

Snow Avalanche Dynamics with RAMMS::Avalanche

RAMMS AG

October, 2026

Perry Bartelt & Marc Christen



Title picture: © WSL

Table of Content

1	Introduction.....	1
1.1	Motivation	2
1.2	RAMMS.....	2
1.2.1	RAMMS::Avalanche - Model Overview and Scope.....	3
1.3	Learning by doing	4
2	Installation and Setup	6
2.1	System requirements	6
2.1.1	GPU acceleration.....	6
2.2	Installation and Licensing	7
2.2.1	Multilingual Installation	7
2.2.2	New Update Procedure	8
2.2.3	Administrative Privileges Requirement	8
2.2.4	Licensing	8
2.3	First-Time Startup and License Registration.....	8
2.3.1	License Transfer.....	10
2.3.2	Module availability.....	10
2.3.3	Registering and updating Avalanche and Debrisflow licenses	11
3	The RAMMS::Avalanche GUI.....	12
3.1	Starting the Graphical Interface.....	12
3.2	Project Creation	13
3.2.1	Understanding the New Project Window	14
3.2.1	DEM Files *.tif and *.asc	17
4	Setup and Theory.....	18
4.1	Preparations	18
4.1.1	Topographic data - Digital Elevation Model (DEM)	18
4.1.2	Project and Scenarios.....	19
4.2	Release information.....	19
4.3	Forest information	20

4.4	Friction information.....	20
4.4.1	Physical friction model.....	20
4.4.2	Cohesion.....	21
4.4.3	Curvature.....	22
4.4.4	Model Assumptions and Known Limitations.....	22
4.4.5	Friction parameters μ and ξ	23
4.5	Automatic MuXi Procedure (μ , ξ).....	24
4.5.1	Define the global parameters.....	24
4.5.2	Define the altitude limit.....	25
4.5.3	Automatic terrain classification	26
4.5.4	Assignment of μ and ξ	27
4.5.5	Calculation parameters.....	28
4.6	Preferences.....	28
4.7	Working with the RAMMS GUI.....	30
4.7.1	Open input- and output-files	30
4.7.2	Visualizing shapefiles, MuXi-files and domain-files.....	30
4.7.3	Hillshade visualization.....	33
4.7.4	Changing maps and orthophotos (aerial images)	34
4.7.5	Moving, resizing, rotating, viewing	35
4.7.6	Colorbar	38
4.7.7	How to save input files and program settings	40
4.7.8	About RAMMS	42
4.8	Preparing a simulation	43
4.8.1	Release area(s).....	43
4.8.2	Potential Release Areas (PRA)	50
4.8.3	Calculation Domain	51
4.8.4	Friction parameters μ and ξ	55
4.9	Running a simulation	59
4.9.1	Run Batch Simulations	65
5	Results.....	67
5.1	Log Files and Project information	68
5.1.1	Output Log File	68
5.1.1	Input Log File	70

5.1.2	Project Region Information	70
5.2	Visualization and analysis of results.....	71
5.2.1	Maximum Results and Dump-Step Results	71
5.2.2	Visualize different parameters.....	72
5.2.3	Line profile and time plot	74
5.2.4	Deposition analysis	78
5.2.5	Google Earth Export	81
5.2.6	Exporting video, GIF animation or image files.....	85
5.3	Avalanche Impact Pressures	86
5.3.1	Swiss Pressure Guidelines for Hazard Mapping	86
5.3.2	Avalanche-Obstacle Interaction Forces	89
5.3.3	Impact load per unit width.....	91
5.3.4	Visualization of Impact Pressures	91
5.3.5	Pressure on Long Walls (Dams)	94
5.4	Stopping criteria.....	96
5.4.1	Momentum-based Stopping Criterion	96
5.4.2	Center-of-Mass Stopping Criterion	98
5.1	Modifying the DEM for subsequent simulations	101
5.1.1	Adding structures to the DEM.....	101
5.1.1	Adding deposited avalanche material to the DEM	105
5.1.2	Sequential multi-surge simulations using batch mode	107
6	References.....	111
	Publications.....	112
7	Appendix	113
7.1	MuXi-Table	113
7.2	Third-Party Software.....	116

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 was developed since 2010 by the RAMMS team at the WSL Institute for Snow and Avalanche Research SLF. However, since July 1st 2024, development, support and licenses are managed exclusively at RAMMG AG. 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 website <https://www.ramms.ch> provides useful information about all RAMMS modules, licenses, frequently asked questions (FAQ), recent software updates or publication lists.

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.

Snow avalanches pose a major risk to infrastructure and settlements in alpine regions. Physically based 3D simulation tools like RAMMS::Avalanche enable engineers to quantify runout extent and impact pressure that historical records alone cannot provide, especially for rare or extreme events.

1.2 RAMMS

The RAMMS (**RA**pid **MA**ss **M**ovements **S**imulation) software system contains five process modules:

- [RAMMS::AVALANCHE](#)
- [RAMMS::EXTENDED](#)
- [RAMMS::DEBRISFLOW](#)
- [RAMMS::ROCKFALL](#)
- [RAMMS::ROCKICE](#) (Beta)

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

CHAPTER 1: INTRODUCTION

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.

1.2.1 RAMMS::Avalanche - Model Overview and Scope

RAMMS::Avalanche is the three-dimensional successor to the Swiss AVAL-1D one-dimensional run-out model (Christen, M. et al. 2002). Both tools share the Voellmy-Salm depth-averaged friction framework, inheriting a parameterization tradition calibrated over several decades against documented Alpine avalanche events (Salm, Burkard & Gubler, 1990). The extension from one-dimensional to three-dimensional terrain-following computation was described in Christen, Kowalski & Bartelt (2008), and Christen, Kowalski & Bartelt (2010, Cold Reg. Sci. Tech. 63), referred to here as ChristenKB2010, which remains the primary peer-reviewed reference for the **RAMMS::Avalanche** modelling framework.

RAMMS::Avalanche simulates dense-flow snow avalanches using the Voellmy-Salm friction model, extended with a cohesion term (Bartelt et al., 2015 [5]) and a curvature correction (Fischer, Kowalski & Pudasaini, 2012 [6]). The model solves depth-averaged continuity and momentum equations on a three-dimensional terrain surface.

Note on **RAMMS::Extended**: **RAMMS::Avalanche** is distinct from **RAMMS::Extended**, which handles mixed and powder-snow avalanche scenarios using the Random Kinetic Energy (RKE) framework. RKE governs granular temperature, dispersive pressure, and density changes relevant to large, fast events with a powder cloud component. Snow entrainment is also part of **RAMMS::Extended**. Users dealing with events where a significant powder cloud is expected, or where the flowing mass substantially increases through snow pick-up during runout, should use **RAMMS::Extended**.

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.

DISCLAIMER

The RAMMS software models are provided "as is" without any warranties, express or implied. RAMMS AG makes no representations or warranties of any kind, including but not limited to the accuracy, reliability, completeness, or suitability of the models for any particular purpose.

Users of the RAMMS software models assume all risk associated with the use of these models. RAMMS AG shall not be held liable for any loss, injury, or damage resulting from the use or reliance on these models, including but not limited to personal injury, property damage, or financial loss.

The RAMMS software models are intended to supplement, not replace, the judgment and expertise of professionals in the fields of geology, engineering, and hazard management. Users should not rely solely on the models for making decisions related to hazard mitigation, land use planning, or emergency response.

In no event shall RAMMS AG be liable for any indirect, incidental, special, consequential, or punitive damages arising out of or in connection with the use or performance of the RAMMS software models, even if advised of the possibility of such damages.

Users are responsible for ensuring that their use of the RAMMS software models complies with all applicable laws, regulations, and standards. RAMMS AG does not guarantee that the models meet any regulatory requirements or industry standards.

RAMMS AG may, at its sole discretion, update, modify, or discontinue the RAMMS software models at any time without notice. Users are responsible for ensuring that they are using the most current version of the models.

By using the RAMMS software models, you acknowledge that you have read, understood, and agree to be bound by the terms of this disclaimer.

Swiss law applies. Court of jurisdiction is Davos. If you encounter problems, please contact info@ramms.ch.

2 Installation and Setup

Before installation, please make sure that you have administrative privileges for the computer where you intend to install RAMMS.

2.1 System requirements

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

- Operating Systems: Windows 10/11 (64-bit)
- RAM (memory): 8 GB (more recommended)
- CPU: > 1 GHz, 8 cores or more recommended
- Disk space: ca. 220 MB needed for the software
- Graphic Card: GPU and OpenGL support recommended (see below)

2.1.1 GPU acceleration

RAMMS can offload the main calculation to a compatible NVIDIA graphics card, which substantially speeds up larger simulations.

