RAMMS::AVALANCHE User Manual

RAMMS

rapid mass movement simulation

A numerical model for snow avalanches in research and practice

User Manual v1.8.0

AVALANCHE

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

Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

Title picture: Vallée de la Sionne, WSL

Manuscript update

October 2022

Table of Content

1 Introduction

In the field of natural hazards there is an increasing need for process models to help understand the motion of geophysical movements. These models allow engineers to predict the speed and mass of hazardous movements in complex terrain. Such models are especially helpful when processing mitigation measures, such as avalanche dams or snow sheds. Hazard mapping is an especially important application in Switzerland and other mountainous countries. An accurate prediction of runout distances, flow velocities and impact pressures in general three-dimensional terrain is the driving motivation for the development of dynamical mass movement models. Although helpful and well-liked by users, one-dimensional models such as AVAL-1D require that the primary flow direction and flow width must be defined by the user in advance. This is often difficult, especially in open terrain, or in terrain consisting of several possible flow channels. Furthermore, flow interaction with catching and deflecting dams cannot be accurately modeled using one-dimensional simulation codes.

RAMMS (Rapid Mass Movement Simulation) is a two-dimensional, state-of-the-art numerical simulation model to calculate the motion of geophysical mass movements (snow avalanches, rockslide, debris flows and shallow landslides) from initiation to runout in three-dimensional terrain. It was designed to be used in practice by hazard engineers who need solutions to real, everyday problems. It is coupled with a user-friendly visualization tool that allows them to easily access, display and analyze simulation results. New constitutive models have been developed and implemented in RAMMS, thanks to calibration and verification at full scale tests at sites such as Vallée de la Sionne. These models allow the application of RAMMS to solve both large, extreme avalanche events as well as smaller mass movements such as hillslope debris flow and shallow landslides.

RAMMS is developed since 2010 by the RAMMS 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 https://ramms.slf.ch provides useful information such as a forum, frequently asked questions (FAQ) or recent software updates. Please visit this web page frequently to stay up to date!

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.

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 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).

1.3 Learning by doing

This manual provides an overview of RAMMS::AVALANCHE. 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 an avalanche 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::AVALANCHE:

- Operating Systems: Windows 8, 10 and 11 (64-bit)
  32-bit systems (Win XP) are not supported 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::AVALANCHE setup file *"ramms_user_setup_64.zip"* from https://ramms.slf.ch/en/modules/avalanche.html (Downloads tab at the bottom of page).

Direct download link: https://ramms.slf.ch/ramms/downloads/ramms_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_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>_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.

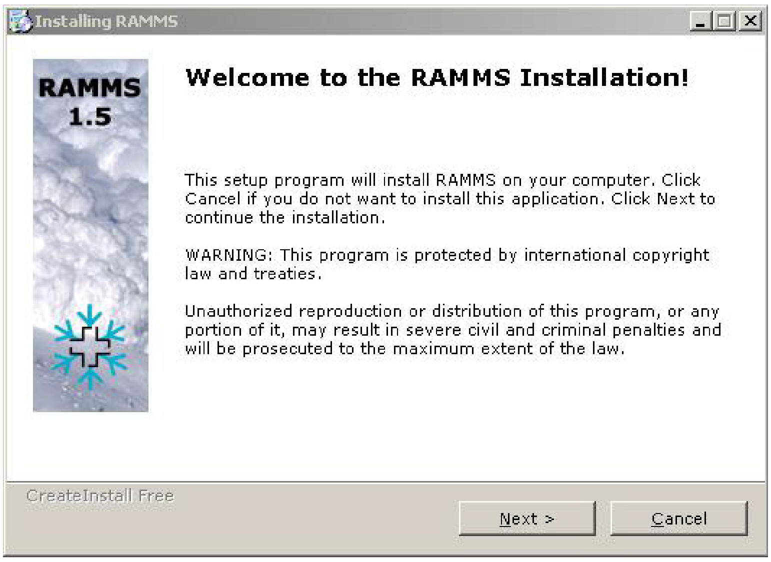


Figure 2-1: Installation - welcome dialog window.

Step 2: Readme

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

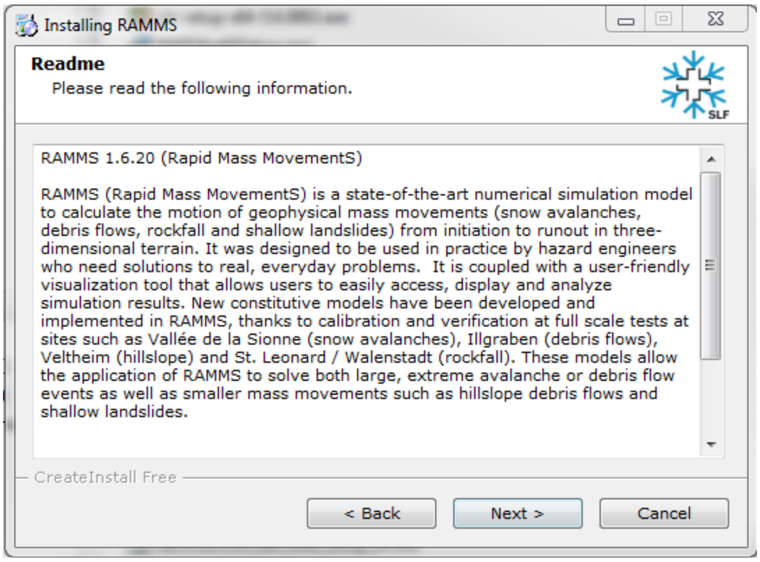


Figure 2-2: Installation - Readme dialog window.

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.

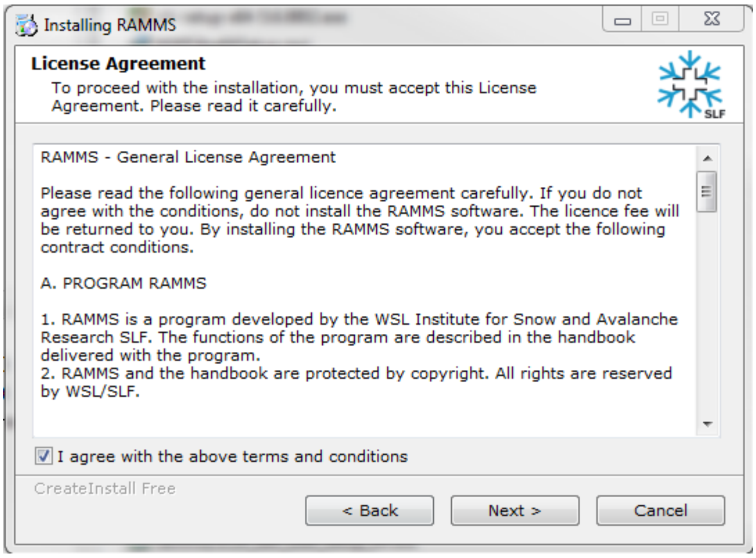


Figure 2-3: Installation - license agreement dialog window.

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. Click *Next* to start the installation process.

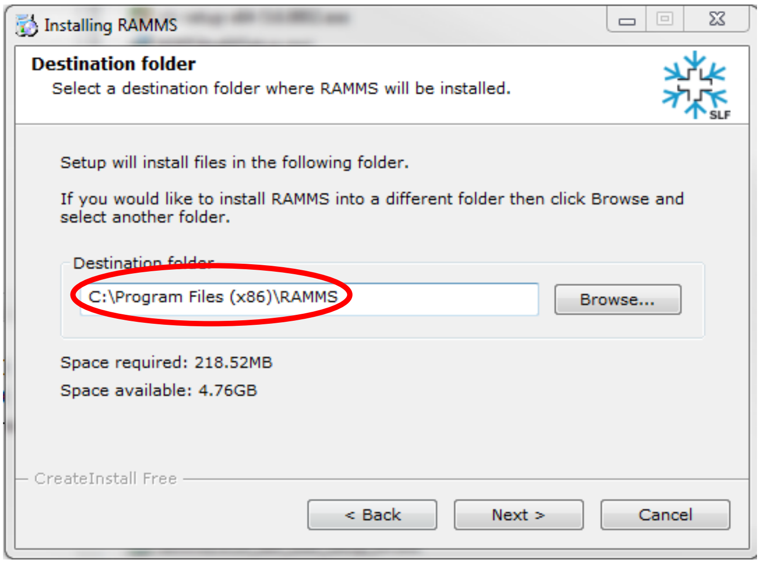


Figure 2-4: Installation - destination directory dialog window.

Step 5: Installing the files

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

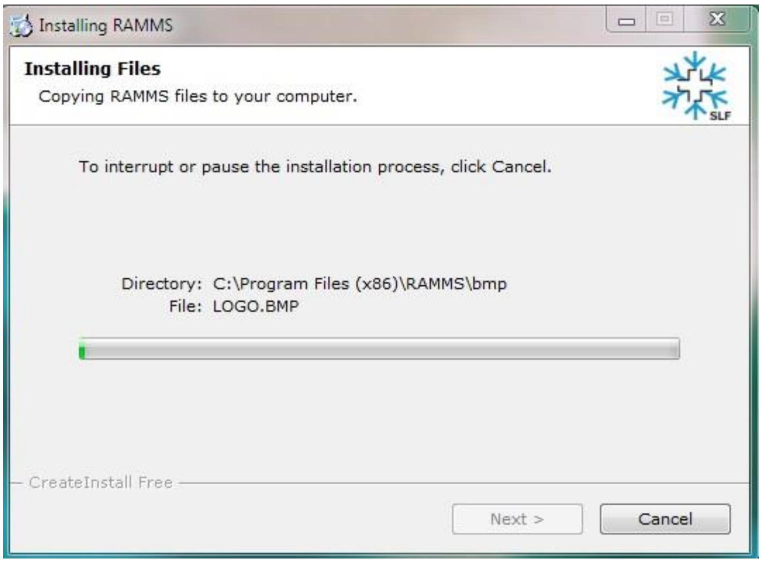


Figure 2-5: Installation - installing files dialog window.

Step 6: Finished installing the files

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

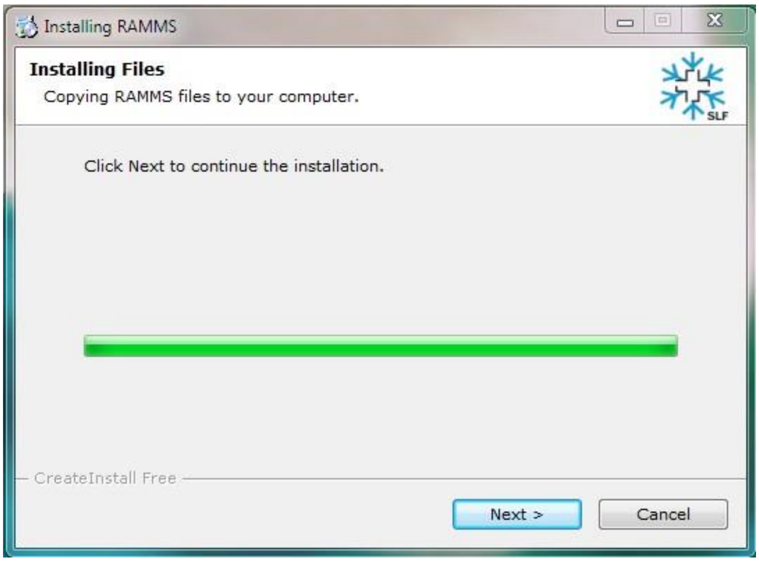


Figure 2-6 : Installation - finished installing files dialog window.

Step 7: RAMMS installation finished!

RAMMS successfully finished the installation. Click *Finish*.

Figure 2-7: Installation - finished installation dialog window.

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.

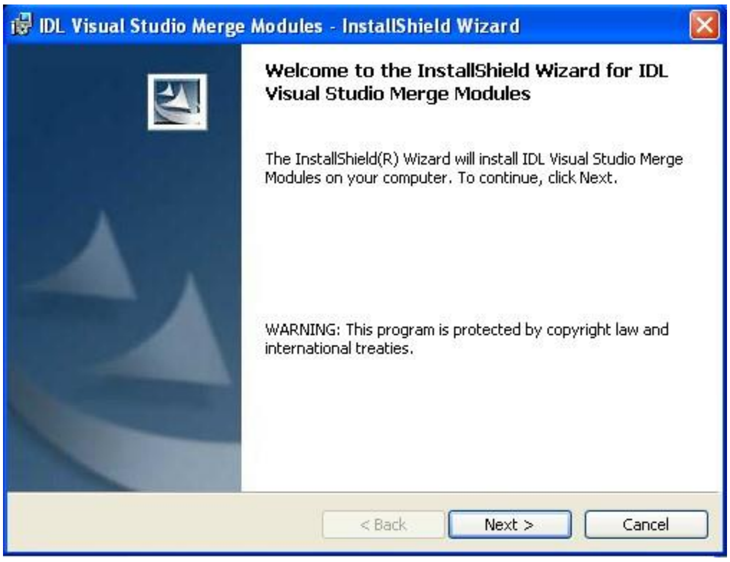


Figure 2-8: IDL Visual Studio Merge Modules - welcome dialog window.

Step 9: Ready to install the program

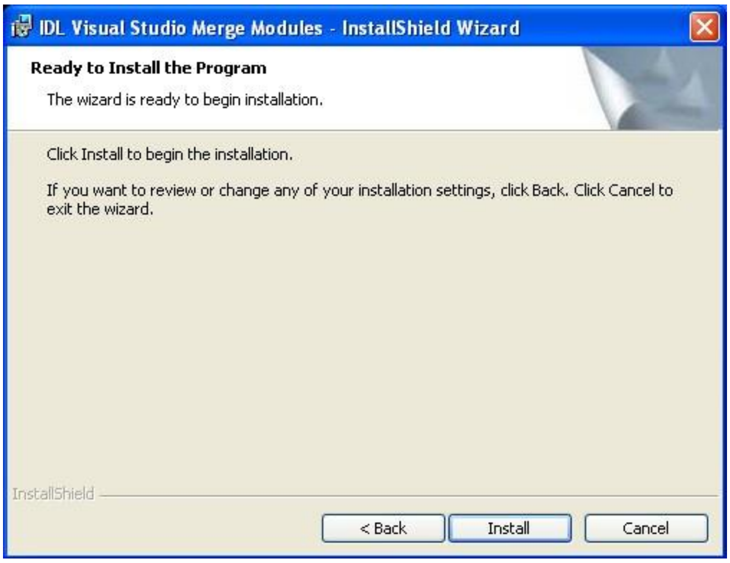


Figure 2-9: IDL Visual Studio Merge Modules - ready to install the program.

Click *Next* to continue.

Step 10: Installing IDL Visual Studio Merge Modules

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

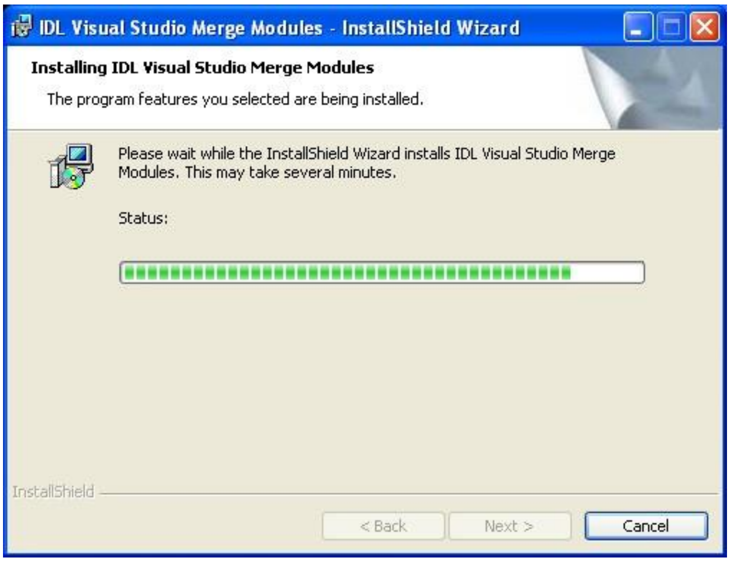


Figure 2-10: IDL Visual Studio Merge Modules - installing...

Step 11: InstallShield Wizard Completed

The wizard completed the installation. Click *Finish*.

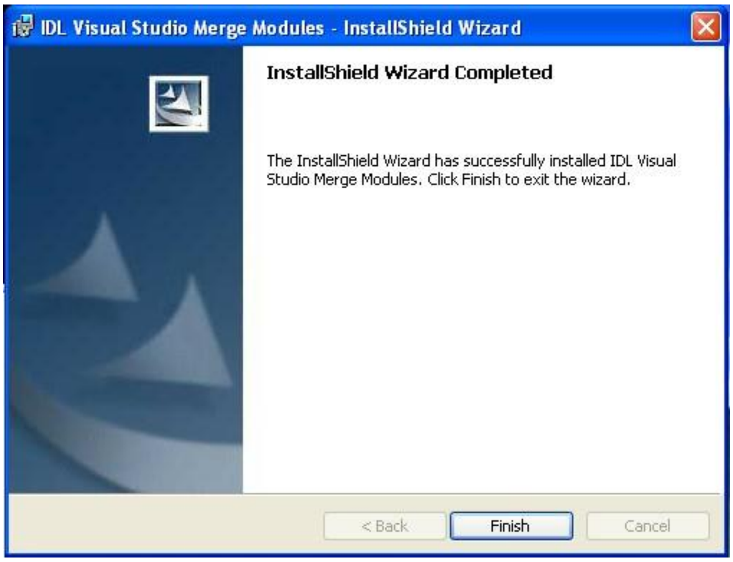


Figure 2-11: Installation - destination directory dialog window.

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*

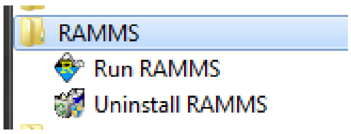


Figure 2-13: RAMMS program group

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 https://ramms.slf.ch. Alternatively, the license can be installed on a Windows Server and accessed by different users (only one at a time) by RDC (Remote Desktop Connection). This only works for one license per module.

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:

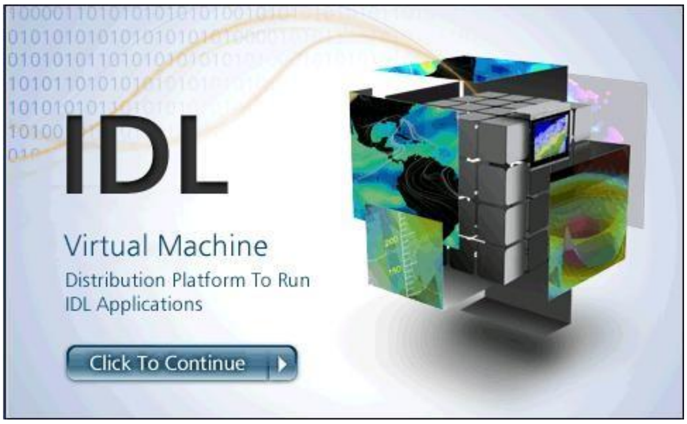


Figure 2-14: RAMMS start window.

Click on the image. It will disappear and RAMMS will start up. The following dialog window appears (Figure 2-15 RAMMS Licensing):

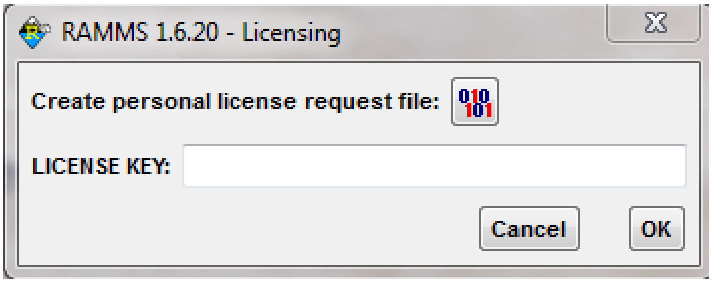


Figure 2-15: RAMMS licensing window

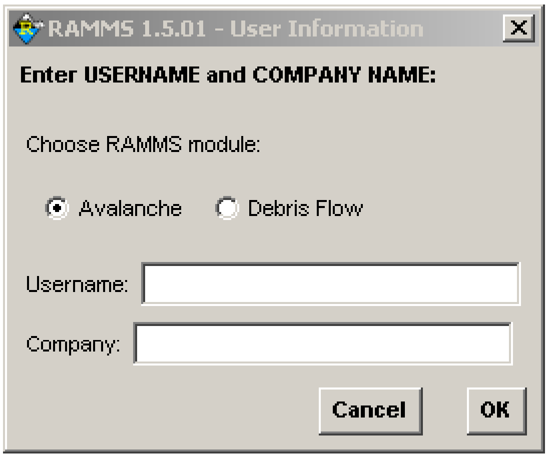


Figure 2-16: Enter user name and company name.

2.4.1 Personal license request file

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.

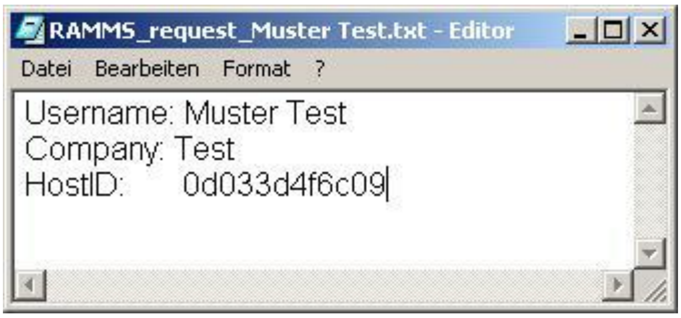


Figure 2-17: Personal license request file RAMMS_DBF_request_TestName.txt

2.4.2 Getting the personal license key

You find order forms on the RAMMS website (*Full License, Demo License or Student License*) at https://ramms.slf.ch/en/licenses.html. 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. You will find all the banc details on this invoice for a banc transfer. Please let us know if you wish to pay by credit card. After we have received your payment, we will send you your personal license key, valid according to the license period you purchased. Your personal license key is named in the following manner:
"AVA_20151013_TestName_TestCompany_RAMMS_TimeLicense.txt". Open the file in a text editor. It should look similar to

Figure 2-18 below. The filename reveals also the end date of your license period, e.g. "20151013" = October 13th 2015.

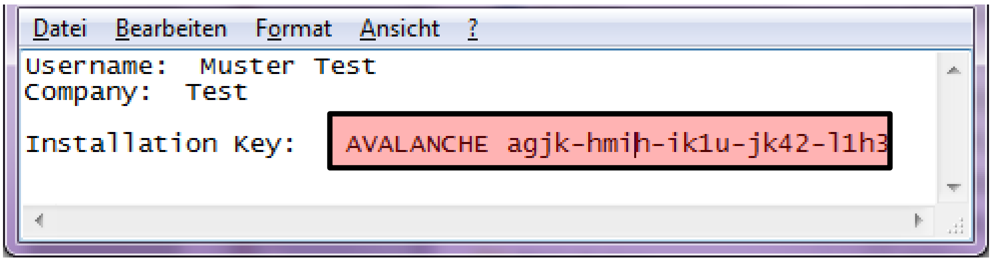


Figure 2-18: Personal license key file RAMMS_license_Muster Test.txt

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: *AVALANCHE agjk-hmih-ik1u-jk42-l1h3*) and paste it in the field *LICENSE KEY* (see Figure 2-15). Notice that there is the prefix AVALANCHE. 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.

2.4.3 License Transfer

If you want to transfer your RAMMS license to a new PC, do the following:

- install RAMMS on the new PC
- generate a new license request file
- send us the new license request file TOGETHER with your old license file by email
- we will then send you the new license file (email)
- uninstall RAMMS from your old PC

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- or GEOTIFF-format), project boundary coordinates and georeferenced maps or orthophotos should be prepared in advance (.tif format and .tfw-file, maps and orthophotos 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:

1. ESRI ASCII grid (Figure 3-1)
2. GEOTIFF (georeferenced information embedded within a TIFF file)
3. ASCII XYZ regular, single space data (Figure 3-2)

ASCII XYZ data (regular and irregular) can be converted within RAMMS into an ASCII or GEOTIFF grid. A wizard will guide you through the conversion process. The following interpolation methods are available: *LINEAR* or *INVERSE DISTANCE.*

The header of an ESRI ASCII grid must contain the information shown below in Figure 3-1.

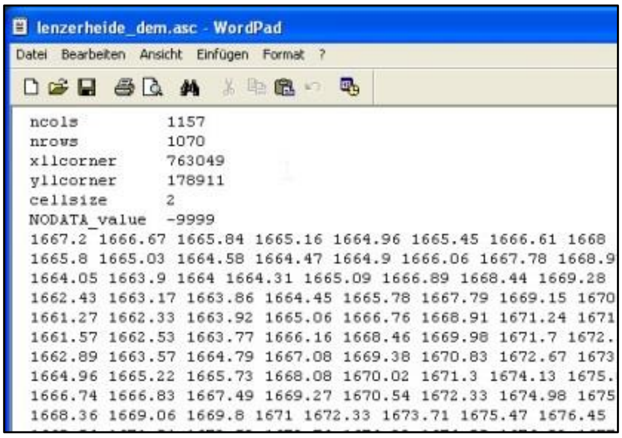


Figure 3-1 : Example ESRI ASCII grid.

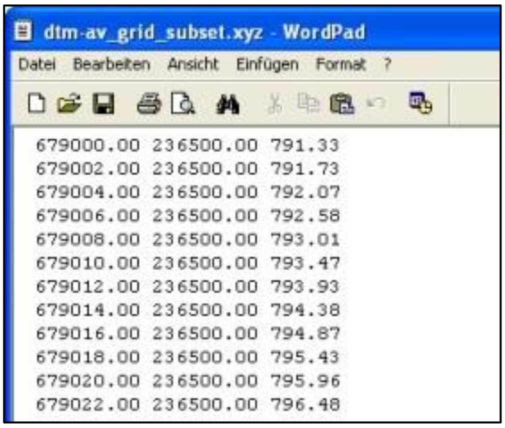


Figure 3-2: Example ASCII XYZ single space data.

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.

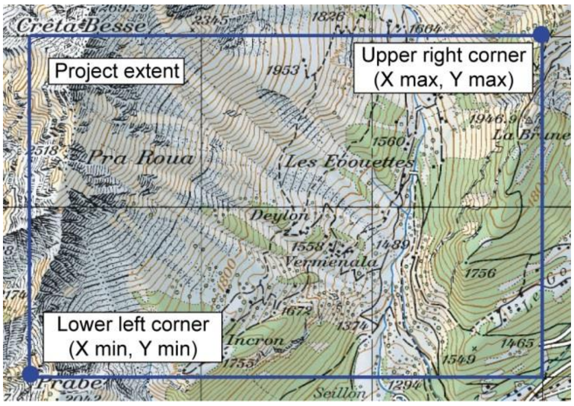


Figure 3-3: The same project extent (area of interest) can be used to calculate different scenarios with different input parameters.

