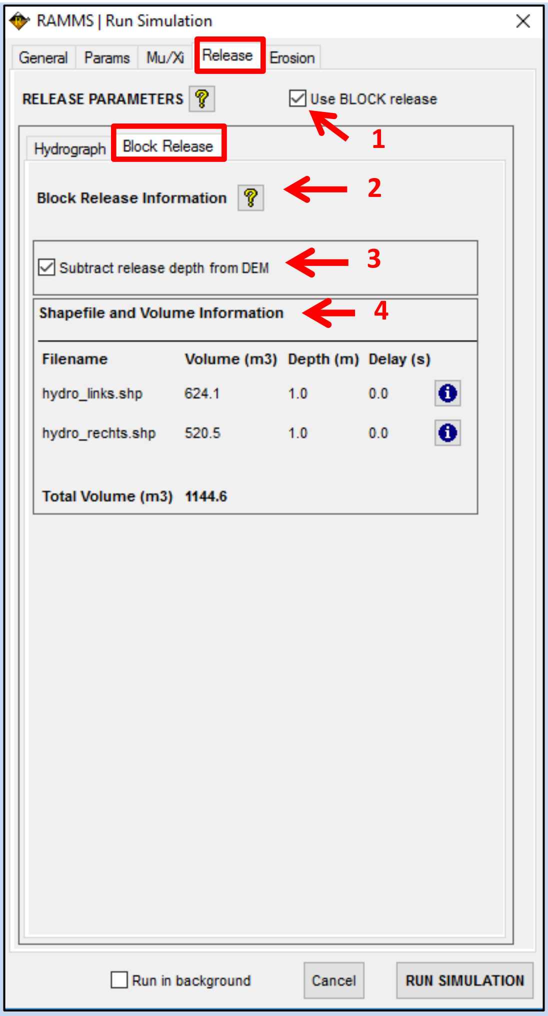


Figure 3.50: Hydrograph Tab

Continuation of exercise 3.5e: How to run a simulation – Erosion

Erosion Tab

(1) **Erosion Shapefile**: Specify your erosion shapefile area.
(2) **Erosion Parameters:** Specify erosion parameters. Use dropdown menus to select parameter values. See section 3.5.3 on page 41 for more details about erosion parameters.
(3) **Add to list button:** Click the button to add your erosion shapefile and parameters to your erosion list.
(4) **Erosion shapefile list:** List of erosion shapefiles and parameters. You can delete single entries or the whole list with the buttons *Delete* and *Clear List*.
(5) **Run Simulation Button**

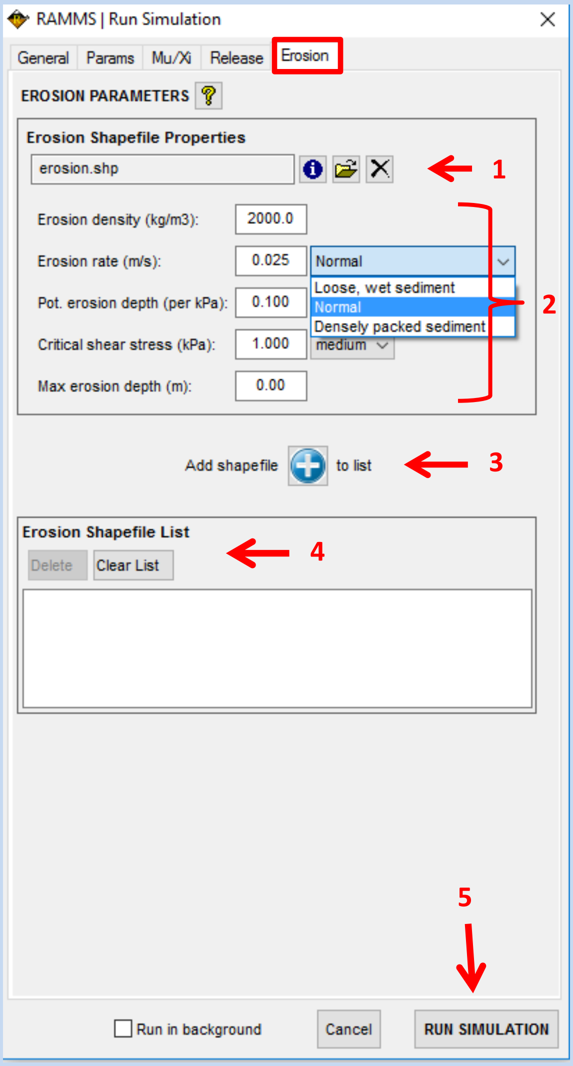


Figure 3.51: Erosion Tab

Continuation of exercise 3.5e: How to run a simulation – Start Simulation

- Click run simulation (Figure 3.51)
- If you want to start multiple simulations automatically (e.g. overnight) use *Track* → *New...* → *Run Batch Simulations*. You can choose how many computational cores the Batch-Mode should use.
- The following window appears, showing the status of the calculation (Figure 3.52) (1) General information of the simulation, (2) output file, (3) starting the calculation (4) for every time step RAMMS calculates the maximal values (height, velocity and pressure) as well as the outflow mass, the moving momentum and the flow volume.

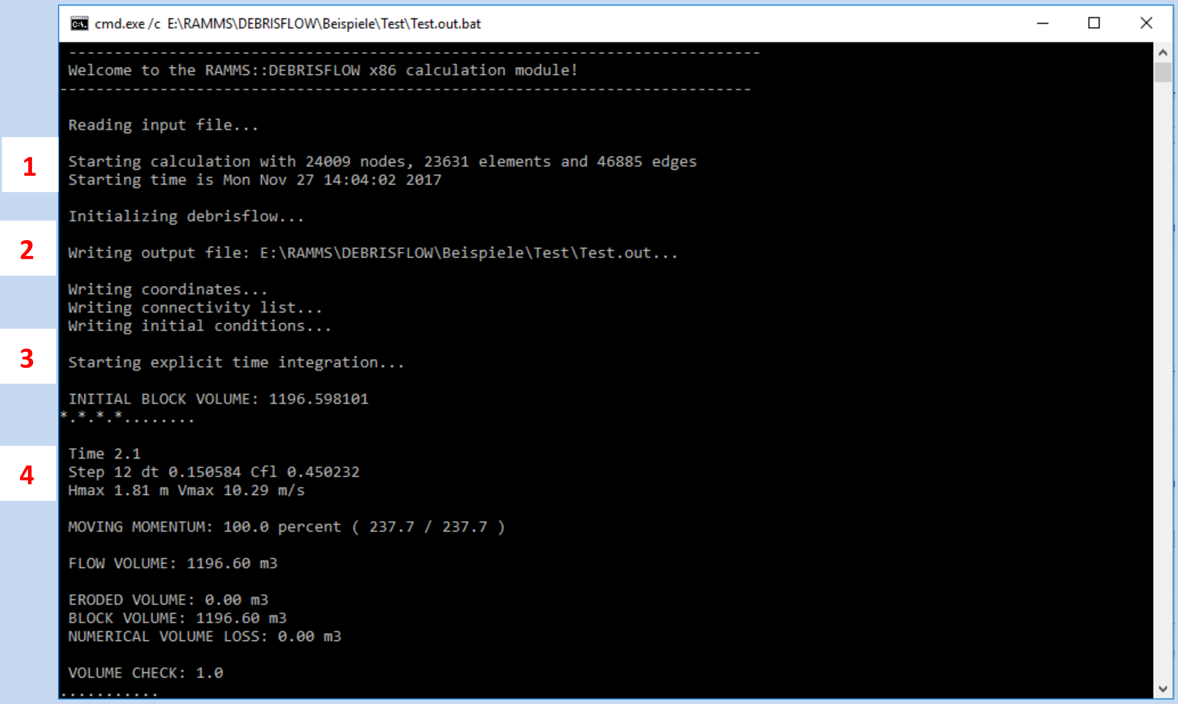


Figure 3.52: Standard output log window.

Figure 3. 53: Background simulation mode window.
Press any button to close the DOS window.

4 Results

Once the simulation is finished, the simulation as well as the output logfile (see Figure 4.3) are opened in RAMMS (if you ran the simulation in background mode, see Figure 3. 53, click any button inside the DOS window to close the window. Afterwards, open the simulation in RAMMS manually).

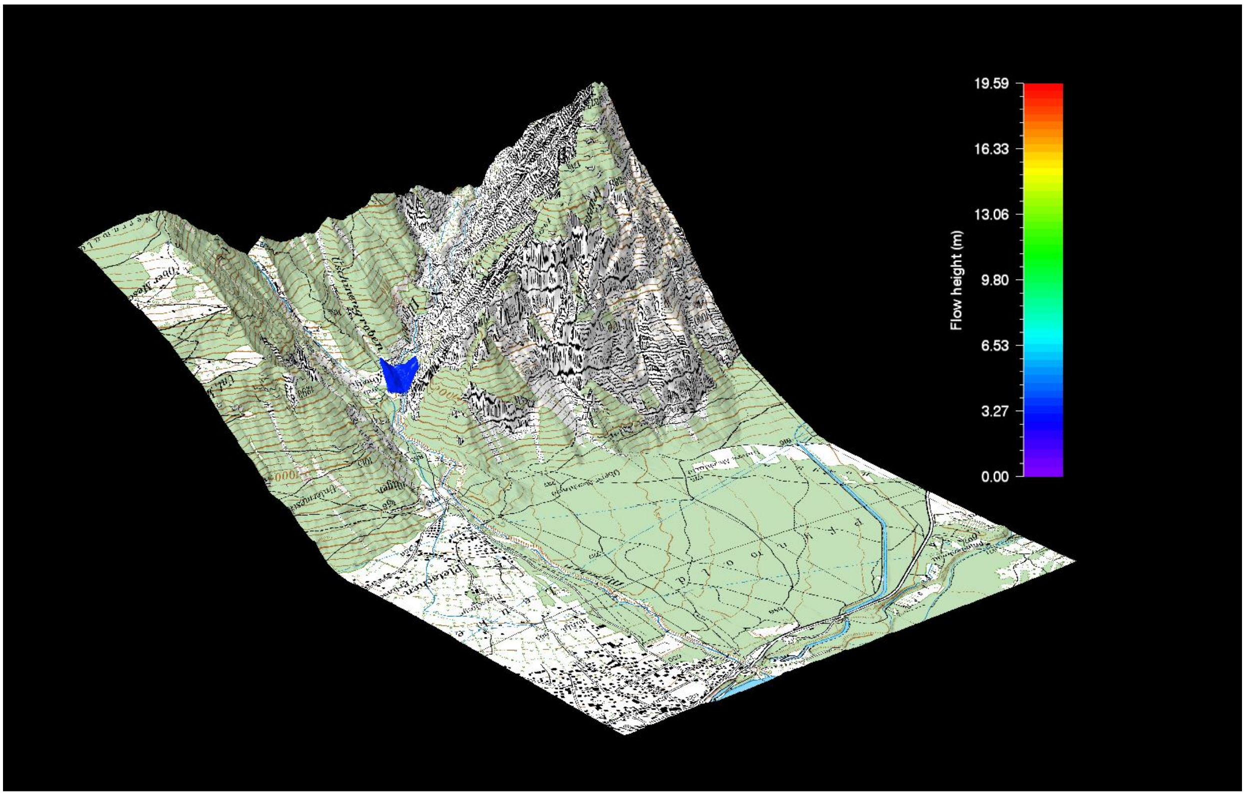


Figure 4.1: Main window in output mode.

If mass flows out of the calculation domain, RAMMS shows an alert (Figure 4.2). To get reliable results you should enlarge your calculation domain (see section 3.5.4).

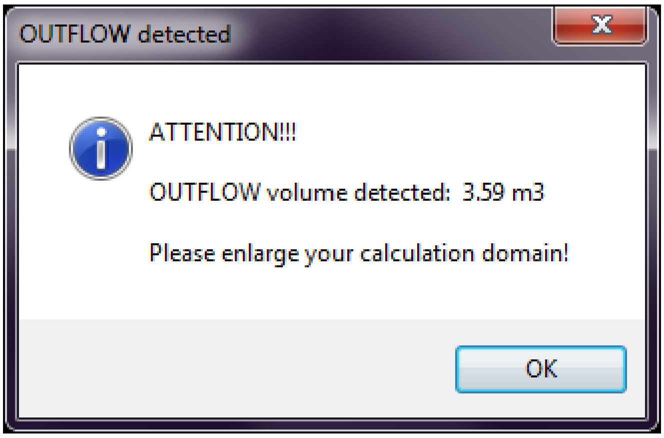


Figure 4.2: Outflow volume alert.

4.1 Project information

Once a scenario within a specific project is calculated it is possible to open the *output logfile* (in output mode) including project settings and information as well as calculation specifications. You can open the project's output log with *Project →Output Log File*. A window as shown in Figure 4.3 opens. This window provides information about your project and is the first thing to look at after running a simulation to check your simulation results.

(1) Information on simulation time and resolution. Be sure the simulation stopped due to *LOW FLUX*. Otherwise the output *TIME END CONDITION* informs you, that your simulation stopped before the debris flow reached the stopping criteria you defined for the simulation (see section 4.2.5 on page 65).

(2) Information on simulation results.

(3) Input logfile (see Figure 4.4).

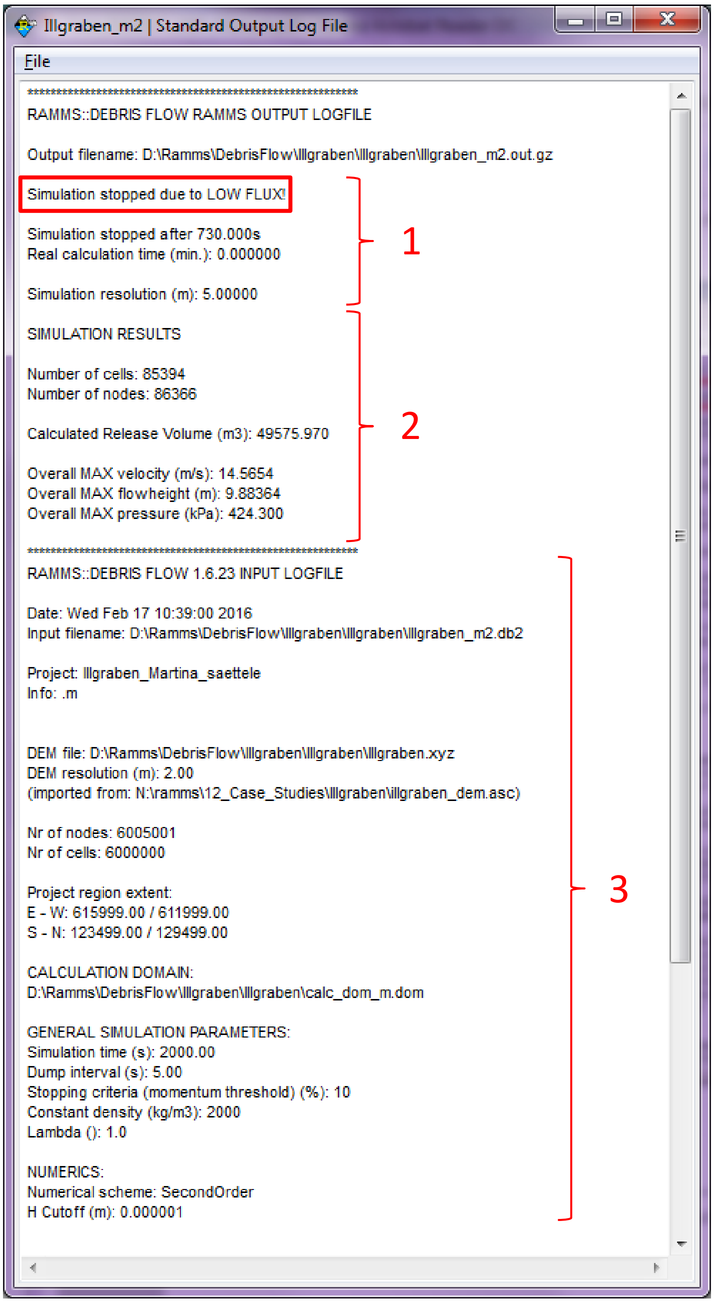


Figure 4.3: Output Logfile.