Requirements

- An NVIDIA GPU based on one of the following architectures:
 - **Pascal:** GeForce GTX 10-series, Quadro P-series
 - **Turing:** GeForce GTX 16-series / RTX 20-series, Quadro RTX/T-series
 - **Ampere:** GeForce RTX 30-series, RTX A-series
 - **Ada Lovelace:** GeForce RTX 40-series, RTX 2000/3500/4000/4500/5000/6000 Ada Generation
 - **Blackwell:** GeForce RTX 50-series, RTX PRO Blackwell series (desktop, laptop, and professional GPUs all supported within these architectures.) GPUs older than Pascal — e.g. Maxwell-based cards such as the Quadro M-series — are not supported.
- A current NVIDIA driver (keep it updated via NVIDIA's own driver update tool).
- Enough free GPU memory (VRAM) to hold the simulation mesh — this is checked automatically at startup.

How it works

By default, RAMMS decides automatically: if a suitable GPU is found and the simulation is large enough to benefit, it runs on the GPU; otherwise it runs on

CHAPTER 2: INSTALLATION AND SETUP

the CPU as before. Nothing needs to be configured for this. To override the automatic choice, and force the CPU or GPU solution, use the following flags in the Additional Preferences:

SOLVER CPU

SOLVER GPU

What to expect

GPU and CPU produce the same result to within ordinary floating-point rounding. A CPU run always reproduces bit-for-bit the same output as before this feature was added.

Speedup depends on simulation size: small simulations may see little or no benefit (and may run on the CPU automatically for this reason), while larger simulations have measured speedups in the range of roughly 2–5×.

2.2 Installation and Licensing

Please download the RAMMS::Avalanche setup file "[ramms_avaldbf_setup.exe](https://www.ramms.ch/ramms-avalanche/ramms_avaldbf_setup.exe)" from <https://www.ramms.ch/ramms-avalanche/> (Downloads at the bottom of page).

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_avaldbf_setup.exe` starts after clicking Yes.
- **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.
- **Start the file "`ramms_avaldbf_setup.exe`".** If you need to do a **silent** installation, add `-silent` when starting the exe-file.

2.2.1 Multilingual Installation

The RAMMS::Avalanche installer is designed to support multiple languages, facilitating a user-friendly installation experience for a diverse user base. Upon launching the installer, users can select their preferred language, which will guide them through the installation process in that language.

CHAPTER 2: INSTALLATION AND SETUP

2.2.2 New Update Procedure

RAMMS::Avalanche incorporates a streamlined update mechanism to ensure users have access to the latest features and improvements.

- **Web Update:** The application can automatically check for available updates online (**Help → Check for Updates**). If an update is found, users can download and install it directly through the application interface.

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 again.

2.2.3 Administrative Privileges Requirement

To install or update RAMMS::Avalanche, administrative privileges are required. This is necessary because the installation process involves modifying system configurations and installing essential components, such as the IDL Visual Studio Merge Modules. Without administrative rights, the installer cannot make these critical changes, and the installation or update will fail.

It's important to note that while administrative privileges are required for installation and updates, they are not needed to run RAMMS::Extended after installation is complete.

2.2.4 Licensing

RAMMS uses a personal-use licensing model. Licenses are time-limited and bound to a single machine via its unique host ID. To obtain a license, generate a license request file from the target machine and send it to RAMMS AG. You will receive a license key that, once entered, activates full functionality on that specific system.

Alternatively, a license can be installed on a Windows Server and accessed via Remote Desktop Connection (RDC). In this setup, the license can be used by different users, but only one user per module can access it at a time.

2.3 First-Time Startup and License Registration

Double-click the RAMMS icon or use **Start → Programs → RAMMS → Run RAMMS** to display the initial splash screen (image below). This screen confirms that the RAMMS application has loaded successfully and is ready to start.

CHAPTER 2: INSTALLATION AND SETUP

Please note that this splash screen is part of the IDL Virtual Machine startup procedure. RAMMS will not continue to the main interface until the splash screen has been clicked. To continue, click once anywhere on the splash screen image:

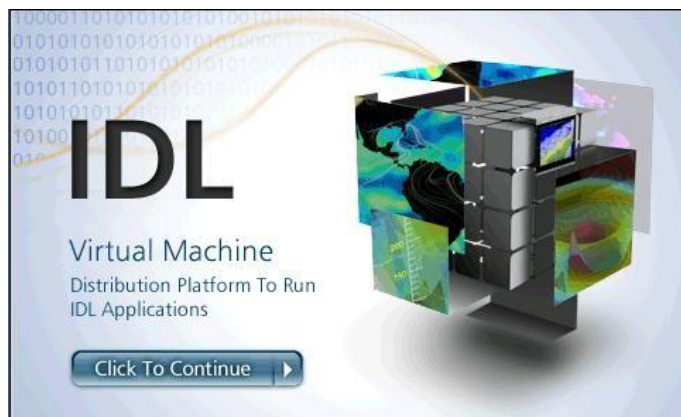


Figure 2-1: IDL Virtual Machine splash screen.

You will then see the *RAMMS / Licensing* window (Figure 2.1):

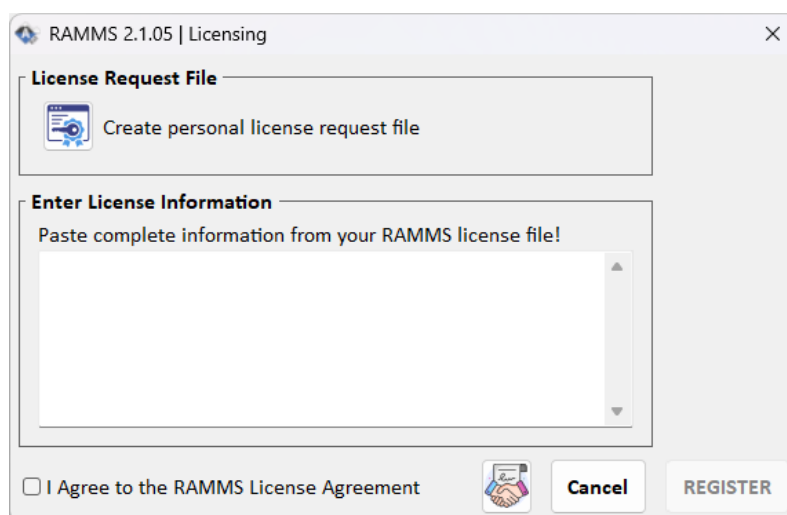


Figure 2-2: Enter the **full contents** of your license file in this dialog box.

Paste the **full contents** of your RAMMS license file into the designated field. Check the box to confirm **I Agree to the RAMMS License Agreement**, then click **REGISTER**. To review the license agreement before registering, click the handshake icon.

Create your personal license request file by clicking the top button, and follow the instructions.

CHAPTER 2: INSTALLATION AND SETUP

2.3.1 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**

2.3.2 Module availability

RAMMS modules can only be used by one user at a time.

If a module is already in use, RAMMS shows the name of the user currently blocking the module.

A module may also remain reserved while a long simulation is running.

If RAMMS is not closed properly, the reservation may remain active for a limited time and is released automatically after the timeout period.

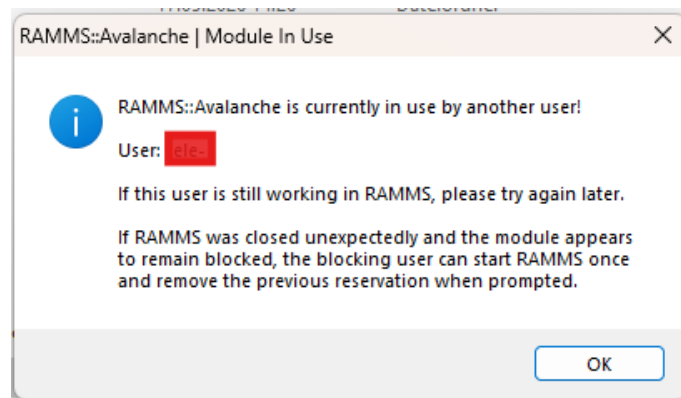


Figure 2-3: RAMMS showing the name of the user currently blocking the module.

2.3.3 Registering and updating Avalanche and Debrisflow licenses

Licenses for **RAMMS::Avalanche** and **RAMMS::Debrisflow** can be registered or updated directly from within either of these modules.

This can be used in two situations:

- **Adding another module:** If, for example, RAMMS::Avalanche is already registered, a RAMMS::Debrisflow license can be added.
- **Updating an existing license:** A new license for the same module can be registered, for example to replace a temporary license with a final license. The existing license information for that module is overwritten.

To register or update a license, select:

Help → Register / Update License

and enter the new license key (see Figure 2-2).

After registering an additional module, close and restart RAMMS. If both RAMMS::Avalanche and RAMMS::Debrisflow are licensed, the **module selection window** is displayed at startup, allowing you to select the module you want to use.



Figure 2-4: Module selection window when both RAMMS::Avalanche (left) and RAMMS::Debrisflow (right) are licensed.

Note: This function is available only for **RAMMS::Avalanche** and **RAMMS::Debrisflow**. Other RAMMS modules, such as **RAMMS::Rockfall**, must be installed and licensed separately.