3.1.3 Release information

The definition of release areas and release heights have a very strong impact on the results of RAMMS simulations. Therefore, we recommend to use reference information such as photography, PGS measurements or field maps to draw release areas. This should be done by people with experience concerning the topographic and meteorological situation of the investigation area.

Users can use any polygon shapefile as a release area, see section 3.5.1 on page 38. Release areas drawn in RAMMS are saved as polygon shapefiles and can be easily imported in GIS-Software (e.g. ArcGIS or QGIS). Shapefiles created in e.g. ArcGIS or QGIS can of course be used in RAMMS.

3.1.4 Friction information

RAMMS employs a Voellmy-fluid friction model, which is based on the Voellmy-Salm approach (we refer to Salm et al. 1990 [3] and Salm 1993 [4]).

Physical friction model

The physical model of RAMMS::AVALANCHE 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

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

where ρ is the density, g the gravitational acceleration, φ the slope angle, h the flow height and **u** the vector $\mathbf{u} = (u_x,u_y)^T$, consisting of the avalanche velocity in the x- and y-directions. The normal stress on the running surface, $\rho hg\cos(\varphi)$, 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 of a calculation domain are constant. However, you have the possibility to add up to two polygons within the calculation domain with different friction parameters (see section 3.5.4 *"How to run a calculation" on page* 50.)

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.

Cohesion

Since Version 1.6.20 the basic Voellmy equation has been modified to include a cohesion (Bartelt et al. 2015 [5]). 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 $S \to 0$ when both $N \to 0$ and $U \to 0$. It increases the shear stress and therefore causes the avalanche to stop earlier, depending on the value of N_0 .

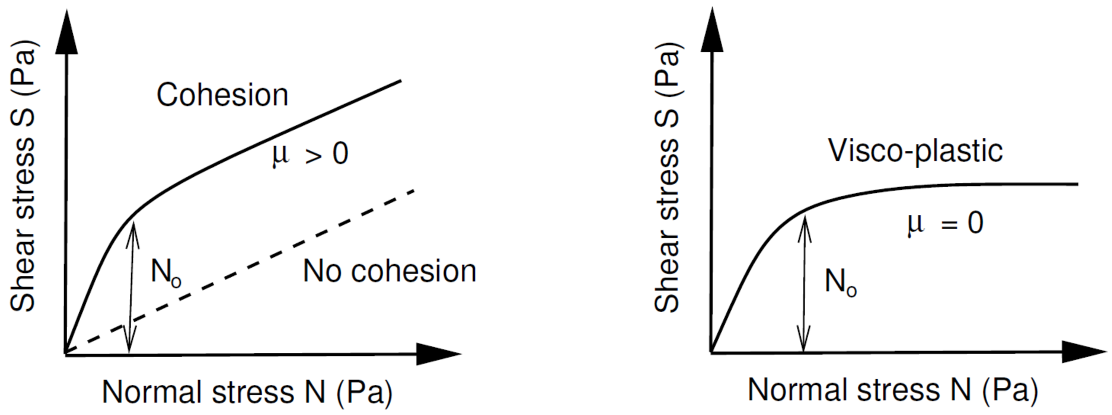


Figure 3-4: Relation between normal and shear stress. Left: Yield stress N_0 serves to increase the shear stress for higher normal pressures. At low normal pressures (small flow heights) the shear stress increases rapidly from $S=0$ to $S=N_0$. The slope of the 'S vs N' relation remains μ, when the normal 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) [6], which was specifically developed for RAMMS. The centrifugal acceleration *f* is both a function of the avalanche velocity and terrain curvature. The acceleration is calculated according to

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

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 avalanche to slow down in tortuous and twisted flow paths. It can change the location of the deposition once the flow leaves the gully. Curvature may be activated/deactivated in the *Run Simulation* window (tab *Params*) or via the menu 'Help → Advanced… → Curvature'.

Friction parameters μ and ξ

RAMMS::AVALANCHE offers a constant and a variable calculation mode. If a calculation is done with constant friction values, of course, no terrain undulations and forest areas are considered. Therefore, we suggest to use the variable friction values if possible. μ and ξ values are saved as ASCII files (called MuXi-files) and can be easily imported in GIS-Software (e.g. ArcGIS)

The creation of a new MuXi-file is demonstrated in the exercise 3.5e *"How to create a new MuXi-file"* on page 49.

Automatic MuXi Procedure

First Step: RAMMS performs a terrain analysis, where slope angles and curvatures are analyzed, resulting in four classification classes: flat, open slope, channeled and gully, see example in Figure below.

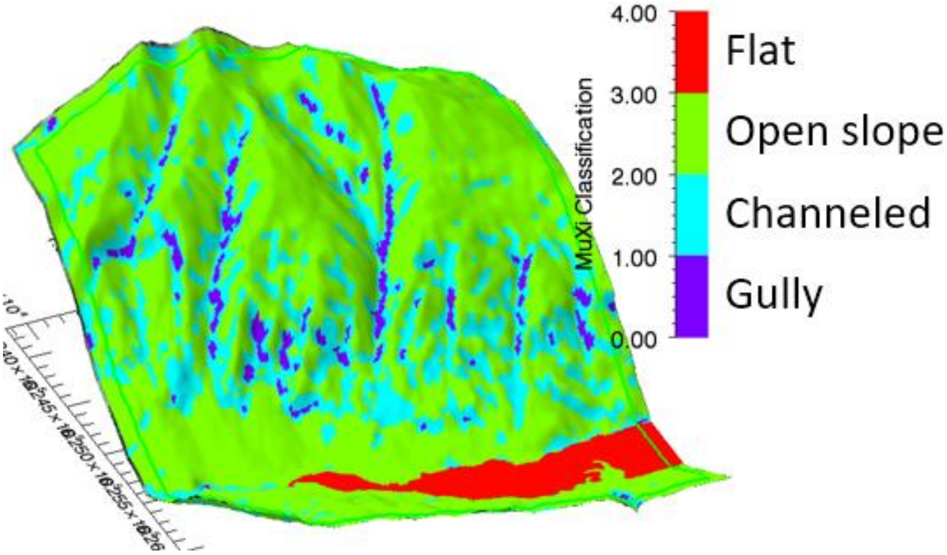
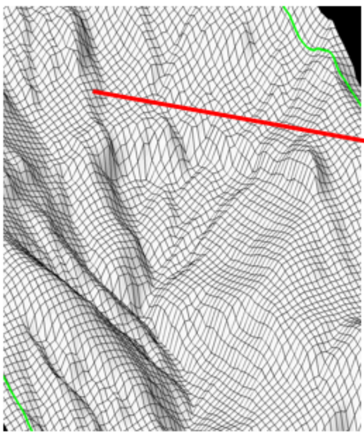


Figure 3-5: Automatic terrain analysis in RAMMS

In the next step, RAMMS uses the classification from above together with forest information, global parameters (*return period* and *avalanche volume*, see Figure 3-8) and altitude limits to assign the μ and ξ values.



Large avalanche (> 60'000 m³)		300-Year		100-Year		30-Year		10-Year	
	Altitude (m.a.s.l.)	μ	ξ	μ	ξ	μ	ξ	μ	ξ
unchannelled	above 1500	0.155	3000	0.165	3000	0.17	3000	0.18	3000
	1000 - 1500	0.17	2500	0.18	2500	0.19	2500	0.2	2500
	below 1000	0.19	2000	0.2	2000	0.21	2000	0.22	2000
channelled	above 1500	0.21	2000	0.22	2000	0.225	2000	0.235	2000
	1000 - 1500	0.22	1750	0.23	1750	0.24	1750	0.25	1750
	below 1000	0.24	1500	0.25	1500	0.26	1500	0.27	1500
gully	above 1500	0.27	1500	0.28	1500	0.29	1500	0.3	1500
	1000 - 1500	0.285	1350	0.3	1350	0.31	1350	0.325	1350
	below 1000	0.3	1200	0.315	1200	0.33	1200	0.345	1200
flat	above 1500	0.14	4000	0.15	4000	0.155	4000	0.165	4000
	1000 - 1500	0.15	3500	0.16	3500	0.17	3500	0.18	3500
	below 1000	0.17	3000	0.18	3000	0.19	3000	0.2	3000
Medium avalanche (25'000 - 60'000		300-Year		100-Year		30-Year		10-Year	
unchannelled	above 1500	0.195	2500	0.205	2500	0.215	2500	0.225	2500
	1000 - 1500	0.21	2100	0.22	2100	0.23	2100	0.24	2100
	below 1000	0.23	1750	0.24	1750	0.25	1750	0.26	1750

Figure 3-6: Automatic assigning of μ and ξ values to grid cells

Altitude limits

The default altitude limits that are used in the Automatic MuXi Procedure, are 1000 and 1500 m.a.s.l., valid for the Alps.

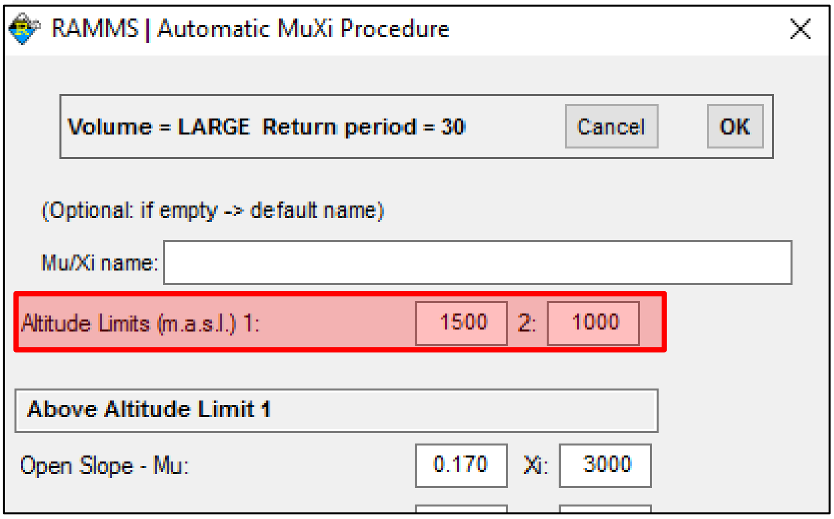


Figure 3-7: Altitude limits for automatic MuXi procedure

The two altitude limits can be interpreted as follows:

- Alt. Limit 1 (higher value): ***Tree line*** of your project region should be higher than this limit
- Alt. Limit 2 (lower value): ***Snow line*** of your project region

Example altitude limits are provided (only suggestions, without warranty) in the table below:

Table 1: MuXi Altitude Limits, examples for different climatic regions

Regions	**RAMMS Altitude Limits**	**Climate**	
		Tree line (m)	**Snow line (m)**
Alps	1500 / 1000	2100	1000
Bansko (BUL)	1500 / 1000	2100	1000
Pyrenees	1700 / 1200	2200	1200
Socchi (RUS)	1700 / 1200	2200	?
Norway (West Coast)	500 / 200	800	100
Juneau (USA)	500 / 200	700	200?
Wyoming (USA)	2000 / 1500	3000	1500?
Chile (Santiago Region)	2500 / 2000	3000	2000
Himalaya (Manali, India)	3000 / 2500	3500	2000

3.1.5 Global parameters

The friction values μ and ξ strongly depend on the global parameters *return period* and *avalanche volume* (see MuXi-table on page 98). Therefore, an appropriate return period has to be defined and the avalanche volume has to be checked under *Input* → *Global Parameters* prior to creating a new MuXi-file (see Figure 3-8 and exercise *"How to create a new MuXi-file"* on page 49).

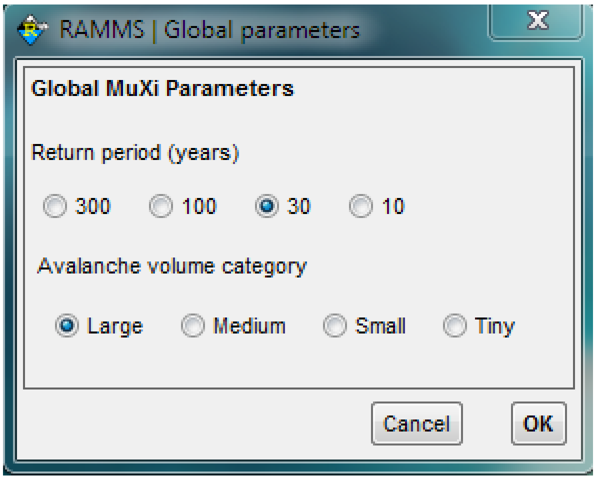


Figure 3-8: RAMMS global parameters.

3.1.6 Forest information

Forest information is not required for a successful simulation, but recommended, because the friction parameters strongly depend on forest information. Forest information can be provided as:

- ESRI ASCII grid (0: no forest, 1: forest)
- Polygon shapefile

If no such files are available, the user can draw a polygon shapefile in RAMMS and import it as forest information (see section 3.5.3 on page 47).

3.1.7 Calculation parameters

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

3.2 Preferences

You can set your RAMMS preferences and place the necessary DEM (Digital Elevation Model) files as well as the forest files, maps and georeferenced orthophotos you wish to use in the appropriate folders defined in the preferences see Figure 3-9 and Figure 3-10 below. This is optional, you do not have to use these folders.

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

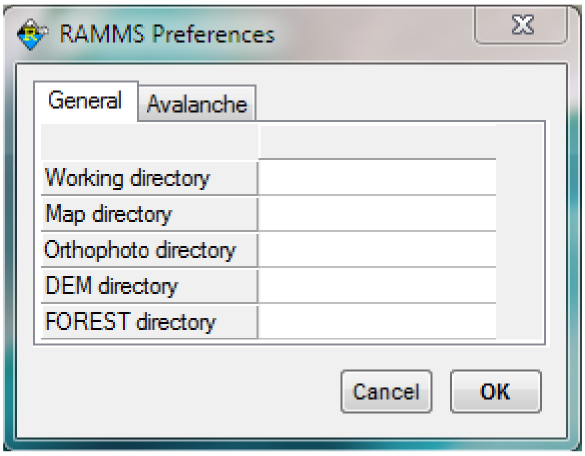


Figure 3-9: General tab of RAMMS preferences.

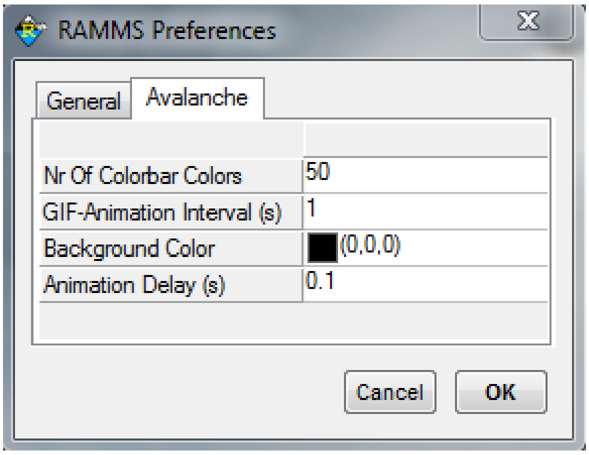


Figure 3-10: Avalanche tab of RAMMS preferences.

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 orthopho-tos (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, see section 3.1.1 on page 21)
FOREST directory	Set the folder where you place your forest-files (formats: ASCII grid or polygon shapefile).

Avalanche Tab

Setting	Purpose
Nr of colorbar colors	Set default nr of colorbar colors.
GIF-Animation Interval [s]	Set interval for GIF animation images.
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.

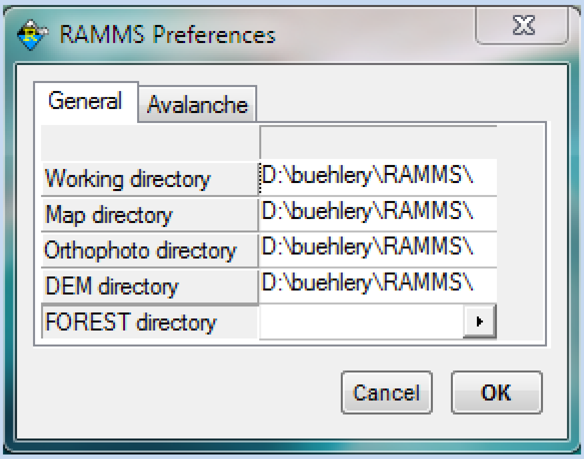


Figure 3-11: RAMMS preferences

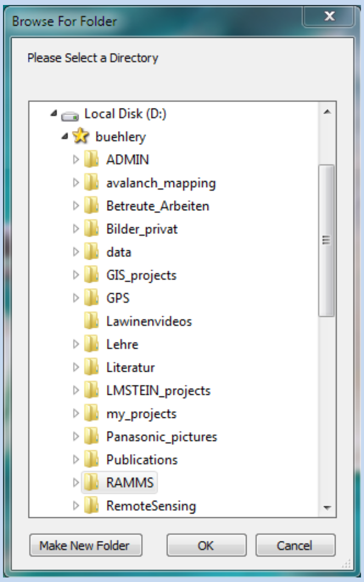


Figure 3-12: Browse for the correct folder.

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.

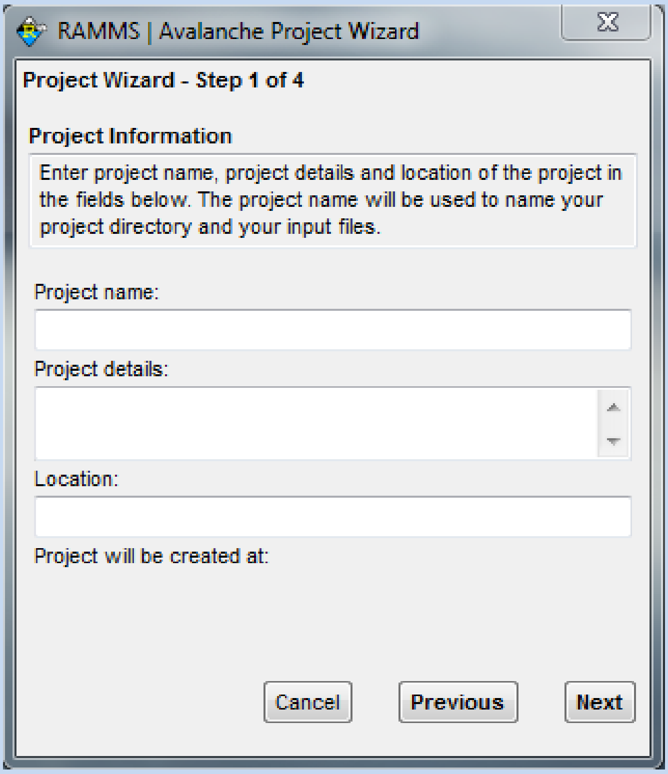


Figure 3-13: RAMMS Project Wizard Step 1 of 4

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)

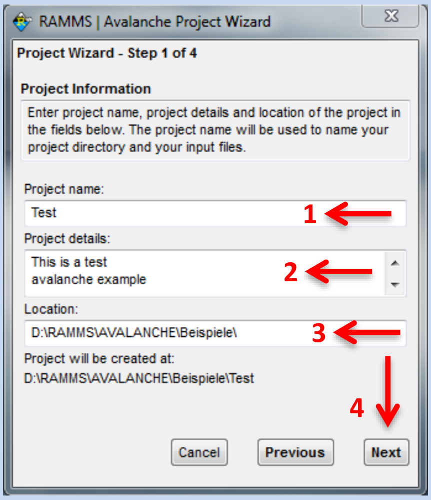


Figure 3-14: Step 1 of the RAMMS Project Wizard Project Information.

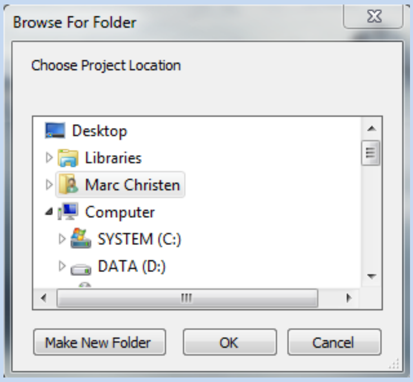


Figure 3-15: Window to browse for a new project location.

Step 2:

- Locate your DEM-file (ASCII or GEOTIFF). Click into the corresponding field to browse for the appropriate file (1).
- The grid resolution of your DEM-file is shown in (2). Change the resolution, if needed (bilinear interpolation).
- Click ***Next*** (3).

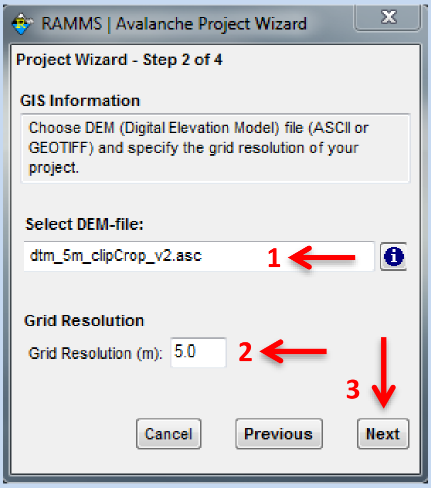


Figure 3-16: Step 2 of the RAMMS Project Wizard: GIS Information.

Continuation of exercise 3.3: How to create a new project

Step 3:

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

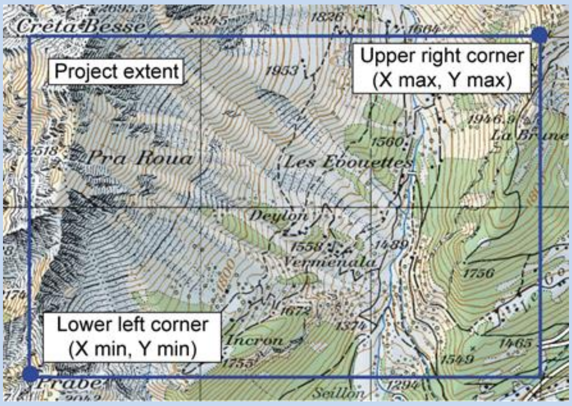


Figure 3-17: Project coordinates: lower left and upper right corner of project area.

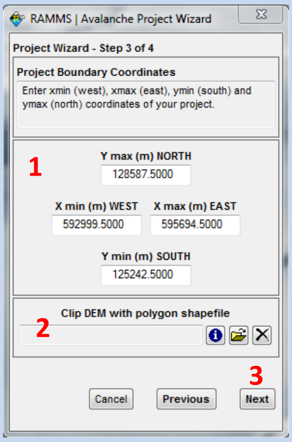


Figure 3-18: Step 3 of the RAMMS Project Wizard: Project Boundary Coordinates.

Step 4:

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

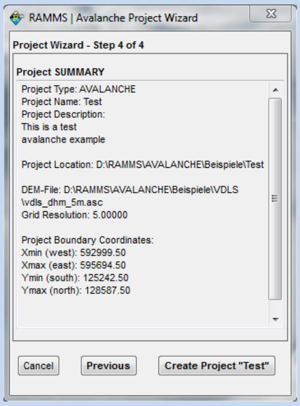


Figure 3-19: Step 4 of the RAMMS Project Wizard: Project Summary.

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.

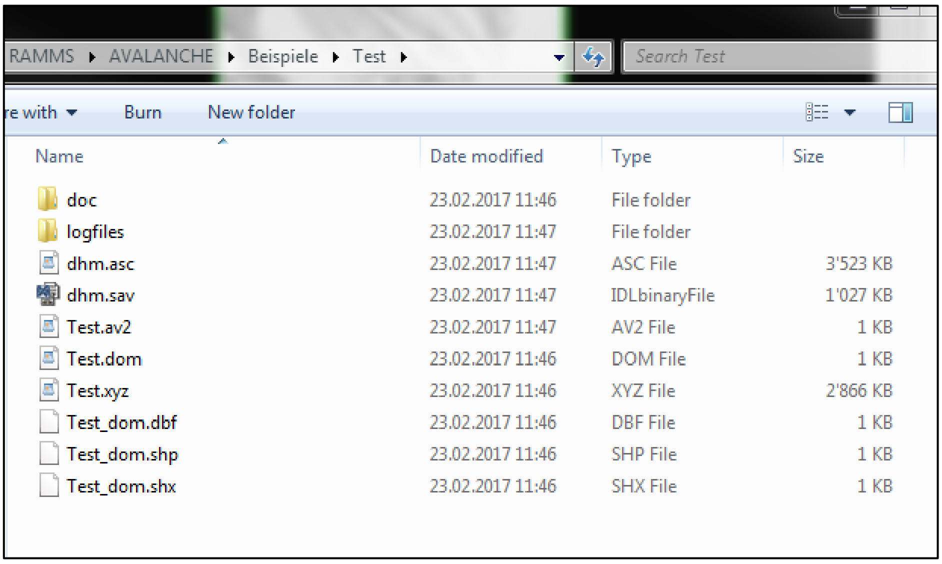


Figure 3-20: Created project files

Table 3.2: Listing of files and directories created with a new RAMMS::AVALANCHE project.

File / Folder	Purpose
doc (folder)	Folder containing input and output log files
logfiles (folder)	Project creation and calculation log files
dhm.tif	GEOTIFF grid with altitude values
dhm.sav	Compressed binary altitude information (used in RAMMS)
_.av2	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 sections and exercises below.

3.4.1 Open input- and output-files

The easiest way to open either input- or ouput-files is by using these toolbar buttons:

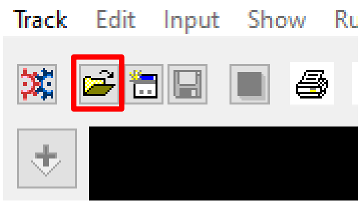


Figure 3-21: Toolbar button to open an input file. You can also use the menu ***Track → Open... → Input File***.

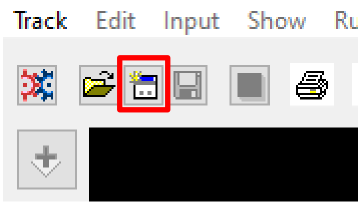


Figure 3-22: Toolbar button to open an output file. You can also use the menu ***Track → Open... → Avalanche Simulation***.

Recent-Menu

The menu ***Track → Recent...*** allows you to directly open your 10 last accessed files (input and out-put), without having to search for them.

3.4.2 Visualizing shapefiles, MuXi-files and domain-files

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

Exercise 3.4a : Visualizing shapefiles, MuXi-files and domain-files

a. *Files* tab in the right panel:

- Click on the *Files* tab in the right *AVALANCHE* panel.
- In the file tree below, you will see your available project files (polygon shapefiles, MuXi ASCII files and domain shapefiles.