The *input logfile* (included in the output logfile), however, can already be opened once a project is created and before a simulation is performed.

There are two ways to view your project settings and information. First you can open your project's input logfile (or output logfile, in *output mode*), or you can check your project's region extent and area in the debris flow panel in the region tab.

You can open the project's input log file with *Project → Input Log File*. The following window opens:

This window provides information about all your project's input specifications, like number of nodes and cells, release areas, which DTM was used, the loaded map and orthophotos as well as your global simulation parameters.

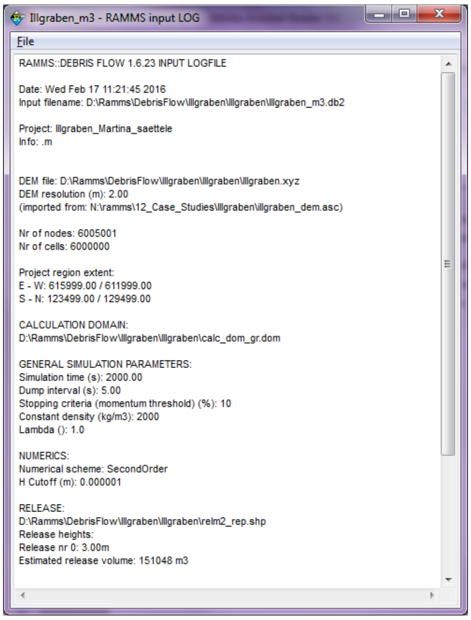


Figure 4.4: RAMMS Project Input Log file.

To view the project coordinates, click the region tab in your debris flow panel. The region tab lists X- and Y-Coordinates of the lower left (minimal values) and upper right (maximal values) corner (these are coordinates you entered when creating the project) as well as the global minimum and maximum altitude (Z value). Additionally, the total region area is shown (in km^2).

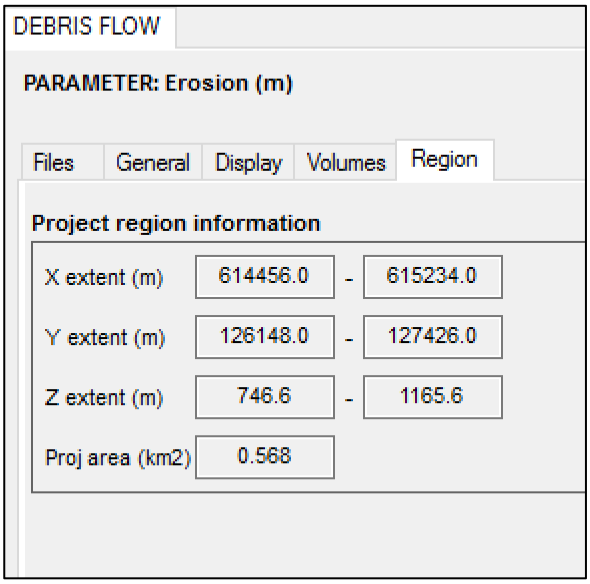


Figure 4.5: Region extent (X-, Y- and Z-Coordinates, total area).

4.2 Visualization and analysis of the results

This section gives a short overview on what is possible in RAMMS to view and analyze the simulation results. The interpretation of the results has to be done by an expert who is familiar with the local as well as with the topographic and meteorological situation of the investigation area.

RAMMS is a model and each model is a simplification of reality, therefore the simulation results should not be analyzed without questioning them. We strongly recommend that all users perform sensitivity studies.

4.2.1 Visualize different parameters

The drop down menu *Results* offers the following functions:

- Flow Height
- Flow Velocity
- Flow Pressure
- Flow Momentum
- Erosion
- Max values (Height, Velocity, Pressure, Momentum, Shear Stress, Erosion)
- DEM Adaptions (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (*μ, ξ*)

These results are all visualized in the topography. See exercise *"Displaying calculation values"* below.

Exercise 4.2a: Displaying calculation values

The maximum values of flow height, velocity and pressure give a good overview of the dimension of the debris flow. You find them under
Results → Max values…

- → **Max flow height**
- → **Max velocity**
- → **Max pressure**

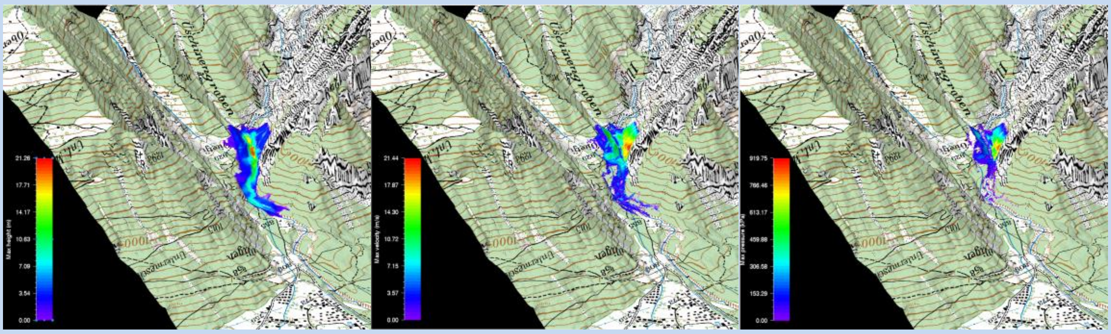


Figure 4.6: Results: Maximum values of flow height (left), velocity (middle) and pressure (right)

The flow height can be visualized exaggerated by a factor. Click ***Help → Advanced… →Additional Preferences… →Edit*** to change the factor of the quasi 3D-visualization of the flow height under the keyword **exaggeration**.

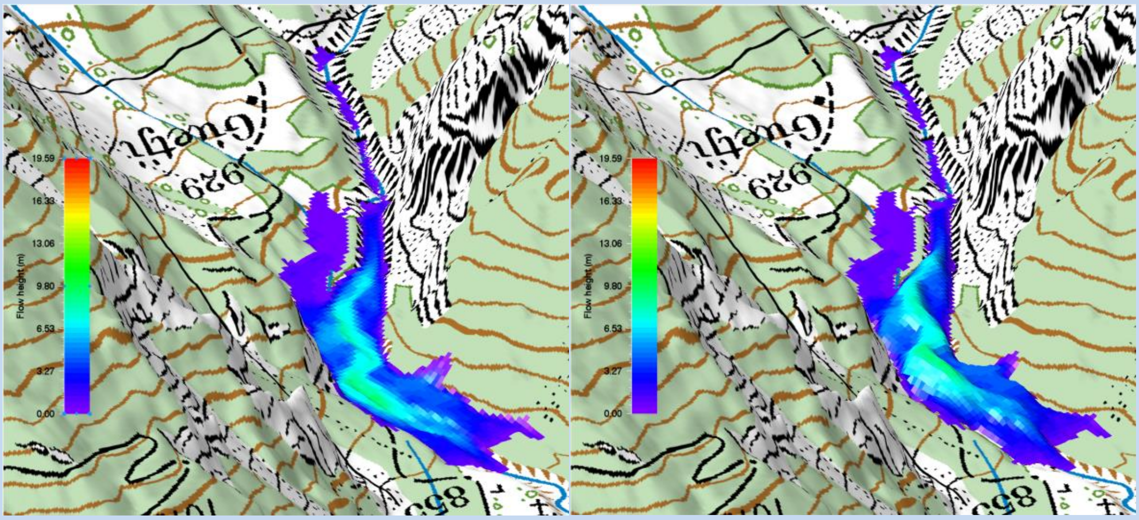


Figure 4.7: Quasi 3D-Visualization of flow height (left: exaggeration 1; right: exaggeration 5).

4.2.2 Line profile plot and time plot

In the horizontal toolbar you find two further functions:

- Line Profile 
- Time Plot

Line profile plot

A line profile is a good analysis tool, if the flow height, velocity or pressure should be known at a specific location. The graph shows the currently active parameter. Every line profile is saved in the file *profile.shp* in the project directory. If you want to keep this line profile, you have to save it, see exercise *"How to draw a line profile"* below.

Time plot

This function provides a time plot at a single point. This is helpful when it is of interest to know the values and maximum values at a specific location (e.g. at a building, dam, or a tree) through time. Every point is saved in the file *point.shp* and a point-info file *point_info.txt* is additionally saved in the project directory. If you want to keep this point, you have to save it, see exercise *"How to create a time plot"* below. The point-info file can be visualized with ***Extras → Point… → View Point Info File*****.**

Flow discharge

From version 1.7.0 on it is possible to calculate the flow discharge when drawing a line profile (cross profile). See line profile exercise below.

Exercise 4.2b: How to draw a line profile

a) Draw a new line profile:

- Switch to 2D mode by clicking 
- Activate the project by clicking on it once, then click  or choose ***Extras → Profile → Draw New Line Profile***
- Define the line profile in the same way you specify a new release area. Finish the line profile with a right-mouse click.
- Calculate flow discharge? Answer with **YES**, and the flow discharge is calculated and shown in the line profile plot, see below (Beware: this can be time consuming!).
- A window opens, displaying the line profile.

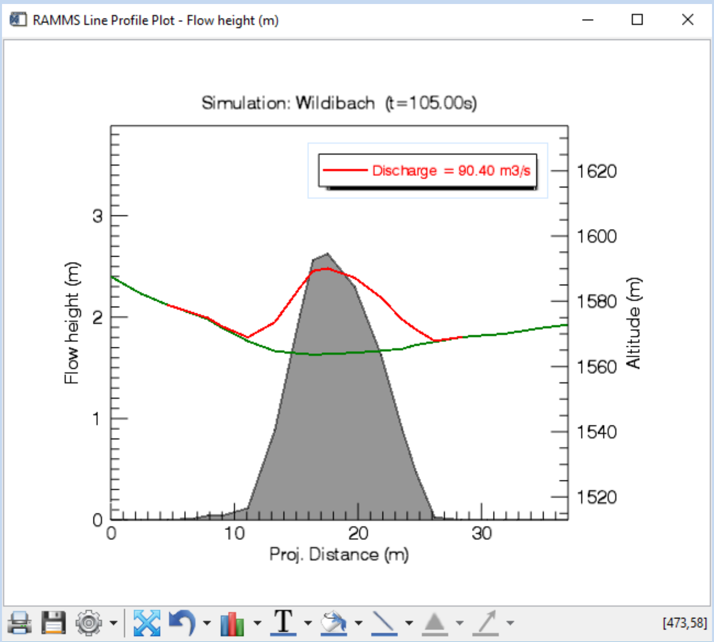


Figure 4.8: Line profile plot, with calculated flow discharge.

- Filled grey area active parameter (scale on left side).
- Red line active parameter (multiplied by 50) added to the track profile (altitude, scale on the right side).
- Black line track profile (altitude, scale on the right side).
- Bottom scale projected profile distance (in m).

- If you change the active parameter, min or max values or the dump-step in RAMMS, the plot is directly updated. You can also start the simulation and then watch the time variations in your line profile plot.
- It makes sense to either draw a profile line perpendicular to the flow direction or draw the line along the flow path. Basically every imaginable path is possible

Continuation of exercise 4.2b: How to draw a line profile

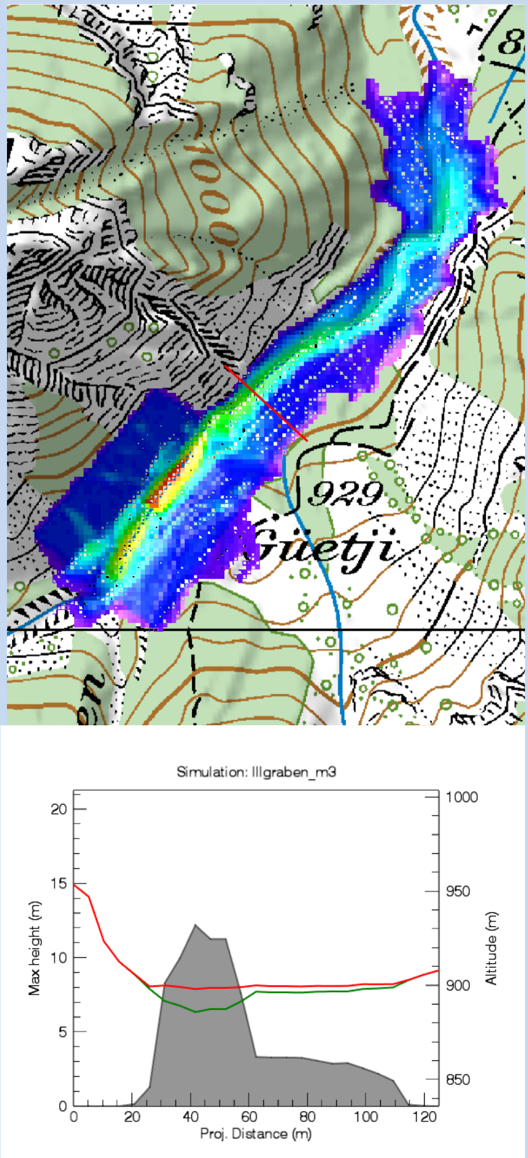


Figure 4.9: Line profile perpendicular to debris flow direction.

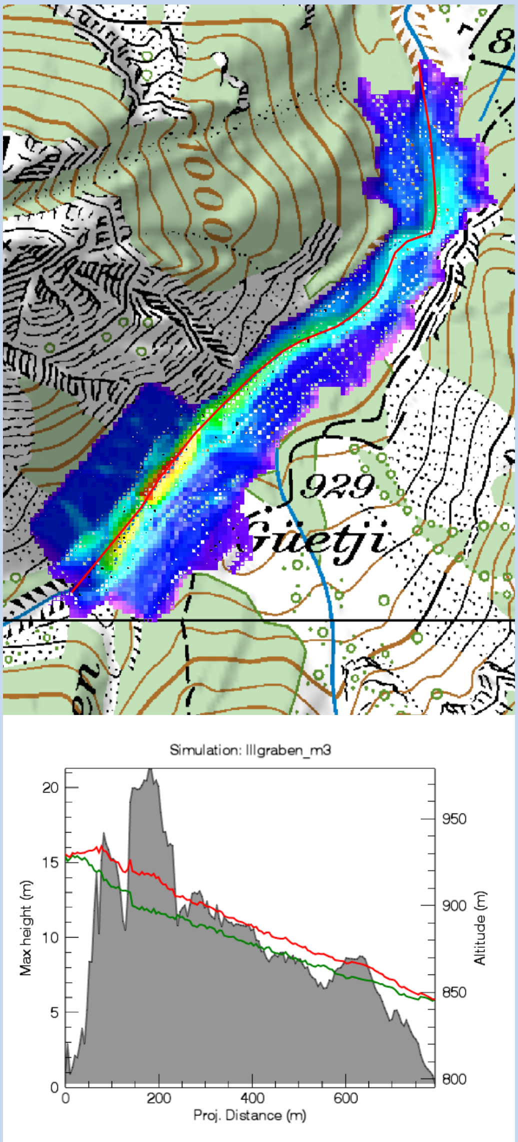


Figure 4.10: Line profile along the debris flow direction.

- To save the coordinates of the points belonging to the line profile, go on ***Extras → Profile → Save Line Profile Points*** and enter a file name.
- To save the line profile parameters (distance in m and the active parameter, e.g. the flow height in m) at the current dump-step, go on ***Extras → Profile → Export Profile Plot Data*** and enter a file name.

Continuation of exercise 4.2b: How to draw a line profile

b) Load an existing line profile:

- Switch to 2D view by clicking
- Activate the project by clicking on it once and click or choose ***Extras → Profile → Draw New Line Profile***
- Click the **middle mouse button** once
- A window pops up and you can browse for the line profile you wish to open

Exercise 4.2c: How to create a time plot

a) Select time plot point:

- Click or choose ***Extras → Point → Choose Point***
- Click into the map at the point where you want to create a time plot.
- A window opens, displaying the time plot at the point of interest (active parameter vs. time).