3 The RAMMS::Avalanche GUI

RAMMS uses 3D and 2D views. Release areas, friction zones, and domain boundaries can only be defined in 2D mode. Each project holds multiple scenarios sharing the same DEM, allowing systematic parameter variation without recreating the topographic base.

3.1 Starting the Graphical Interface

When the user clicks the **RAMMS::Avalanche** icon, the **Graphical User Interface (GUI)** appears,

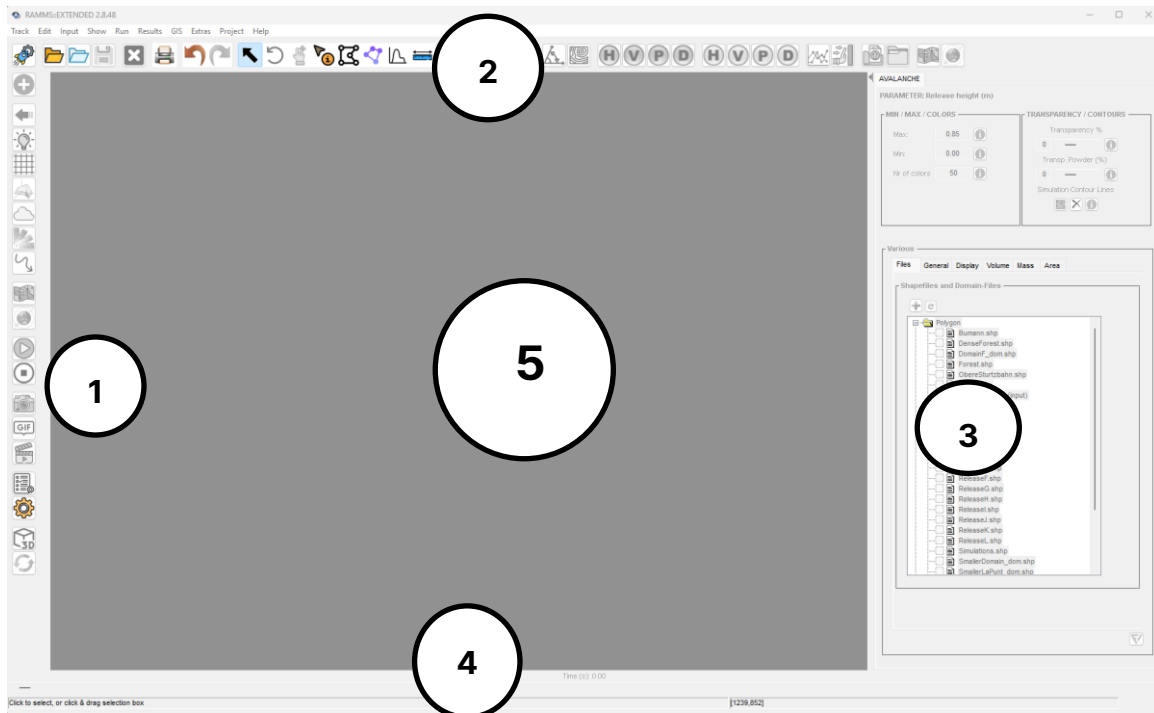


Figure 3-1: RAMMS::Avalanche GUI at startup. The interface is structured into five main regions.

providing a structured and user-friendly workspace. The interface is divided into five distinct regions, see Figure above:

1. **Display, Animation and Preference Icons (Left Side):** A vertical list of icons for quick access to display features.
2. **Action Toolbar (Top Side):** A horizontal row of icons, including key functions for project management and data input/output handling.
3. **Tab Field (Right Side):** A dynamic space for working with multiple tabs, used for file management, user information and display options.
4. **Time Slider & Information Displays (Bottom Side):** A timeline control and key important user information.

CHAPTER 3: RAMMS::AVALANCHE GUI

5. **Input/Output Graphics Display:** In this window the input/output graphics is displayed in two-dimensional, or full three-dimensional mode.

At this point, users have **three primary options**, conveniently located in the upper action toolbar:

1. **Create a New Project** – Begin a fresh project from scratch.
2. **Open an Input File** – Load an existing input dataset for processing.
3. **Open an Output File** – Access previously generated results.

3.2 Project Creation

	Create Project	Creates a project, selects a DEM
	Open Input File	Opens an existing input file
	Open Output File	Opens an existing output file

Every **RAMMS::Avalanche** workflow begins with the **creation of a project**. A project is inherently linked to a specific terrain model, which is defined by a **Digital Elevation Model (DEM)**. To establish a new project, users must consider three essential elements:

1. **Project Name:** A unique identifier for the project.
2. **Project Location, or Home Directory:** The directory on the user's computer where project files will be **stored**.
3. **Digital Elevation Model (DEM):** A topographical dataset that forms the foundation of the simulation.

CHAPTER 3: RAMMS::AVALANCHE GUI

Initiating a New Project

To create a **new project**, click the "**Create Project**" icon, located on the upper toolbar. This icon is easily recognizable, featuring a rocket ship in front of a cogwheel. Upon selecting this option, the **Create Project Wizard** will appear.

The user can also launch the **Create Project Wizard** via the pull-down menu by navigating to **Track → New → Project Wizard**.

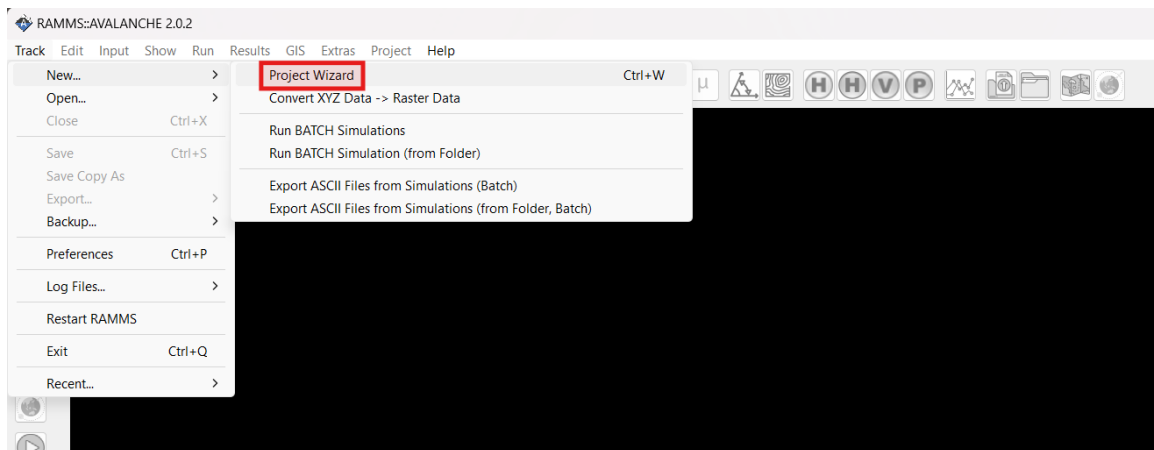


Figure 3-2: Activating the project wizard via pull-down menus.

3.2.1 Understanding the New Project Window

The **Create Project** interface is divided into three distinct sections (see Figure 3-3), each serving a crucial role in the project setup process:

1. Project Name & Home Directory (Upper Section)

- Users must define a project name and select a home directory where all simulation files will be stored.

2. Project Description (Middle Section)

- This section allows users to input descriptive text about the project.
- While often overlooked, this feature can be highly valuable—especially when revisiting simulations years later. Documenting important details ensures continuity and understanding for future analysis.

3. DEM File Selection (Lower Section)

- Here, users import the DEM file, which defines the terrain for the simulation.
- Once selected, **RAMMS::Avalanche** automatically analyzes and reports the DEM resolution, typically 2m or 5m. The **Grid Resolution** tab in the **DEM Input** section allows users to adjust the resolution of a DEM to better suit their simulation needs. High-resolution DEMs **are often specified, but their level of detail may be excessive for avalanche dynamics simulations**, leading to unnecessary computational complexity.