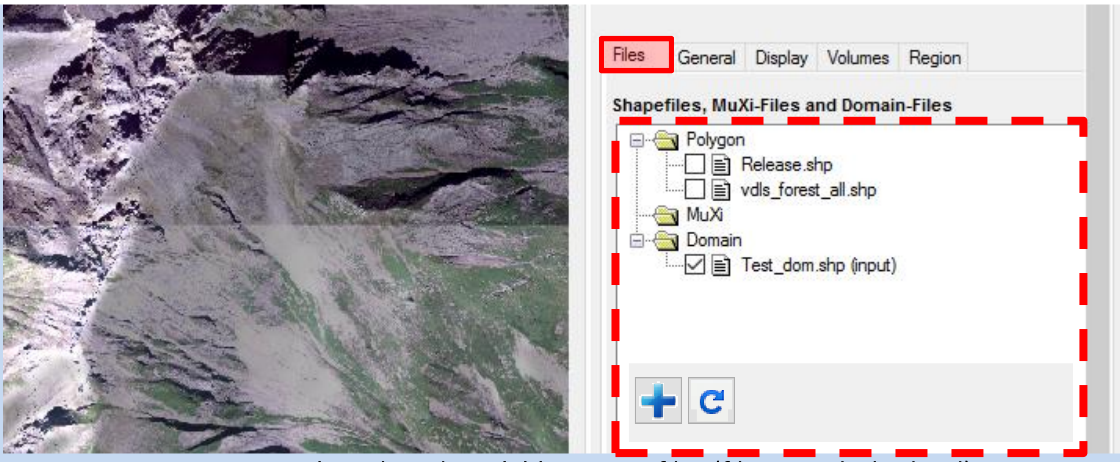


Figure 3-23: Files tab and available project files (file-tree, dashed red). With the **blue + button**, files from external directories can be added to the file-tree. Refresh the tree with the refresh-button.

- **Click the checkbox** next to a filename and the file will be shown in your visualization.

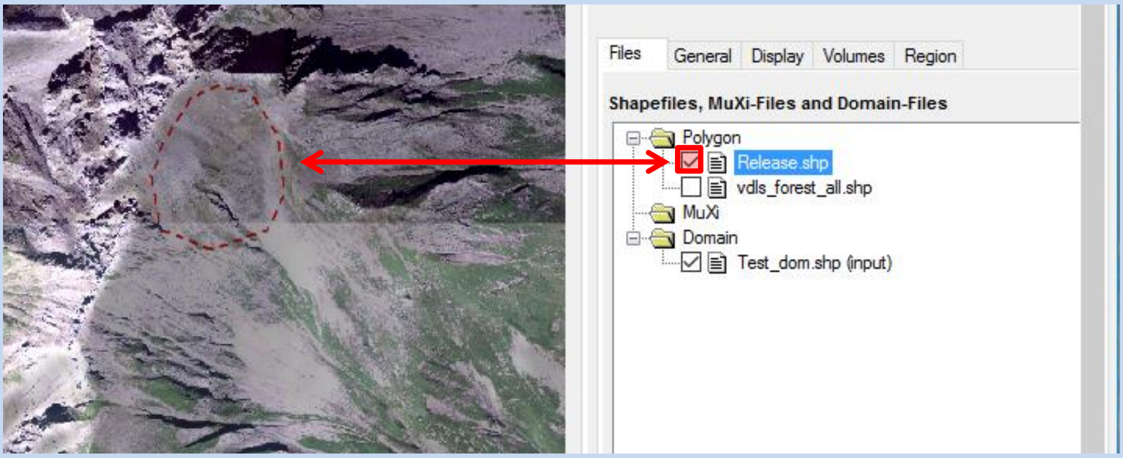


Figure 3-24: Selected file (Release.shp) on the right is shown in the visualization.

- 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. These properties are only saved within this RAMMS Session. Right-click on a filename and choose ***Shapefile properties***:

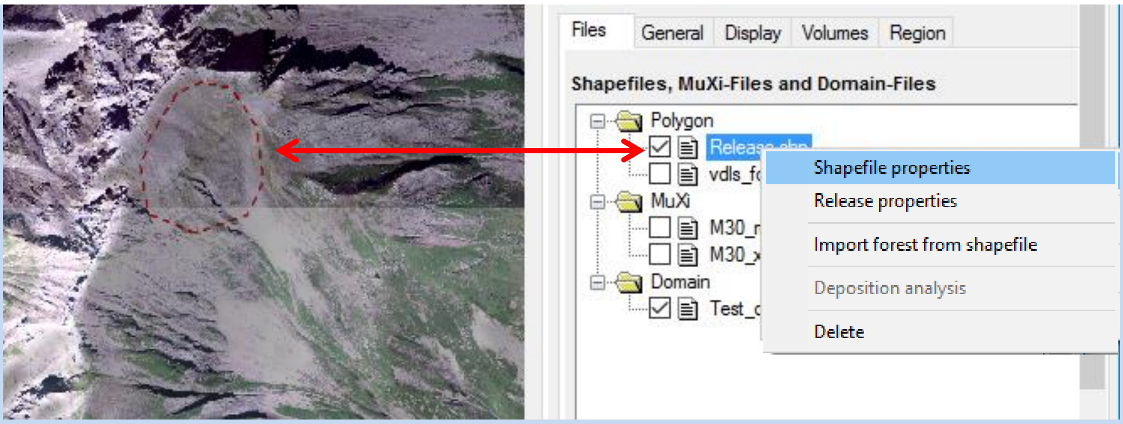


Figure 3-25: Right-click menus Shapefile properties, Release properties, Import forest from shapefile, Deposition analysis (only output) and Delete.

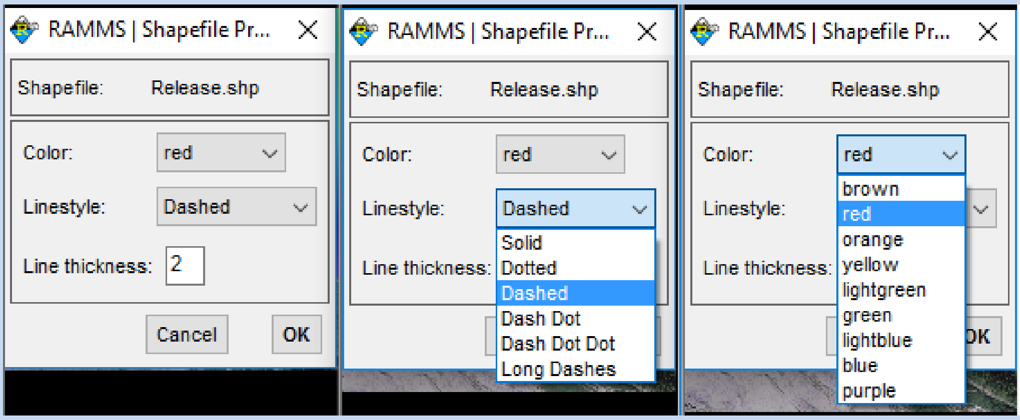


Figure 3-26: Use ***Shapefile properties*** to change line thickness, color or linestyle.

Release properties

- Please see section 3.5.1 on page 38 on how to specify release area properties.

Import forest from shapefile

- Please see section 3.5.3 on page 47 on how to import forest from a shapefile.

Deposition analysis

- This function is only available in output mode. Please see section 4.2.3 on page 65 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 shape-file 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)

3.4.3 Hillshade visualization

Use *Extras* → *Create Hillshade Image* to create a hillshade visualization. For this we follow the instructions from ArcGIS at

https://desktop.arcgis.com/en/arcmap/10.3/tools/spatial-analyst-toolbox/how-hillshade-works.htm

to calculate the hillshade representation of your DEM, see Figures below.

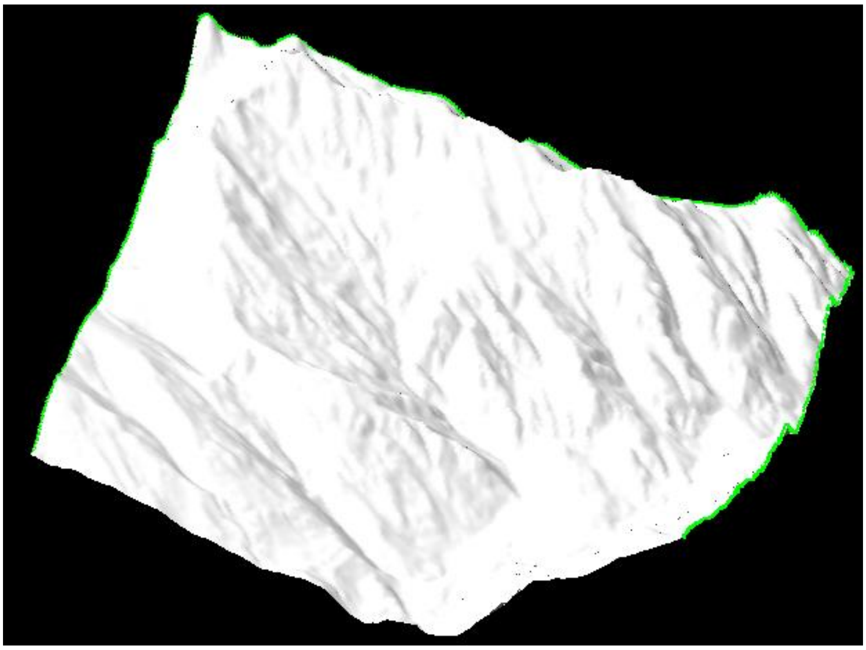


Figure 3-27 DEM surface visualization (with shadows) after creating a new project in RAMMS

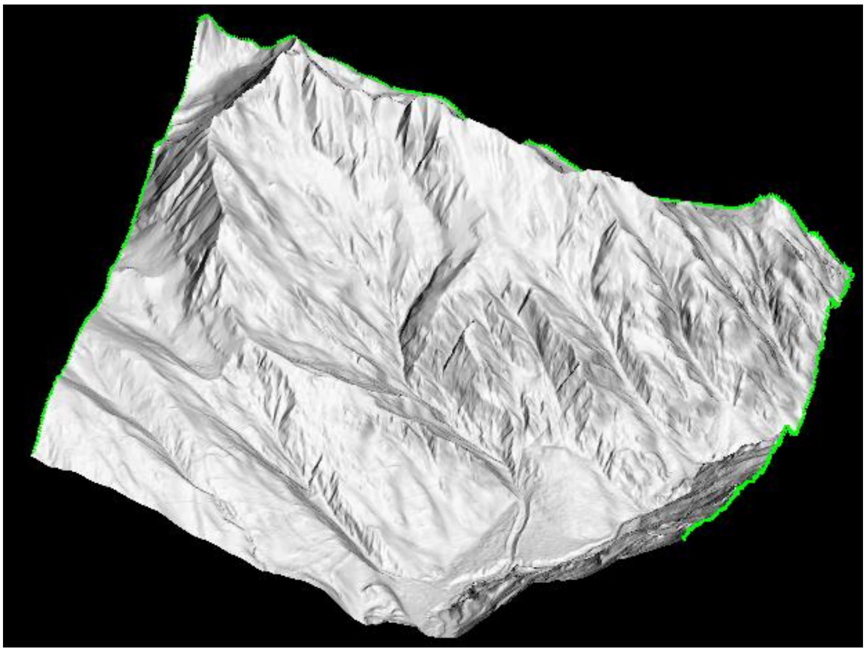


Figure 3-28 Visualization after creating and adding the hillshade image to RAMMS

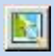
3.4.4 Changing maps and orthophotos (aerial images)

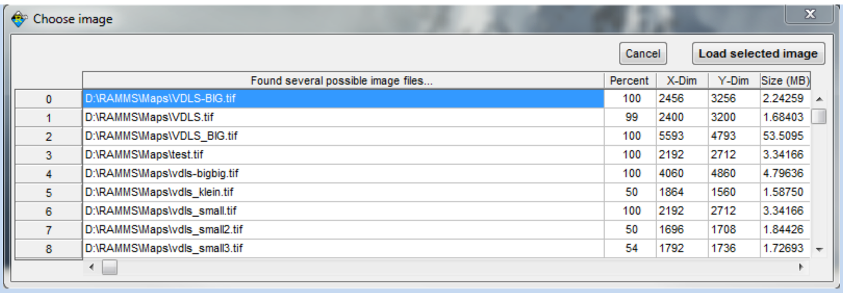
It is possible to change the map or orthophoto of a project anytime. Take into account, that the corresponding .tfw-file (world-file) has to be in the same folder as the actual map (.tif). If this is not the case, the map will not be found!

To check which map and orthophoto are currently loaded in the project, open the project input (or output) log (*Project →Input Log File*). Next to *map image* and *ortho image* you will find the location and name of the loaded map and image, 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:



	Found several possible image files...	Percent	X-Dim	Y-Dim	Size (MB)
0	D:\RAMMS\Maps\VDLS-BIG.tif	100	2456	3256	2.24259
1	D:\RAMMS\Maps\VDLS.tif	99	2400	3200	1.68403
2	D:\RAMMS\Maps\VDLS_BIG.tif	100	5593	4793	53.5095
3	D:\RAMMS\Maps\test.tif	100	2192	2712	3.34166
4	D:\RAMMS\Maps\vdls-bigbig.tif	100	4060	4860	4.79636
5	D:\RAMMS\Maps\vdls_klein.tif	50	1864	1560	1.58750
6	D:\RAMMS\Maps\vdls_small.tif	100	2192	2712	3.34166
7	D:\RAMMS\Maps\vdls_small2.tif	50	1696	1708	1.84426
8	D:\RAMMS\Maps\vdls_small3.tif	54	1792	1736	1.72693

Figure 3-29: Window to choose map image.

Information on the image cover ratio (Percent), image dimensions (X- 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 set 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 orthophotos:

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

3.4.5 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.

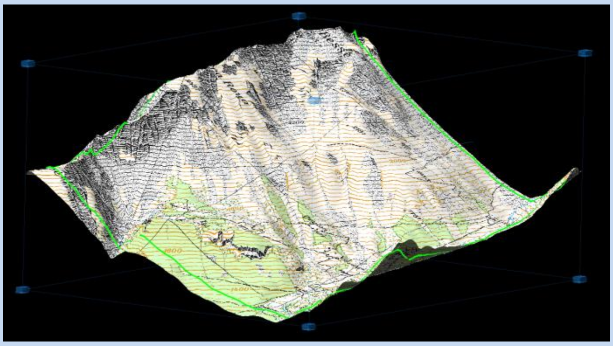


Figure 3-30: *Active* project with lines and corners for resizing.

- 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 100%.

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 axes until the cursor changes from to . Otherwise a freehand rotation in any direction is possible.

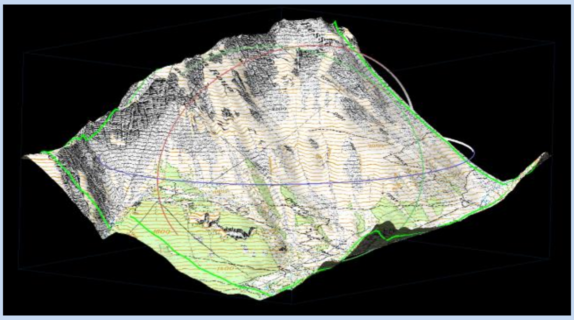


Figure 3-31: *Active* project with rotation axes.

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

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

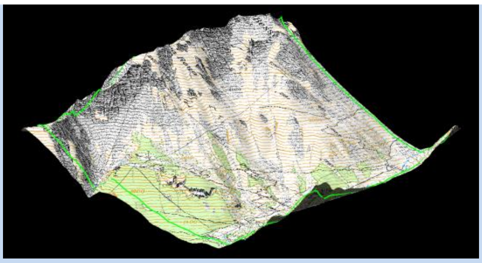


Figure 3-32: 3D view of example model.

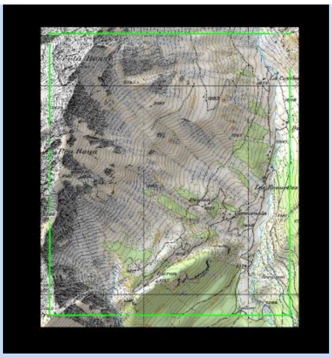


Figure 3-33: 2D view of example model.

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 polygon shapefile
- Release area information
- Draw new domain
- Measure distance and angle

OUTPUT:

- Draw new polygon shapefile
- Draw new line profile
- Measure distance and angle

3.4.6 Colorbar

As soon as a parameter is shown in the project, the colorbar appears in the panel 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 *AVALANCHE* (right of the map window).

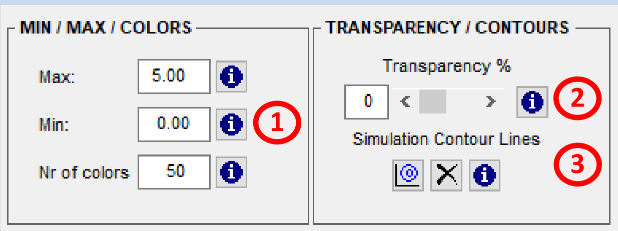


Figure 3-34 Colorbar and visualization properties

- 1) Simply type a new value into the respective field and hit the return key on the key-board. The display will then be refreshed.
- 2) To view the underlying topography or image, you can change the transparency by en-

tering a value or moving the slider.

- 3) Simulation contour lines: Contour lines of simulation results can be visualized. Default contour levels are:
  - Flow height (m): 1.0 2.0 3.0 5.0 10.0 (H_CONTOUR_LEVELS)
  - Velocity (m/s):1.0 5.0 10.0 20.0 30.0 40.0 50.0 (V_CONTOUR_LEVELS)
  - Pressure (kPa): 1.0 3.0 10.0 30.0 100.0 (P_CONTOUR_LEVELS)

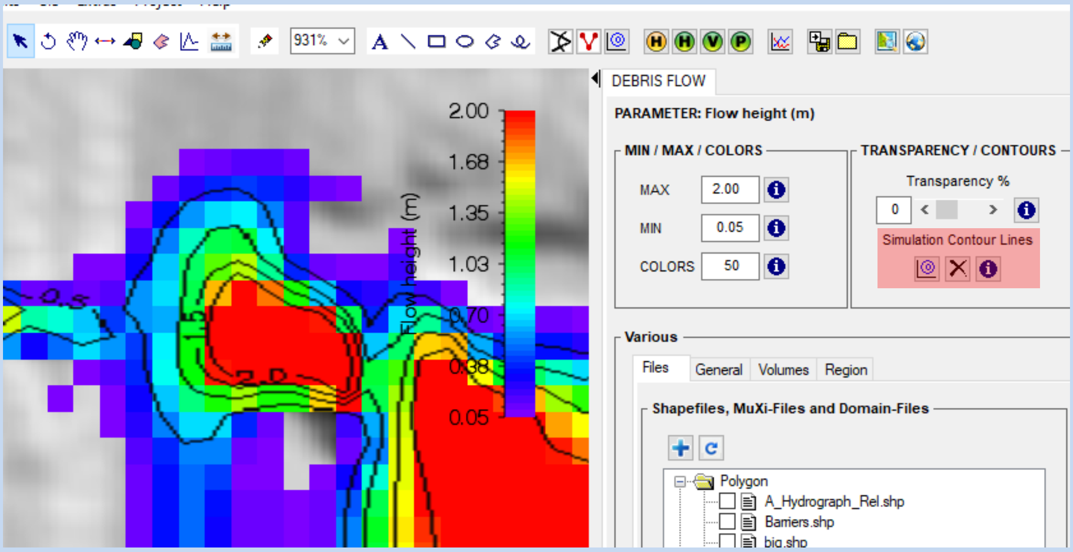


Figure 3-35: Simulation contour lines

You can change these contour levels by adding the following lines in the Add. Preferences (before the END tag):

Example:
H_CONTOUR_LEVELS 0.5 1.0 1.5 2.0
V_CONTOUR_LEVELS 5.0 10.0 50.0
P_CONTOUR_LEVELS 3.0 10.0 30.0

Changing colorbar color and position:

- 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 → Avalanche Tab → Background Color***.

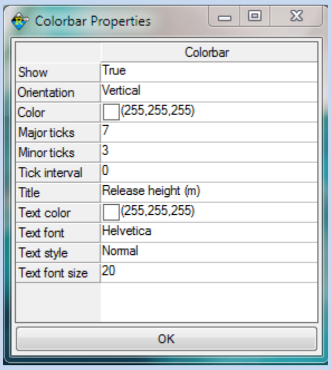


Figure 3-36: The Colorbar Properties window.

3.4.7 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::AVALANCHE Project Wizard (see Figure 3-13 on page 23). The created input file has the suffix *.av2.

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 avalanche input file (*.av2).
- 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 load 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/avalanche simulation

- Go to ***Track*** → ***Open...*** → ***Avalanche Simulation***, click or use ***Ctrl+A***.
- A window opens to browse for an avalanche 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.

3.4.8 About RAMMS

Some information about the RAMMS installation on your computer is found here: ***Help*** → ***About RAMMS***. If you click on the Info-Button left of the OK-Button, RAMMS will display the RAMMS-License Agreement.

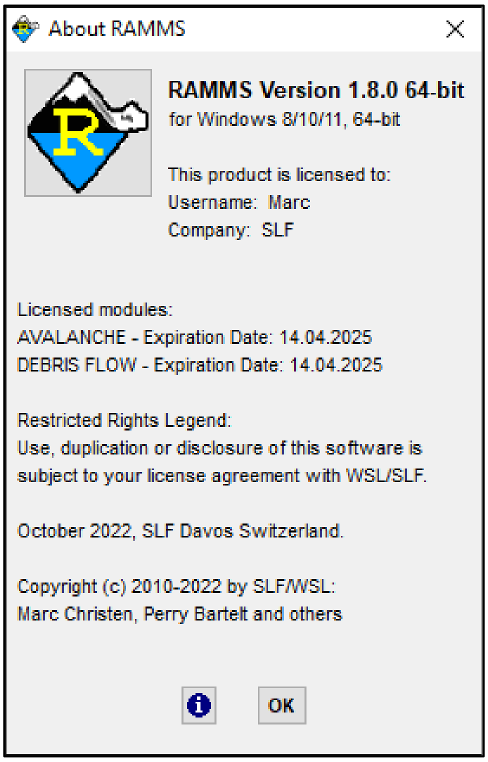


Figure 3-37: About RAMMS

3.5 Running a simulation

To run a calculation or a specific scenario within a newly created project (creating a project see section 3.3 on page 23) it is necessary to define

- release area(s),
- a calculation domain
- and friction parameters μ and ξ.

The definitions of release area(s) and release heights as well as the set of friction parameters μ and ξ have a strong impact on the results of RAMMS simulations, the definition of a smaller calculation domain is especially useful to keep the number of calculation grid cells as small as possible. The exercises below show you how to create a release area, a calculation domain and a MuXi-file. Details on the friction model used in RAMMS::AVALANCHE are given in section 0. on page 16.

3.5.1 Release area(s)

There are different possibilities to include release area(s) into the project. Since Version 1.7.0 it is possible to specify more than one release shapefile. The following table gives an overview of the possibilities RAMMS offers. For further explanations see the exercises below.

Create a new release area (polygon shapefile)	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	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.

The definitions of release areas and release heights have a very strong impact on the results of RAMMS simulations. Therefore, we recommend to use reference information such as photography, GPS measurements or field maps to draw release areas. This should be done by people with experience concerning the topographic and meteorological situation of the investigation area. Release areas can only be drawn in 2D mode.

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.

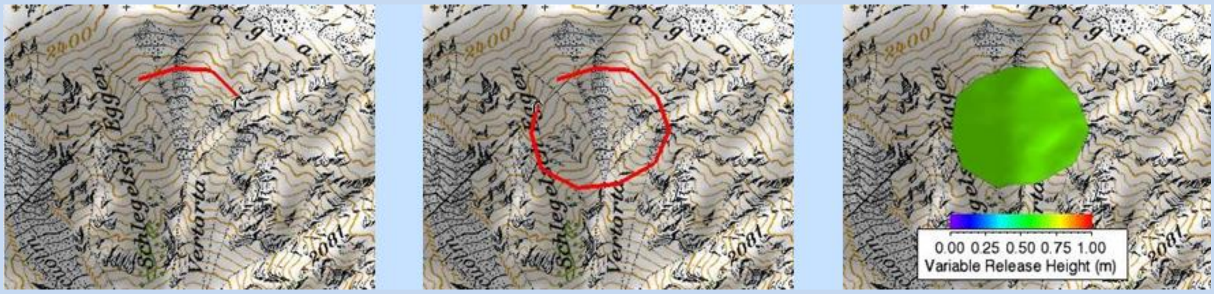


Figure 3-38: Drawing a new release area.

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.

The polygon area will now be created and opened directly, as well as the colorbar.

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

Visualize shapefiles by clicking the checkbox next to the filenames in the file-tree (**1**, right panel), use the 'Add data' button (**2**) or drag & drop shapefile(s) from the Windows file explorer onto the RAMMS topography.

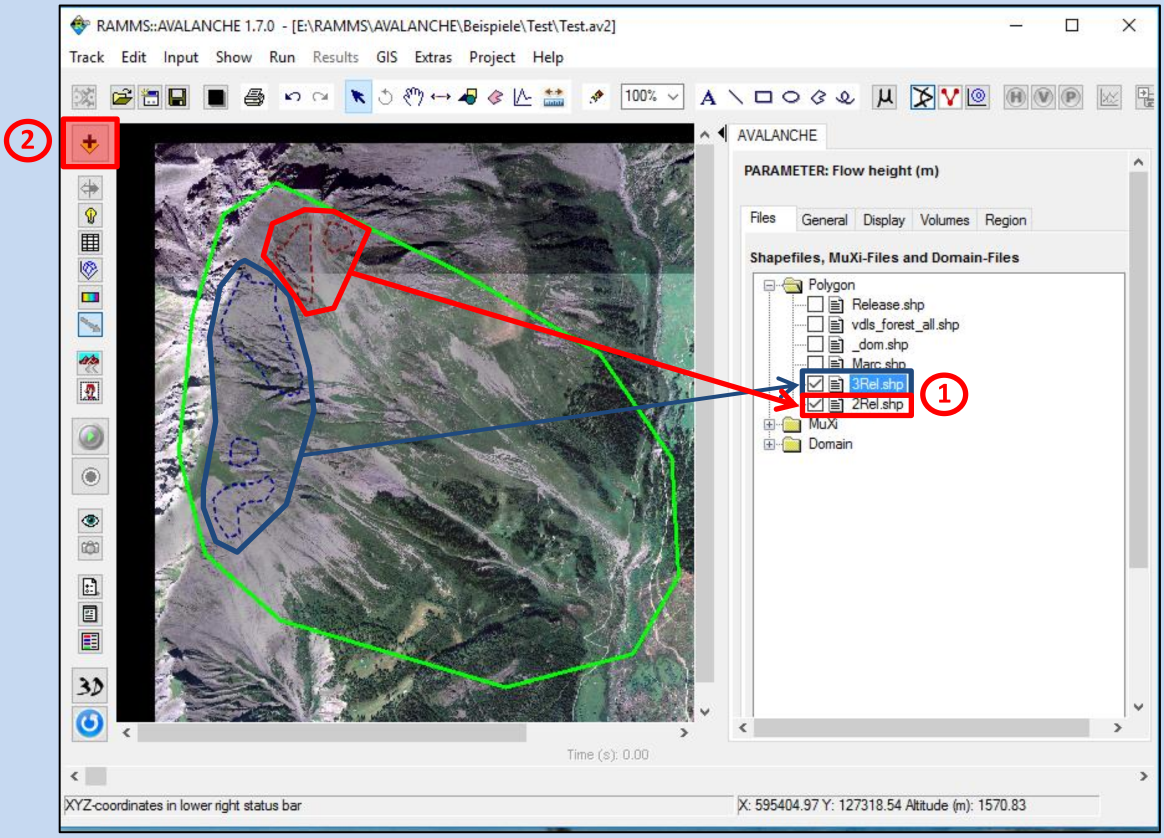


Figure 3-39: How to visualize polygon shapefiles

In Figure 3-39 we selected two polygon shapefiles: *3Rel.shp*, consisting of three polygons, and *2Rel.shp* with two polygons. For every polygon, we can specify a release depth and/or release delay. Not every polygon must contain a release depth, only the ones we want to release, see exercises below.