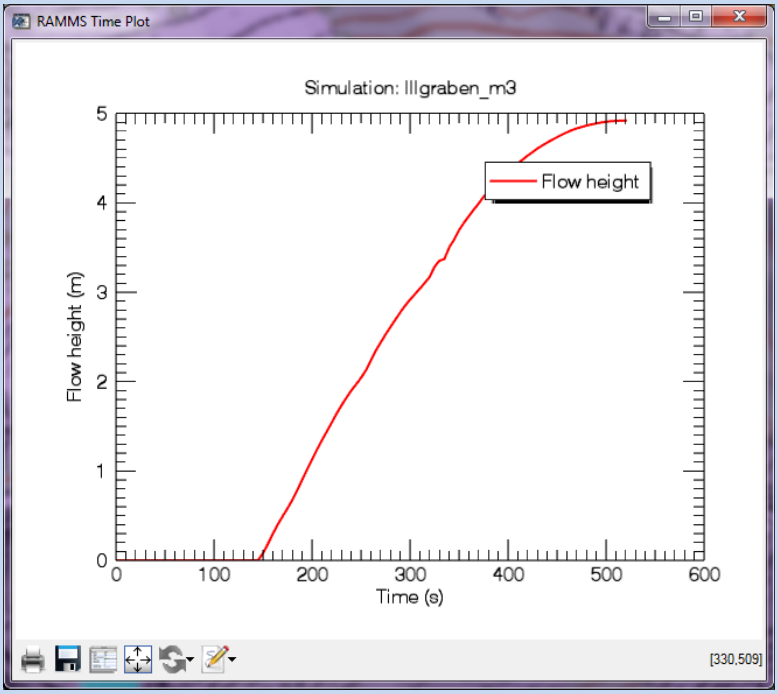
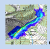


Figure 4.11: Time plot window.

- To save the point coordinates, choose ***Extras → Point → Save point Location*** and enter a file name
- To save the time plot data (time in s and the active parameter, e.g. the flow height, for every dump-step), choose ***Extras → Point → Export Point Plot Data*** and enter a file name.

Continuation of exercise 4.2c: How to create a time plot

b) Load a time plot:

- To reopen the time plot graph window of the last selected point, go on ***Extras → Point → Create Point Time Plot***
- To open an arbitrary time plot that was saved any time before, click .
- Click the **middle mouse button** once.
- A window pops up and you can browse for the time plot file you wish to open.

c) Enter point coordinates and get a time plot:

- Go to ***Extras → Point → Enter Point Coordinates (X/Y)***
- Enter X-coordinates of your point of interest. Click **OK**.
- Enter Y-coordinates of your point of interest. Click **OK.**
- The time plot opens.

4.2.3 Deposition analysis

A deposition analysis (flow height) for a region of interest (ROI) can be done in the following way:

- right-click the shapefile you want to analyse
- choose *Deposition analysis*

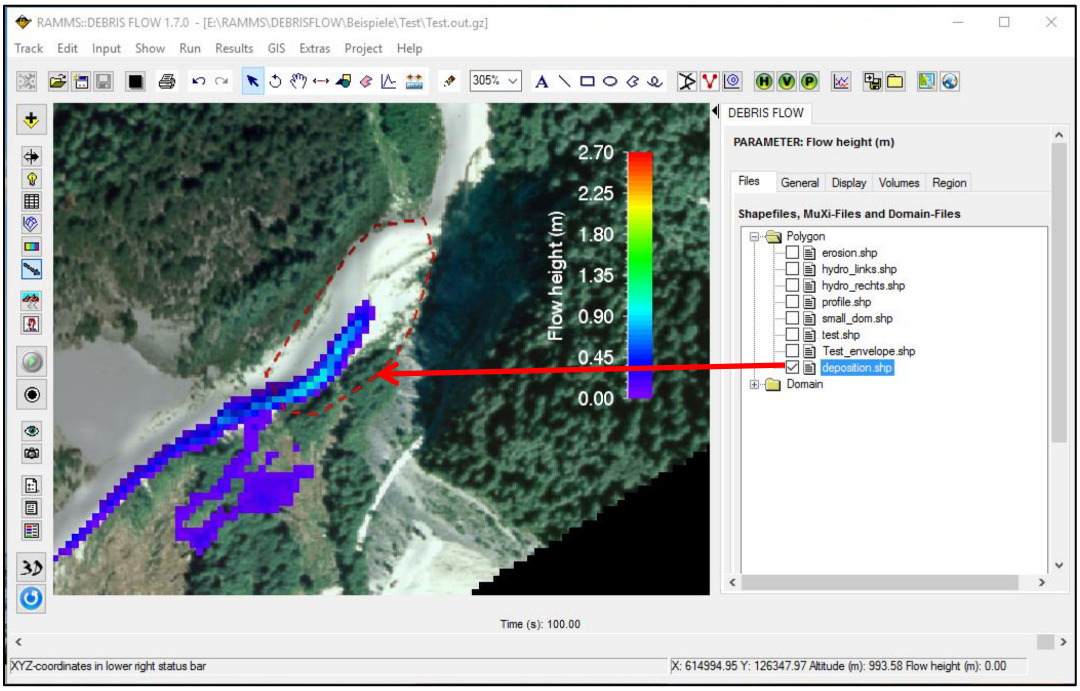


Figure 4.12: Deposition analysis of region of interest.

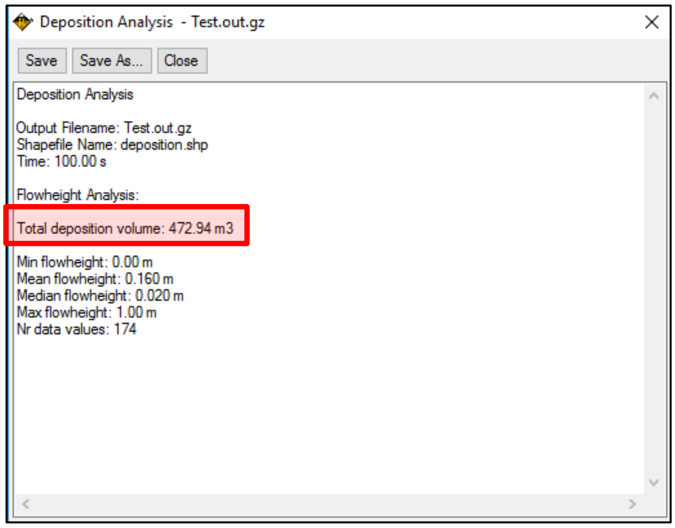


Figure 4.13: Result of a deposition analysis. Total deposition volume (m^3) as well as some statistical values are shown (min, mean, max).

4.2.4 Creating an image or a GIF animation

Image

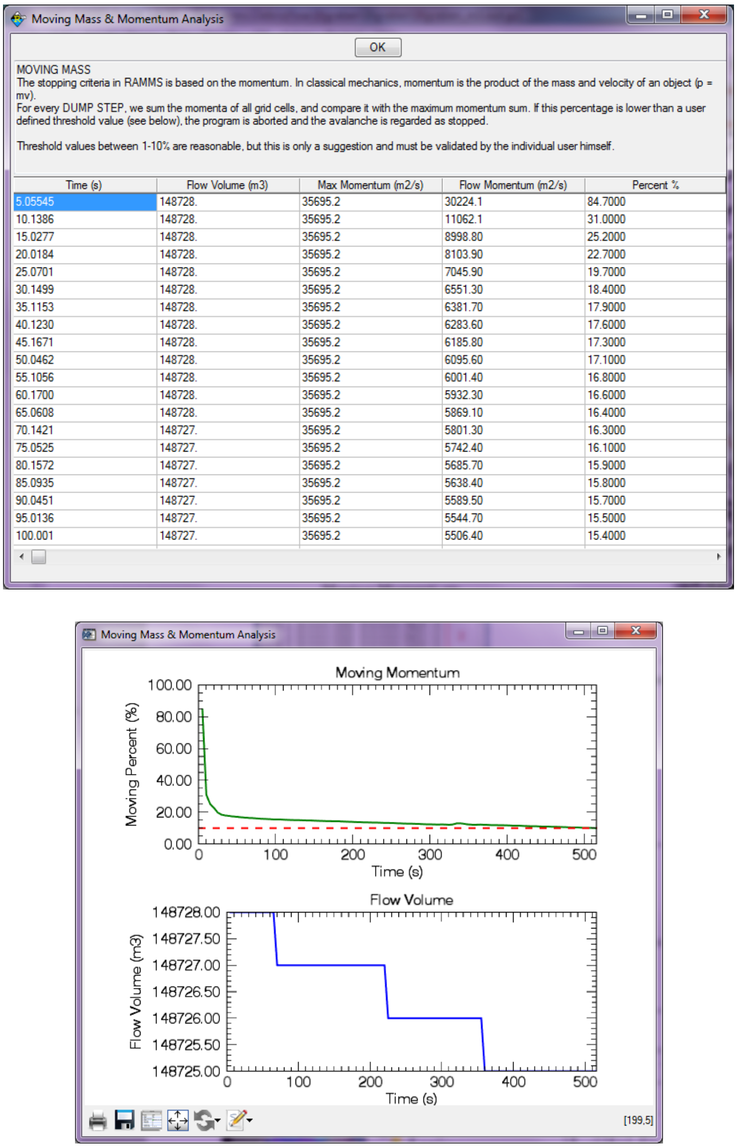
It is possible to export your results as an image in different formats (e.g. .png, .jpg, .gif, .tif etc.). Click  or choose ***Track*** → ***Export...*** → ***Image File*** and define a file name with the corresponding extension. An image of the visible part in the viewer will then be exported.

GIF animation

Creating a GIF animation is only possible in output mode. Click  or choose ***Track*** → ***Export...*** → ***GIF Animation***. Enter a file name and location and wait until the simulation stopped. As soon as the simulation finished, the GIF animation file is saved. In the *Preferences (debris flow tab)* you can define the interval for the GIF animation (GIF animation interval [s]).

4.2.5 Stopping mechanism

Check the output logfile under ***Project → Output Logfile*** to verify your simulation stopped due to low flux (see Output Logfile on page 51).Otherwise enlarge the end time of your simulation (see exercise *"Run a calculation"* on page 45). To check the stopping of your simulation click ***Results → Summary of Moving Mass***. A window similar to Fig. 4.12 opens which shows the summary of moving mass. For every dump-step, RAMMS summed up the momenta of all grid cells, and compared it with the maximum momentum sum. If this percentage is smaller than a user defined threshold value (see page 48), RAMMS aborts the simulation and the debris flow is regarded as stopped.



Time (s)	Flow Volume (m3)	Max Momentum (m2/s)	Flow Momentum (m2/s)	Percent %
5.05545	148728.	35695.2	30224.1	84.7000
10.1386	148728.	35695.2	11062.1	31.0000
15.0277	148728.	35695.2	8998.80	25.2000
20.0184	148728.	35695.2	8103.90	22.7000
25.0701	148728.	35695.2	7045.90	19.7000
30.1499	148728.	35695.2	6551.30	18.4000
35.1153	148728.	35695.2	6381.70	17.9000
40.1230	148728.	35695.2	6283.60	17.6000
45.1671	148728.	35695.2	6185.80	17.3000
50.0462	148728.	35695.2	6095.60	17.1000
55.1056	148728.	35695.2	6001.40	16.8000
60.1700	148728.	35695.2	5932.30	16.6000
65.0608	148728.	35695.2	5869.10	16.4000
70.1421	148727.	35695.2	5801.30	16.3000
75.0525	148727.	35695.2	5742.40	16.1000
80.1572	148727.	35695.2	5685.70	15.9000
85.0935	148727.	35695.2	5638.40	15.8000
90.0451	148727.	35695.2	5589.50	15.7000
95.0136	148727.	35695.2	5544.70	15.5000
100.001	148727.	35695.2	5506.40	15.4000

Figure 4.14: Summary of moving mass.

The stopping criteria in RAMMS is based on the momentum. In classical mechanics, momentum p (SI unit kgm/s, or, equivalently, Ns) is the product of the mass and velocity of an object (p = mv). Threshold values between 1-10% are reasonable, but this is only a suggestion and has to be empirically determined for each test case.

Stopping criteria with large threshold values (e.g. >10%) may result in unrealistic early stopping of a simulation.

Small threshold values however may lead to numerical diffusion of the simulation results and very slow creeping of the debris flow material and velocity oscillations.

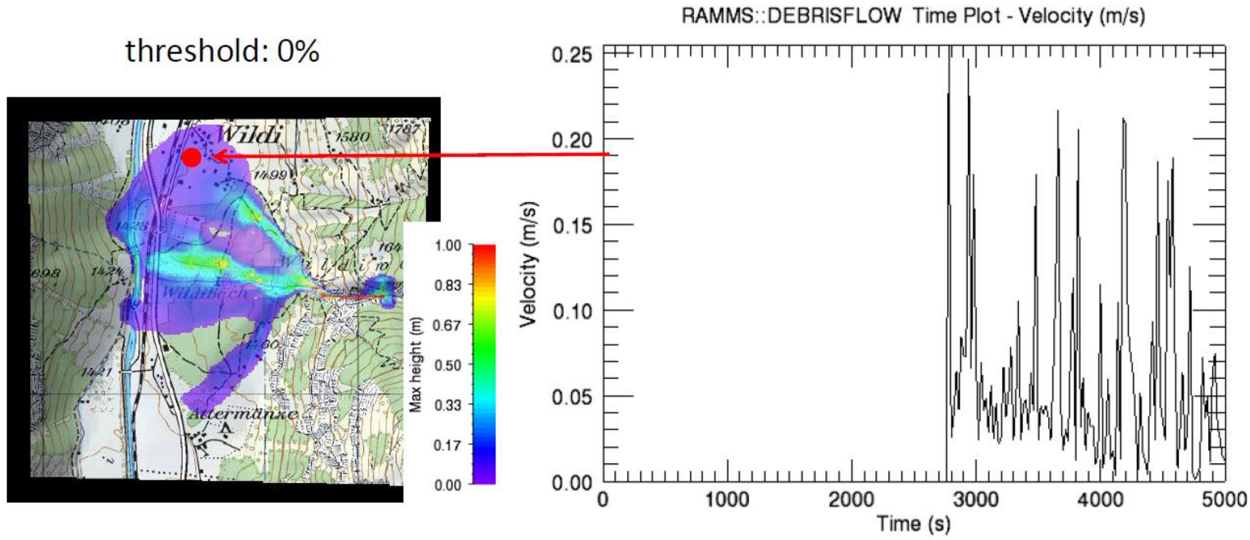


Figure 4.15: Stopping behaviour of a RAMMS simulation. Small threshold values may lead to unlikly slow creeping of the material. In the example shown in the figure above the stopping criteria is set to 0%.

Because block release simulations (the entire debris flow material starts to flow at the same time) usually show larger values for the total maximum momentum than hydrograph simulations (controlled inflow of debris flow material into calculation domain), the threshold values for block release simulations could be set smaller than the threshold values for hydrograph simulations in some situations (see Fig. 4.14 and 4.15 and section 5.2).

Whether or not a flow stops depends on terrain (slope angle in runout), total flow volume and friction values and should always be evaluated by an expert. In case of doubt on how to choose threshold values we recommend running a simulation with a 1% threshold and checking the summary of moving mass for numerical diffusion and analyzing the debris flow runout (flow height and flow velocity) with time plots (section 4.2.2).