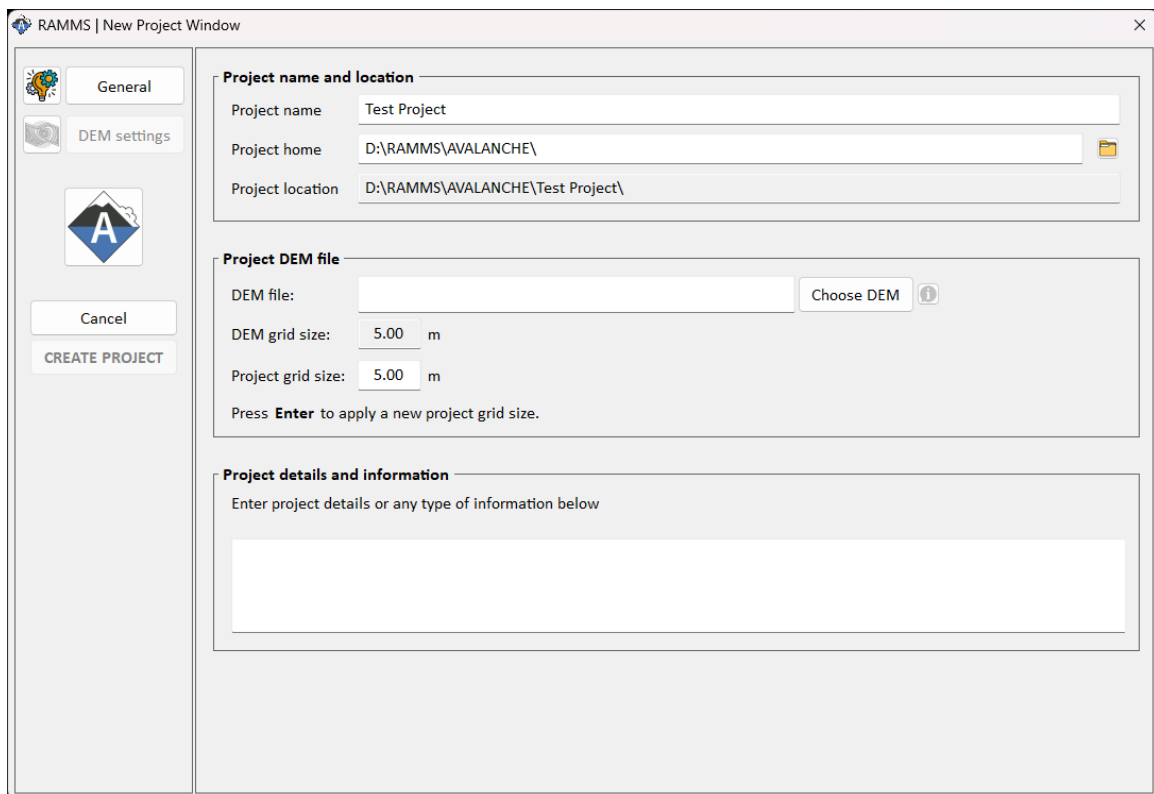


Figure 3-3: The RAMMS::Avalanche New Project Window. The menu is divided into three sections. In the upper section a project name and home directory is defined. In the lower section the DEM is specified by the user. In the middle section project information ca

To modify the **DEM resolution**, simply click on "**Grid Resolution**", enter the desired resolution, and apply the change. For instance, if the original DEM has a resolution of **2m**, it can be resampled to a coarser resolution, such as **5 meters**, using **bilinear interpolation**. Similarly, if the DEM resolution is too coarse (e.g., **30 meters**), it can be refined to a finer resolution (e.g., **5 meters**) using the same functionality. This flexibility ensures that the DEM is appropriately scaled for efficient and accurate simulations.

By carefully setting up a project with a clear name, an appropriate directory, and an accurate DEM, users establish a solid foundation for successful simulations in **RAMMS::Avalanche**.

CHAPTER 3: RAMMS::AVALANCHE GUI

The **DEM Settings** menu is accessed from the **Create Project** menu (see Figure below) and serves two main purposes: **providing key spatial information**—such as the maximum **North, South, East, and West** coordinates—and **allowing users to clip the DEM** if needed.

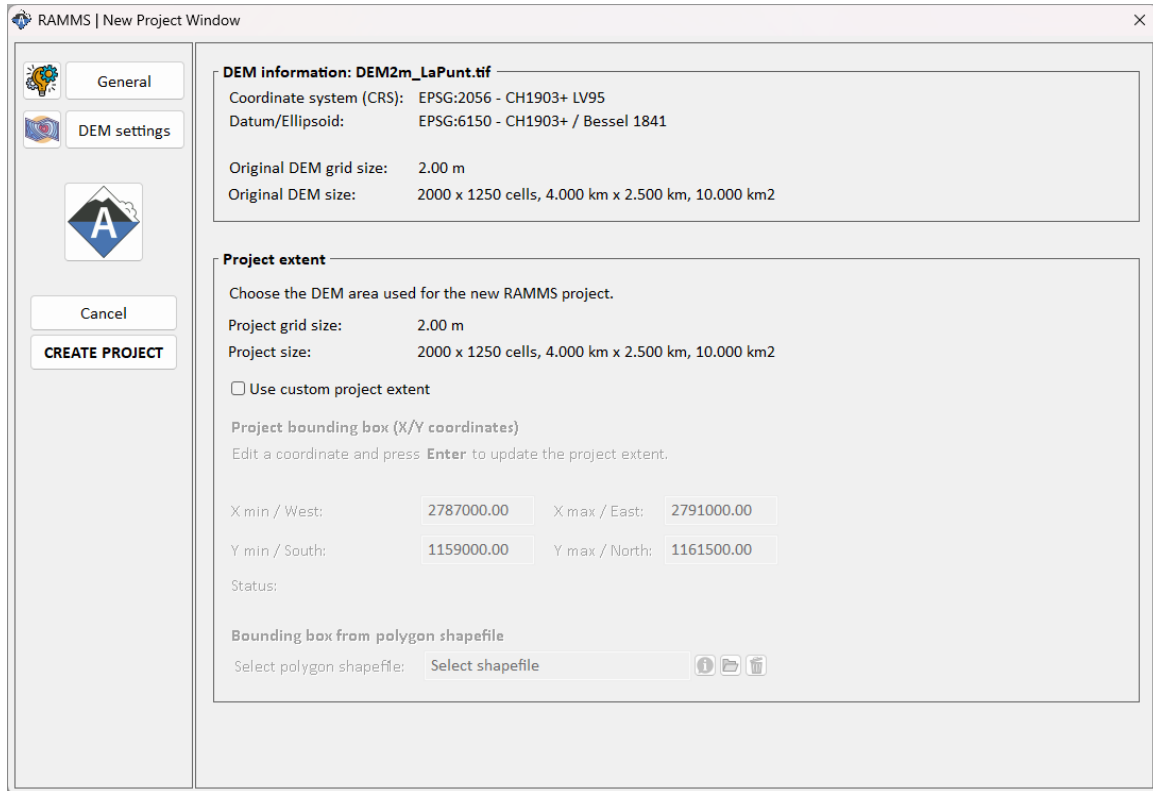


Figure 3-4: The DEM Settings menu. With this menu, the DEM can be reduced by clipping, manually or by using a polygon shapefile.

Clipping the DEM can be done in one of two ways:

1. **Manual Coordinate Entry** – Users can manually define the clipping boundaries by entering exact **N, S, E, and W** coordinates. This is done by activating the corresponding icons in the **DEM Settings** menu, after which the GUI prompts the user to input the desired values.
2. **Shapefile-Based Clipping** – Alternatively, users can clip the DEM using a **polygon shapefile**. This method is often preferred, particularly when refining the computational domain from existing simulations and when a suitable domain file from a previous calculation is available (because the project is being created, often, a domain file does not exist.)

By offering these two clipping options, the **DEM Settings** menu provides flexibility in tailoring the DEM to the specific needs of the project.

3.2.1 DEM Files *.tif and *.asc

RAMMS::Avalanche supports **two different types of file formats for DEM files**: *.asc and *.tif. The **ASC format**, an ASCII Grid format developed by Esri, is a plain text file that stores elevation data in a structured grid with human-readable values, making it easy to edit but often larger in size. It includes metadata such as cell size, coordinates, and NoData values. On the other hand, the **TIFF format**, specifically GeoTIFF (*.tif), is a raster format that supports **georeferencing** and can store elevation data in a more compact, binary form. GeoTIFFs are widely used due to their efficiency, ability to store multiple bands, and compatibility with GIS software. While ASC files are simpler and useful for certain applications, GeoTIFFs are generally preferred for large-scale DEM storage and processing due to their superior compression, metadata handling, and performance.

4 Setup and Theory

A complete RAMMS::Avalanche setup requires a georeferenced DEM (5m recommended), a defined release area with release depth, friction parameters (μ , ξ), and a calculation domain. Validate all inputs visually in 2D before running — incorrect release geometry is the most common source of unrealistic results.

4.1 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 must 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.

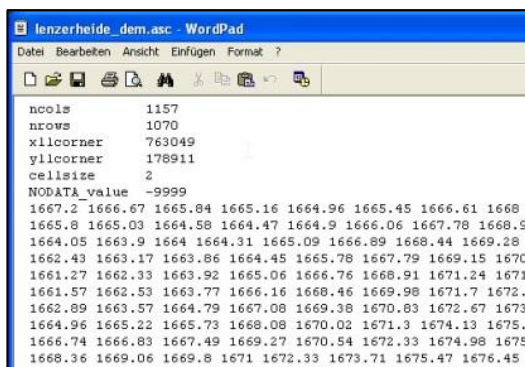
4.1.1 Topographic data - Digital Elevation Model (DEM)

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 (**Error! Reference source not found.**)
2. GEOTIFF (georeferenced information embedded within a TIFF file)
3. ASCII XYZ regular, single space data (**Error! Reference source not found.**)

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*.