Once polygon area(s) are created or loaded, you have to specify the release height(s). 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 (Figure 3-40) yields information about release area, mean slope angle, mean altitude and estimated release volume. And, most importantly, the release height can be entered, see exercise below. Do this for every release area you wish to release. You can specify one release area, or multiple release area(s), see exercise below.

Exercise 3.5c : Specify release depth(s) and view release information

- Switch to 2D mode by clicking .
- Activate the project by clicking on the map once.
- Click on the **View/Edit release area** button **(1)**, choose **Input → Release area... → Details/Edit release area** or right-click the filename and choose **Release properties.**
- Then click into the release area you want to get information on **(2)**. A red polygon is drawn around the selected release area. The following window appears:

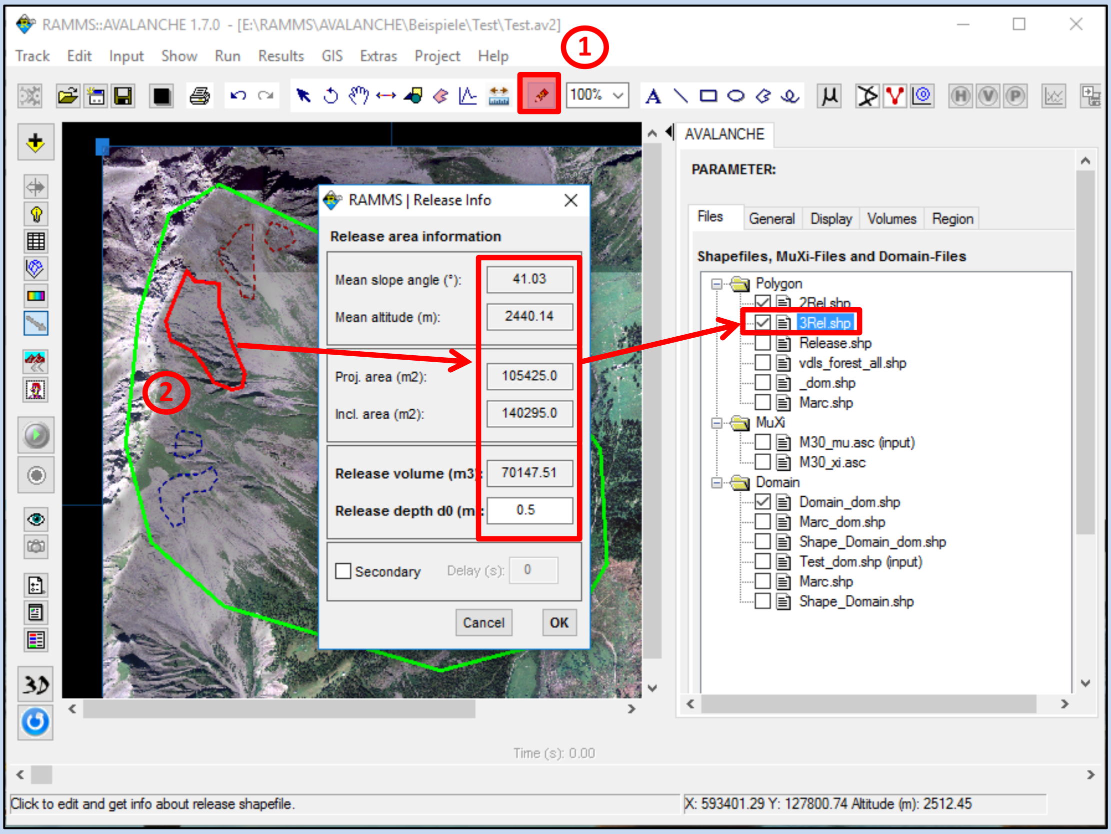


Figure 3-40: **View/Edit release area**

- Enter a release depth in the field *Release depth d0 (m)*. The corresponding release volume is updated automatically.
- Additionally, the following release area parameter are shown: *mean slope angle, mean altitude, projected area and inclined (real) area*.
- A release *delay in (s)* can be specified in the last line, see next Figure.

Remark: The estimated release volume is very accurate for the grid resolution of your input project. If you calculate a different simulation resolution, the estimation can differ from the calculated release volume.

In Figure 3-40 we selected the northern-most polygon of the shapefile *3Rel.shp* and assigned a release depth of 0.5m.

Let’s assign a release delay (secondary avalanche release) for one of the polygons in *3Rel.shp*.

Therefore, we again click on **(1)** and choose the middle polygon of 3Rel.shp **(2)**, see Figure 3-41 below. We then specify a release depth of 0.3m and a delay of 10s **(3)**.

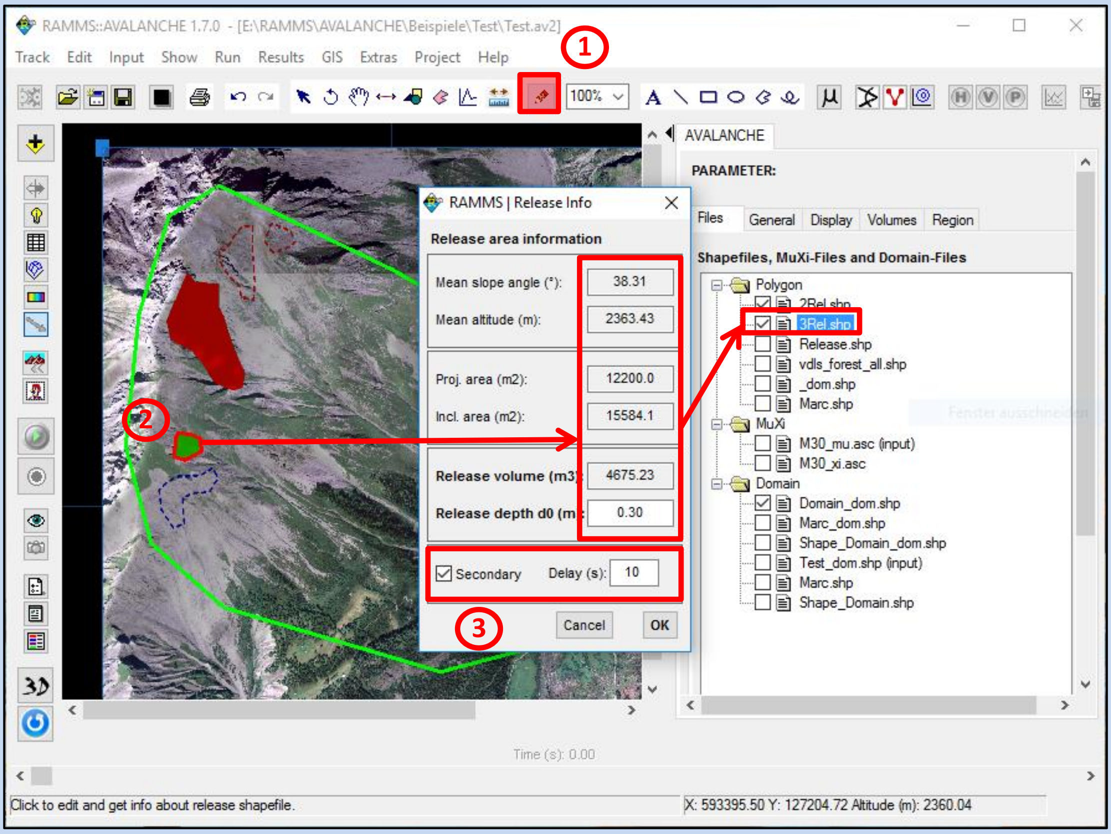


Figure 3-41: About

We do not specify a release depth for the last polygon of 3Rel.shp.

In a next step we can specify a release height for one of the two polygons of shapefile *2Rel.shp*, see below.

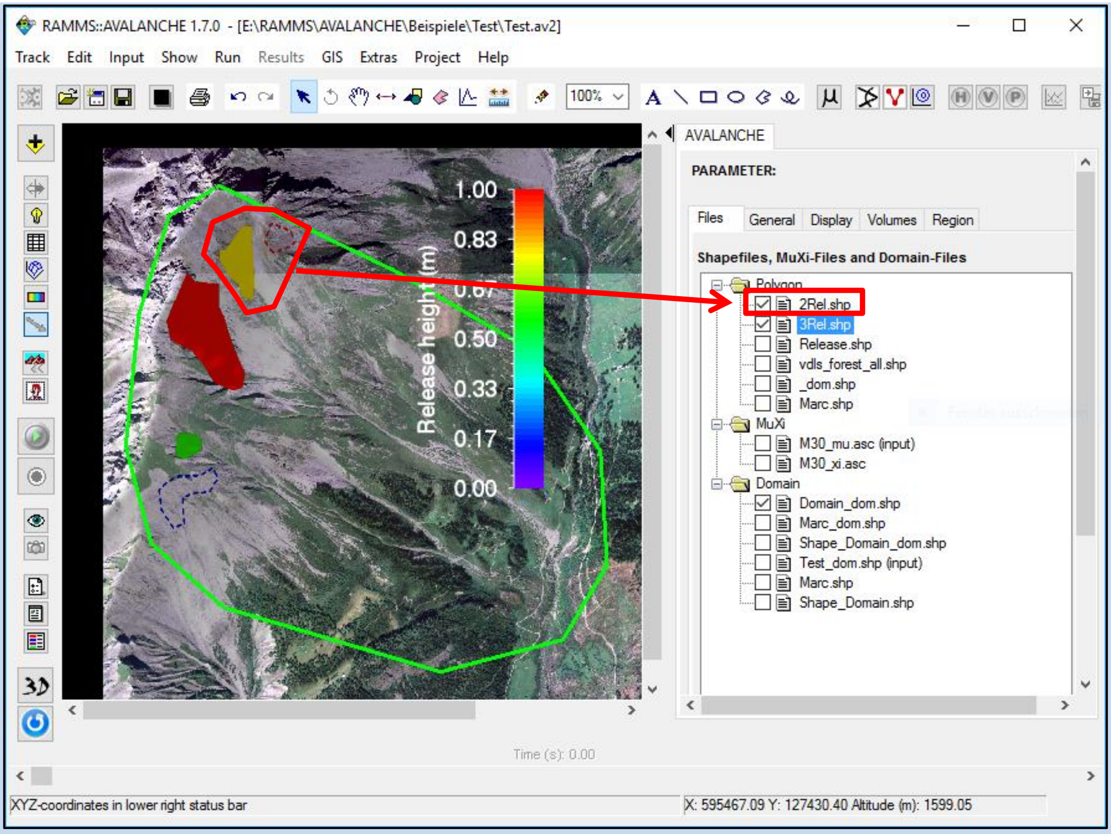


Figure 3-42: About

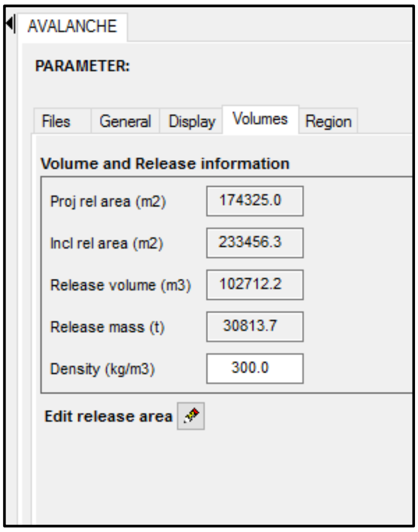


Figure 3-43: Release area and volume information

Additional release information is found in the *Avalanche panel*, tab *Volumes*, see Figure 3-43 above.

3.5.2 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 release area (see section “Release area(s)” on page 38). We strongly recommend using smaller calculation domains especially if you calculate with small cell sizes (e.g. < 5m).

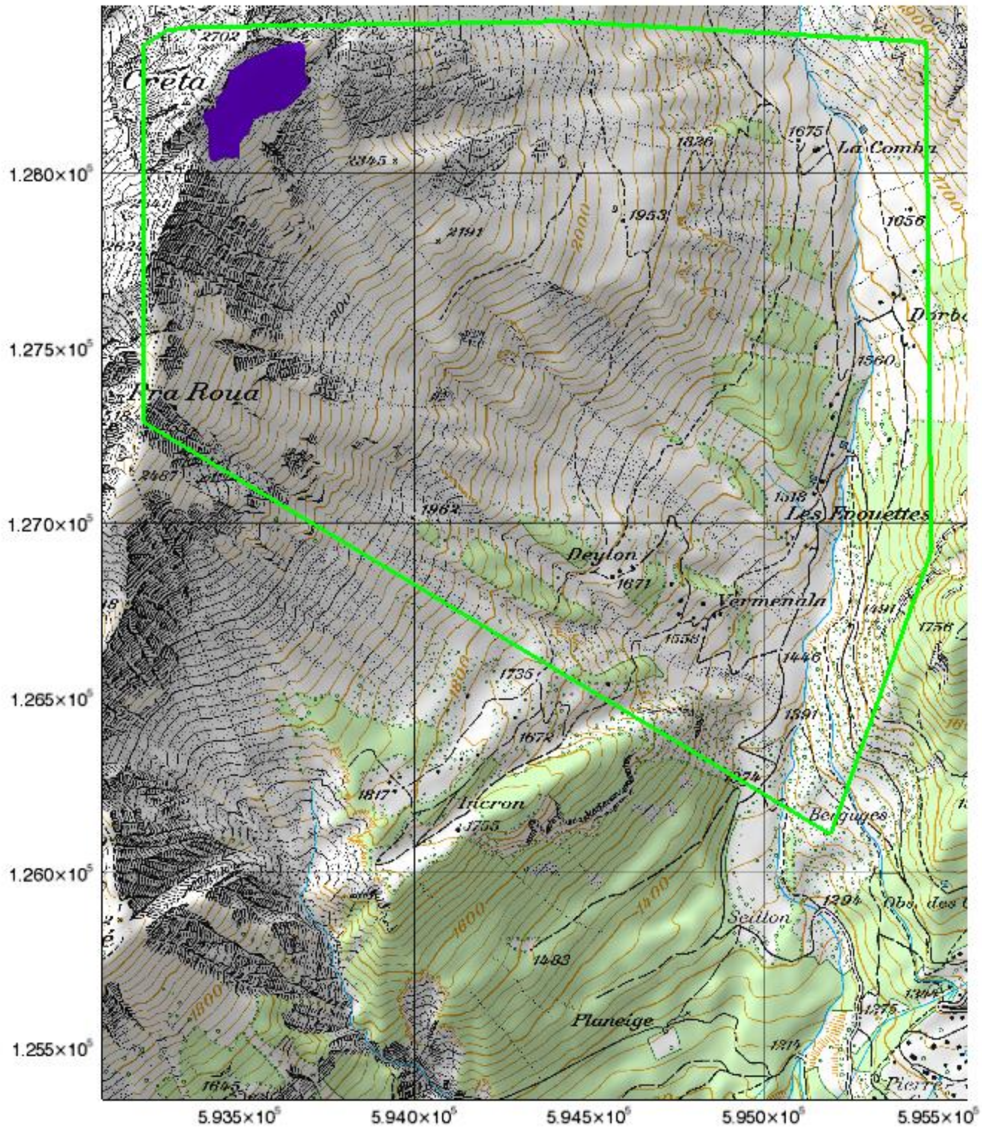


Figure 3-44: Calculation domain in green encloses the area of interest and reduces calculation time in comparison with the default rectangular domain which is automatically generated.

Exercise 3.5d : Finding an optimized calculation domain

- Open your input file.

- Draw a rough calculation domain as explained above, see Figure 3-45 below.

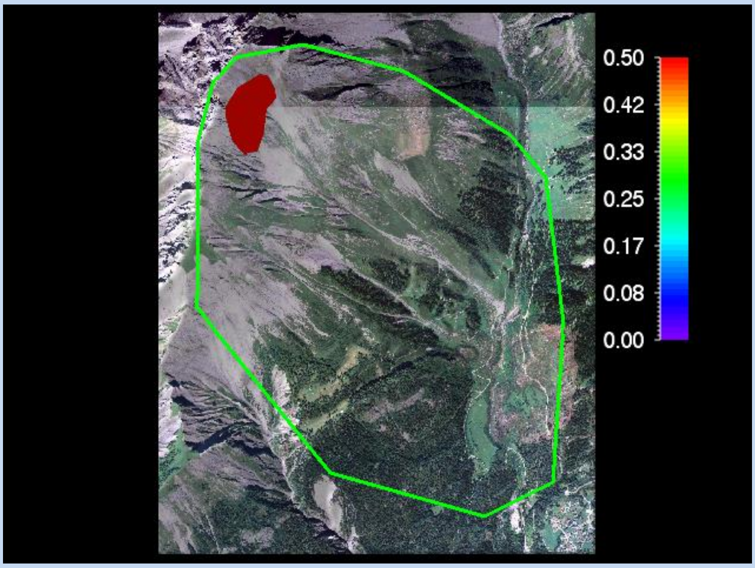


Figure 3-45: Input file with big calculation domain

- Do a rough calculation with a simulation resolution of 10m. Use constant μ and ξ values of e.g. μ=0.2 and ξ=2000.
- Wait for the simulation to finish. The simulation result will be displayed.
- Click the *Max Flow Height* button.

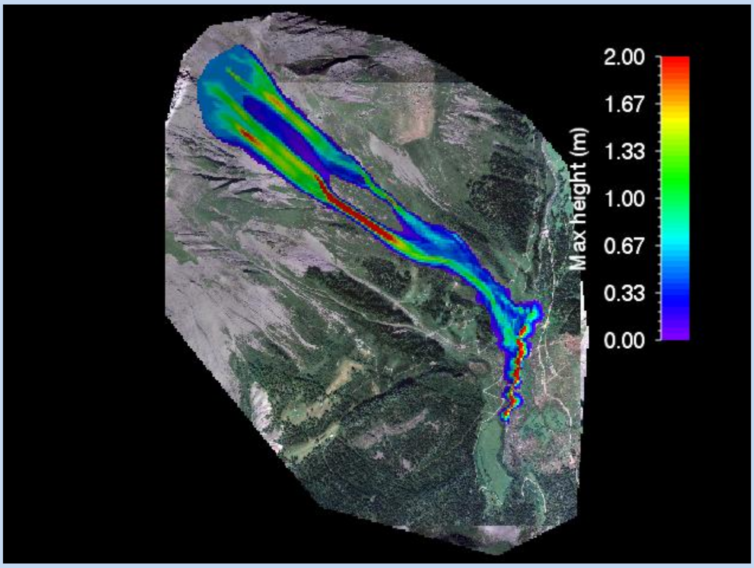


Figure 3-46: Max Flow Height of a 10m simulation with constant mu and xi

- Click *GIS* → *Export…* → *Create Envelope Shapefile*
- A question pops up: *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 Shapefile* of your output result. Beware: the MIN-value of your colorbar will be used as the lower limit for the *Envelope Shapefile*!
- 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.

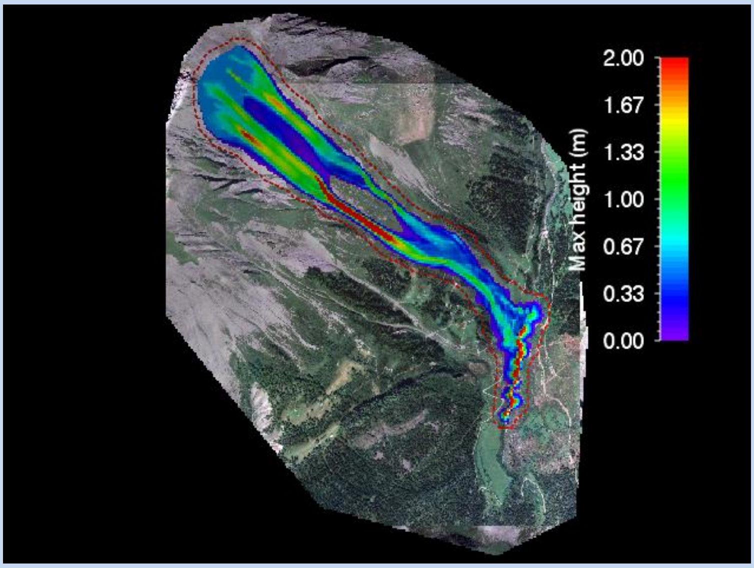


Figure 3-47: Envelope shapefile of Max Flow Height extent

- 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 3-48 below.

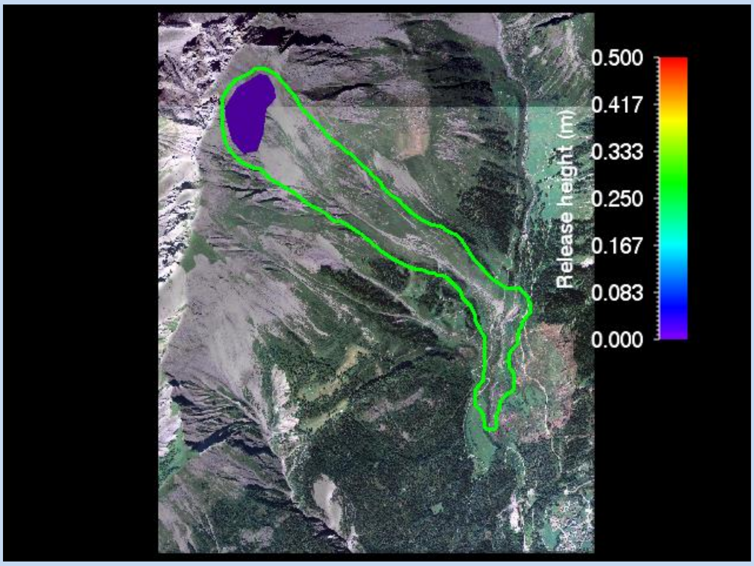


Figure 3-48: Input file with optimized calculation domain (envelope shapefile)

- Now redo your simulation with a simulation resolution of 5m and a MuXi-file.

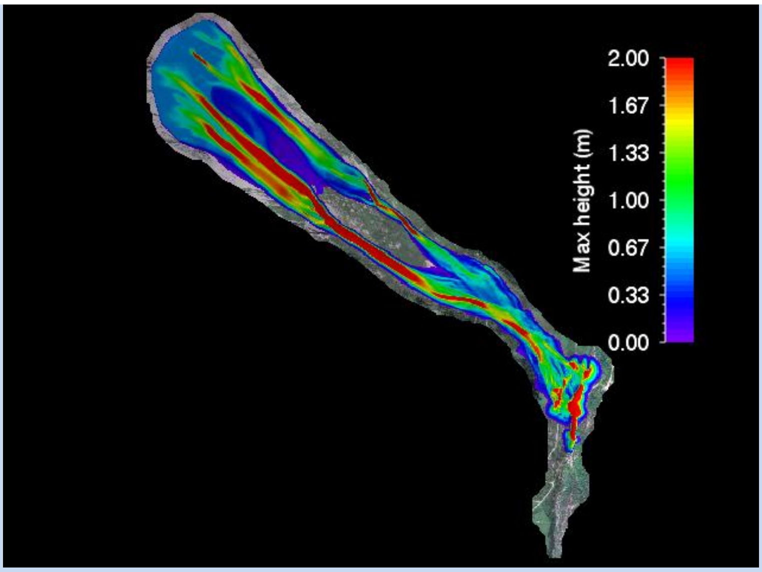


Figure 3-49: Max Flow Height result of a 5m simulation with variable MuXi-file

- In this example, the much smaller new calculation domain saves **70% !!!!!** of the computational time.

3.5.3 Friction parameters µ and ξ

Forest area

It is sometimes necessary to take forest areas into account, when running a simulation with variable friction parameters (µ and ξ). Beware: If your scenario is an extreme scenario, then it might not be appropriate to use forest information, as a forest would be completely destroyed by an extreme avalanche.

There are different ways to consider forest information:

(1) Import a digital forest file (ASCII grid or forest shapefile): Use ***Input → Forest… → Import Forest from SHAPEFILE*** or ***Input → Forest… → Import Forest from ASCII Grid.***
(2) Draw a forest file manually: See exercise below.

Exercise 3.5d: How to create a FOREST file

- Switch to 2D mode by clicking .
- Activate the project by clicking on the map once.
- Click or choose ***Input → Polygon Shapefile… → Draw New Polygon Shapefile***.
- Trace the forest outline by creating as many FOREST area polygons as necessary (proceed as in section "Release area(s)" on page 38) and name your new forest shapefile accordingly. The shapefile will be shown as a red dashed line in the GUI.
- Then right-click the shapefile in the Files-tree and choose *Import forest from shapefile*, see below.