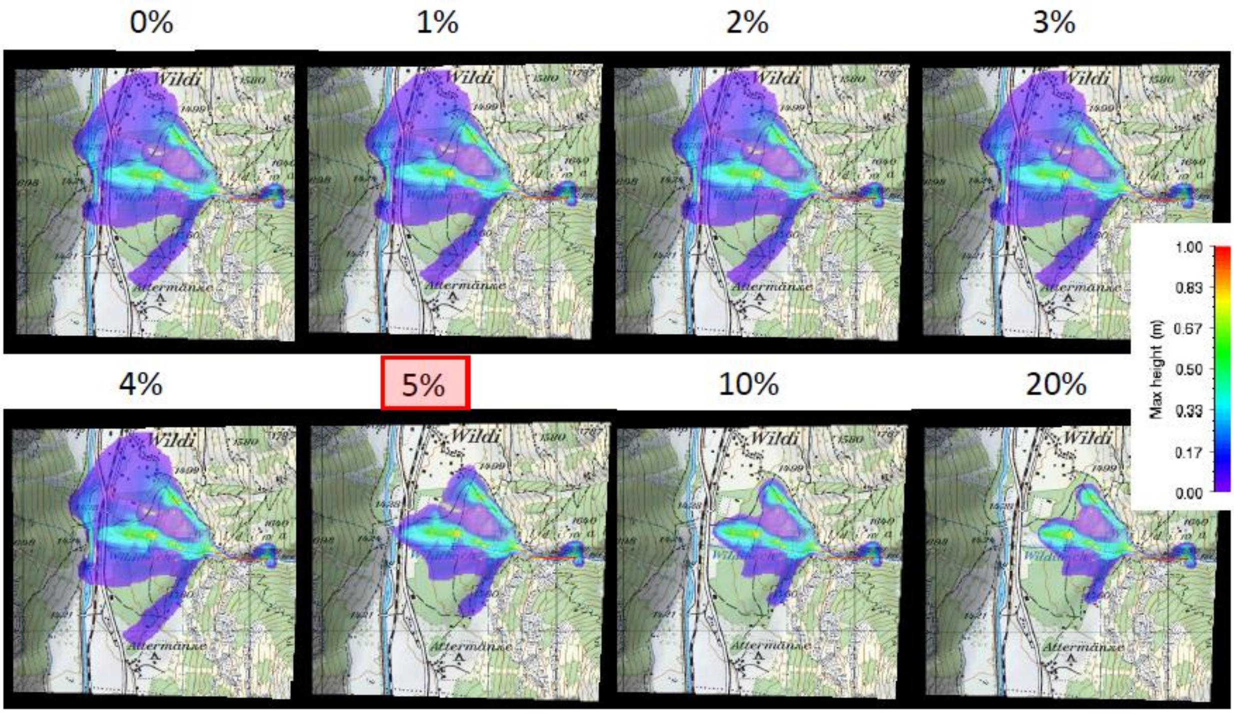


Figure 4.16: Stopping behaviour of a hydrograph RAMMS simulation. In this example threshold values <5% lead to numerical diffusion of the simulation results. A threshold value of 5% seems to be appropriate in this cas.

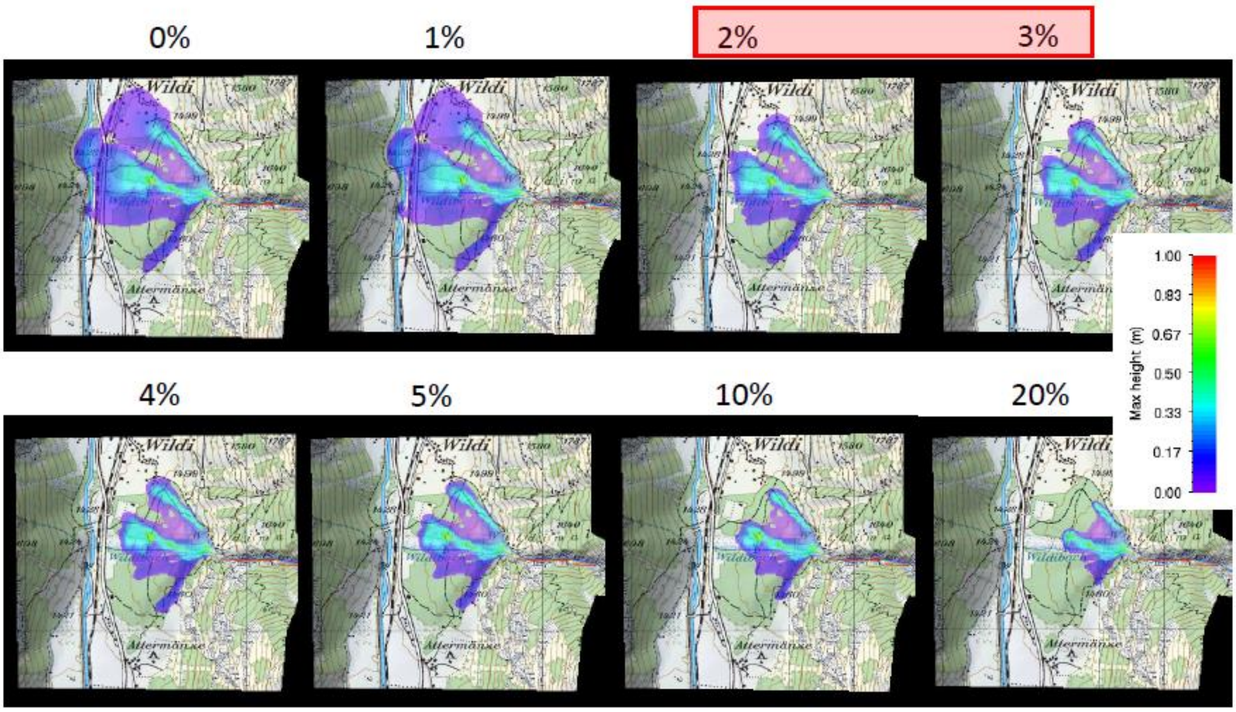


Figure 4.17: Stopping behaviour of a block release RAMMS simulation. In this example threshold values <2% lead to numerical diffusion of the simulation results. A threshold value of 2-3% seems to be appropriate in this cas.

4.3 Adding structures or deposition to DEM

The option to adding structures or deposition to DEM must be used with great care and should not be used to design deflecting dams. Deflecting or catching dams can neither be designed directly with RAMMS nor can the residual risk below dams be calculated directly with RAMMS. RAMMS takes important factors in dam design such as energy dissipation, dam geometry or debris flow deposits in front of a dam not properly into account. Dams have to be designed using well known standard engineering procedures, e.g. Johannesson et al. 2009, Rudolf-Miklau and Sauermoser 2011 and Rickenmann 1995. RAMMS is well suited to calculate the key input factors for dam design such as flow height and velocity. The dam-option should however only be used to try to visualize the influence of guiding or small deflection of the debris flow mass. RAMMS cannot be used directly to evaluate if the height of a deflecting dam is sufficient for a certain scenario or not (see explanations below).

4.3.1 Creating a dam

RAMMS offers the possibility to simulate the presence of a deflecting dam by increasing the altitude at the position where a dam is considered. This option helps the user to design mitigation structures and to test its influence on potential flow paths near populated areas.

Exercise 4.3a: How to create a new DEM to simulate a dam

- Create a new polygon area where a dam is supposed to be built (Figure 4.18).
- Create a second, inner polygon, if you wish to have a two-stage dam.
- Go on ***GIS → Add DAM to DEM…***
  You have two options***… → Enter Relative Dam Height*** or ***… → Enter Dam Elevation***
- You will be asked to “Open dam file (*.shp)”. Select the shapefile you want to use as the outer edge of the dam.
- The question pops up, if you want to “Open 2nd dam shapefile (inner polygon)?”
  → Click No to continue with the next step
  → Click Yes to choose a 2nd dam file (*.shp).
- Next step is to enter the total elevation height or the total relative height of the dam in meters. This is the elevation of the dam crest.
- If you loaded an outer polygon file, you will be asked to enter the intermediate height (m) (height of the outer polygon file) as well.
- Finally you have to “Enter new XYZ name”. Your new xyz-file with the topographic information, containing the “dam”, is created in your project directory.

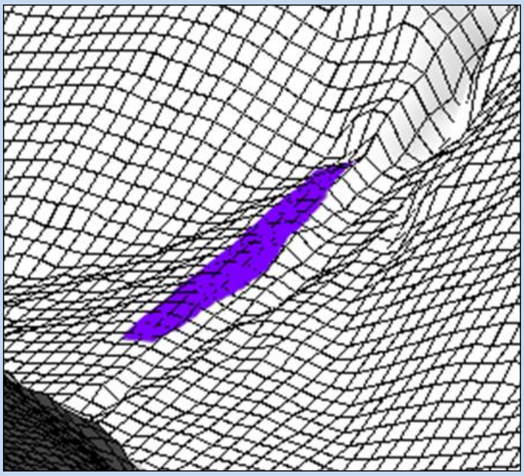


Figure 4.18: Release area where a dam is supposed to be built.

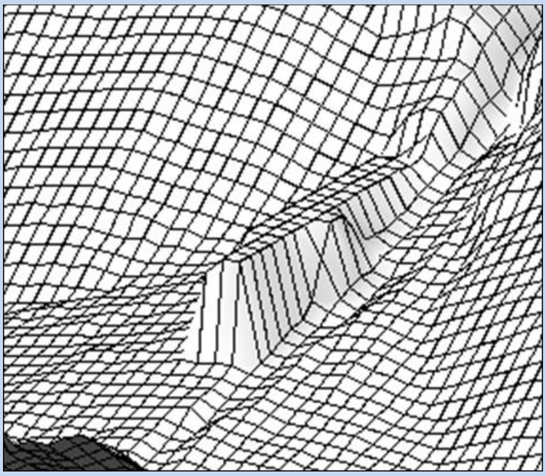


Figure 4.19: Dam.

To run a simulation based on the new created xyz-file, all you have to do is to choose the new xyz-file in the ***Run Simulation*** window, see below:

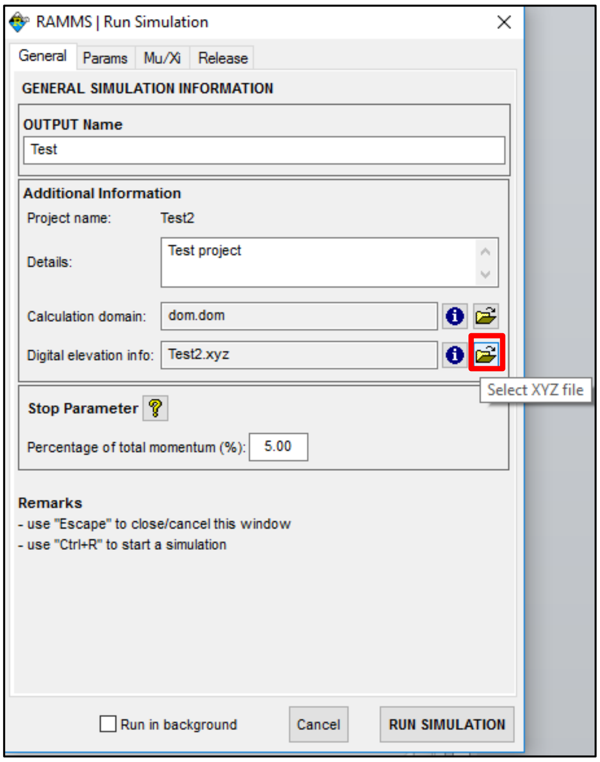


Figure 4.20: Select new xyz-file with dam information.

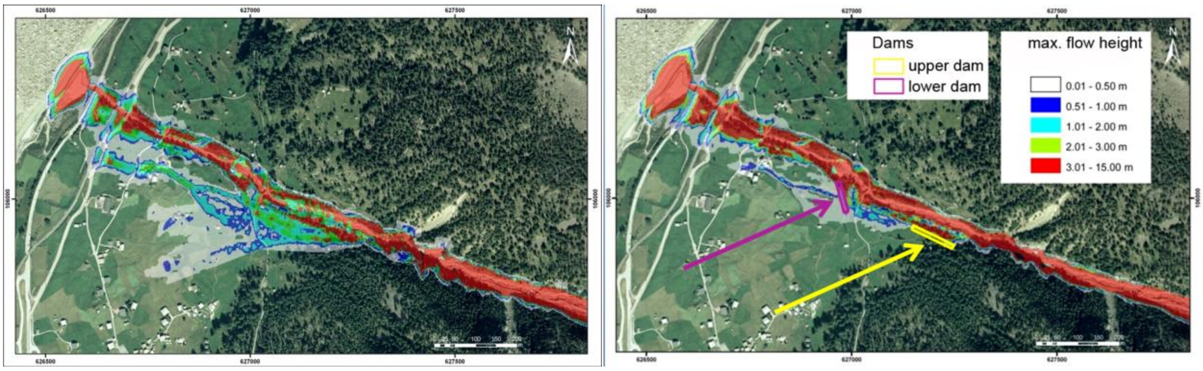


Figure 4.21: Simulation without mitigation measures (left) and with two dams built in RAMMS (right).

While RAMMS is able to simulate the effect of a dam lying lateral to the direction of flow quite well, there might occur numerical problems if a dam lies perpendicular to the direction of flow (see Figure 4.22).

- Because there is no energy dissipation due to collision with dams implemented in RAMMS, unrealistically large flow velocities and flow heights may be simulated in front of a dam.
- The numerical solver used in RAMMS incorporates information from neighboring cells. The effect of dams with only one cell as dam side wall may therefore be difficult to simulate.

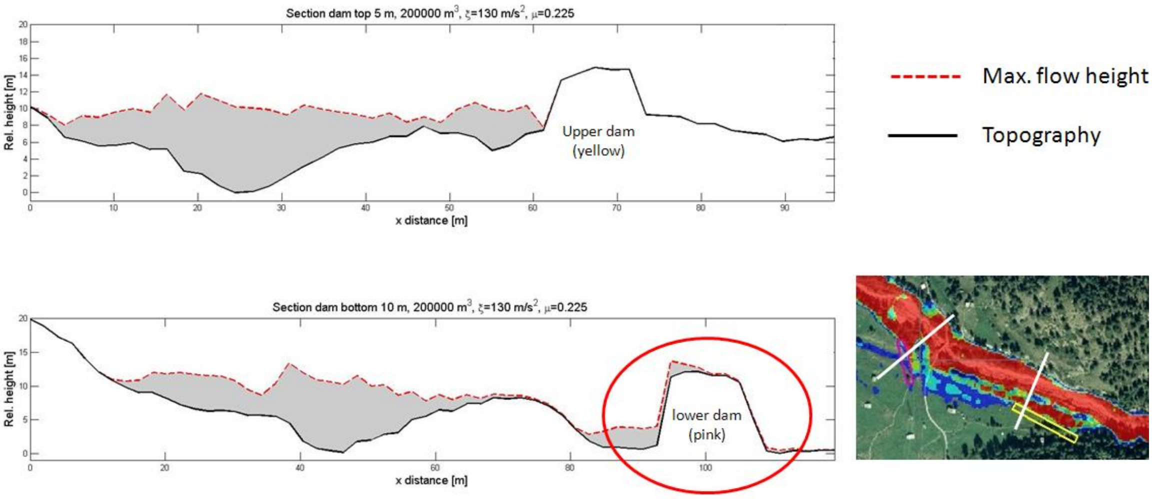


Figure 4.22: Comparison of the profiles of two mitigation measures in RAMMS.

If you encounter problems with the simulation of mitigation measures as described, we suggest creating a DEM including a dam in GIS, ideally using progressively increasing side walls as shown in Figure 4.23.

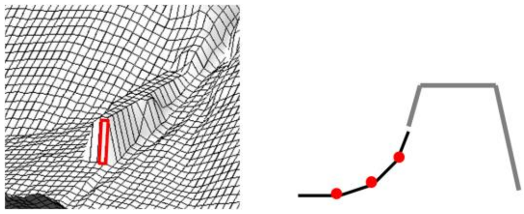


Figure 4.23: Dam with gradually rising side walls.

The interpretation of RAMMS simulations including mitigation measures such as dams (see Figure 4.22 and Figure 4.23) has to be done by experts. In addition we recommend to always check the simulation results with engineering approaches.

4.3.2 Creating a new DEM with deposited debris material

It is possible to simulate multi-surge debris flow events to estimate how deposits from earlier surges can influence the flow of subsequent surges. One has to assume that the deposits from an initial surge are not entrained by subsequent surges. To do this, in the output mode, users can select the option to add the flow height of a debris flow to the DEM at any arbitrary dump-step. Finally, a new project can be created based on the updated DEM.