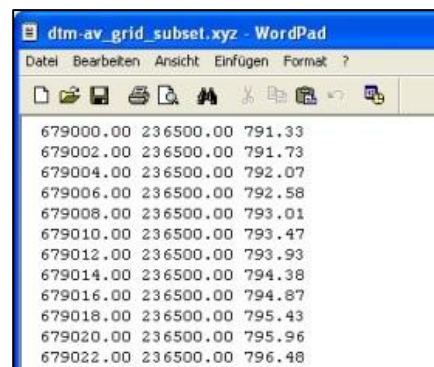
The header of an ESRI ASCII grid must contain the information shown below in **Error! Reference source not found.**



```
lenzerheide_dem.asc - WordPad
Datei Bearbeiten Ansicht Einfügen Format ?

ncols      1157
nrow       1070
xllcorner  763049
yllcorner  178911
cellsize   2
NODATA_value -9999
1667.2 1666.67 1665.84 1665.16 1664.96 1665.45 1666.61 1668
1665.8 1665.03 1664.58 1664.47 1664.9 1666.06 1667.78 1668.9
1664.05 1663.9 1664 1664.31 1665.09 1666.89 1668.44 1669.28
1662.43 1663.17 1663.86 1664.45 1665.78 1667.79 1669.15 1670
1661.27 1662.33 1663.92 1665.06 1666.76 1668.91 1671.24 1671
1661.57 1662.53 1663.77 1666.16 1668.46 1669.98 1671.7 1672.
1662.89 1663.57 1664.79 1667.08 1669.38 1670.83 1672.67 1673
1664.96 1665.22 1665.73 1668.08 1670.02 1671.3 1674.13 1675.
1666.74 1666.83 1667.49 1669.27 1670.54 1672.33 1674.98 1675
1668.36 1669.06 1669.8 1671 1672.33 1673.71 1675.47 1676.45
```

Figure 4-1: Example ESRI ASCII grid.



```
dtm-av_grid_subset.xyz - WordPad
Datei Bearbeiten Ansicht Einfügen Format ?

679000.00 236500.00 791.33
679002.00 236500.00 791.73
679004.00 236500.00 792.07
679006.00 236500.00 792.58
679008.00 236500.00 793.01
679010.00 236500.00 793.47
679012.00 236500.00 793.93
679014.00 236500.00 794.38
679016.00 236500.00 794.87
679018.00 236500.00 795.43
679020.00 236500.00 795.96
679022.00 236500.00 796.48
```

Figure 4-2: Example ASCII XYZ single space data.

CHAPTER 4: RESULTS

4.1.2 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.

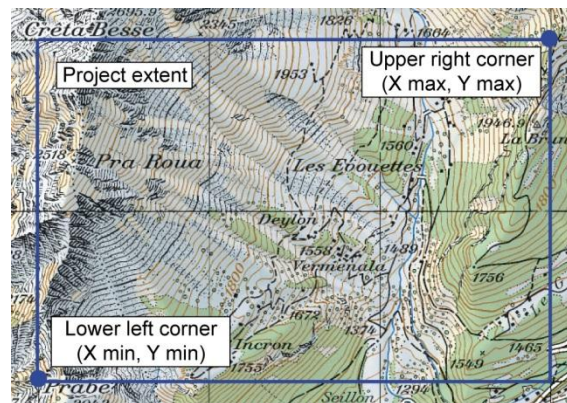


Figure 4-3: The same project extent (area of interest) can be used to calculate different scenarios with different input parameters.

4.2 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.

Estimating Release Depth and Volume

Release depth is one of the two most sensitive inputs to a **RAMMS::Avalanche** simulation, alongside the friction parameter μ and ξ . The statement 'use reference information' requires practical elaboration. The following steps reflect Swiss engineering practice:

Terrain-based empirical estimates: Swiss engineering practice (Salm, Burkard & Gubler, 1990 [3]) provides empirical formulae for release depth as a function of return period, elevation, and terrain aspect. These are the appropriate starting values for sites without documented events and are the basis of the MuXi table return period categories.

CHAPTER 4: RESULTS

Release area delineation: The polygon used as a release area should be based on field-mapped terrain features: documented avalanche crowns, tension fractures, and mapped starting zones from avalanche cadastres. Maggioni et al. (2002, 2003) document the relationship between terrain parameters (slope angle, aspect, curvature) and release area extent, providing a literature basis for terrain-based release zone identification.

RAMMS provides a new **PRA** (Potential Release Area) feature, which helps to delineate release areas (see section 4.8.2).

Users can use any polygon shapefile as a release area, see section 4.8.1. 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.

4.3 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 4.8.4). Forest information should be defined before running the automatic muxi procedure.

4.4 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]).

4.4.1 Physical friction model

The friction law was originally described by Voellmy (1955), who divided snow resistance into a dry-Coulomb term proportional to the normal force (coefficient μ) and a turbulent term proportional to the square of the flow velocity (coefficient ξ). The model was formalized for engineering application by Salm (1993) [4] and provides the theoretical basis for the AVAL-1D model and for RAMMS::Avalanche as described in Christen, Kowalski & Bartelt (2008). The depth-averaged numerical scheme used to solve the governing equations on three-dimensional terrain is described in Sartoris & Bartelt (2000) and Bartelt (1999): an explicit, terrain-following Godunov-type finite difference approach on a structured grid. [Reference: Voellmy, A. (1955) *Über die Zerstörungskraft von Lawinen*. Schweizerische Bauzeitung 73; Sartoris, G. & Bartelt, P. (2000) *Upwinded finite difference schemes for dense snow avalanche modelling*. Commun. Numer. Meth. Engng. 16.]

CHAPTER 4: RESULTS

The frictional resistance S (Pa) is then

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

where ρ is the density, g the gravitational acceleration, ϕ the slope angle, h the flow height and $\mathbf{u}$ 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, $\rho h g \cos(\phi)$, 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 per grid cell within 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 “How to run a calculation”)

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.

4.4.2 Cohesion

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 ($\mu = \text{constant}$), see Figure 4-4. 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

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

where N_0 is the yield stress of the flowing material. Unlike a standard Mohr-Coulomb type relation this formula ensures that $S \rightarrow 0$ when both $N \rightarrow 0$ and $U \rightarrow 0$. It increases the shear stress and therefore causes the avalanche to stop earlier, depending on the value of N_0 .

CHAPTER 4: RESULTS

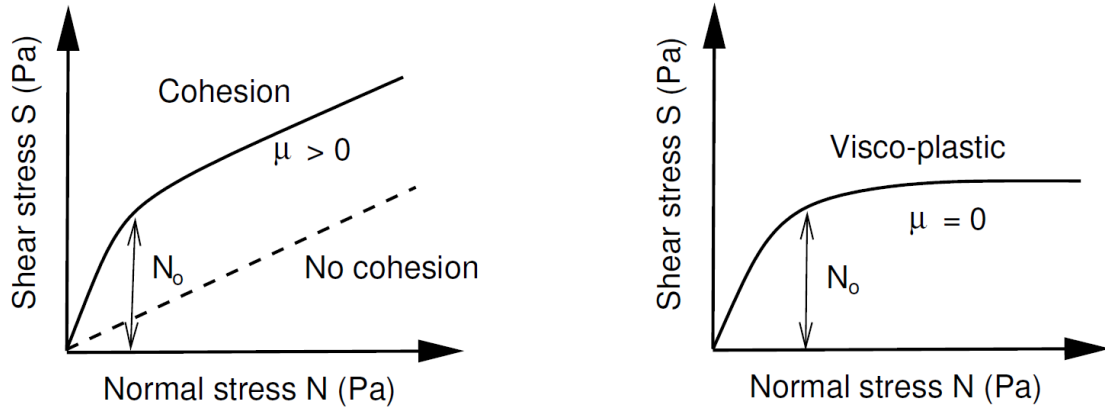


Figure 4-4: Relation between normal and shear stress. Left: Yield stress N_0 serves to increase the shear stress for higher normal pressures. At low normal pressures (small flow heights) the shear stress increases rapidly from $S=0$ to $S=N_0$. The slope of the 'S vs N' relation remains μ , when the normal pressures are large. Right: If $\mu=0$, we have a visco-plastic behaviour.

4.4.3 Curvature

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

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

The matrix $\mathbf{K}$ describes the track curvature in all directions, including the track "twist". The centrifugal force is then

$$F = \rho h f$$

which is added to the normal force N . Typically this increases the friction, causing the avalanche to slow down in tortuous and twisted flow paths. It can change the location of the deposition once the flow leaves the gully. Curvature may be activated/deactivated in the *Run Simulation* window (tab *Params*) or via the menu 'Help → Advanced... → Curvature'.