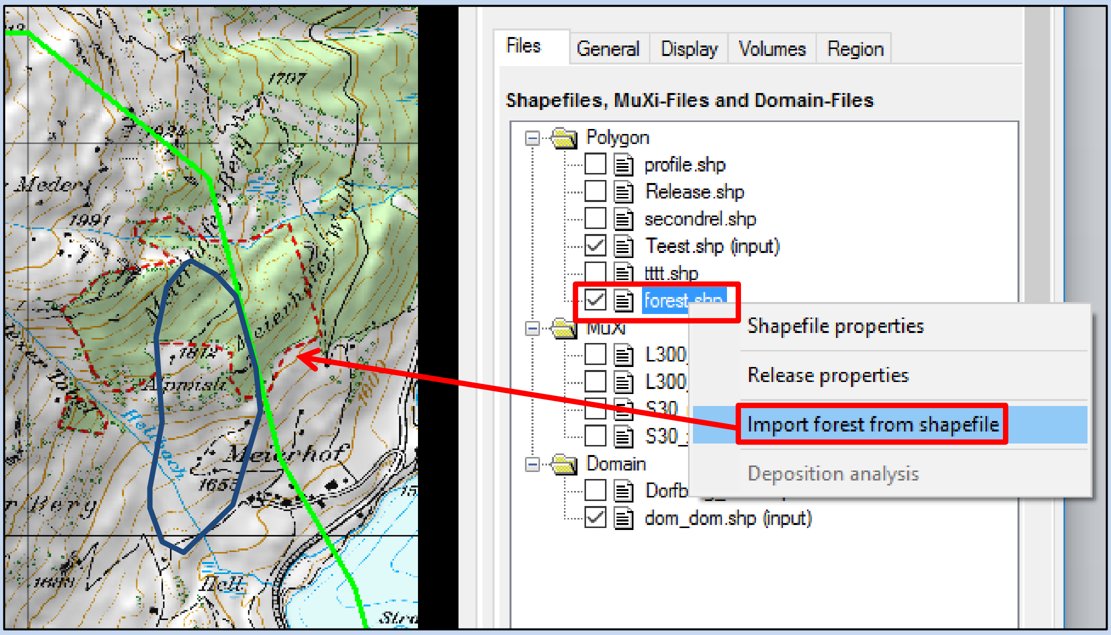


Figure 3-50: Import forest from shapefile

- You are asked, if you want to import the created FOREST file into your project. Click **yes**, if you want to use the newly created FOREST (ignore the next point in this case). Otherwise click **no** and import the FOREST file later, as explained in the next point.
- Import the new FOREST shapefile: Choose ***Input → Forest… → Import Forest Area from SHAPEFILE***, then select your FOREST shapefile.
- This new FOREST information is not automatically taken over in existing MuXi-files. Therefore, recreate existing MuXi-files if needed. If you create a new MuXi-file with ***Input → Friction Values… → Create new MuXi File (Automatic Procedure)***, the forest will now be considered.

MuXi-file

In RAMMS::AVALANCHE you can automatically generate a μ and ξ file based on topographic data analysis, forest information and global parameters. The following exercise shows how to create and load MuXi-files for a RAMMS simulation with variable friction parameters.

Exercise 3.5e: How to create a new MuXi-file

- Choose ***Input*** **→** ***Friction Values…*** **→** ***Create new MuXi File (Automatic Procedure)*** or click the button .
- A window pops up where you have to define an appropriate return period and check your avalanche volume. You can also define these global parameters under (***Input*** **→** ***Global Parameters***, see more information on page 20).
- In the following window (Automatic MuXi Procedure) you can enter a file name (optional). If you leave it empty, then RAMMS will use a default filename, e.g. L30 (for Large avalanche, 30-year return period). Unless you know better, leave the values as they are.

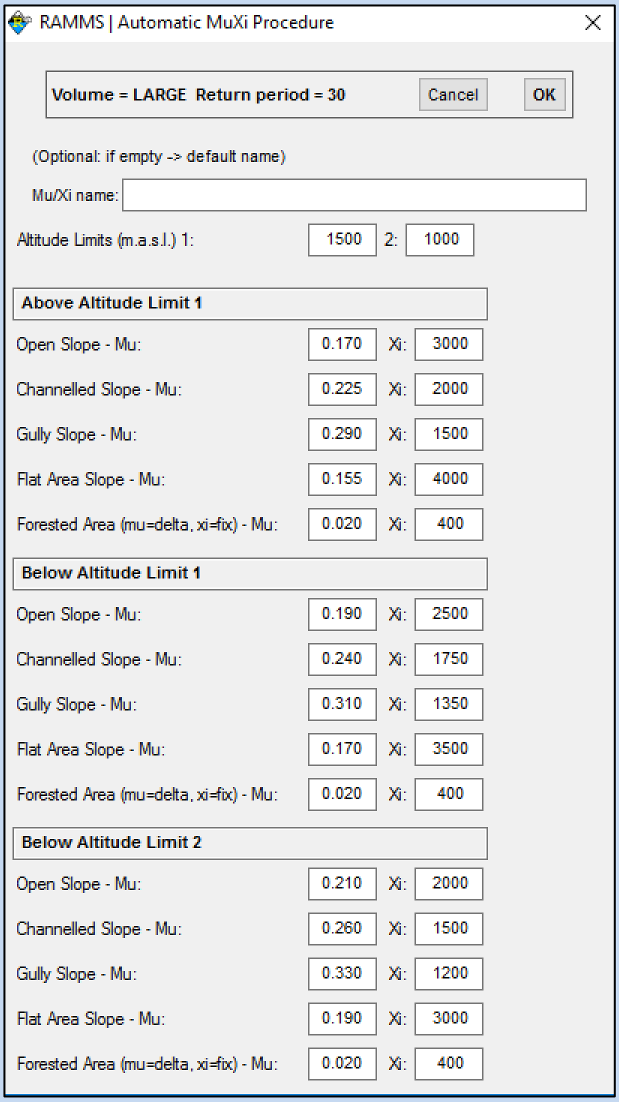


Figure 3-51: RAMMS Automatic MuXi Procedure.

- Click **OK**.
- If this is the first MuXi-file for this project, or if you changed or removed a forest cover or if you changed the altitude limits when entering the file name, RAMMS will start a terrain classification. Otherwise, RAMMS will skip the terrain classification (the classification is saved in the file muxi_class.asc in the logfiles folder).
- The MuXi-file will be visualized after its creation. The μ- and ξ-values are saved in two ASCII-files (L30_mu.asc and L30_xi.asc respectively). Only the region within the calculation domain will be visualized.
- You can switch between the release area (if already loaded), and the μ and ξ values in the choose Visualization area in the avalanche panel.

Exercise 3.5f: How to load an existing MuXi-file

- Choose ***Input*** → ***Friction Values…*** → ***Load existing MuXi File***
- A window opens to browse for an existing MuXi-file.
- Click **open** and the file will be loaded.

3.5.4 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 a calculation domain (section 3.5.2). A MuXi-file is necessary as well. Below you find two examples, one for running a constant calculation (constant release height and constant friction parameters μ and ξ) and one for using variable friction parameters..

Exercise 3.5g: How to run an avalanche calculation

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

General Tab:

- **SCENARIO Name**
  - (1) Scenario output name: Choose a meaningful output filename, add parameter information to the filename to recognize the output file.
  - (2) Save Max Values Only: If you want to save disk space and if the max values are all you are interested in, then use this feature. After a simulation finished, all ASCII files (deposition, max flowheight, max velocity and max pressure) are exported automatically. The simulation results cannot be visualized in RAMMS.

- **Additional Information**
  - (3) Project name.
  - (4) Project info: Add valuable project information to this field.

(5) Calculation domain.

(6) Digital elevation info: Digital elevation model (DEM) file.

- **Stop Criteria**

(7) Momentum stopping criterion, see section 4.2.6 on page 70.

(8) Center of mass stopping criterion, see 4.2.6 on page 70.

- **Remarks**

(9) “Escape” and “Ctrl+R” can be used to cancel resp. start a simulation.

(10) 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.

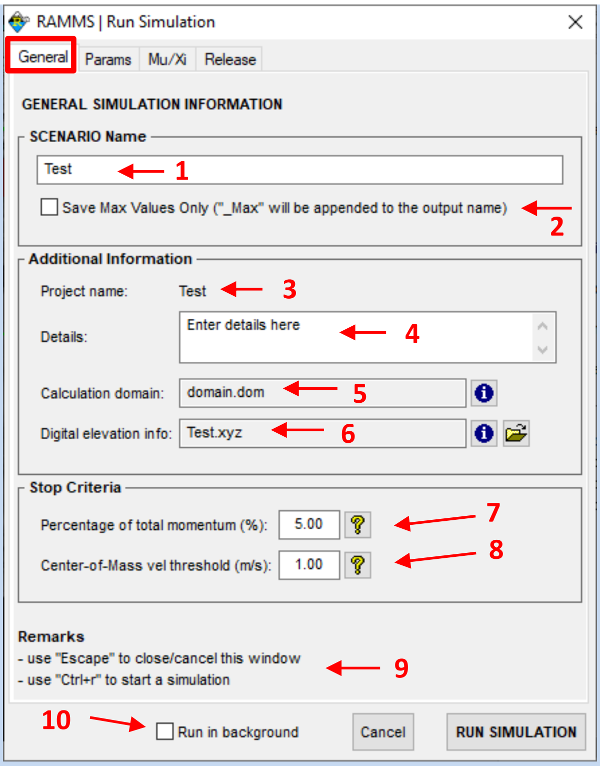


Figure 3-52 General Information

Continuation of exercise 3.5g: How to run a constant avalanche calculation

Params Tab

- **Simulation Parameters**
  (1) Sim resolution: Suggested simulation resolution for large avalanches: **5m**. For small avalanches, you can try better resolutions, between **2–5m**. If your DEM resolution is different than your simulation resolution, then RAMMS performs a bilinear interpolation. High resolution grids will extend your calculation time.

  (2) End time: Simulation end time. Default time = 300s.

  (3) Dump-step: The dump-step interval defines the resolution of the animation of your simulation but has no effect on the simulation results. Suggested: **2-5s**.

  (4) Density: Keep the default value for density if no further information on the avalanche density is available (300 kg/m^3).

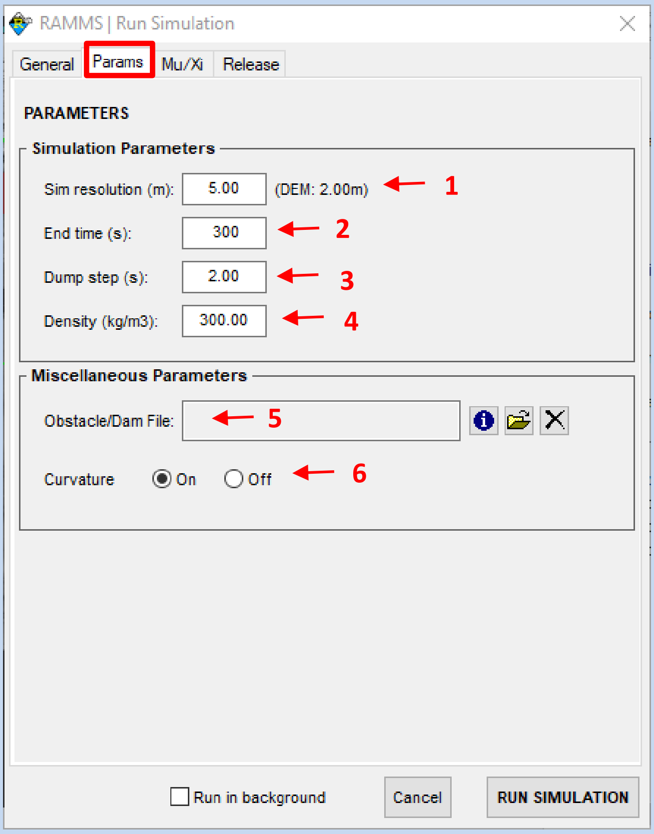


Figure 3-53: Parameter Tab

- **Miscellaneous Parameters**
  (5) Obstacle/Dam File: Draw polygons of areas, where no avalanche-flow should pass (houses, deflecting dams, obstacles). The flow is then deflected.

  (6) Curvature: Switch *Curvature* On or Off. 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. Typically, this increases the friction, causing the flow to slow down in tortuous and twisted flow paths.
  See https://ramms.slf.ch/en/modules/avalanche/theory/friction-parameters.html for more information.

Continuation of exercise 3.5g: How to run a constant avalanche calculation

Mu/Xi FRICTION PARAMETERS

(1) **Constant:** For a calculation with constant MuXi-values, click **Constant**. Enter μ and ξ values below. Choose *Help → RAMMS Manuals… → Friction Parameter Table (PDF)* or see friction value table on page 98 for an idea of μ and ξ.

(2) **Variable:** For a calculation with variable MuXi-values (recommended), click **Variable**. You should have created a MuXi-file before starting a variable MuXi calculation.

(3) **Cohesion**: For dry avalanches, values between 0-100Pa are ok (suggested: 50Pa). For wet avalanches, slightly higher values (up to 200Pa) can be used. See https://ramms.slf.ch/en/modules/avalanche/theory/friction-parameters.html for more information.

(4) **Define additional MuXi areas:** You can specify up to two additional polygon areas where you can change the MuXi-values. Make sure you have good reasons to change the μ and ξ values there.

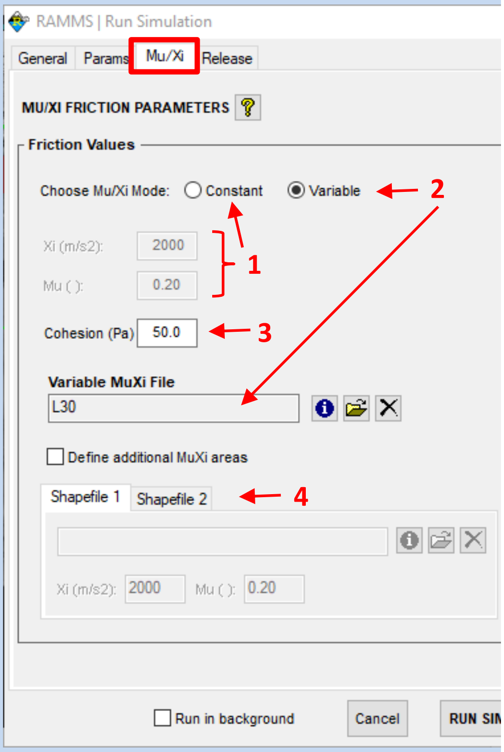


Figure 3-54: Friction values Mu and Xi.

RELEASE PARAMETERS

(1) **Filename:** List of all the release shapefiles in use.

(2) **Volume (m³):** The estimated release volume per shapefile is indicated in the second row.

(3) **Depth (m):** Release depth per shapefile.

(4) **Delay (s):** Release delay per shapefile. In this example, the shapefile “secondrel.shp” has a start delay of 10s.

(5) **Total Volume (m³):** Sum of all the release volumes from above.

(6) **Run Simulation Button**

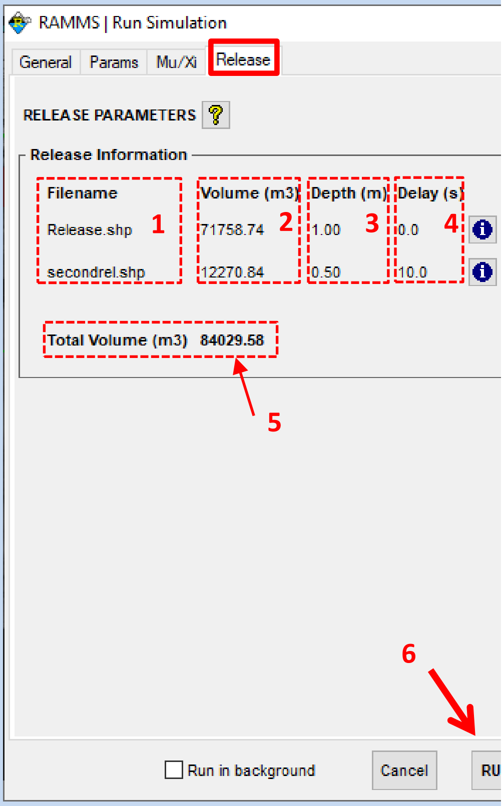


Figure 3-55: Release information.

Continuation of exercise 3.5g: How to run an avalanche calculation

Run Simulation

- Click run simulation (Figure 3-55)
- The following window appears, showing the status of the calculation (Figure 3-56) General information of the simulation (1), output filename (2), starting the calculation (3), for every dump step RAMMS outputs max flow height (Hmax) and velocity (Vmax) (4), moving momentum (%, 5) as well as flow volume, outflow volume (if it exists) and numerical volume loss.

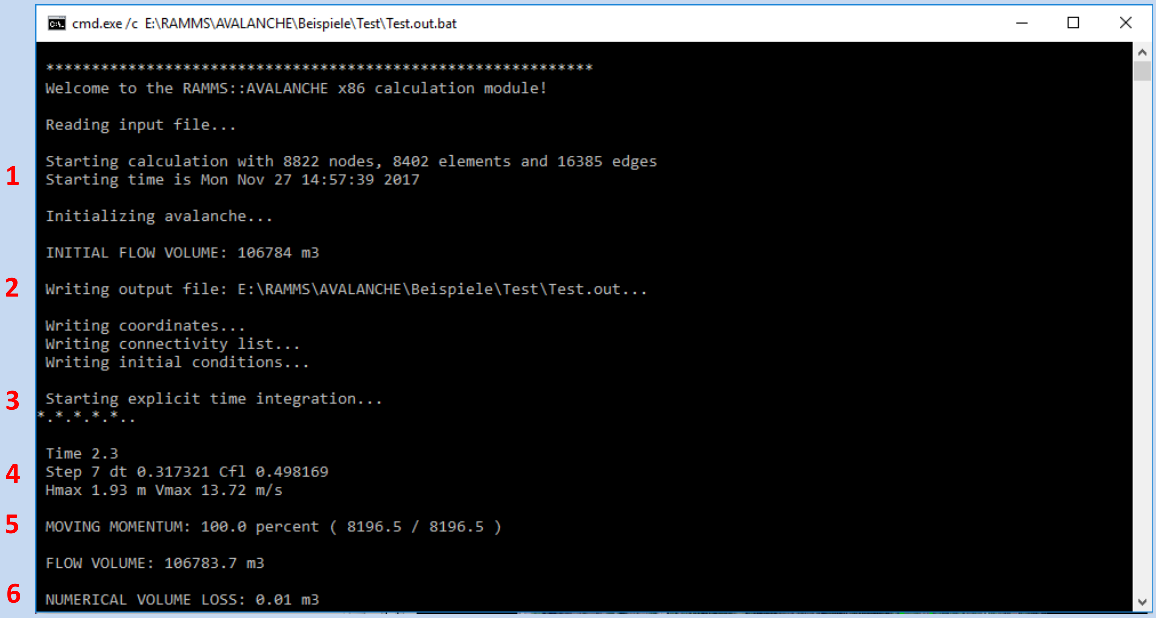


Figure 3-56: Calculation status window.

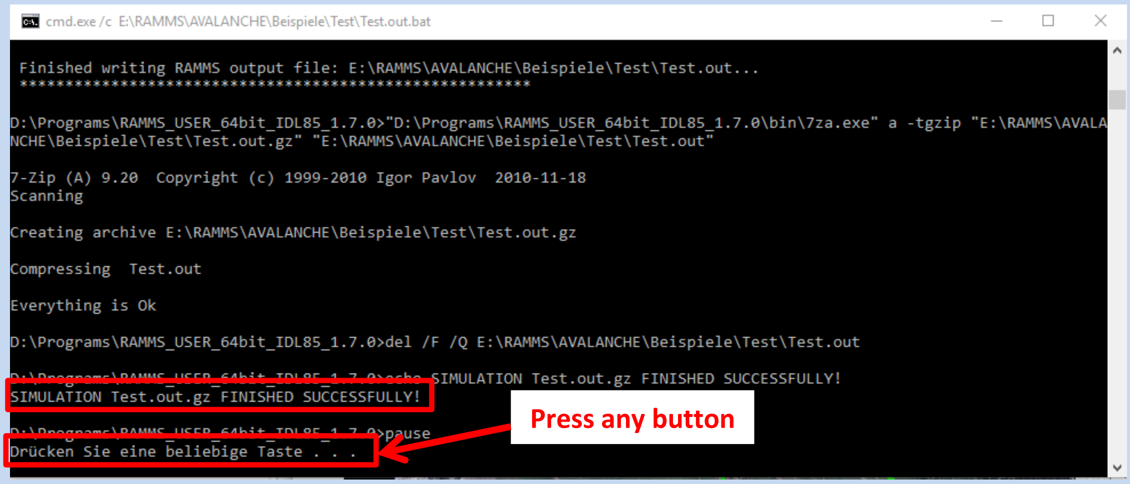


Figure 3-57: Background simulation mode window.
Press any button to close the DOS window.

3.5.5 How to run BATCH calculations

If you want to start several simulations automatically (e.g. overnight) use *Track* → *New…* → *Run Batch Simulations*. You can choose how many computational cores the Batch-Mode should use. It's even possible, to create a new directory for every single batch-simulation.

Please do the following to prepare input files for Batch-Simulations:

- Open an input file from your project, and open the "Run Simulation" window (the big green button on the left vertical toolbar).
- In the "Run Simulation" window, go through all the tabs, and set all the necessary input specifications.
- Then, instead of starting a simulation, click "Cancel" (this will close the "Run Simulation" window), and then click the "Save Copy As" Button (or *Track* → *Save Copy As*), and save a new input file (enter a meaningful name, .av2 file).
- Do the procedure above for every scenario you want to calculate. Then start the batch-process and choose your input file. You can also choose input files from different directories (projects).

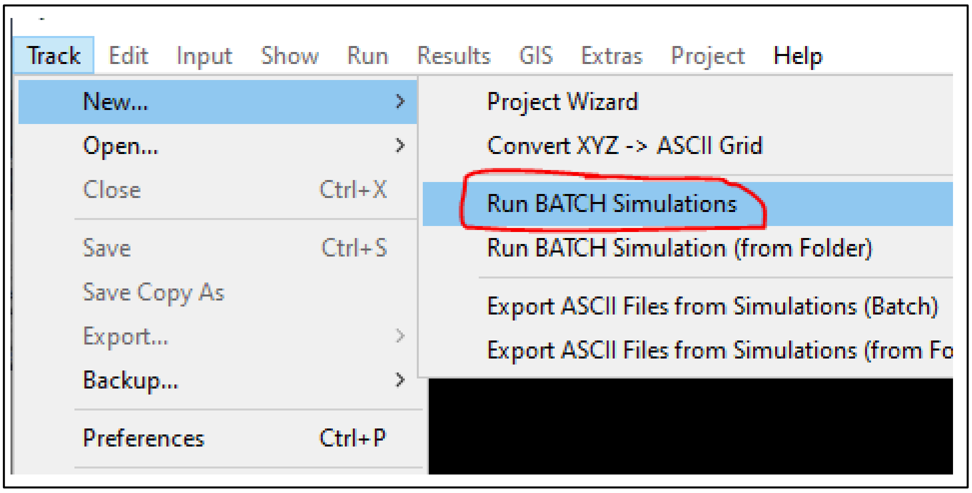


Figure 3-58: Batch-Simulations

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 3-57, click any button inside the DOS window to close the window. Afterwards, open the simulation in RAMMS manually).

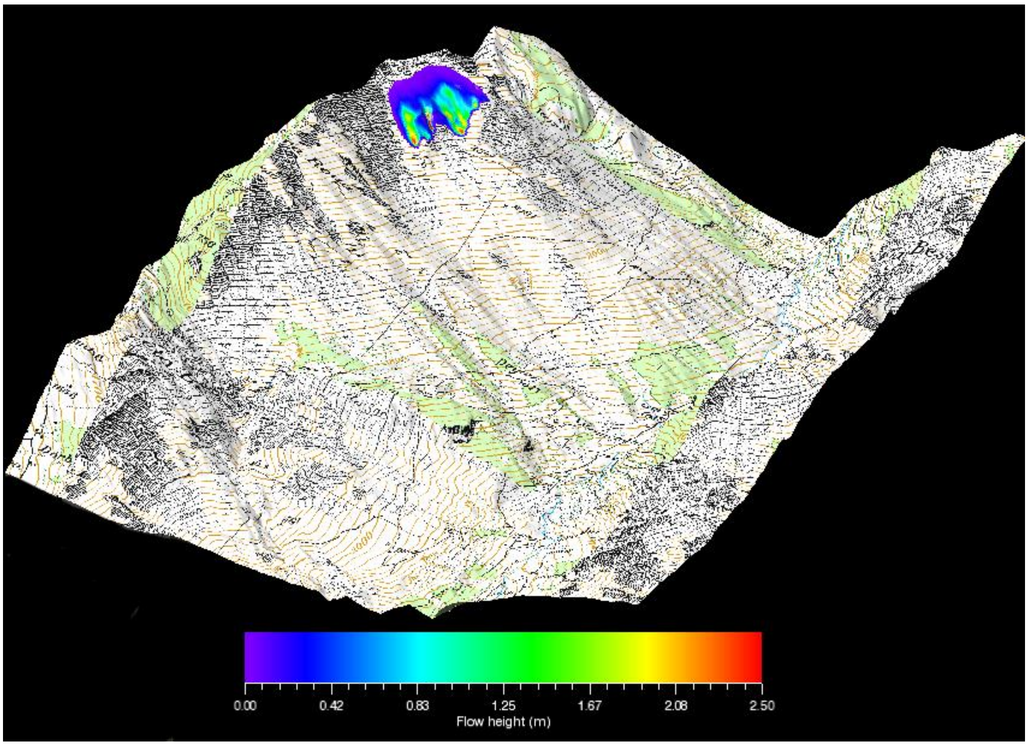


Figure 4-1: Main window in output mode.

If the avalanche 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 3.5.2).

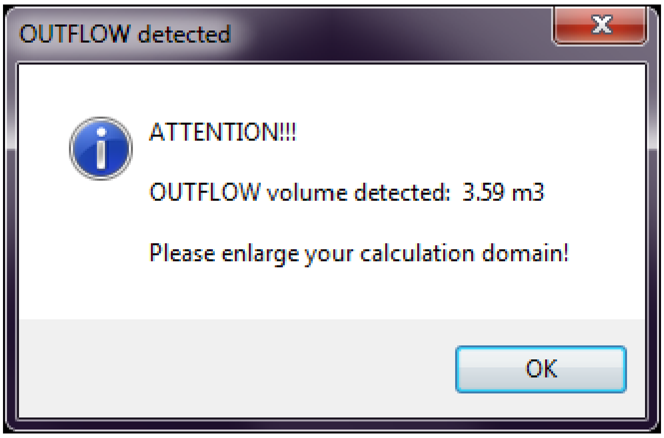


Figure 4-2: Outflow volume alert.

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*** *or* ***CENTER of MASS***. Otherwise the output ***TIME END CONDITION*** informs you, that your simulation stopped before the avalanche reached the stopping criterion you defined for the simulation (see section 4.2.6 on page 70).

(2) Information on simulation results.

(3) Input logfile (see Figure 4-4).