Exercise 4.3b: How to add debris flow deposition to new DEM

- The deposition height is the flow depth at the end of a simulation when the flow is considered to have stopped moving (alternatively, earlier dump-steps may be used if there are reasons to believe the flow should have stopped earlier). So first view the results at the last time step or a different time step, if desired.
- Go to ***Results → DEM Adaptations → Add Deposition to DEM***
- Enter a new name for the new xyz-file.
- The new xyz-file, containing the deposition information, is created. To run a simulation based on this new xyz-file, just choose the xyz-file in the Run Simulation window, see Figure 4.20.

5 Applications

RAMMS::DEBRISFLOW can be initiated using either a block release (e.g. landslide release) or a hydrograph (flow discharge as a function of time). In some cases it may not be entirely clear which method is most suited. In this section we describe some practical differences in model results which may occur depending on which method is used to start the flow in the model.

This chapter presents several topics relevant for the application of RAMMS::DEBRISFLOW in practice. Several examples of how to use RAMMS to simulate channelized debris flow (flow paths of debris flow limited by topography, such as a mountain torrent) using either a block release or an *input hydrograph* are summarized. Model calibration and the question how to calculate model impact pressure are discussed.

5.1 Calibration

The following section is based on simulations run in 2011 with RAMMS::DEBRISFLOW v1.4.

The calibration of the Voellmy friction model is one of the most important steps to get realistic and useful results. For the calibration procedures a well-documented historical event is required. The historical event should be similar to the problem where calibration is required. Ideally, information should include flow heights and velocities at different locations in the torrent as well as the material composition, information concerning different flow paths and the initial conditions such as the total and the initial volumes. In practice, it is difficult to collect exact data on flow heights and velocities from field studies. However, field data can provide estimates of total volume, flow paths, flow heights and material composition and sometimes even flow velocities.

Here, an example is shown , which was used for the calibration of RAMMS::DEBRISFLOW for the specific case at Dorfbach, close to Rana (VS, CH). See Deubelbeiss and Graf, 2013. The following data was used to calibrate the model:

Table 5.1: Parameters for calibration for the case study of Dorfbach, Randa (CH).

Field investigations	
Cross sectional analysis	Heights of levees or heights of marks on constructions, estimation of velocity (splashing, superelevation)
Flow paths	Tracks of boulders, rocks and mud
Deposition of material	Lobes, levees and debris flow heads
Estimation of total volume	Retention basin in the runout zone plus deposited material in torrent and receiving river
Photographs	
Release area	Geometry of release area
Flow Paths	Deposited material analyzed by aerial photographs

Field observations and investigations provided an estimate of a total volume of approximately 10'000 m^3. The aim was to calibrate the model by comparing flow velocities and flow heights at two locations (Figure 5.1a). The estimated values obtained from the field observations at these locations are:

Table 5.2: Field estimations of height and flow velocity for the case study of Dorfbach, Randa (CH).

	Max flow height [m]	**Velocity [m/s]**
Location 1	2-3	3-4
Location 2	2-3	1-2

Find the best-fit Voellmy friction coefficients (procedure)

To find the best- fit Voellmy friction coefficients (dry-Coulomb type friction μ and viscous-turbulent friction ξ) we suggest the following calibration procedure:

1. Define μ which – as a first guess – is set to $\tan(\alpha)$ is the slope angle in the deposition zone. For the case in Dorfbach we defined $\mu \approx 0.2$ $(\alpha \approx 15°)$.
   Values of μ **normally range between 0.05 and 0.4.** Values of μ larger than 0.4 rarely provide useful simulation results.
2. ξ is more delicate to calibrate and we suggest starting with an initial guess of ξ =200 m/s^2. ξ describes the turbulent behavior of the flow. Typically small values of ξ are reported for granular flows while relatively large ξ values are sometimes associated with muddy flows.

Table 5.3: Suggestions for setting the viscous-turbulent friction parameter ξ.

	Granular flow (solid-dominated)	**Mud flow (fluid-like)**
Viscous-turbulent friction, ξ [m/s^2]	100-200 *)	200-1'000 *)
***These values are only suggestions not fixed definitions.**		

If the type of flow (granular or muddy) is not known, we suggest the following initial values of the Voellmy friction coefficients for the calibration:

Table 5.4: General suggestion for the initial values of the Voellmy friction coefficients used for the calibration procedure.

Initial values of the Voellmy friction coefficients used for a calibration procedure with unknown flow type		
Dry-Coulomb type friction	μ [ ]	0.2
Viscous-turbulent friction	ξ [m/s^2]	200

3. Vary μ around the initial definition with steps of ±1 and ξ with steps of ±100 m/s^2. After comparing this initial results with field observations, start with a fine tuning and incrementally change μ and ξ around the initially-found best-fit values. To match the simulation results

with the flow heights and velocities from field observations at the given locations we found the following best-fit Voellmy friction coefficients for Dorfbach:

Table 5.5: Best-fit Voellmy friction coefficients for the case study of Dorfbach, Rand (CH).

Dry-Coulomb type friction	μ [ ]	0.225
Viscous-turbulent friction	ξ [m/s^2]	130

Table 5.6: Resulting maximum flow heights and velocities for total volumes of 10’000 and 5’000 m^3 and μ=0.225 and ξ=130 m/s^2

Volume: 10’000 m^3	Max. height [m]	Max. velocity [m/s]
Location 1	2.7	3.8
Location 2	2.5	2.5
Volume: 5’000 m^3		
Location 1	2.2	2.6
Location 2	1.1	2

The best-fit Voellmy friction coefficients are now used to perform several simulations (Figure 5.1). We performed an additional simulation with a small total volume (5’000 m^3) to better model the observed flow path. This modification is justified because there are two main locations where deposition was observed during the event of June 7, 2011. One location is out of Figure 5.1 around an elevation of 1’730 ma and the second location coincides with location number (3) in the center of the channel. The simulation using a total volume of 10’000 and 5’000 m^3 show the following flow heights and velocities at the two locations 1 and 2:

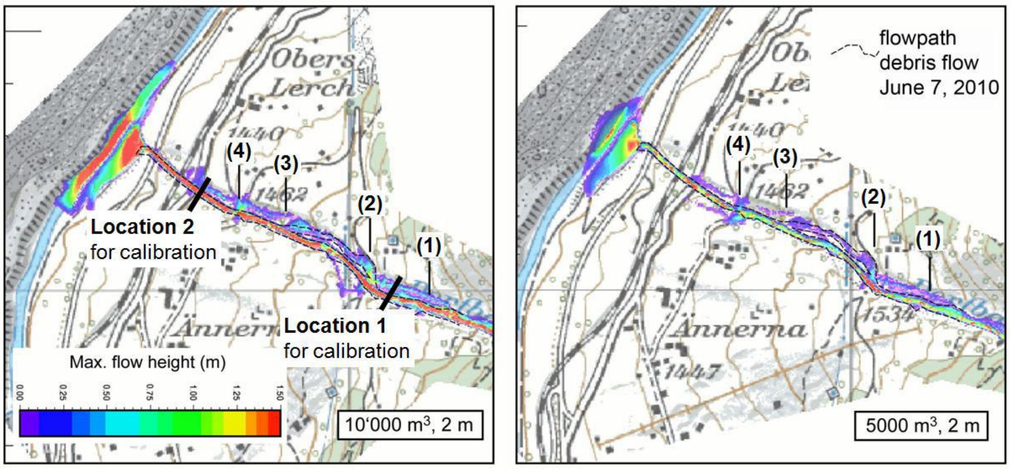


Figure 5.1: Simulations with best-fit parameters μ=0.225 and ξ=130 m/s^2 for (a) a volume of 10’000 and (b) 5’000 m^e and a DEM resolution of 2m. The dashed line indicates the flow path of the event from June 7, 2010.

Both simulations with volumes of 10’000 and 5’000 m^3 show locations where the flow leaves the channel (Figure 5.1, locations 1-2), while in reality the debris flow should simply follow the channel. Specifically, these are locations indicated by numbers (1) to (4). A breakout, where the flow left the channel, was observed only at location (2) on the orographic right side of the channel.

These differences might be caused for the following reason: For these simulations we use a resolution of the DEM (2010) of 2m. Additional simulations using a DEM-resolution of 1m and a volume of 5’000 m^3 could reproduce the flow path at location (1), (3) and (4). At location (2) the outbreak disappeared in the simulation, although it occurred in reality. This result indicates that a resolution of 2m cannot reproduce the exact flow paths because not all vertical curves are visible on a 2m-DEM and therefore material can break out of the channel. The missing outbreak at location (2) for the 2m-DEM can be explained as following: first, it may indicate that at this location the volume was larger than 5’000 m^3 and only after location (2) more material was deposited and second, an aspect which will be discussed in the next section (5.2), the use of a block release rather than an input hydrograph resulted in slightly underestimated flow heights and velocities.

To conclude, the simulations using a volume of 10’000 m^3 indicate a slightly too high volume estimation for the lower part of the torrent. However, to calibrate the Voellmy friction coefficients it was still possible to match the flow heights and velocities observed in the field. The small differences in the flow path between the simulation and reality can be explained by no ideally resolved terrain.

5.2 Input hydrograph in comparison with block release

The following section is based on simulations run in 2011 with RAMMS::DEBRISFLOW v.1.4.

With an input hydrograph it is possible to constrain inflow of the debris material into the calculation domain at a specific location. This is in contrast to a block release simulation where a release area with an initial depth is defined for which the entire block of material is accelerated by gravity given the frictional resistance, e.g. simulating debris flow initiation as an instantaneous landslide failure. Flow depth is a value that can be measured with an observation station or estimated by geomorphic evidence. Assuming that the profile and the velocity are known at the measurement location one can calculate the discharge hydrograph there. Other options are presented in section 3.5.2.

5.2.1 Comparison between a block release and an input hydrograph

Simulations with large volumes and a block release usually have larger maximal discharge values (maximum heights) compared to simulations using a hydrograph (Figure 5.2a). The reason is that for a block release the release area is defined and the initial release height has to be adjusted according to the total volume, specified by the user. This block begins moving as soon as the simulation is started. Hence, extremely large and possibly unrealistic initial heights can occur (e.g. we use an initial height of 13.2 m for simulations with a total volume of 200'000 m^3 at Dorfbach, see studies by Deubelbeiss and Graf, 2013). In many cases it would be more reasonable to assume a realistic initial release height and adjust the release area to match the corresponding total volume. This effect inverts for small volumes: the inflow velocity of a hydrograph simulation may be larger than what results from a block release simulation at the location of the input hydrograph (Figure 5.2b). The inverse effect is also visible in the flow paths (Figure 5.3). The larger flow heights and resulting velocities of a block release simulation are represented by the larger extent compared to a hydrograph simulation.

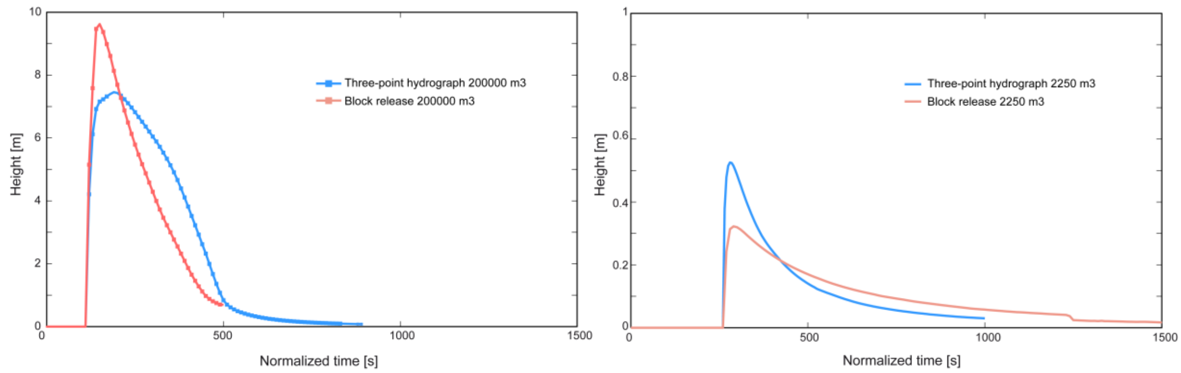


Figure 5.2: Comparison of discharge curves of a simulation with a block release and a hydrograph for (a) a volume of 200'000 m^3 and (b) a volume of 3'350 m^3. For all simulations μ=0.225 and ξ=130 m/s^2. The discharge curves are measured at the location shown in Figure 5.3.

Figure 5.3: Comparison of the inindated areas of a simulation with a vlock release and a hydrograph (a) for a volume of 200’000 m^3 and (b) a volume of 2’250 m^3. For all simulations μ=0.225 and ξ=130 m/s^2. Blue: hydrograph, red: block release. The red points indicate the location of the measured discharge curves (Figure 5.2).

5.2.2 Discussion

Two general problems appear when comparing the input hydrograph and block release: the friction parameters for a large design event may be different than those for the calibration event, and differences in peak discharge. Large debris flows are often expected to have disproportionally longer runout in comparison with smaller flows (e.g. less friction, e.g. smaller μ values). However it is difficult to generalize about how much the friction parameters should be changed as a function of event volume. The situation is further complicated by the observation that very large debris flows tend to take place as a series of surges rather than one large surge, and the effective flow topography may change due to deposition (or erosion) during the surge. The problem with the unusually large flow depths and the input hydrograph was described earlier in this section. Another aspect of that problem is that sediment entrainment often takes place along the flow path, a process which is NOW implemented in RAMMS::DEBRISFLOW. The new erosion module in RAMMS predicts the depth of erosion of sediment caused by debris flows. Using this option, it is possible to predict the increase in volume of a debris flow as it travels along a channel (see sections 3.5.3 and 5.3).

5.2.3 Summary

Using an input hydrograph rather than a block release certainly enhances the simulation results if data are available. In this case we can assume that the input parameters are well known and represent the measured field data (at least at the measurement location). This way large initial release heights and the resulting large momenta can be avoided.

Channelized debris flows, which typically follow a torrent to the runout zone are more accurately simulated by input hydrographs. Additionally, the length of the flow path to be simulated can be reduced resulting in shorter simulation times. (Be sure the entire hydrograph lies within the calculation domain.) For hillslope debris flows, which typically occur in open, unchannelized topographies, a block release achieves better results.

5.3 Erosion example

An example of applying the RAMMS debris-flow erosion algorithm is presented below. In this case (Illgraben, Switzerland), channel-bed erosion can only occur between check-dams installed along the channel, not at the check dams themselves, so four individual polygons were drawn and included in one shape file. The flow is started as a hydrograph (bar-shaped release) and the computational domain has been made relatively narrow to decrease computational time.

Step 1 (not shown): Setup and calibrate RAMMS without considering entrainment. In this case RAMMS was calibrated for an example in the literature (Berger et al., 2011) where we know the travel time between two of the check dams. The purpose of the first calibration–without erosion–is to help narrow-down the choice of the friction parameters μ and ξ for fine-tuning the model after erosion is considered.

Step 2: Define where erosion can take place. Select the menu option ***Input*** –> ***Polygon shapefile*** –> ***Draw New Polygon Shapefile*** and draw the first polygon. After closing the first polygon with a left-mouse-click, choose the option ***Add polygon areas***, and repeat this procedure until all desired polygons have been drawn. After adding the final polygon (in this case the fourth one) click on the "No" button and name the polygon (Figure 5.4). Once this step is completed, the erosion properties within this shapefile can be set in a different dialog box (Step 3, Figure 5.5). Although not illustrated here, it is possible to create multiple shape files with different erosion parameters within each shape file.