4.4.4 Model Assumptions and Known Limitations

The depth-averaged continuum approach of RAMMS::Avalanche has well-documented limitations that practitioners should be aware of when applying the model for engineering assessments:

Depth-averaged flow (no vertical velocity profile): RAMMS::Avalanche cannot resolve impact pressure distribution with height - only mean depth-averaged pressure is computed. This is relevant for structural design of snow sheds and walls where the pressure varies significantly with height above the ground.

CHAPTER 4: RESULTS

Fixed terrain (no deposition feedback between phases): Deposit from a first simulation phase is not automatically included in the terrain for a second phase without explicit DEM modification. Sequential avalanche events on the same path require manual terrain updating between simulations. This can also be done with batch simulations, see section 5.1.2.

Dense-flow module only: RAMMS::Avalanche does not simulate powder-cloud dynamics. For events where a significant powder cloud is expected to dominate hazard, RAMMS::Extended should be used instead.

4.4.5 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. QGIS).

The creation of a new MuXi-file is explained in the section below, and demonstrated in the exercise 3.5e *“How to create a new MuXi-file”* in section 4.8.4.

4.5 Automatic MuXi Procedure (μ , ξ)

The **Automatic MuXi Procedure** generates spatially distributed friction parameters μ and ξ for a RAMMS::Avalanche simulation. The assigned friction values depend on several characteristics of the avalanche and the terrain:

- terrain classification,
- elevation,
- forest cover,
- avalanche return period, and
- avalanche volume.

RAMMS combines this information to assign appropriate μ and ξ values to the individual computational grid cells. Before starting the procedure, the avalanche release area and release volume should be defined. Forest areas should also be delineated where applicable, since different friction parameters are used in forested terrain.

4.5.1 Define the global parameters

When the Automatic MuXi Procedure is started, RAMMS first displays the **Global Parameters** dialog (Figure 4-5). Here, the **return period** and **avalanche volume category** used for the friction parameter selection are specified. The available return periods are 10, 30, 100, and 300 years. The avalanche volume is classified as Tiny, Small, Medium, or Large.

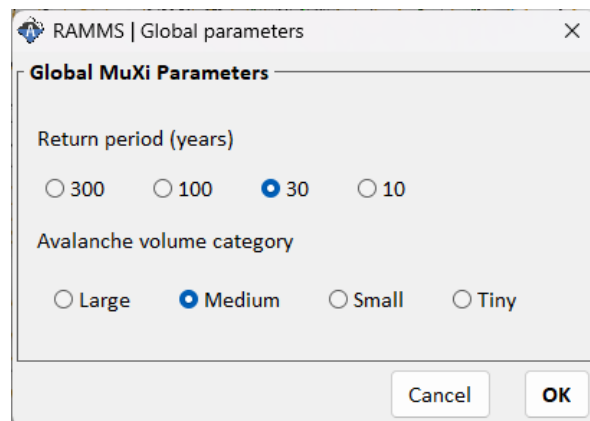


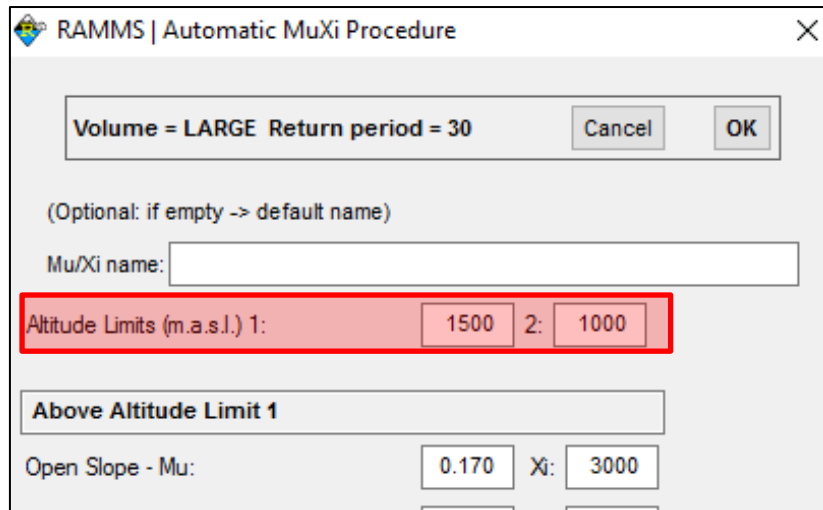
Figure 4-5: RAMMS global parameters.

The μ and ξ values depend strongly on both parameters. The selected avalanche volume category should therefore be consistent with the release volume defined for the simulation.

CHAPTER 4: RESULTS

4.5.2 Define the altitude limit

After confirming the global parameters, the actual **Automatic MuXi Procedure** dialog appears. There, the altitude limits used by the Automatic MuXi Procedure are specified (Figure 4-6). These limits account for the variation of the friction parameters μ and ξ with elevation and divide the terrain into three elevation ranges.



RAMMS | Automatic MuXi Procedure

Volume = LARGE Return period = 30 Cancel OK

(Optional: if empty -> default name)

Mu/Xi name:

Altitude Limits (m.a.s.l.) 1: 2:

Above Altitude Limit 1

Open Slope - Mu: Xi:

Figure 4-6: Altitude limits for automatic MuXi procedure

For projects in the Alps, the default values are **1500 m a.s.l.** for Altitude Limit 1 and **1000 m a.s.l.** for Altitude Limit 2.

The following reference elevations can be used as guidance:

- **Upper altitude limit (Alt. Limit 1):** should be selected below the regional **tree line**; the tree line of the project region should therefore lie **above** this altitude.
- **Lower altitude limit (Alt. Limit 2):** should approximately correspond to the regional **snow line**.

For projects outside the Alps, the altitude limits should be adapted to the local climatic and vegetation conditions.

CHAPTER 4: RESULTS

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

The values in Table 1 are intended as general guidance. Appropriate limits should be selected according to the local conditions of the project area.

4.5.3 Automatic terrain classification

RAMMS then analyzes the terrain based on the digital elevation model. Local slope and terrain curvature are evaluated and each computational grid cell is assigned to one of four terrain classes: **flat, open slope, channeled, or gully**.

CHAPTER 4: RESULTS

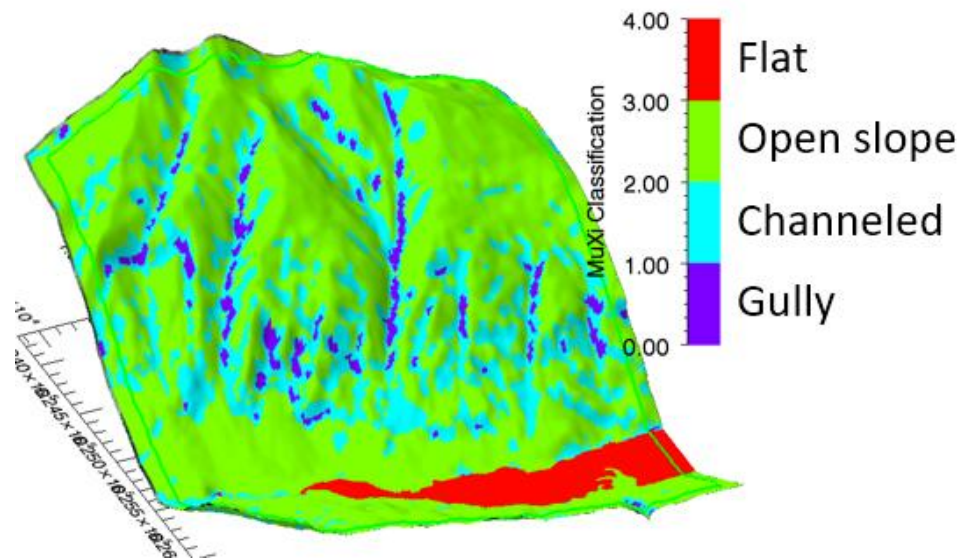


Figure 4-7: Example of the automatic terrain classification used by the MuXi procedure. RAMMS distinguishes between flat terrain, open slopes, channeled terrain, and gullies.

4.5.4 Assignment of μ and ξ

Based on the terrain classification, RAMMS assigns μ and ξ values to the computational grid cells. The assigned friction parameters depend on the **terrain class, elevation range, forest information, return period, and avalanche volume category**.

Figure 4-8 illustrates the principle of this assignment. For each grid cell, RAMMS combines the terrain classification and elevation with the selected global parameters and the available forest information to determine the corresponding μ and ξ values from the MuXi parameterization.