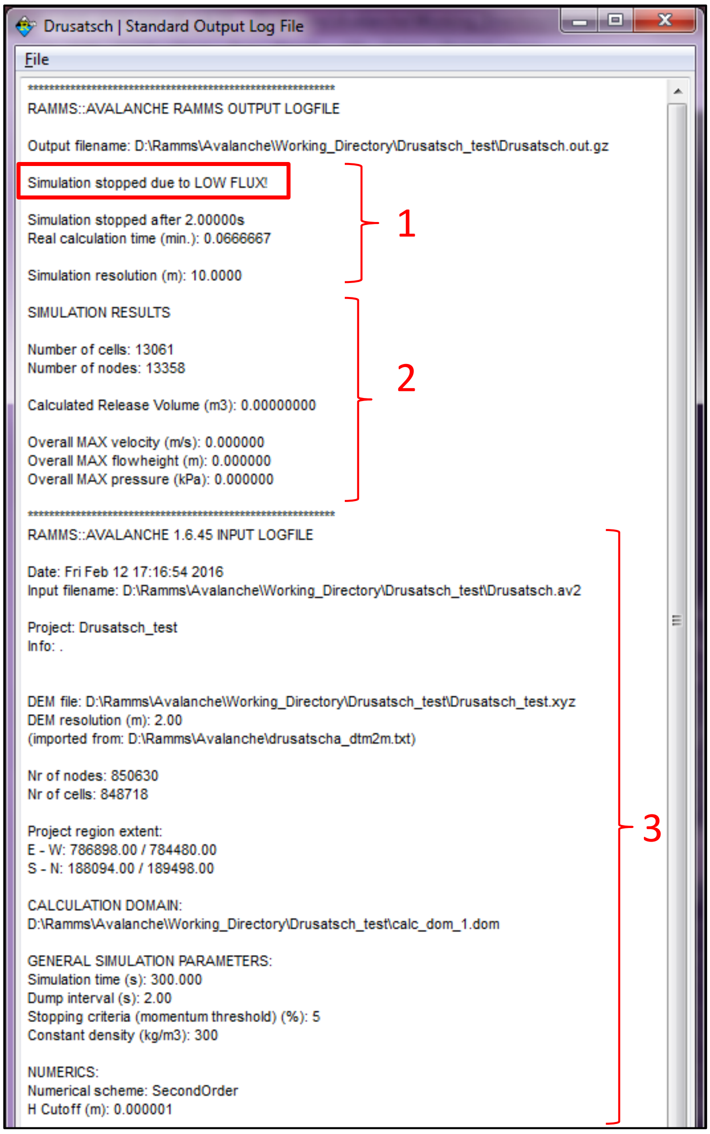


Figure 4-3: Output Logfile.

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 avalanche panel (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 DEM was used, the loaded map and orthophotos as well as your global simulation parameters.

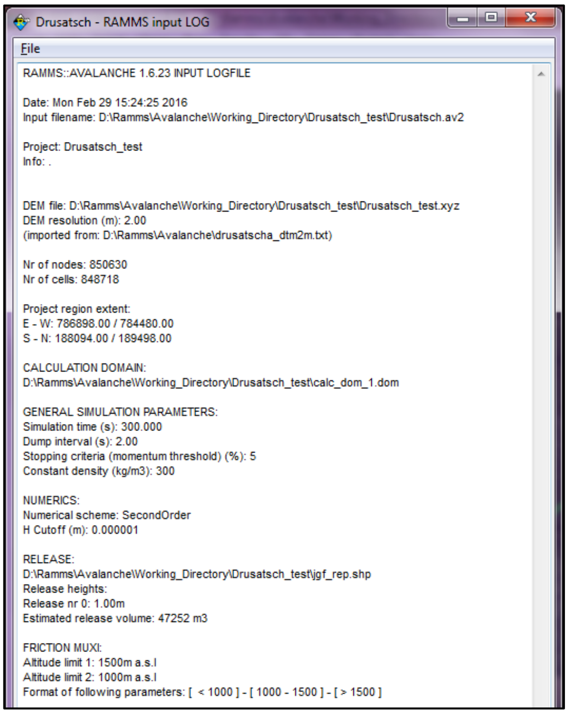


Figure 4-4: RAMMS Project Input Log file.

To view the project coordinates, click the region tab in your avalanche 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).

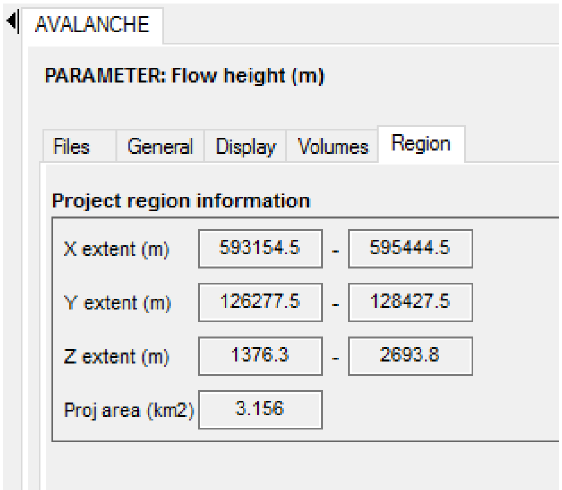


Figure 4-5: Region extent (X-, Y- and Z-coordinates, total projected area).

4.2 Visualization and analysis of 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
- Max values (Height, Velocity, Pressure, Momentum, Shear Stress)
- DEM Adaptions (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (μ, ξ)
- Cell area (m^2)

These results are all visualized by a color-plot in the topography. See exercise *"4.2a 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 avalanche. You find them under
Results → Max values…

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

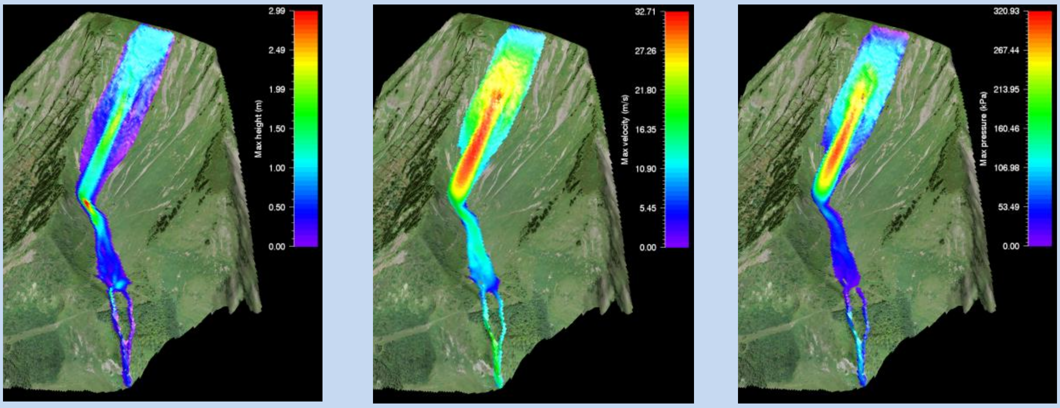


Figure 4-6: Results: Maximum values of flow height (left), velocity (middle) and pressure (right)

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**.

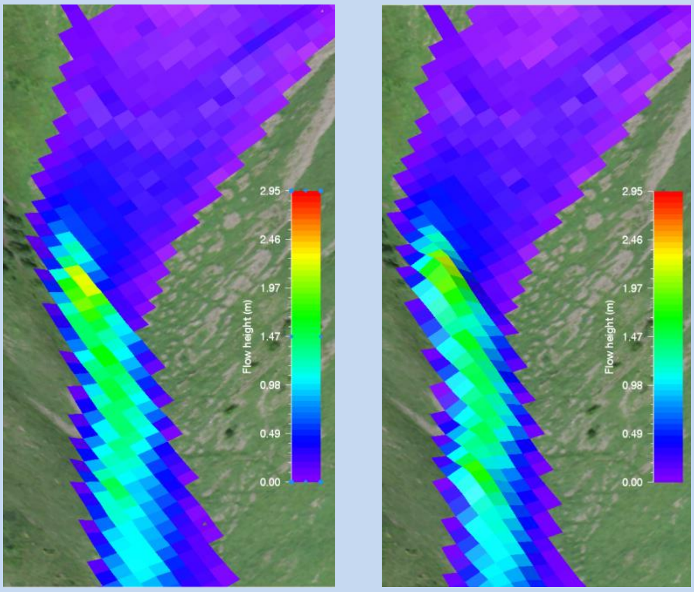


Figure 4-7: Quasi 3D-Visualization of flow height (left: exaggeration 1; right: exaggeration 5).

4.2.2 Line profile and time plot

In the horizontal toolbar you find two further functions:

- Line Profile 
- Time Plot

Line profile

A line profile is a good alternative to the color plot if the avalanche snow 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 *"4.2b 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 *"4.2c How to create a time plot"* below. The point-info file can be visualized with ***Extras → Point… → View Point Info File*.**

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-click on the mouse button.
- A window opens, displaying the line profile.

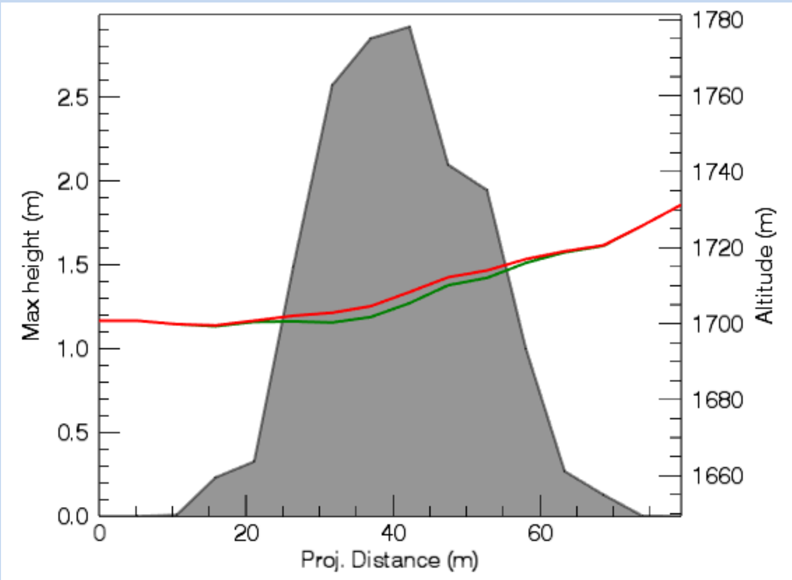


Figure 4-8: Line profile plot.

- 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

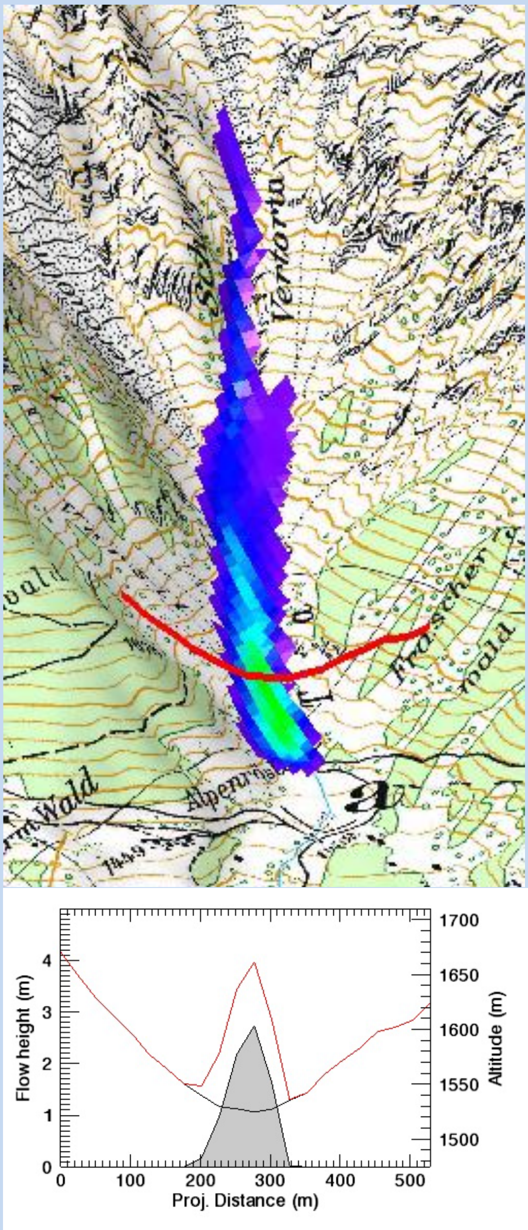


Figure 4-9: Line profile perpendicular to flow direction.

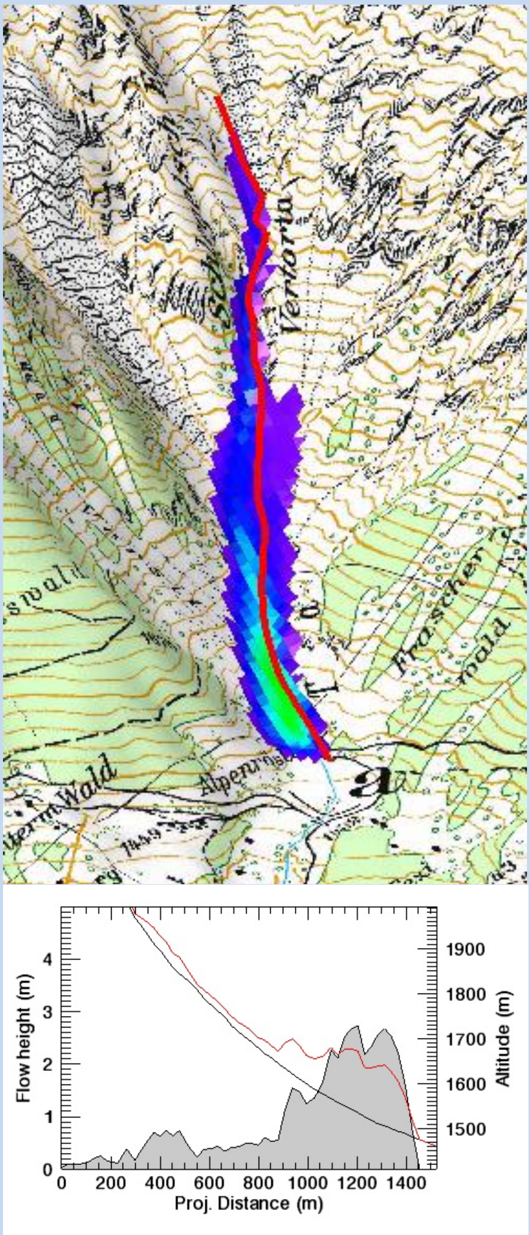


Figure 4-10: Line profile along the flow direction.

- 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). Flowheight is always combined with Velocity, whereas Velocity, Pressure or Momentum are always combined with Flowheight.

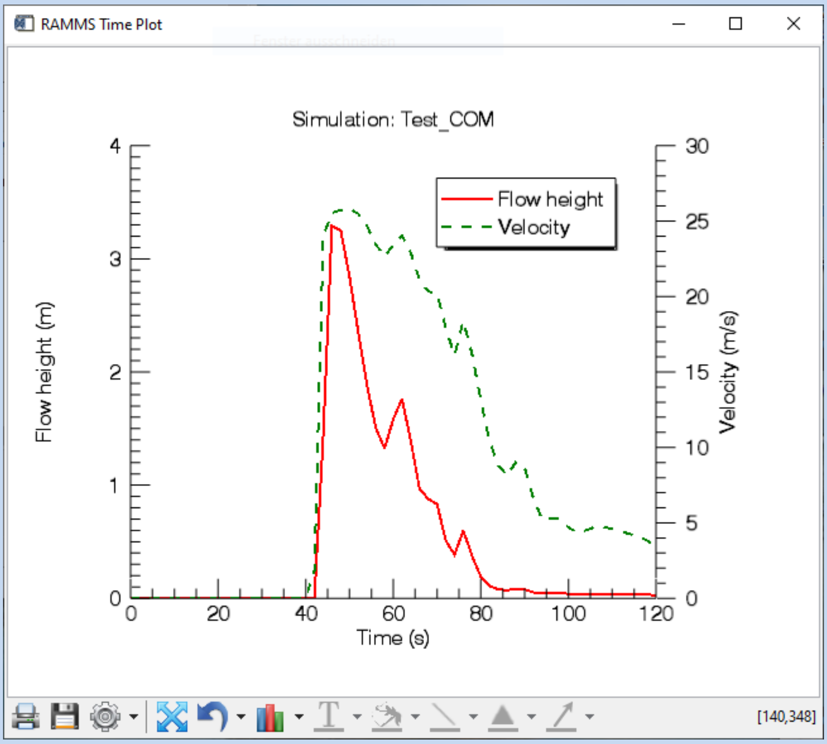


Figure 4-11: Time plot window.

- 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 analyze
- choose *Deposition analysis*

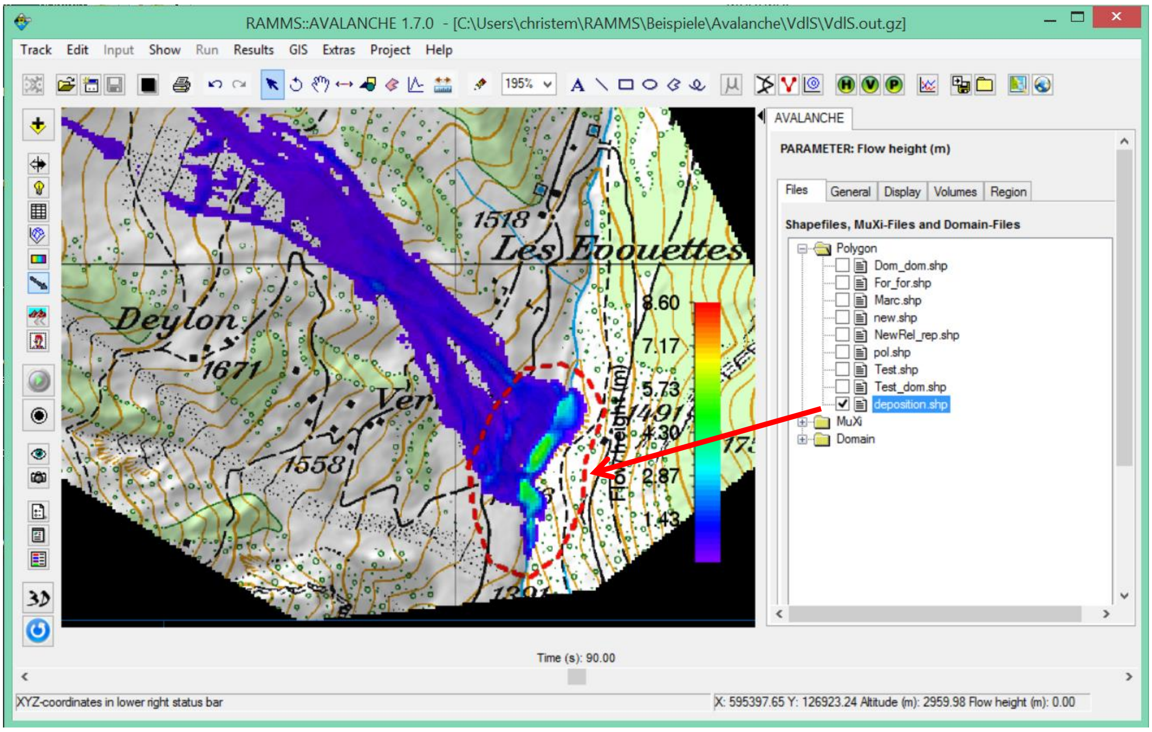


Figure 4-12: Deposition analysis of region of interest.

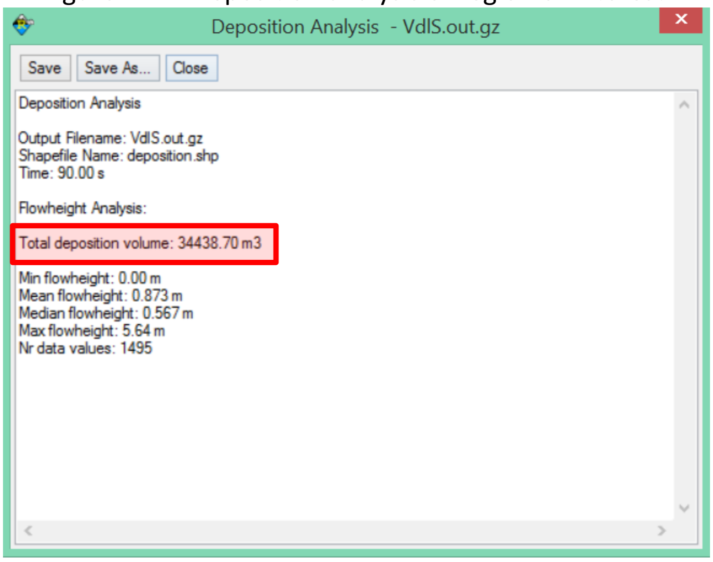


Figure 4-13: Result of a deposition analysis. Total deposition volume (m^3) as well as some statistical values are shown (min, mean, max).

4.2.4 Google Earth Export

You need to have the Google Earth Software installed on your PC/laptop, otherwise you cannot use this feature. The use of Google Earth is for free.

It's possible to export your result to Google Earth. The default settings are for Switzerland. If your project region is within Switzerland, you are lucky, and all you have to do is visualize the result you want, and then using ***Extras → Google Earth… → Export* Result to Google Earth**.

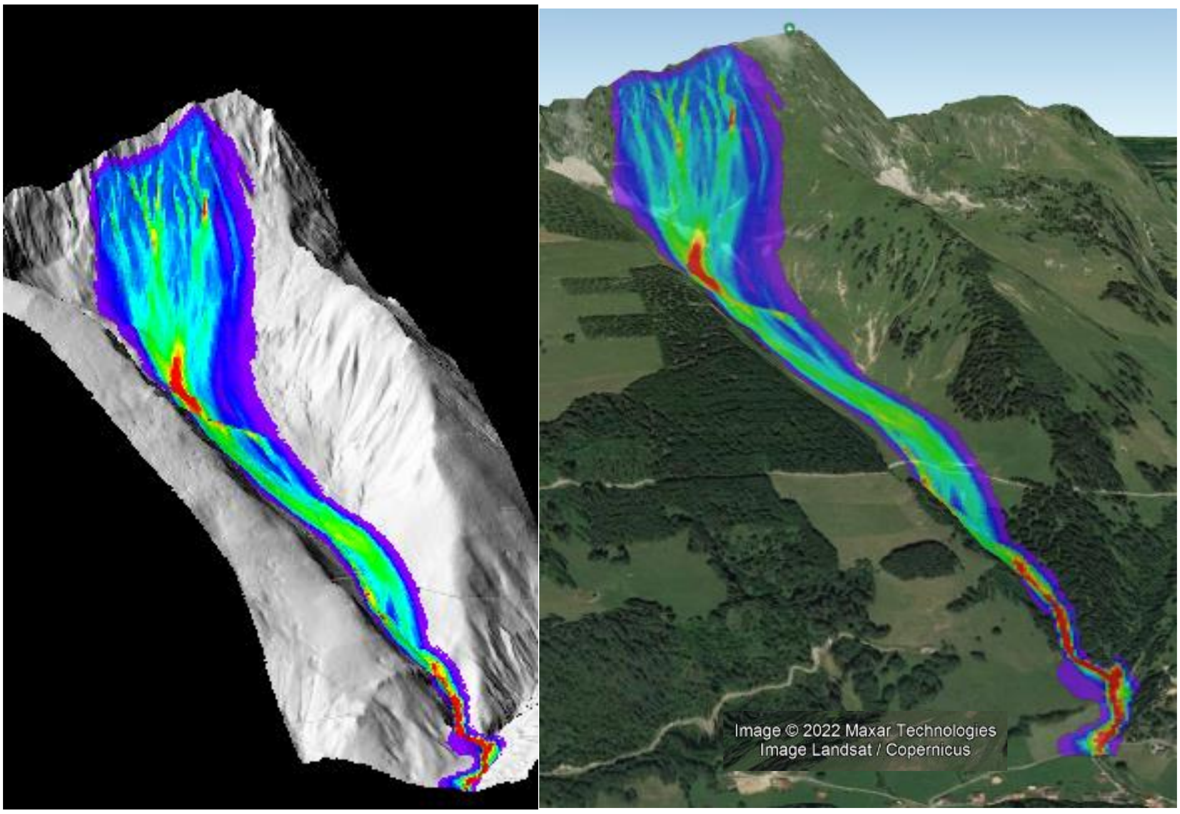


Figure 4-14: Left: Visualization in RAMMS; Right: Exported result in Google Earth (© Google Earth)

If your project region is not within Switzerland, then the following other projections are supported in RAMMS:

- UTM
- State Plane

Click ***Extras → Google Earth… → Google Earth Option*** to open the following window:

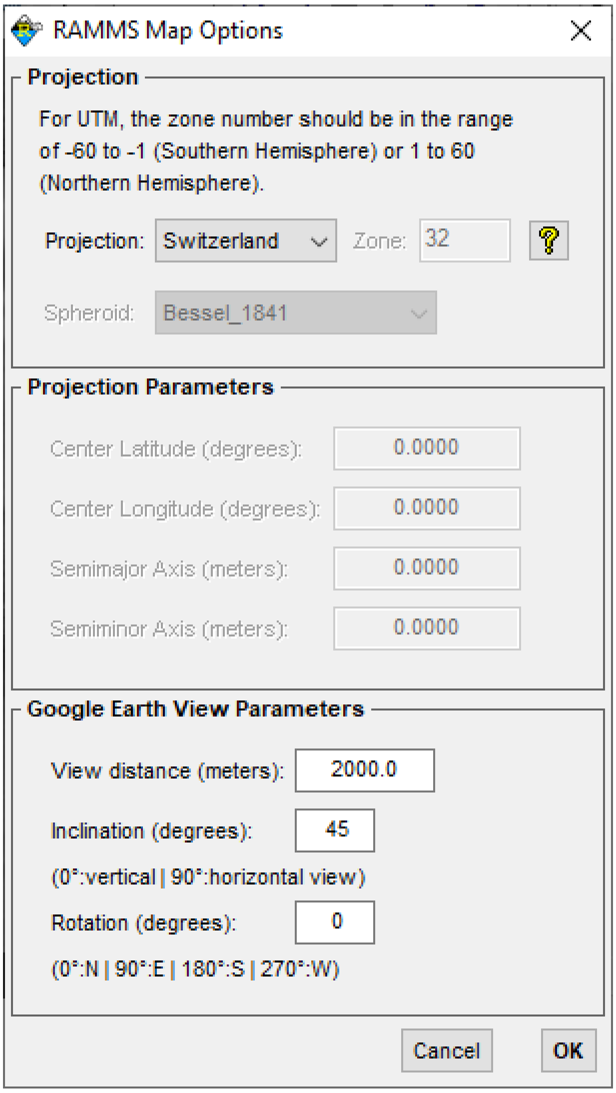


Figure 4-15: Google Earth Options. Choose *Projection* and *Spheroid (Datum)* of your project region

If you choose a UTM-projection, then it's possible to set the parameters in the *Projection Parameters* section. The section *Google Earth View Parameters* are parameters for the initial visualization in Google Earth. Change them as you like.

The Figure below shows the drop-down menus for the *Projection* and the *Spheroid (Datum)* fields.