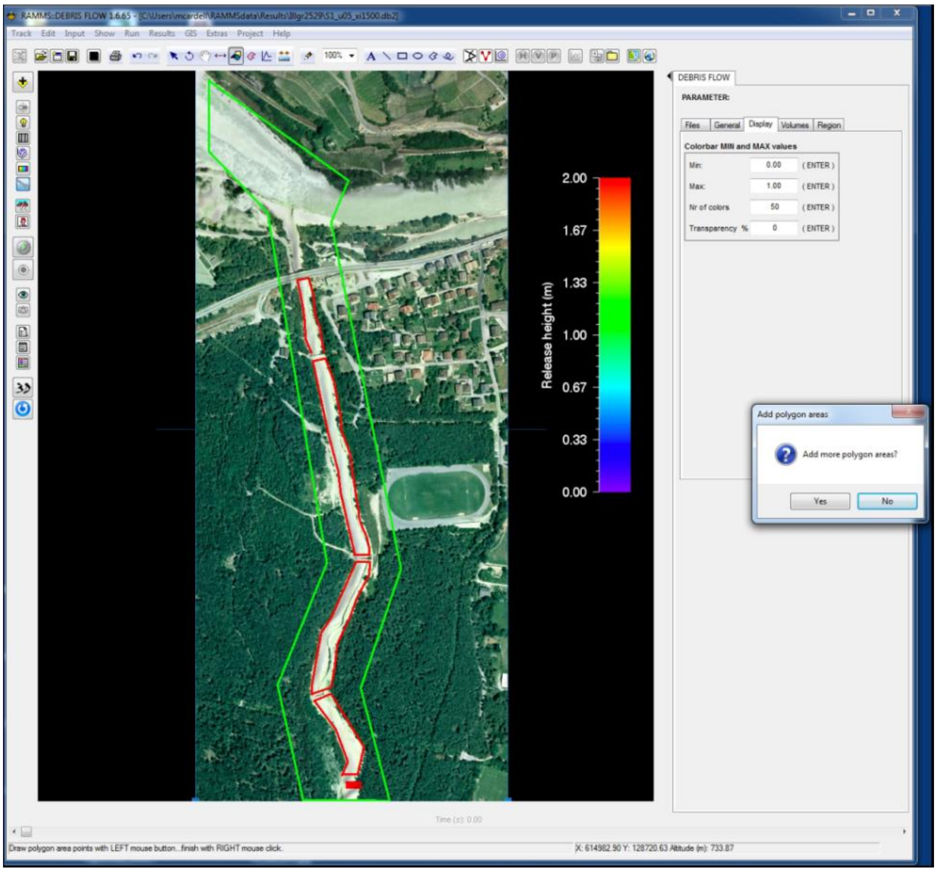


Figure 5.4: Adding erosion polygon areas.

Step 3: Set erosion parameters. To set the erosion parameters used during the simulation, select ***Run*** simulation and then click on the ***Erosion*** tab. Click on the folder icon and select the polygon shapefile (in this case *erodible_bed.shp*, Figure 5.5). After selecting the erosion parameters, click on the button ***Add shapefile to list.*** The default values are proposed, however they can be edited if desired.

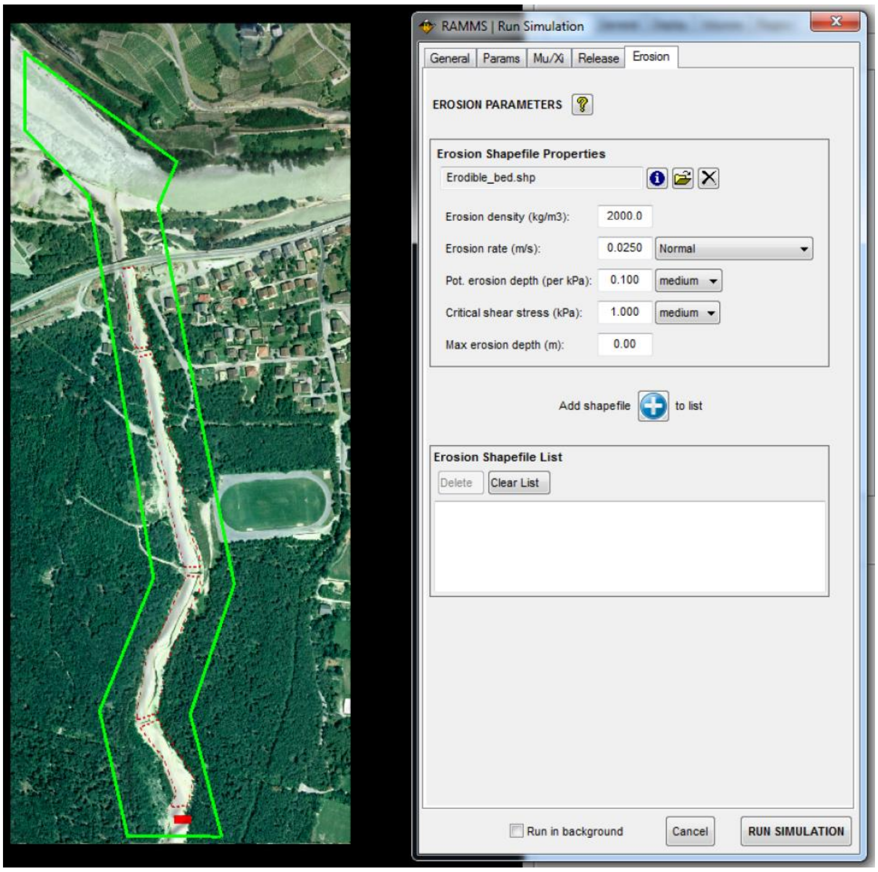


Figure 5.5: Erosion parameters.

One advantage of the new Erosion Module in RAMMS is that the user can start the model with realistic starting conditions, e.g. either a block release (landslide) scenario based on field observations or a realistic hydrograph. Then the model will entrain sediment along the flow path, thereby increasing in size. The main disadvantage of the new model is that you cannot specify the event volume before the simulation starts, because the volume of the debris flow will increase as it entrains sediment along the flow path. If precise final volume scenarios are necessary, it may be necessary to run the model several times, iteratively adjusting the initial landslide size, until the desired volume is achieved (see Frank et al., 2017, for an example).

The predicted channel-bed erosion for this case is illustrated in Figure 5.6. Here one can clearly see (in comparison with Figure 5.4) that erosion has taken place only within the user-specified polygons within the erosion shape file, and not on the check dams separating the individual polygons.

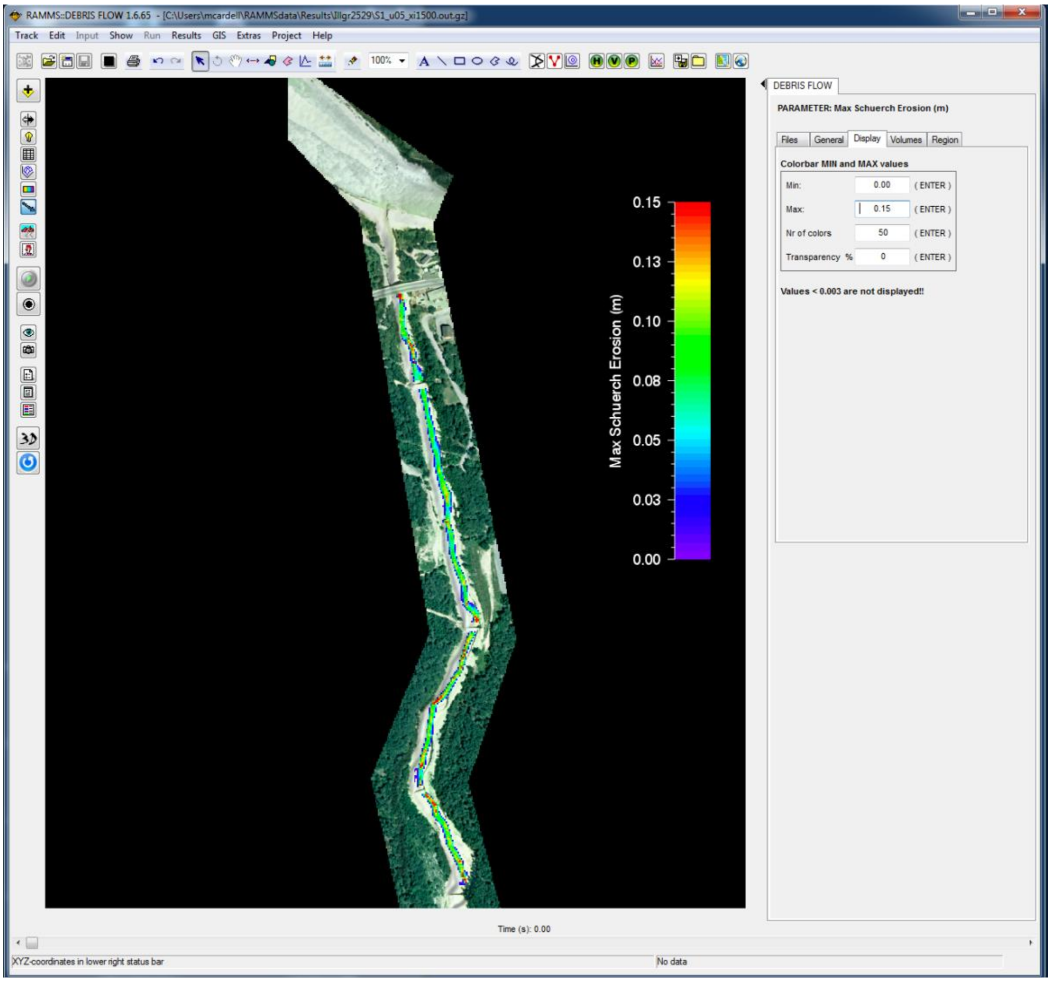


Figure 5.6: Simulated channel-bed erosion.

In this case the model was then re-calibrated with small adjustments to the turbulent friction parameter, and calibrated (for flow speed) to be within 1 second of observed travel time. A comparison of the hydrographs (output from a point placed under the bridge at the top of Figure 5.7, in comparison with the flow height measured in the field) illustrates that the steepness of the front, when including debris-flow entrainment, is somewhat more realistic. However the case where entrainment is not considered is somewhat slower in travel speed, for the same friction parameters.

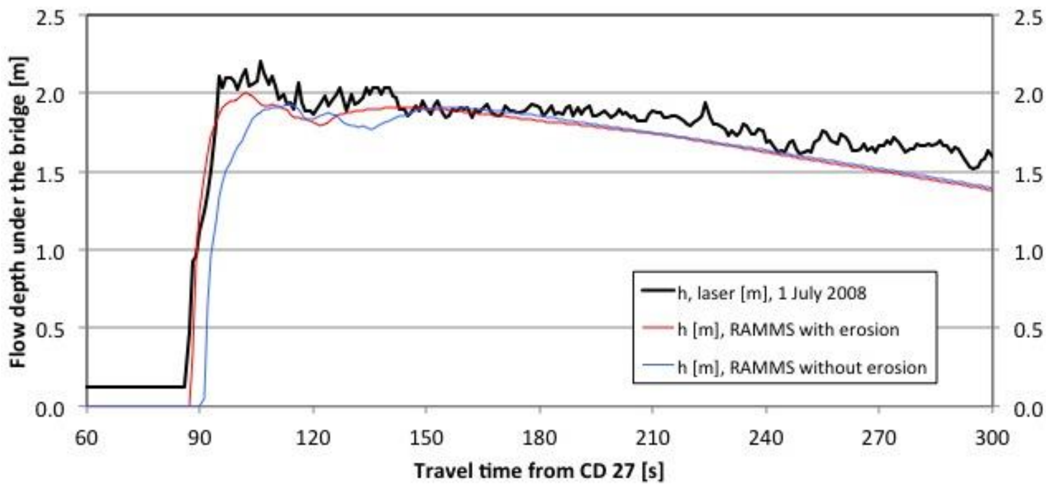


Figure 5.7: Front steepness comparison; laser data, RAMMS with erosion and RAMMS without erosion.

6 Program overview

RAMMS is a windows-based program that relies on drop-down menus and dialog boxes to set the model parameters, run calculations and view results. Toolbar buttons are also available and provide short-cuts of the menu paths; moving the cursor over a button results in a short explanation, appearing in a text box below the cursor ('tooltip'). For functions not available in the current context, the menus and buttons are deactivated and cannot be used.

6.1 The Graphical User Interface (GUI)

The graphical user interface (GUI) (Fig. 6.1) consists of menu bar, horizontal and vertical toolbar, main window, time step slider, right and left status bar, colorbar and panel. They will be explained in the following sections.

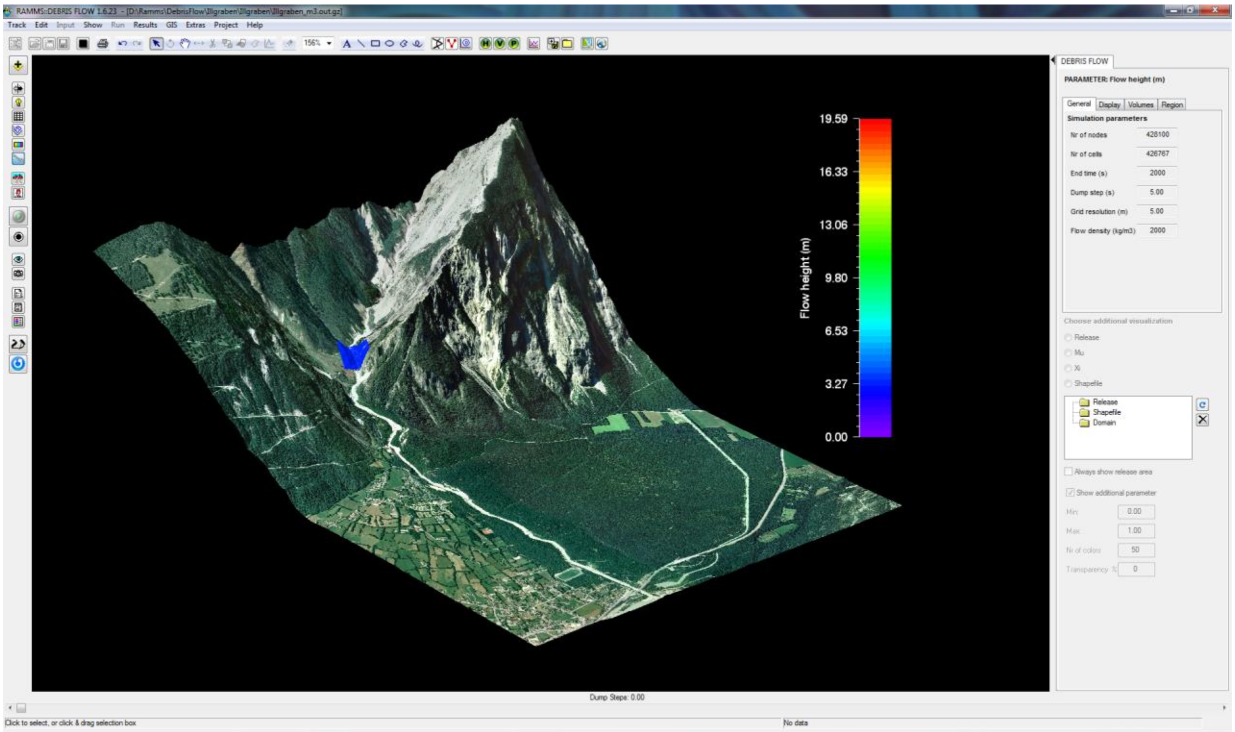


Figure 6.1: Graphical user interface (GUI)

6.1.1 The menu bar

Track

Similar to the Microsoft Windows *File* menu, *Track* is used to open, close, save, print, backup and export files.

New	• **Project Wizard**	Start a new project, guided by the wizard (Ctrl + w)
	• **Convert XYZ → ASCII grid**	Convert (regular) laser scanning data into an ESRI ASCII grid
	• **Run BATCH simulations**	Possibility to start many simulations automatically (e.g. overnight) You can choose how many computational cores the Batch-Mode should use (quasi parallel simulations, saves computational time).
	• **Export ASCII Files from Simulations (Batch)**	Automatically export all ASCII files (max height, max velocity, max pressure, deposition) from multiple output files.
Open	• **Input File**	Open an existing input file (*.db2) (Ctrl + O)
	• **Debris Flow Simulation**	Open existing debris flow simulation (*.out.gz) (Ctrl + A).
Close		Close active file (input or output)
Save		Save active file (Ctrl + S)
Save Copy As		Save a copy of the active file (e.g. test.db2) under a new name (e.g. simulation1.db2, works only in input mode). Useful for Batch-Simulations.
Export	• **Export Image File**	Create an image of the active window in a chosen format. You can choose the desired image format using the file extension (e.g. .png, .jpg, .gif, .tif, etc.).
	• **Export GIF Animation**	Create a GIF animation if the simulation (only in output mode). Change GIF animation interval [s] in the preferences.
Backup	• **Backup RAMMS Version**	Make a backup of the current RAMMS Version.
	• **Backup Active Project**	Backup your active project. The user will be asked if he wants to include output files in the backup. This function is useful when having problems with a simulation. Make a backup and send the zip-file together with some explanations to ramms@slf.ch. Make sure that all your input data (release area shapefiles, domain files, etc…) is in the project folder.
	• **Backup User Defined Files/Folders**	Backup any folder or files you want.
Preferences		Change RAMMS preferences
Log files	• **RAMMS Logfile (current)**	Show active RAMMS logfile.