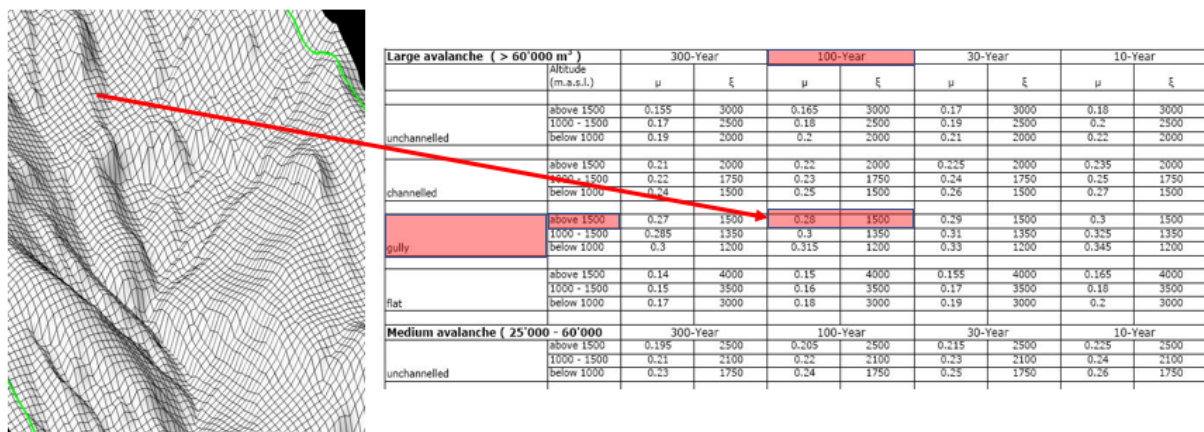


Figure 4-8: Automatic assignment of μ and ξ values based on terrain classification, elevation, forest information, return period, and avalanche volume.

CHAPTER 4: RESULTS

4.5.5 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*.

4.6 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 4-9 and Figure 4-10 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**.

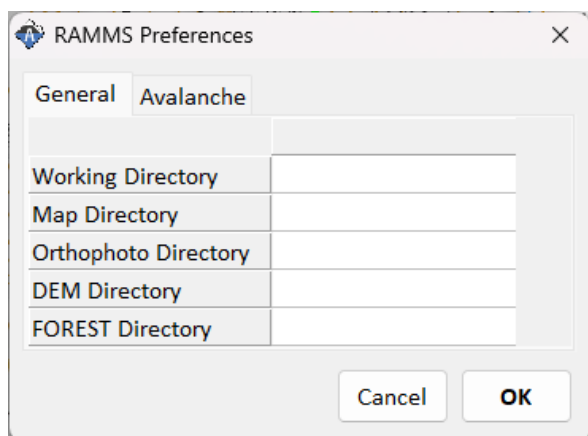


Figure 4-9: General tab of RAMMS preferences.

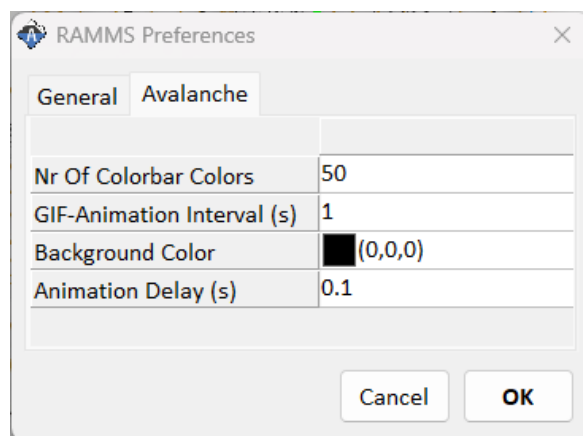


Figure 4-10: Avalanche tab of RAMMS preferences.

General Tab

Setting	Purpose
Working Directory	Set your working directory.
Map Directory	Set the folder where you place your georeferenced digital maps (consists of a .tif file and a corresponding .tfw file (world-file)).
Orthophoto Directory	Set the folder where you place your digital georeferenced orthophotos (aerial picture, consists of a .tif file and a corresponding .tfw file (world-file)).
DEM Directory	Set the folder where you place the Digital Elevation Models
FOREST directory	Set the folder where you place your forest-files (formats: ASCII grid or polygon shapefile).

CHAPTER 4: RESULTS

Avalanche Tab

Setting	Purpose
Nr of colorbar colors	Set default number 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.

- 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.

4.7 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.

4.7.1 Open input- and output-files

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

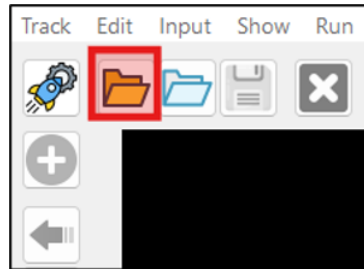


Figure 4-11: Toolbar button to open an input file. You can also use the menu **Track → Open... → Input File**.

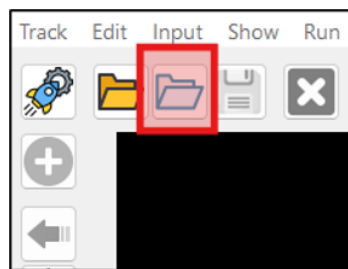


Figure 4-12: Toolbar button to open an output file. You can also use the menu **Track → Open... → Avalanche Simulation**.

Recent-Menu

The menu **Track → Recent...** allows you to directly open your 30 last accessed files (input and output), without having to search for them.

4.7.2 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.

CHAPTER 4: RESULTS

- In the file tree below, you will see your available project files (polygon shapefiles, MuXi ASCII files and domain shapefiles).

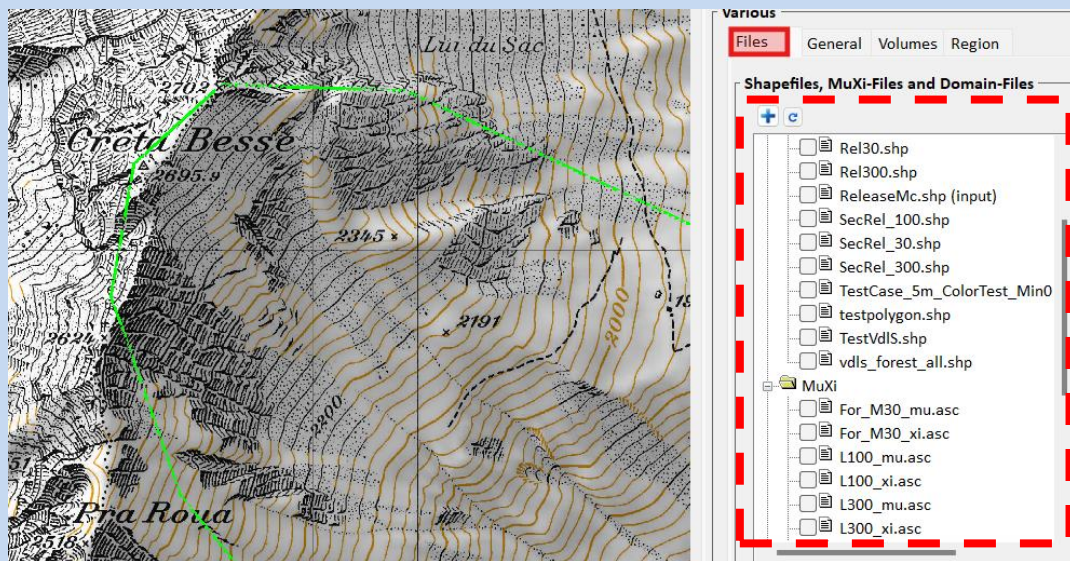


Figure 4-13: Files tab and available project files (file-tree, dashed red).
With the **blue + button**, files from external directories can be added to the file-tree. Refresh the tree with the refresh-button.

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

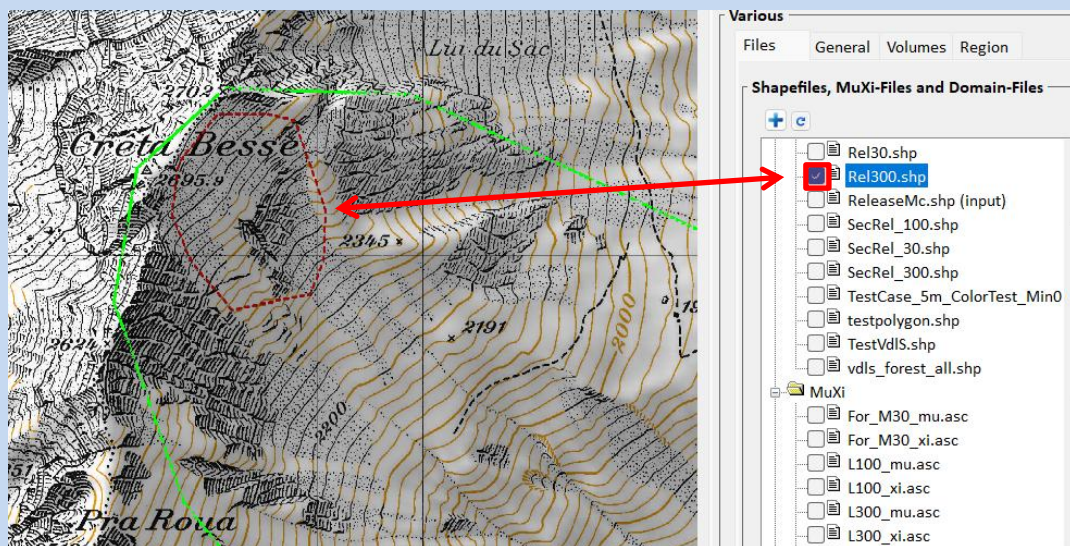


Figure 4-14: Selected file on the right is shown in the visualization.