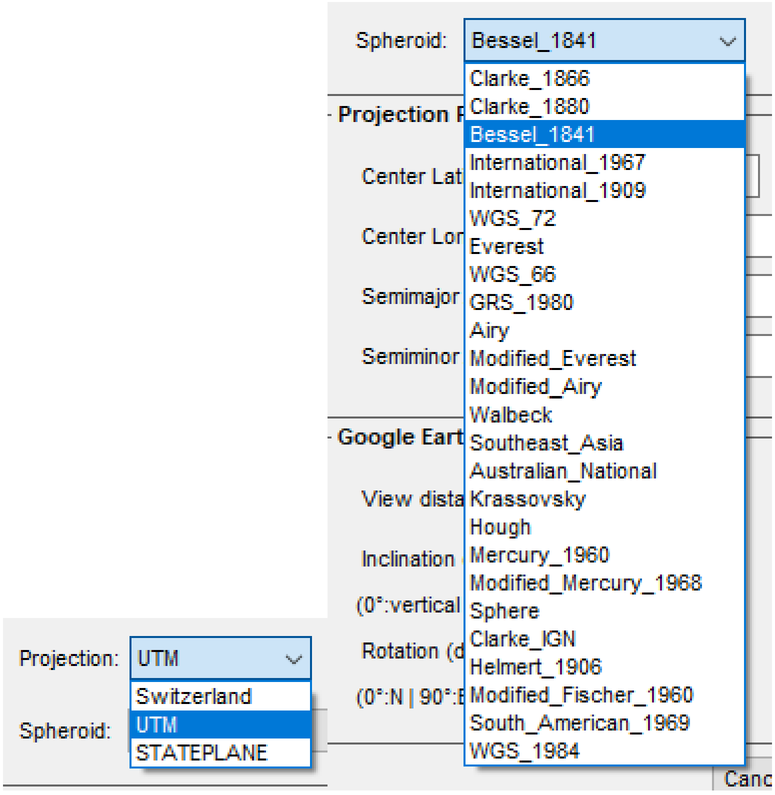


Figure 4-16: *Google Earth Projection* and *Spheroid (Datum)* drop-down menus

Click the Help-Button  to get help about the projections and how to specify *Zone*, *Spheroid* and *Projection Parameters* (only for UTM), such that your result is visualized correctly.

4.2.5 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* in the *avalanche tab* you can define the interval for the GIF animation (GIF animation interval [s]).

4.2.6 Stopping criteria

There are two stopping criteria available in RAMMS:

1. Standard stopping criterion: Momentum-based, *"Percentage of total momentum (%)"*
2. New stopping criterion: Center-of-mass based, *"Center-of-Mass vel threshold (m/s)"*

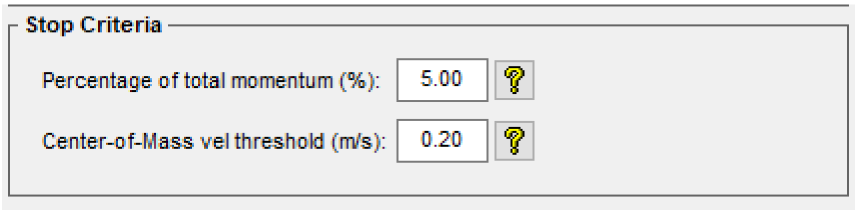


Figure 4-17: The two stopping criteria available in RAMMS

The stopping criteria can be changed when starting a new simulation, see also Figure 3-52 on page 51. You can define both stopping criteria, and the simulation will then stop when one of them is reached, either the momentum- or the center-of-mass-criterion. In the following, both criteria are explained in more details.

Momentum-based stopping criteria

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. For every dump-step, RAMMS sums up the momenta of all grid cells (dump-step-sum), thus finding a maximum momentum sum (max-momentum-sum). Then the dump-step-sum is compared to the max-momentum-sum. If this percentage is smaller than a user defined threshold value (see Figure above and on page 51), RAMMS aborts the simulation and the avalanche is regarded as stopped.

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 avalanche material and velocity oscillations.

To check the stopping of your simulation, click ***Results → Summary of Moving Mass***. A window similar to Figure 4-18 opens which shows the summary of moving mass.

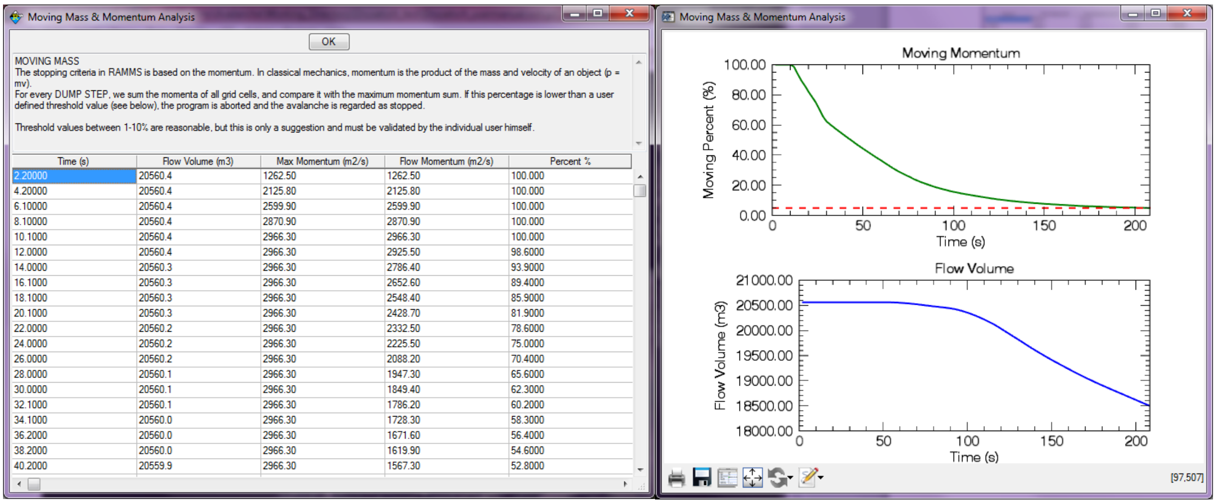


Figure 4-18: Summary of Moving Mass information windows

Whether or not an avalanche 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 avalanche runout (flow height and flow velocity) with time plots (section 4.2.2).

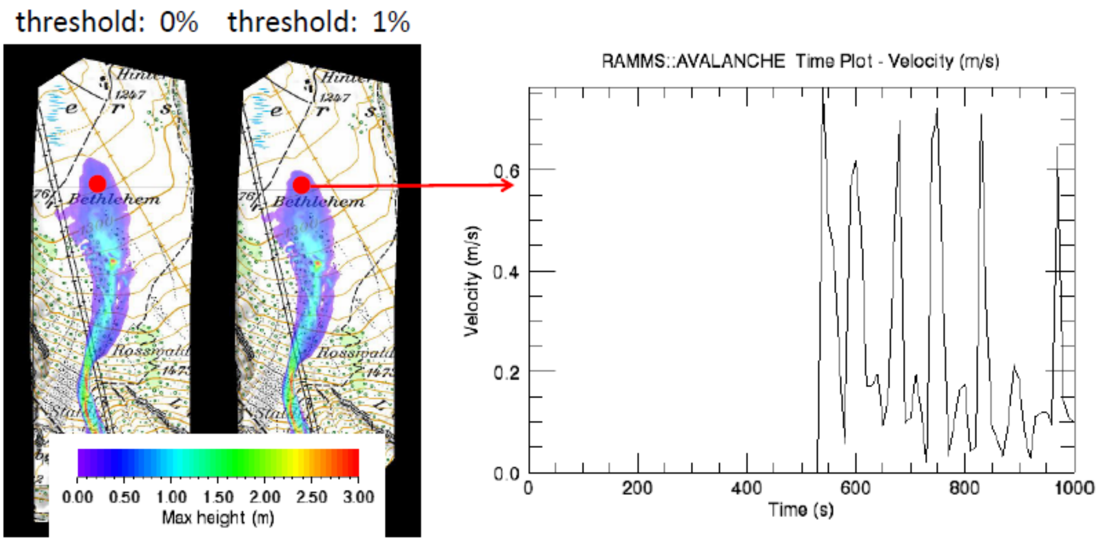


Figure 4-19: Stopping behaviour of a RAMMS simulation. Small threshold values may lead to unlikly slow creeping of the material. In the example shown in the figure above the stopping criteria is set to 0%.

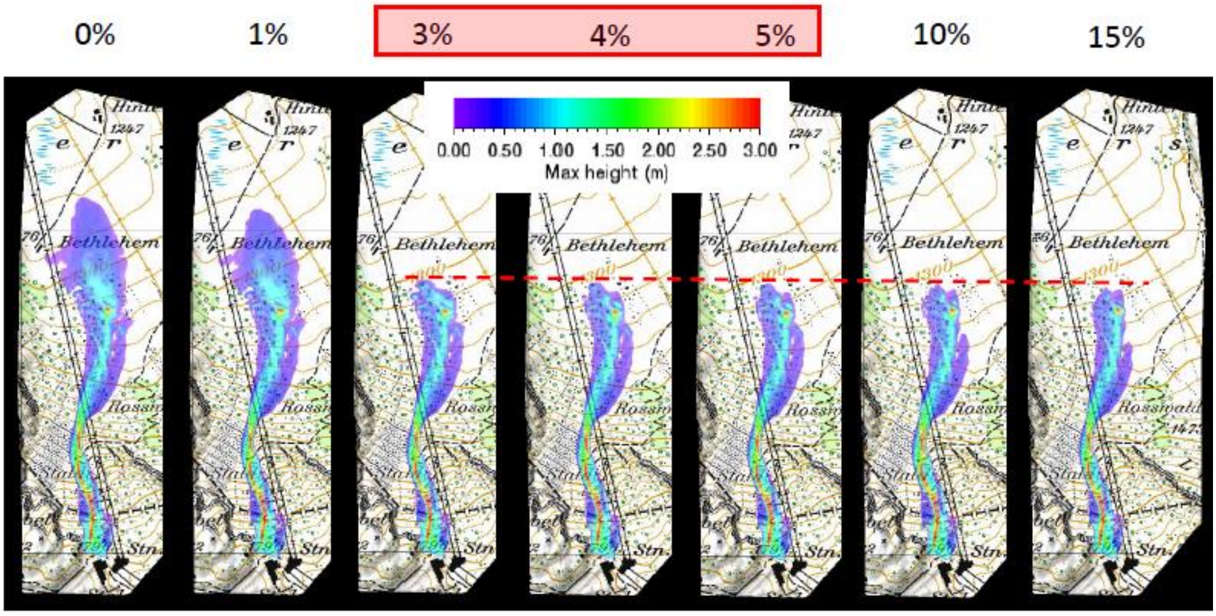


Figure 4-20: Stopping behaviour of a RAMMS simulation. In this example threshold values <2% lead to numerical diffusion of the simulation results. Threshold values between 3-5% seem to be appropriate in this case.

Check the output logfile under ***Project → Output Logfile*** to verify your simulation stopped due to low flux (see Figure 4-3 on page 57). Otherwise enlarge the end time of your simulation (see exercise *"Run a calculation"* on page 50).

Center-of-mass stopping criteria

This stopping criterion was introduced in Version 1.8.0. The idea of this new stopping criterion is the following: Following the center-of-mass (COM) of the flow. RAMMS is calculating the COM every 2s, and from the covered distance the velocity, with which the COM is moving.

The user can define a velocity threshold (in m/s), see Figure 3-52 on page 51, and as soon as the COM is moving slower than this threshold, RAMMS considers the flow as stopped. With this approach, numerical diffusion and slow creeping movements at the front can be prevented.

This criterion should only be used, if you simulate one single avalanche problem (catchment). If you define two or more release areas (for different catchments), then this approach makes no sense. In this case, set the threshold value to zero. In any case it's suggested to do simulations with and without the center-of-mass stopping criteria, and to critically analyze results.

During a calculation, the center-of-mass travel-speeds are also written to the standard-output (the black DOS command prompt), see Figure below. You can open the standard-output-logfile with ***Extras → View Simulation Standard Output Log.***

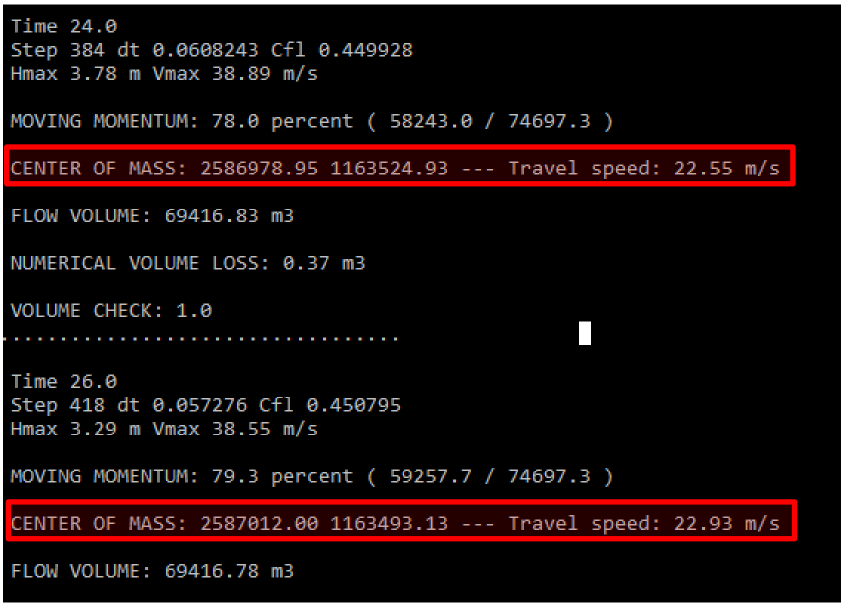


Figure 4-21: Center-of-mass travel speeds

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 snow 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 [1], Rudolf-Miklau and Sauermoser 2011 [2]. 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 avalanche 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

RAMS 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 polygon shapefile where a dam is supposed to be built (Figure 4-22).
- 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.

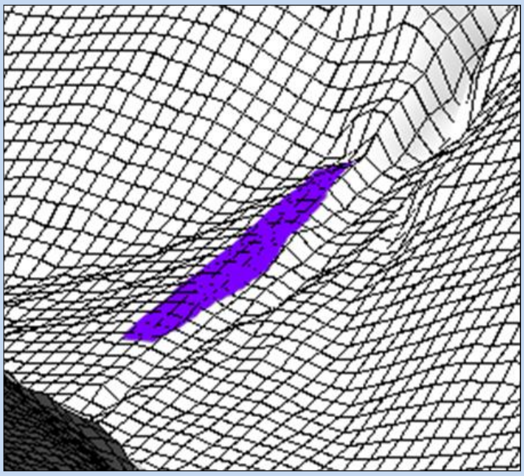


Figure 4-22: Polygon area where a dam is supposed to be built.

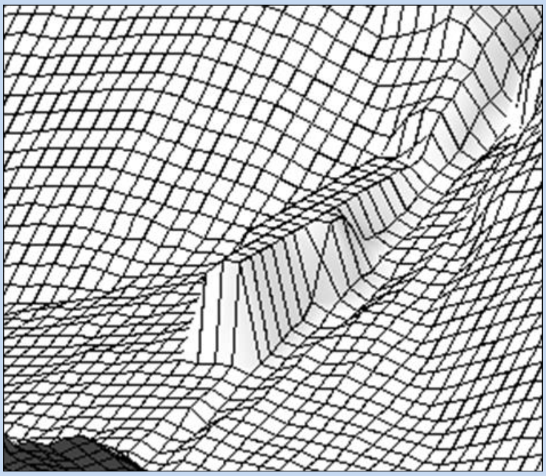


Figure 4-23: New DEM with dam at location of polygon shapefile.

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

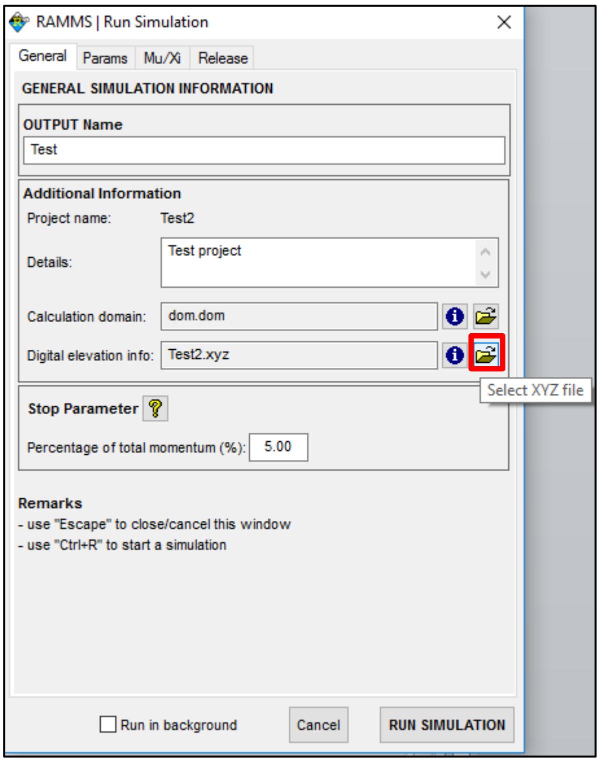


Figure 4-24: Select new XYZ-file with dam information.

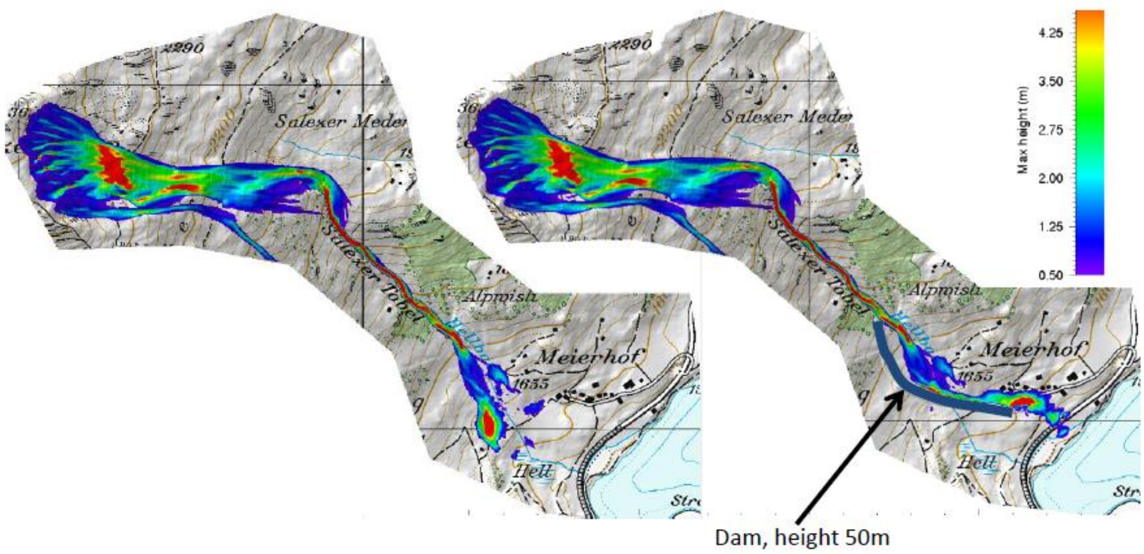


Figure 4-25: Simulation without (left) and with (right) a dam.

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.

- 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.

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 4-26.

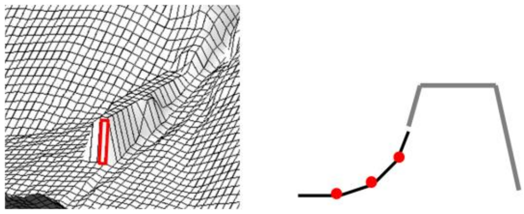


Figure 4-26: Dam with gradually rising side walls.

The interpretation of RAMMS simulations including mitigation measures such as dams 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 avalanche deposition

In case you wish to simulate an avalanche overflowing a previous avalanche, you should consider the deposition of the previous avalanche, because the path of the second avalanche will be influenced by the modified terrain. In RAMMS one has to assume that the deposits from an initial avalanche are not entrained by a subsequent avalanche. To do this, in the output mode, users can select the option ***Add Deposition to DEM*** to add the flow height of an avalanche to the DEM at any arbitrary dump-step. A new xyz-file (with the updated topographic information) will be created.

Exercise 4.3b: How to add avalanche deposition to new DEM

- The deposition height is the flow depth at the end of a simulation when the avalanche 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 → 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 4-24.

5 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 path; 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.

5.1 The Graphical User Interface (GUI)

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

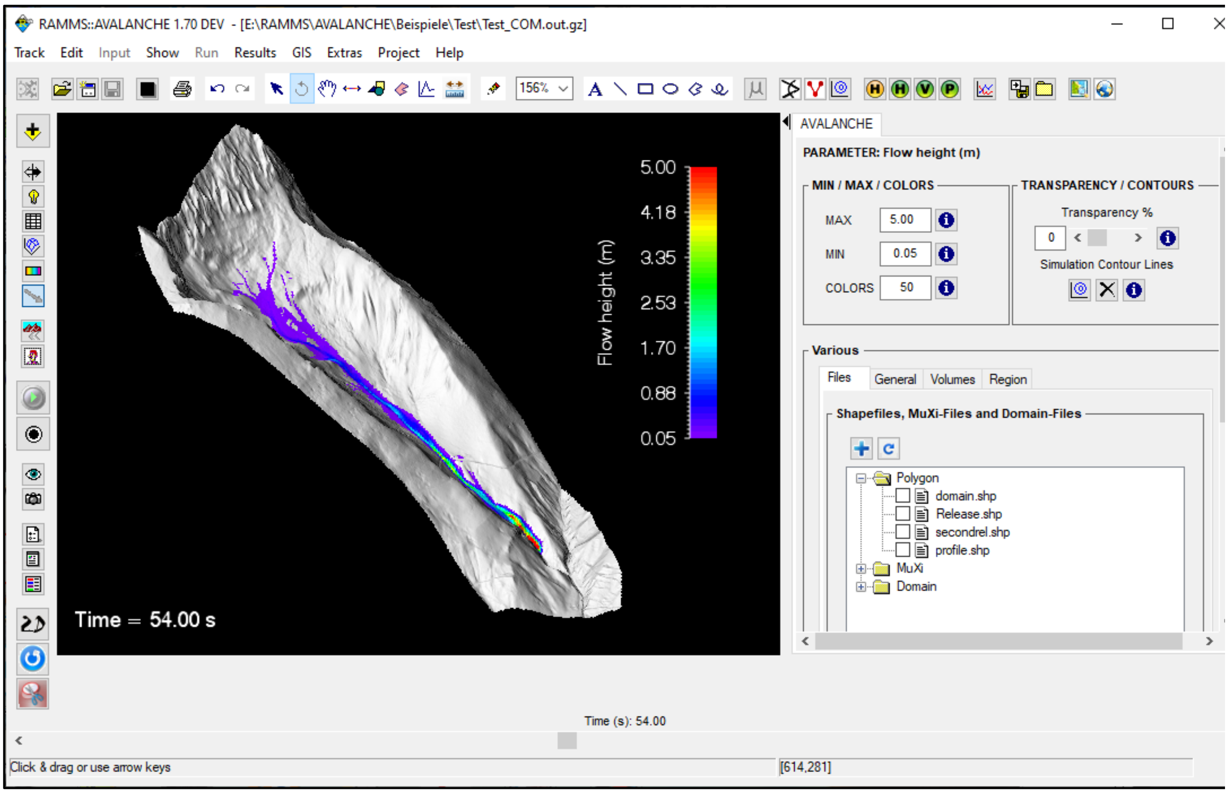


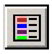
Figure 5-1: Graphical user interface (GUI)

5.1.1 The menu bars

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 Data → Raster Data**	Convert regular or irregular XYZ data (e.g. laser scanning data) into a raster grid (ESRI ASCII or GEOTIFF).
	• **Run BATCH simulations**	Possibility to start simulations automatically (e.g. overnight) You can choose how many computational cores the Batch-Mode should use (quasi parallel simulations, saves computational time).
	• **Run BATCH simulations (from Folder)**	Choose a folder with input files for a batch simulation (see above).
	• **Export ASCII Files from Simulations (Batch)**	Automatically export all ASCII files (max height, max velocity, max pressure, deposition) from multiple output files.
	• **Export ASCII Files from Simulations (from Folder, Batch)**	Automatically export all ASCII files (max height, max velocity, max pressure, deposition) from all output files from a folder.
Open…	• **Input File**	Open an existing input file (*.av2) (Ctrl + O)
	• **Avalanche Simulation**	Open an existing avalanche 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.av2) under a new name (e.g. simulation1.av2, works only in input mode).
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.).
	• **GIF Animation**	Create a GIF animation (only in output mode). Change GIF animation interval (s) and delay (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/she 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 (Ctrl + P)	
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.	
Restart RAMMS		Restart RAMMS.	
Exit		Exit RAMMS (Ctrl + Q).	
Recent…		Shows a list of up to 10 of your recent input and output files.	

Edit

This menu is used to edit colorbar, axes and dataspace properties.

Colorbar Properties	Edit the colorbar properties.	
Get Colorbar	Get back your colorbar, if lost.	
Dataspace Properties	Edit your dataspace properties.	
Show Dataspace Axes	Shows or hides dataspace axes of the project region. The axes are only visible if the background color is NOT set to black.	
Colorbar White Color	Checkbox. If checked, the colorbar text-color is white (default), otherwise black.	

Input

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

Global Parameters		Set return period and avalanche volume. These parameters are used to calculate a friction MuXi-File.
Calculation Domain…	• **Draw New Domain**	Draw a new calculation domain. 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 calculation domain). This works only in 2D mode. Choose points **clockwise**.
	• **Load Existing Domain**	Load an existing calculation domain (*.shp). Any polygon shapefile can be used as calculation domain (Ctrl+D).
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. Choose points **clockwise**.
	• **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 a release area to define the release depth and to view release area information. This works only in 2D mode.
	• **Details/Edit Overlap Release Areas**	Same as above, but used for overlapping release areas.
Forest…	• **Show Active Forest Cover**	If forest is considered, the corresponding shapefile is dis-played. If your project uses no forest cover at the moment, RAMMS will tell you so.
	• **Import Forest Area From SHAPEFIL**	You can import any polygon shapefile using this function.
	• **Import Forest Area From ASCII grid**	If a forest ASCII grid is available, it can be imported using this function (0 = no forest, 1 = forest).
	• **Remove Active Forest Cover**	Remove the active forest raster data from the project.
Friction Values…	• **Load an Existing MuXi File**	Load afore created MuXi-file (*_mu.asc or *_xi.asc).
	• **Create New MuXi File (Automatic Procedure)**	The DEM is analyzed, classified and according to altitude, slope and curvature information, return period and avalanche volume, a new MuXi-file is created.
	• **Show MuXi Classifica-tion**	Shows the result of the MuXi-classification.
	• **Advanced… → Remove MuXi Classification File**	Remove the MuXi-classification file.

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 Velocity Arrow	OUTPUT | Show/hide velocity vectors
Show Domain	INPUT | Show/hide calculation domain

Run

This menu is active only in input mode.

Run Avalanche Calculation	Opens the *Run Simulation* window to change parameters and to start the calculation of an avalanche simulation (F8).