	• **RAMMS Logfile (last session)**	If RAMMS crashed, open this logfile and copy/paste the content into an email to ramms@slf.ch.
Exit		Exit RAMMS (Ctrl + Q).

Edit

This menu is used to edit colorbar and dataspace properties.

Colorbar Properties	Edit the colorbar properties.
Get Colorbar	Get back your colorbar, if lost.
Edit Dataspace Properties	Edit your dataspace properties.
Show Dataspace Axes	Shows or hides dataspace axes of the project region. The axes can only be visible if the background color is set to white
Colorbar White Color	Checkbox. If checked, the colorbar text-color is white (default), otherwise black.

Input

Menu used to specify the global parameters, the calculation domain, release area, friction parameters and forest cover. This menu is active only in input mode.

Calculation Domain	• **Draw New Calculation Domain**	This activates the button to draw a new calculation domain. The mouse cursor changes to an arrow.
	• **Load Existing Domain**	Load an existing calculation domain (*.dom) drawn and saved before.
Polygon Shapefile…	• **Draw New Polygon Shapefile**	This activates the button to draw new polygon shapefiles. The mouse cursor changes to an arrow. Select points with the left mouse button, finish with a right mouse button click (the final right mouse button click is NOT a point of your polygon). This works only in 2D mode.
	• **Load Existing Polygon Shapefile**	Load an existing polygon shapefile.
Release Area	• **Details/Edit Release Areas**	The mouse cursor changes to an arrow and you can select release area to define the release height and to view release area information. This works only in 2D mode

Show

This menu enables and disables the different visualizations. A little arrow indicates if the visualization is enabled or disabled.

Show Lights	Show/hide light effects
Show Grid	Show/hide computational grid
Show Map	Show map
Show Image	Show orthophoto/image
Show Visualization	Show/hide release area (input mode) or simulation results (output mode)
Show Arrow	OUTPUT | Show/hide point arrow of time plot
Show Colorbar	Show/hide colorbar
Show Bottom Color	Show/hide 0-color
Show Velocity Arrow	OUTPUT | Show/hide velocity vector
Show Domain	INPUT | Show/hide calculation domain

Run

This menu is active only in input mode.

Run Calculation	Opens the *Run Simulation* window to change parameters and to start the calculation of a debris flow simulation.

Results

This menu contains the results functions and is only active in output mode.

Flow Height		Shows flow height of the debris flow every time step.	
Flow Velocity		Shows flow velocity of the debris flow for every time step.	
Flow Pressure		Shows flow pressure of the debris flow for every time step.	
Flow Momentum		Shows flow momentum of the debris flow for every time step.	
Erosion		Shows flow momentum of the debris flow for every time step.	
Max Values	• **Max Flow Height**	Displays the maximum flow height for each cell.	
	• **Max Velocity**	Displays the maximum velocity for each cell.	
	• **Max Pressure**	Displays the maximum pressure for each cell.	
	• **Max Flow Momentum**	Displays the maximum momentum for each cell.	
	• **Max Shear Stress**	Displays the maximum shear stress for each cell.	
	• **Max Erosion**	Displays the maximum erosion depth for each cell.	
Add Deposition to DEM		Adds the deposition of a debris flow simulation to a new DEM.	
Summary of Moving Mass		Summarizes the Moving Mass.	
Mu		Displays the friction parameter µ for this simulation.	
Xi		Display the friction parameter ξ for this simulation.	
Grid Cell Area		Display the grid cell area for each cell (m^2).	

GIS

This menu contains GIS functions.

Add Data		Add data (shapefiles, MuXi-ASCII files) to the visualization.	
Export …	• **Results As Shapefile**	Export the active result to an ESRI GIS shapefile for later use in a GIS program.	
	• **Results As ASCII Grid**	Displays the maximum velocity for each cell.	
	• **Envelope Shapefile**	Create an envelope shapefile from the active result.	
	• **Envelope Shapefile from ASCII File**	Create an envelope shapefile from an ASCII file. User can specify an ASCII file (e.g. max flow height).	
Add Dam to DEM		Adds a dam to the DEM. You have to specify relative dam height or absolute dam elevation.	
Show Slope Angle (°)		Display the slope angles.	
Show Curvature (1/m)		Display the curvatures.	
Show Contour Plot		Display a contour plot.	
Resample Slope/Curvature		Normally, slope an curvature are calculated for a grid resolution of 10m. You can change this resolution by using this function.	

Extras

Add/Change or Remove map		Add or change the topographic map of your project. The maps can be located in your project directory, or in your distribution's 'Map' folder, see section **Fehler! Verweisquelle konnte nicht gefunden werden.** for details. If not, you can browse for the maps.	
Add/Change or Remove Imager		Add, change or remove the image used for visualization of your project. The images can be located in your project directory, or in your distribution's 'IMAGE' folder, see section **Fehler! Verweisquelle konnte nicht gefunden werden.** for details. If not, you can browse for images.	
Point …	• **Choose Interactively**	This activates the button to select a point. The mouse cursor changes to an arrow. Select the point with the left mouse button. This works only in 2D mode.	
	• **Enter Coordinates (X/Y)**	Enter the coordinates of a point you are interested in.	
	• **Create Time Plot**	Create a time plot of a selected point.	
	• **View Info File**	View point info file.	
	• **Save Point Location**	Save point location as a point shapefile.	
	• **Export Time Plot Data**	Export time plot data as a txt-file.	
Profile …	• **Draw New Line Profile**	This activates the button to draw a line profile. The mouse cursor changes to an arrow. Select the points of the line profile with the left mouse button, finish with a right mouse click. This works only in 2D mode.	
	• **Save Line Profile Points**	Save your line profile as a polyline shapefile.	
	• **Export Line Profile Plot Data**	Export the line profile plot data as a txt-file.	
Save Active Position		Save your current state of view, as well as the enabled and disabled visualizations.	
Reload Position		Reload your saved position.	
Google Earth …	• **Export Result to Google Earth**	This function exports release areas and your results to Google Earth. If your project location is within Switzerland (default), you can use this function without changing any options. If not, see Map Options.	
	• **Map Options**	Enter map options if you want to export your result from a location outside of Switzerland	
	• **Map Options Help**	Get help about Google Earth Map Options.	
View input file		Opens the input file in a window.	
View Simulation Standard Output Log		Opens the simulations standard output log in a window (the black DOS window you see when a simulation is running).	

Project

This menu contains the project input and output logfiles.

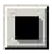
Input Log File	Displays the input logfile.	
Output Log File	Displays the output logfile. The input logfile is appended to the output logfile.	
Open Project Folder (Windows Explorer)	Opens project Folder in Window Explorer from within RAMMS.	

Help

RAMMS User Manual (pdf)		RAMMS User Manual in pdf format.
Friction Parameter Table (pdf)		Table of MuXi friction values in pdf format.
License Agreement		RAMMS License Agreement
RAMMS Homepage		Opens the RAMMS homepage at http://ramms.slf.ch in a web browser.
Update...		Download RAMMS updates manually or directly from the web.
Update…	• **Web Update**	Start web update procedure. RAMMS checks online if there is an update available.
	• **Get Update Manually (download to local folder)**	Download the update to a local folder.
	• **Install Update from local folder**	Install the update from a local folder.
Register New RAMMS Module		Register your new RAMMS::DEBRIS FLOW license here.
Advanced…	• **Color Tables – View Available Color Tables**	Choose a different type of color scheme for your colorbar.
	• **Additional Preferences - Edit**	Only for experts. Please contact ramms@slf.ch if you have questions about the additional preferences.
	• **Reset General Preferences**	Reset your general preferences (working directory, map directory etc.).
	• **Install C++ Libraries**	It is possible, that the Visual C++ Redistributable libraries for Visual Studio 2015 (x64) are not installed on your PC/laptop. These libraries are needed to run RAMMS. In case they are missing, you are not able to run simulations. Run this function to install these libraries.
	• **Logging**	Checkbox. Switch logging on or off.

	• **AutoWebUpdate**	Checkbox. Switch AutoWebUpdate on or off. If AutoWebUpdate is on, then RAMMS will check for updates whenever you start RAMMS.
	• **Hardware Rendering**	Checkbox. Switch hardware rendering on or off. If hardware rendering is switched on, then all graphical rendering is done by your hardware, otherwise by IDL (RAMMS). It is suggested to switch hardware rendering on.
	• **Curvature**	Checkbox. Switch curvature on or off. See section **Fehler! Verweisquelle konnte nicht gefunden werden.** n page **Fehler! Textmarke nicht definiert.** for more information about curvature.
	• **Technical Support Information**	If you have a problem using RAMMS, please send us the information from the *Technical Support Information* together with any error screenshots from RAMMS.
RAMMS Changelog		Information about the RAMMS releases in pdf format.
About RAMMS		About RAMMS information

6.1.2 Horizontal toolbar

	Project wizard: open debris flow wizard for creating a new debris flow project. (Ctrl + W)
	Open input file. (Ctrl + O)
	Open simulation. (Ctrl + A)
	INPUT | Save copy as: save the active file under a new name.
	INPUT and OUTPUT | Close: close the active file.
	Print: displays the Windows print manager.
	Undo, Redo.
	Arrow (move and resize), Rotate, Move.
	Simulation Results: Choose this function and move the arrow over the topography → x-, y- and z-Coordinates of the mouse position are shown in the lower right status bar (see Figure 6.11 on page 100).
	OUTPUT | If you move the arrow over the simulation data, the active parameter is shown as well (see right value in the figure below). If you click once with the left mouse button at a point of interest, a new window pops up called 'RAMMS::DEBRISFLOW Time Plot <Active Parameter>'.
	INPUT, 2D | Draw new polygon shapefile: specify new polygon-points by clicking the left mouse button, finish with a right mouse click. The user is asked if he wants to draw more polygons. At last, he has to specify a new filename for the polygon shapefile.
	INPUT, 2D |Create new domain area: specify a new domain polygon by clicking with left mouse button, for the last polygon-point click the right mouse button to finish. A dialog box will then ask the user for a new domain name (e.g. test).
	OUTPUT, 2D | Line Profile: Select the topography, until the Line-Profile-Button is active. Click the button and then move the cursor to the start point of your profile. Click the left mouse button and move the cursor to the next position of your profile. At the end position of your profile click the right mouse button. A new window pops up called 'RAMMS::DEBRISFLOW Line Profile Plot Active Parameter'. This line profile plot is linked to your simulation. If you change the parameter or if you change the max-value in the debris flow panel, the changes are adapted in the line profile plot!
	INPUT, 2D | View and Edit Release Areas.
	Zoom tools.
	Annotation tools, text, line, rectangle, oval, polygon, freehand. They can be activated and deactivated in the additional preferences. *Preferences → Advanced… → Edit → Annotations*
	Interpretation of the input DEM: Slope Angle, Curvature and Contour Plots. Remove visualization by clicking the button again.

	OUTPUT | Show Maximum values of the simulation results: Max. Flow Height, Max. Flow Velocity and Max. Pressure.
	OUTPUT, 2D |create a time plot for the last point location.
	OUTPUT | export the results to ASCII grid.
	Open project folder in Windows Explorer.
	Add/change maps/orthophotos.

6.1.3 Vertical Toolbar

	Add shapefile (*.shp).
	Switch to input file of an already open simulation.
	Show/hide lights.
	Show/hide mesh.
	INPUT | Show/hide release area (or active parameter).
	OUTPUT | show/hide simulation.
	Show/hide colorbar.
	OUTPUT | Show/hide velocity vector arrow.
	Show map.
	Show image.
	INPUT | Run Simulation.
	OUTPUT | Animate Simulation / Continue Simulation. Stop/Pause Simulation ( ).
	OUTPUT | End Simulation: skip to last dump-step of simulation.
	Create a screenshot of the main window.
	OUTPUT | Create GIF animation.
	Change RAMMS Additional Preferences.
	Edit dataspace properties.
	Change RAMMS preferences (e.g. working directory).
	Change view to 2D / Change view to 3D (3D).
	Refresh visualization (if stuck).

6.1.4 Main window

The RAMMS GUI (**G**raphical **U**ser **I**nterface) consists of two main regions, see Figure 6.2:

1. Main visualization window
2. Information panel, see section below.

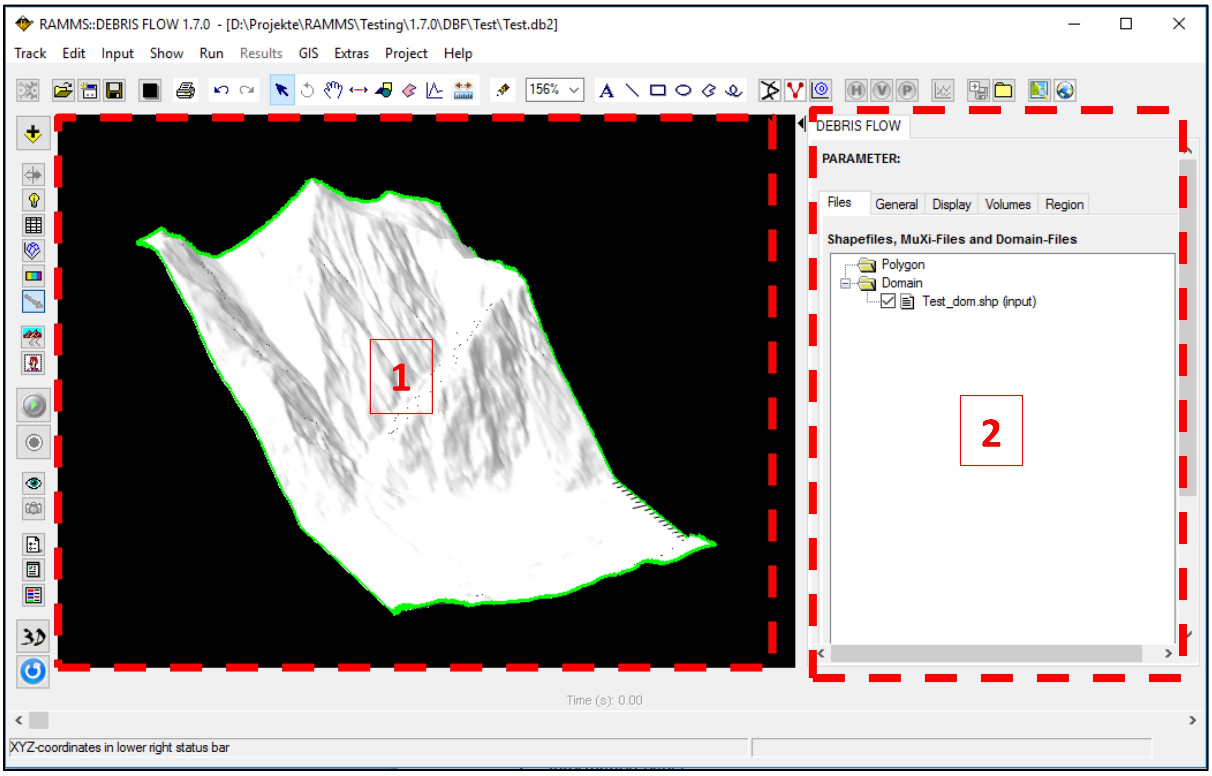


Figure 6.2: RAMMS GUI.