Results

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

Flow Height		Shows flow height of the avalanche every dump step.
Flow Velocity		Shows flow velocity of the avalanche for every dump step.
Flow Pressure		Shows flow pressure of the avalanche for every dump step.
Flow Momentum		Shows flow momentum of the avalanche for every dump step.
Max Values…	• **Max Flow Height**	Displays the maximum flow height.
	• **Max Velocity**	Displays the maximum velocity.
	• **Max Pressure**	Displays the maximum pressure.
	• **Max Flow Momentum**	Displays the maximum momentum.
	• **Max Shear Stress**	Displays the maximum shear stress.
Add Deposition to DEM		Adds the deposition of an avalanche simulation to the DEM. A new DEM file is created.
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 grid cell (m^2).

GIS

This menu contains miscellaneous GIS functions.

Add Data		Add data (shapefiles, MuXi-ASCII files) to the visualization.	
Export …	• **Result as Shapefile**	Export the active result to an ESRI GIS shapefile for later use in a GIS program.	
	• **Result as Raster Data**	Export the active result to a GEOTIFF (default) or ASCII raster file..	
	• **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		Resamples slope/curvature plots to a user defined resolution.	

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 3.2 for details. If not, you can browse for maps.	
Add/Change or Remove Image		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 3.2 for details. If not, you can browse for images.	
Create Hillshade Image		This will calculate a hillshade visualization of your DEM. It's then saved as a TIFF image that you can use in RAMMS.	
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 parameters. 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.
RAMMS License Agreement		RAMMS License Agreement
RAMMS FAQ		Opens the RAMMS FAQ website at https://ramms.slf.ch in a web browser.
RAMMS Website		Opens the RAMMS website at https://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.
Update/Register License Key		Edit your existing license file or add/register a new license key (module).
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 (Admin privileges required).

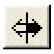
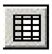
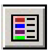
	• **Logging**	Checkbox. Switch logging ON or OFF.
	• **AutoWebUpdate**	Checkbox. Switch AutoWebUpdate ON or OFF. If AutoWebUpdate is ON, then RAMMS will check for updates whenever you start RAMMS.
	• **Hardware Rendering**	Checkbox. Switch hardware rendering ON or OFF. If hardware rendering is switched ON, then all graphical rendering is done by your hardware, otherwise by IDL (RAMMS). It is suggested to switch hardware rendering OFF.
	• **Curvature**	Checkbox. Switch curvature ON or OFF. See section 3.1.4 on page 16 for more information about curvature. Default is ON.
	• **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		Show Information about the RAMMS releases in pdf format.
About RAMMS		*About RAMMS* information window.

5.1.2 Horizontal toolbar

	Project wizard: open avalanche wizard for creating a new avalanche 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 5-11 on page 96).
	OUTPUT | If you move the arrow over the simulation data, the active parameter is shown as well (see Figure 5-11 on page 96). If you click once with the left mouse button at a point of interest, a new window pops up called 'RAMMS::AVALANCHE 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/she wants to draw more polygons. At last, he/sh has to specify a new filename for the polygon shapefile.
	INPUT, 2D |Draw new calculation domain: specify a new calculation domain polygon by clicking with left mouse button, finish with a right mouse click. A dialog box will then ask the user for a new domain name (e.g. domain), and a polygon shapefile is saved.
	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. Finish with a right mouse button click. A new window pops up called 'RAMMS::AVALANCHE 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 avalanche panel, the changes are adapted in the line profile plot too. When animating the simulation, the line profile is animated too.
	2D |Measure distance and angle: Click with left mouse button; distance and angle between clicks is shown in the lower right status bar. Finish with a right mouse click.
	INPUT, 2D | View and Edit Release Areas.
100%	Zoom tools.
A	Annotation tools, text, line, rectangle, oval, polygon, freehand. They can be activated and deactivated in the additional preferences. *Preferences → Advanced… → Edit → Annotations*

	INPUT | Create new MuXi File (Automatic Procedure).
	Analysis of the input DEM: Slope Angle, Curvature and Contour Plots. Remove visualization by clicking the button again.
	OUTPUT | Show dump step values of the simulation result: Flow Height.
	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.

5.1.3 Vertical toolbar

	Add data to visualization (*.shp, *.asc).
	OUTPUT | Switch to corresponding input file.
	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 | Open *Run Simulation* window.
	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 ().
	Refresh visualization (if stuck).

5.1.4 Main window

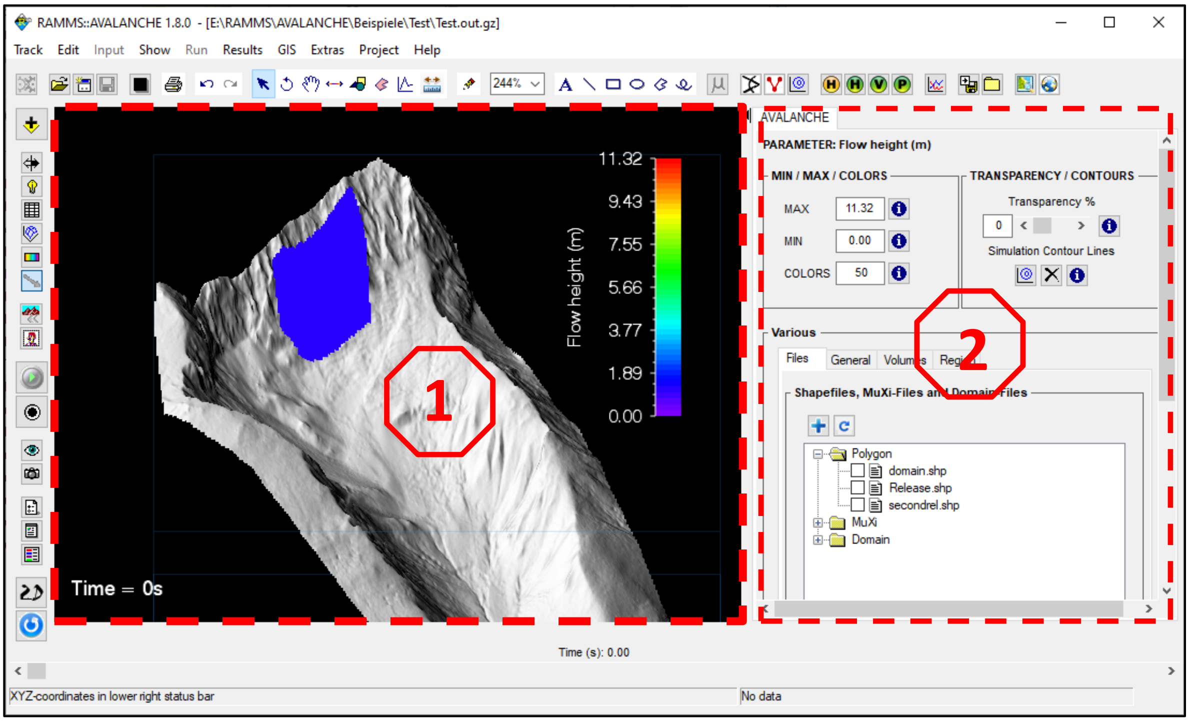


Figure 5-2: Main visualization window and information panel.

The RAMMS GUI (**G**raphical **U**ser **I**nterface) consists of two main regions, see Figure 5-2:

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

5.1.5 Panel

An AVALANCHE panel is displayed on the right side of the RAMMS GUI (Figure 5-2), and consists of the display part, where visualization and colorbar settings can be set (MIN/MAX/COLORS and TRANSPARENCY/CONTOURS, and four tabs (*Files*, *General, Volumes* and *Region*). Always confirm with ***ENTER*** (return key) when changing a value! Additionally, the *PARAMETER* line states the visible parameter (e.g. Release height (m) in the Figure below, red box). Simulation contour lines can only be shown for an output result (see Figure 3-35).

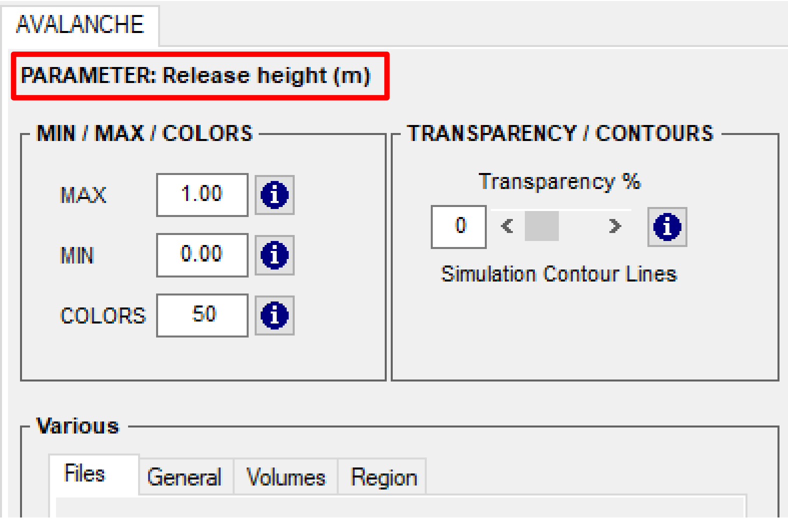


Figure 5-3: AVALANCHE panel with four tabs

The Min and Max values as well as the number of colors influence directly the colorbar and the visualization. The transparency changes the visibility of the result: 0% means no transparency, 100% means total transparency, see Figure below. The colorbar is divided into N (nr. of colors) different colors.

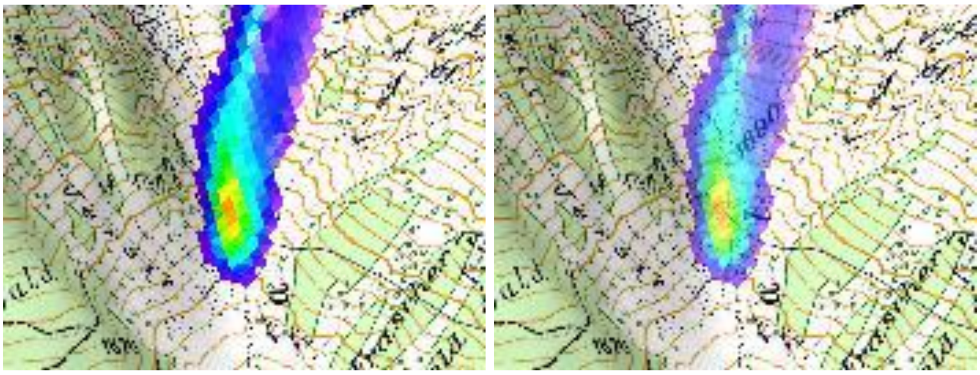


Figure 5-4: No transparency (left) and 40% transparency (right) of simulation result.

Files tab

The *Files* tab (Figure 5-5) shows a file tree with nodes for polygon shapefiles (*Polygon, *.shp*), MuXi-files (*MuXi*, *.asc) and calculation domain files (*Domain*, *.shp). See section 3.4.2 on page 27 on how to use the Files tab.

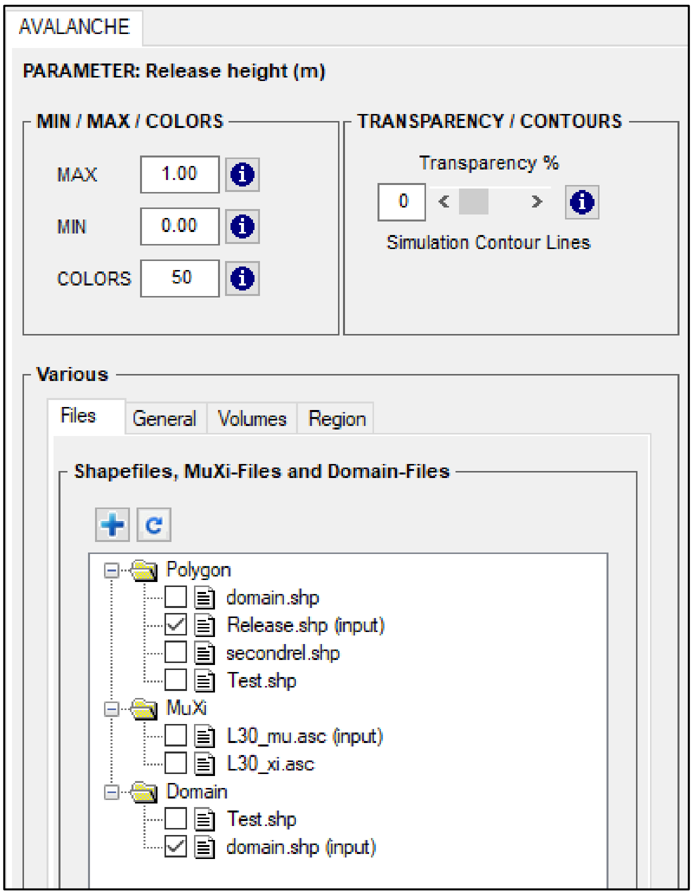


Figure 5-5: Avalanche panel – Files tab

General tab

The *General* tab (Figure 5-6) 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 ~50'000 grid cells remaining (see *Visualization Resampling Remarks* below, red box). This does not influence any simulation at all, it simply makes the users life easier to zoom and rotate the topography.

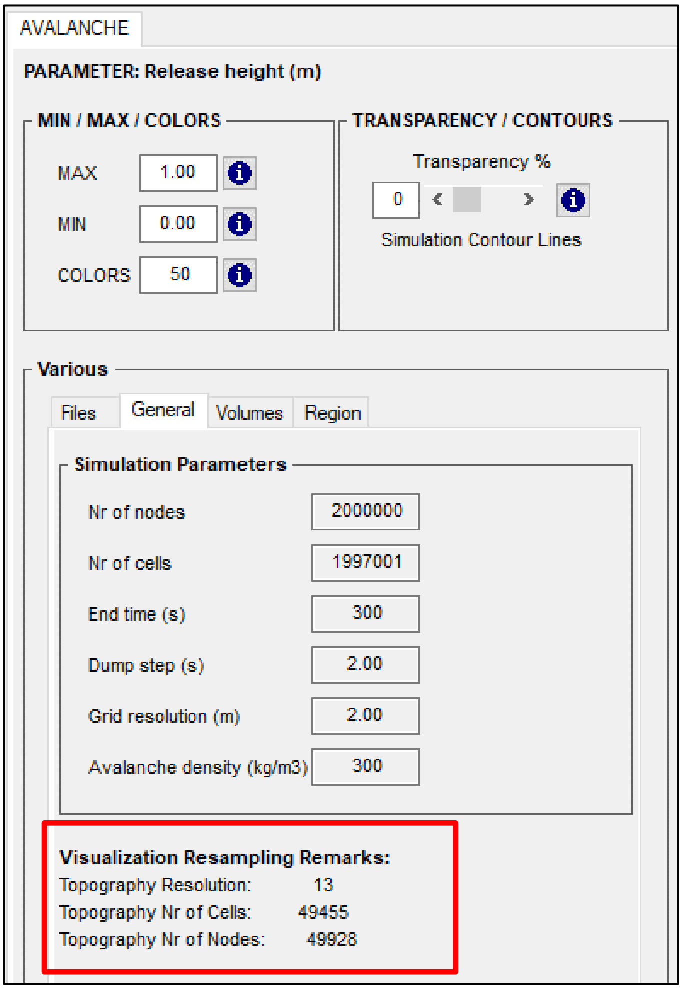


Figure 5-6: Avalanche panel – General tab

Volumes tab

The *Volumes* tab (Figure 5-7) gives the user information about

- projected release area (m^2)
- inclined (3D) release area (m^2)
- release volume (m^3, estimated in input mode)
- Flow volume (m3)
- Outflow volume (m3), volume flowing out of the calculation domain

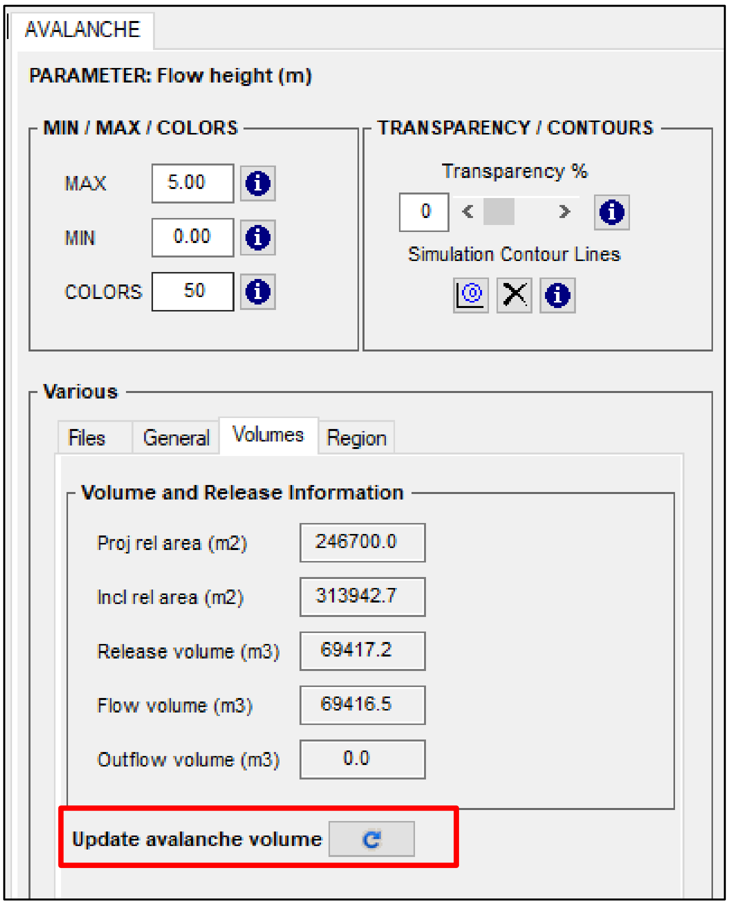


Figure 5-7: Avalanche panel – Volumes tab

Click the *Update avalanche volume* button (in output mode, red box), if the flow volume is not up-dated automatically.

Region tab

The *Region* tab (Figure 5-8) gives information about min, max and diff X-, Y-coordinates and the altitude limits as well as information about the region area in km^2.

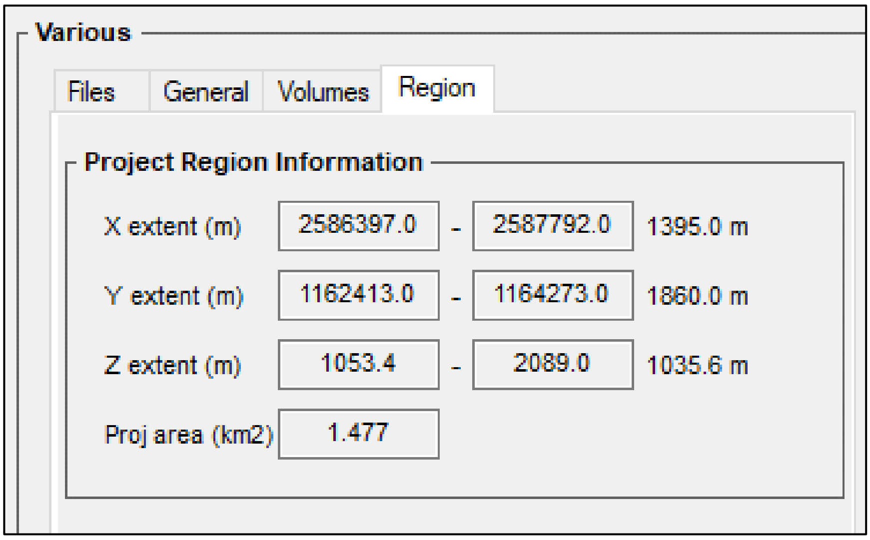


Figure 5-8: Avalanche panel – Region tab

5.1.6 Time step slider

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

Figure 5-9: The active time (20s) is shown in the time slider.

5.1.7 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 5-10: Status information shown in the left status bar.

5.1.8 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 5-11: Position information and simulation results in the right status bar.

5.1.9 Colorbar

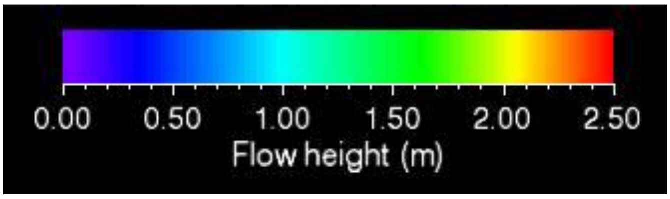


Figure 5-12: Colorbar

In general, the colorbar appears at the right edge of the main window (Figure 5-2) and can be moved and resized (see section 3.4.6 on page 34).

6 References and further reading

6.1 References

Maps and aerial images

➔ Topographic base maps and aerial images (Source: Federal Office of Topography)

Literature

[1] Johannesson et al., 2009: The design of avalanche protection dams. Recent practical and theoretical developments. European Commission. Directorate General for Research, 2009.

[2] Rudolf-Miklau, F. and Sauermoser, S., 2011: Handbuch Technischer Lawinenschutz. Ernst & Sohn GmbH&Co.

[3] 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.

[4] Salm, B., 1993: Flow, flow transition and runout distances of flowing avalanches. In: Annals of Glaciology 18, 221-226.

[5] Bartelt, P.; Vera Valero, C.; Feistl, T.; Christen, M.; Bühler, Y.; Buser, O., 2015: Modelling cohesion in snow avalanche flow. Journal of Glaciology, 61, 229: 837-850. doi: 10.3189/2015JoG14J126

[6] Fischer, J.-T.; Kowalski, J.; Pudasaini, S.P., 2012: Topographic curvature effects in applied avalanche modeling. Cold Regions Science and Technology, Volumes 74–75, May 2012, Pages 21-30.

6.2 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. Visit a module-page, and check the *Publication*-tab at the bottom of the page.

7 Appendix

7.1 MuXi-Table

The following friction parameters (μ and ξ values) are used in RAMMS. Return period and volume category can be changed in ***Input → Global Parameters***.

Large avalanche (> 60'000 m³)		300-Year		100-Year		30-Year		10-Year	
	Altitude (m.a.s.l.)	μ	ξ	μ	ξ	μ	ξ	μ	ξ
unchannelled	above 1500	0.155	3000	0.165	3000	0.17	3000	0.18	3000
	1000 - 1500	0.17	2500	0.18	2500	0.19	2500	0.2	2500
	below 1000	0.19	2000	0.2	2000	0.21	2000	0.22	2000
channelled	above 1500	0.21	2000	0.22	2000	0.225	2000	0.235	2000
	1000 - 1500	0.22	1750	0.23	1750	0.24	1750	0.25	1750
	below 1000	0.24	1500	0.25	1500	0.26	1500	0.27	1500
gully	above 1500	0.27	1500	0.28	1500	0.29	1500	0.3	1500
	1000 - 1500	0.285	1350	0.3	1350	0.31	1350	0.325	1350
	below 1000	0.3	1200	0.315	1200	0.33	1200	0.345	1200
flat	above 1500	0.14	4000	0.15	4000	0.155	4000	0.16	4000
	1000 - 1500	0.15	3500	0.16	3500	0.17	3500	0.18	3500
	below 1000	0.17	3000	0.18	3000	0.19	3000	0.2	3000

Medium avalanche (25 - 60'000 m³)		300-Year		100-Year		30-Year		10-Year	
unchannelled	above 1500	0.195	2500	0.205	2500	0.215	2500	0.225	2500
	1000 - 1500	0.21	2100	0.22	2100	0.23	2100	0.24	2100
	below 1000	0.23	1750	0.24	1750	0.25	1750	0.26	1750
channelled	above 1500	0.25	1750	0.26	1750	0.27	1750	0.28	1750
	1000 - 1500	0.27	1530	0.28	1530	0.285	1530	0.295	1530
	below 1000	0.28	1350	0.29	1350	0.3	1350	0.31	1350
gully	above 1500	0.32	1350	0.33	1350	0.34	1350	0.35	1350
	1000 - 1500	0.33	1200	0.34	1200	0.355	1200	0.36	1200
	below 1000	0.36	1100	0.37	1100	0.38	1100	0.39	1100
flat	above 1500	0.17	3250	0.18	3250	0.19	3250	0.2	3250
	1000 - 1500	0.19	2900	0.2	2900	0.21	2900	0.22	2900
	below 1000	0.21	2500	0.22	2500	0.23	2500	0.24	2500

forested area (mu=delta, xi=fix)	0.02	400	0.02	400	0.02	400	0.02	400

RAMMS::Avalanche 1.4 Friction Parameters

Small avalanche (5 - 25'000 m³)		300-Year		100-Year		30-Year		10-Year	
	Altitude (m.a.s.l.)	μ	ξ	μ	ξ	μ	ξ	μ	ξ
unchannelled	above 1500	0.235	2000	0.245	2000	0.25	2000	0.26	2000
	1000 - 1500	0.25	1750	0.26	1750	0.265	1750	0.275	1750
	below 1000	0.265	1500	0.275	1500	0.285	1500	0.295	1500
channelled	above 1500	0.28	1500	0.29	1500	0.3	1500	0.31	1500
	1000 - 1500	0.3	1350	0.31	1350	0.315	1350	0.325	1350
	below 1000	0.31	1200	0.32	1200	0.33	1200	0.34	1200
gully	above 1500	0.37	1200	0.38	1200	0.39	1200	0.4	1200
	1000 - 1500	0.38	1100	0.39	1100	0.4	1100	0.41	1100
	below 1000	0.4	1000	0.41	1000	0.42	1000	0.43	1000
flat	above 1500	0.215	2500	0.225	2500	0.23	2500	0.24	2500
	1000 - 1500	0.23	2250	0.24	2250	0.245	2250	0.255	2250
	below 1000	0.245	2000	0.255	2000	0.26	2000	0.27	2000

Tiny avalanche (< 5'000 m³)		300-Year		100-Year		30-Year		10-Year	
unchannelled	above 1500	0.275	1500	0.28	1500	0.285	1500	0.29	1500
	1000 - 1500	0.29	1400	0.295	1400	0.3	1400	0.305	1400
	below 1000	0.3	1250	0.31	1250	0.32	1250	0.33	1250
channelled	above 1500	0.31	1250	0.32	1250	0.33	1250	0.34	1250
	1000 - 1500	0.33	1180	0.34	1180	0.345	1180	0.355	1180
	below 1000	0.34	1050	0.35	1050	0.36	1050	0.37	1050
gully	above 1500	0.42	1050	0.43	1050	0.44	1050	0.45	1050
	1000 - 1500	0.43	1000	0.44	1000	0.45	1000	0.46	1000
	below 1000	0.44	900	0.45	900	0.46	900	0.47	900
flat	above 1500	0.26	1750	0.265	1750	0.27	1750	0.275	1750
	1000 - 1500	0.27	1600	0.275	1600	0.28	1600	0.285	1600
	below 1000	0.28	1500	0.285	1500	0.29	1500	0.295	1500

forested area (mu=delta, xi=fix)	0.02	400	0.02	400	0.02	400	0.02	400

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