6.1.5 Panel

A DEBRIS FLOW panel is displayed on the right side of the RAMMS GUI (Figure 6.2), and consists of five tabs (*Files*, *General, Display, Volumes* and *Region*).

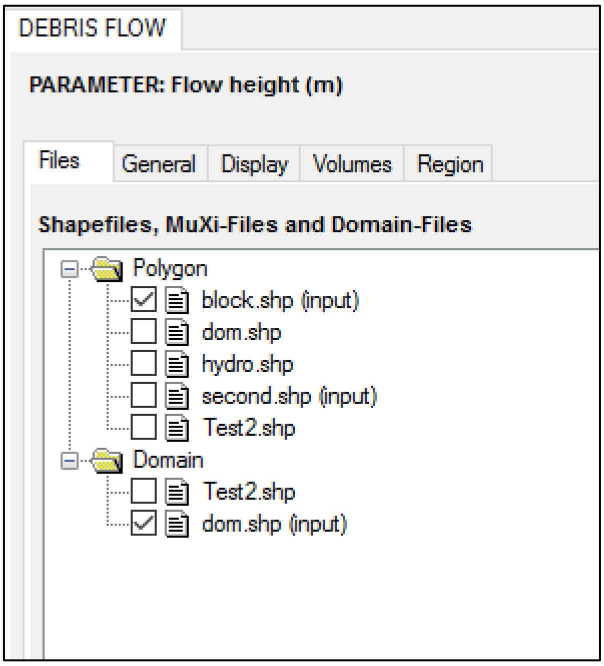


Figure 6.3: Debris flow panel – Files tab

Files tab

The *Files* tab (Figure 6.3) shows a file tree with nodes for polygon shapefiles (*Polygon, *.shp*) and calculation domain files (*Domain*, *.shp). See section 3.4.1 on page 25 on how to use the Files tab.

General tab

The *General* tab (Figure 6.4) shows important simulation parameters, such as: nr. of nodes, nr. of cells, end time (s), dump-step (s), grid resolution (m) and density (kg/m^3). In input mode, for handling and visualization purposes, the topographic information is resampled, such that there are only ca. 50'000 grid cells remaining (see *Visualization Resampling Remarks* in Figure 6.4, red box). This does not influence any simulation at all, it simply makes the users life easier to zoom and rotate the topography.

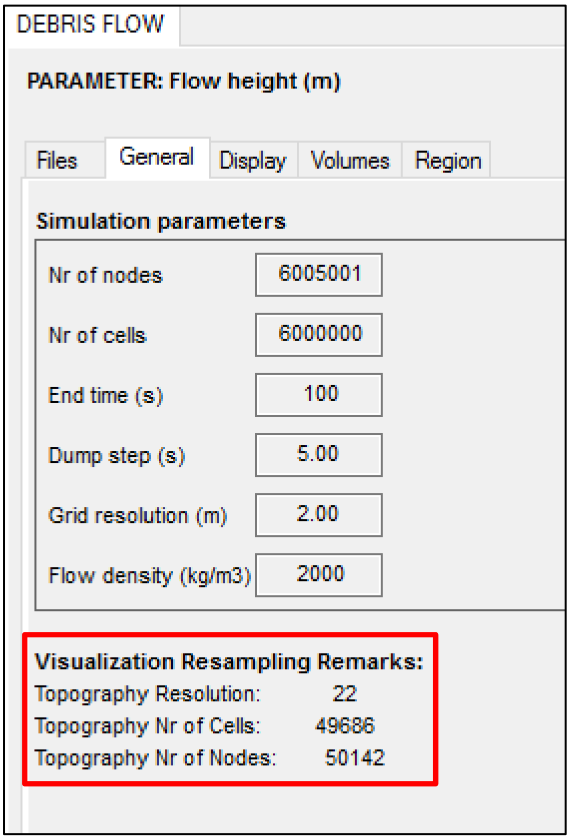


Figure 6.4: Debris flow panel – General tab

Display tab

The *Display* tab (Figure 6.5) shows parameters that are important for the display of results and polygon shapefiles, such as Min_Value, Max_Value, Nr_of_Colors and Transparency. Always confirm with ***ENTER*** (return key) when changing a value! Additionally, the *PARAMETER* line states the visible parameter (e.g. Flow height (m) in Figure 6.5, red box).

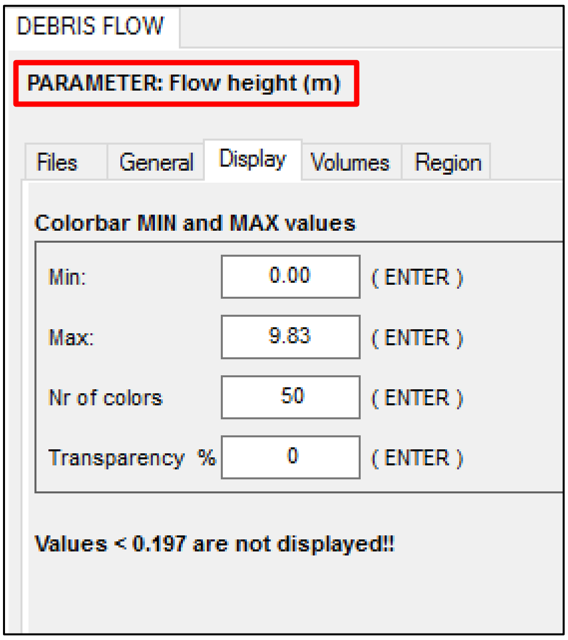


Figure 6.5: Debris flow panel – Display tab

The min and max values as well as the number of colors influence directly the colorbar and the visualization. The transparency of the simulation results can be changed on the debris flow panel, tab display. 0% means no transparency, 100% means total transparency, see figure below (Figure 6.6). The colorbar is divided into n (nr. of colors) different colors, where the lowest color is normally not displayed. The bottom line informs the user of the range of values that are not displayed in the current visualization (only in output mode).

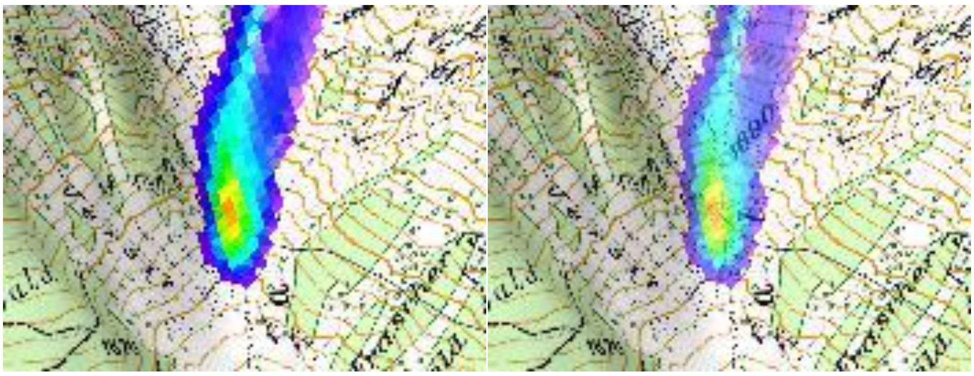


Figure 6.6: No transparency (left) and 40% transparency (right) of simulation result.

Volumes tab

The *Volumes* tab (Figure 6.7) gives the user information about

- projected release area (m^2)
- inclined (3D) release area (m^2)
- release volume (m^3, estimated in input mode)
- release mass (t, input) / flow volume (m^3, output)

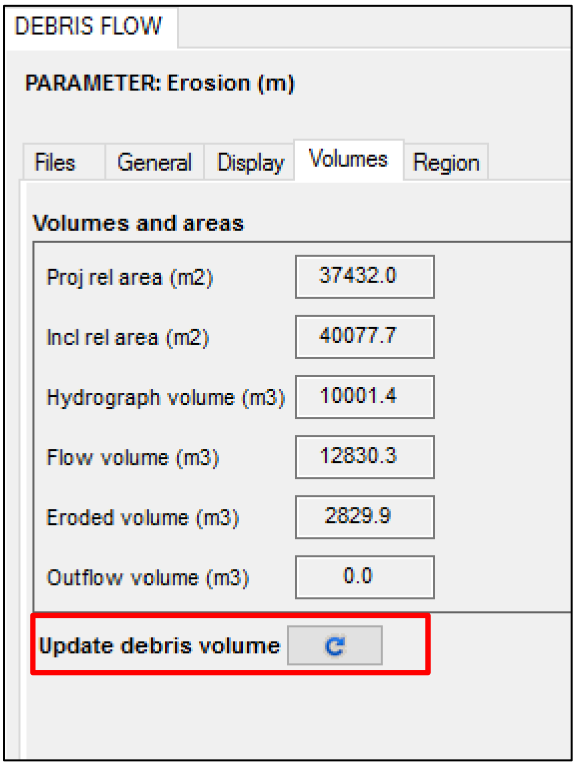


Figure 6.7: Debris flow panel – Volumes tab

Region tab

The *region* tab (D) gives information about min and max X-, Y-coordinates and the altitude limits as well as information about the region area in km^2.

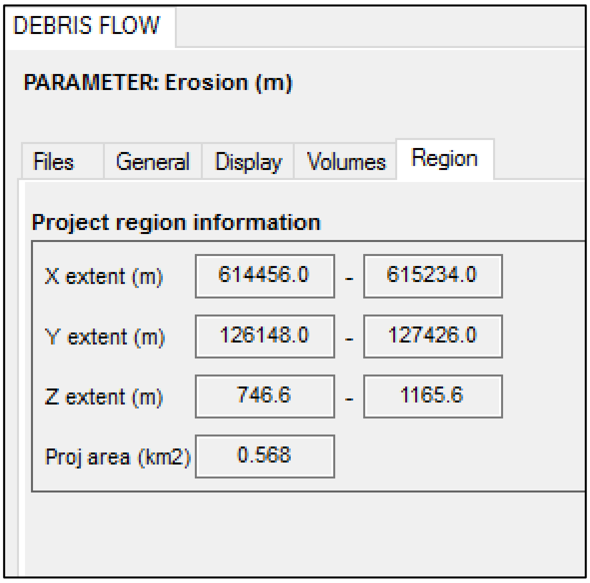


Figure 6.8: Debris flow panel – Region tab

6.1.6 Time step slider

The time step slider can be moved manually to change the active time step (only in output mode).

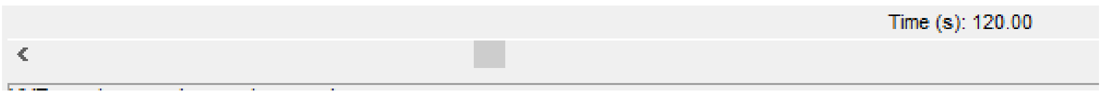


Figure 6.9: The active time (20s) is shown in the time slider .

6.1.7 Left status bar

The left status bar is used to display status information for operations or informational messages pertaining to the currently selected surface or manipulators.

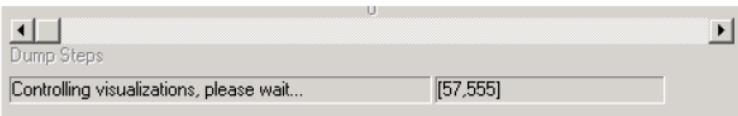


Figure 6.10: Status information shown in the left status bar.

6.1.8 Right status bar

The right status bar is used to display the position of the cursor within the surface and additional simulation results at the position of the cursor.

Figure 6.11: Position information and triangle simulation results in the right status bar.

6.1.9 Colorbar

In general, the colorbar appears at the right edge of the main window (Fig. 6.1) and can be moved and resized (see section 3.4.4 on page 30).

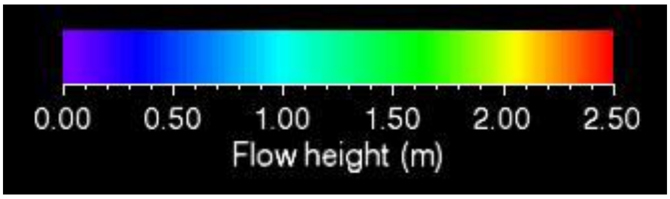


Figure 6.12: Colorbar

7 References and further reading

7.1 References

Maps and aerial images

➔ All topographic base maps and aerial images are reproduced © 2015 swisstopo (JA100118).

Literature

- **Ayotte D. and Hungr O. 2000**: Calibration of a runout prediction model for debris flow and and Naeser, Rotterdam, 505-514.
- **Badoux A., Graf C., Rhyner J., Kuntner R. and McArdell B.W. 2009**: A debris-flow alarm system for the Alpine Illgraben catchment: design and performance. In: Natural Hazards, 49(3), 517-539.
- **Berger C. 2010**: Debris flow entrainment and sediment transfer processes at the Illgraben catchment, Switzerland. Ph.D. thesis, University of Bern, Inst. of Geological Sciences, Bern, Switzerland.
- **Berger C., McArdell B. W. and Schlunegger F. 2011**: Direct measurement of channel erosion by debris flows, Illgraben, Switzerland, J. Geophys. Res., 116, F01002, doi: 10.1029/2010JF001722.
- **Bovis M.J. and Jakob M. 1999**: The role of debris supply conditions in predicting debris flow activity. In: Earth Surface Processes and Landforms, 24, 1039-1054.
- **D'Agostino V. and Cesca M. 2008**: Ricostruzione di alcuni dei principali eventi torrentizi avvenuti in Provincia di Trento. Technical report, Trento Autonomous Province.
- **Deubelbeiss Y. and Graf C. 2013**: Two different starting conditions in numerical debris flow models - Case study at Dorfbach, Randa (Valais, Switzerland). In: Graf, C. (Red.) 2013: Mattertal - ein Tal in Bewegung. Publikation zur Jahrestagung der Schweizerischen Geomorphologischen Gesellschaft 29. Juni - 1. Juli 2011, St. Niklaus. Birmensdorf, Eidg. Forschungsanstalt WSL, 125-138.
- **Hürlimann M., Rickenmann D. and Graf C. 2003**: Field and monitoring data of debrisflow events in the Swiss Alps. In: Canadian Geotechnical Journal, 2003, 40(1), 161-175.
- **Johannesson et al.** 2009: The design of avalanche protection dams. Recent practical and theoretical developments. European Commission. Directorate General for Research, 2009.
- **Mizuyama T., Kobashi S. and Ou G**. **1992**: Prediction of debris-flow peak discharge. Proceedings of the international Symposium Interpraevent, Bern, Switzerland, 4, 99-108.
- **Rickenmann D. 1995**: Beurteilung von Murgängen. In: Schweiz. Ingenieur und Architekt 113(48), 1104-1108.
- **Rickenmann D. 1999**: Empirical relationships for debris flows. In: Natural Hazards, 19, 47-77.
- **Rudolf-Miklau F. and Sauermoser S. 2011**: Handbuch Technischer Lawinenschutz. Ernst & Sohn GmbH&Co.
- **Salm B., Burkard A. and Gubler H. 1990**: Berechnung von Fliesslawinen: eine Anleitung für Praktiker mit Beispielen. Mitteilung 47, Eidg. Institut für Schnee- und Lawinenforschung SLF.

- **Salm B. 1993**: Flow, flow transition and runout distances of flowing avalanches. In: Annals of Glaciology 18, 221-226.
- **Varnes D.J. 1978**: Slope movement types and processes. In: Land-slides Analysis and Control, edited by Schuster, R.L. and Krizec,R.J., Spec. Per. Natl. Res. Counc. Transp. Res. Board, 176, Natl. Acad. of Sci., Washington D.C., 11-33.

7.2 Publications

The development of RAMMS is based on scientific findings published in international scientific journals. A list of the most important scientific publications about RAMMS and its applications can be found on our homepage at http://ramms.slf.ch, section RESOURCES-Publications.

List of Figures

List of tables

Third-Party Software

The following third-party software components are used in RAMMS:

7-zip:

- We sometimes use *7za.exe* to zip data.
- *7-zip* is licensed under GNU LGPL.
- The source code of *7-zip* is available at www.7-zip.org.

Mtee:

- *Mtee* is a Win32 console application that sends any data it receives to stdout and to any number of files.
- *Mtee* is released under MIT License https://ritchielawrence.github.io/mtee/.

Index