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

CHAPTER 4: RESULTS

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**:

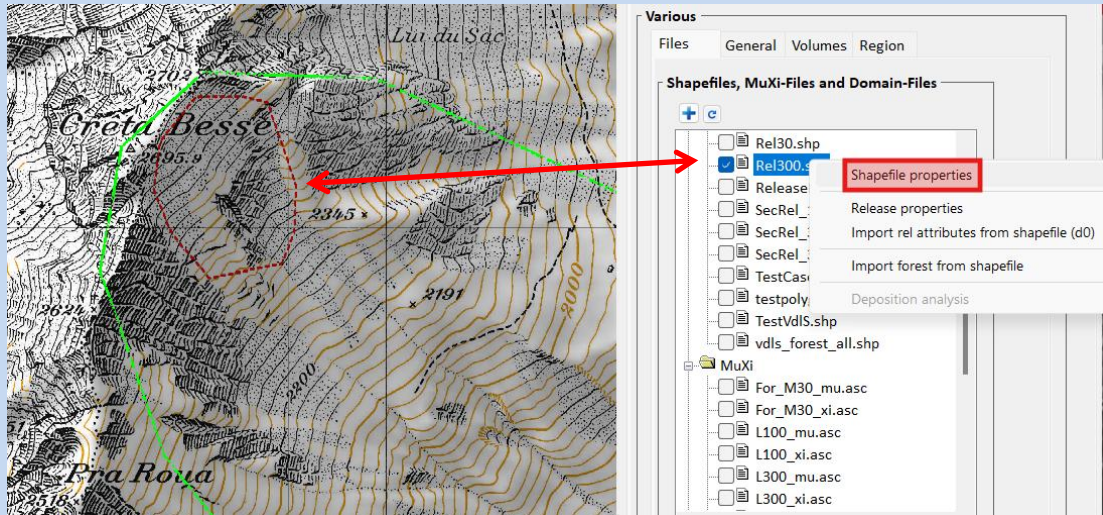


Figure 4-15: Right-click menus Shapefile properties, Release properties, Import forest from shapefile, Deposition analysis (only output) and Delete.

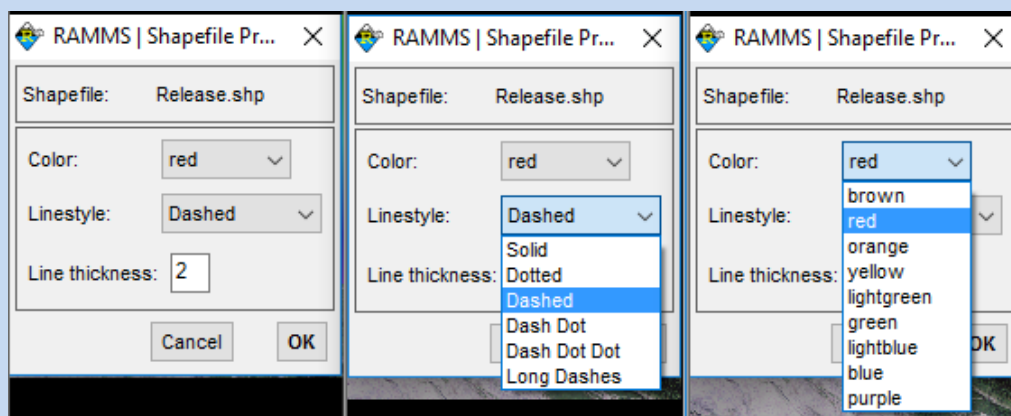


Figure 4-16: Use **Shapefile properties** to change line thickness, color or linestyle.

Release properties

- Please see section 4.8.1 on how to specify release area properties.

Import release attributes (d0 / delays)

- See section 4.8.1 on how to import shapefile attributes as d0 and release-delays.

CHAPTER 4: RESULTS

Import forest from shapefile

- Please see section 4.8.4 on how to import forest from a shapefile.

Deposition analysis

- This function is only available in output mode. Please see section 5.2.4 on how to do a deposition analysis.

b. Adding files to the project

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

- Add data: Use the button  or the menu 'GIS – Add data' to add a shapefile. If this shapefile is located outside of your project directory, it will be added to the files-tree.
- Add files from folder: Use the button  (Add files from external directory) below the file-tree to add all the files from an external directory to the file-tree. These files are added during this RAMMS session. After you exit and restart RAMMS, you have to add the files again.
- Drag & Drop: see next section.

c. Drag & Drop:

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

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

4.7.3 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-hill-shade-works.htm>

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

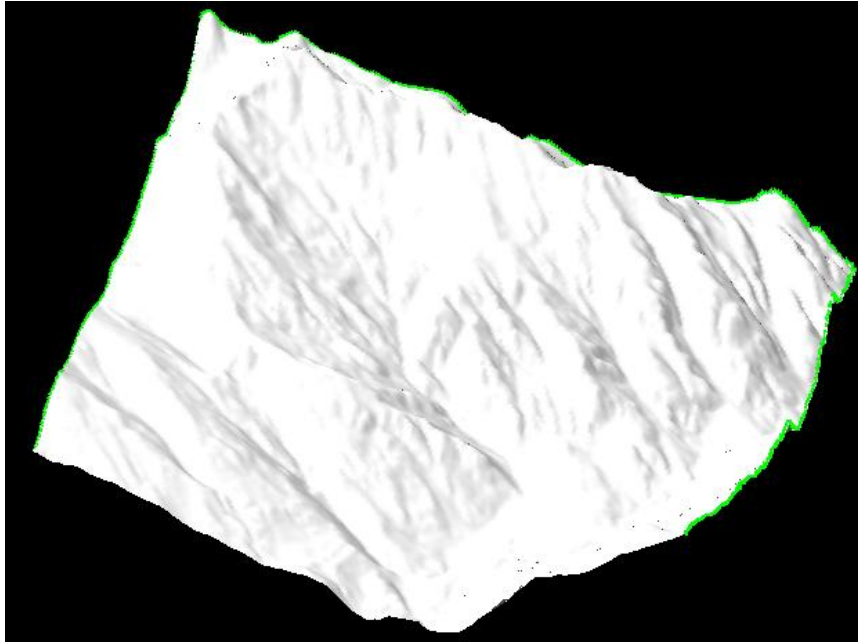


Figure 4-17 DEM surface visualization (with shadows) after creating a new project in RAMMS

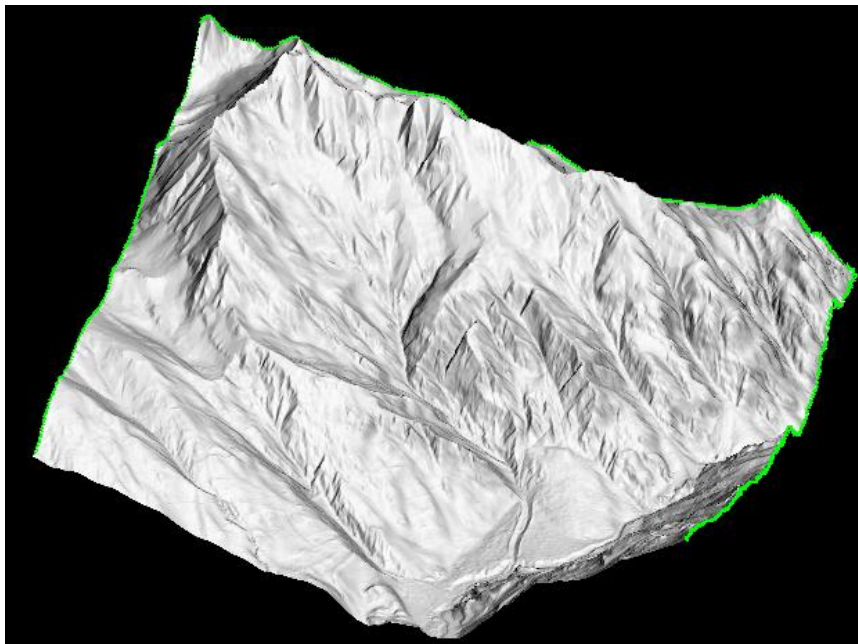


Figure 4-18 Visualization after creating and adding the hillshade image to RAMMS

4.7.4 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. If only a GeoTIFF file is available (and the tfw-file is missing), you can use **Extras → Create TFW from GeoTIFF (World File)** to create the tfw-file from the GeoTIFF file, without using GIS.

CHAPTER 4: RESULTS

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:

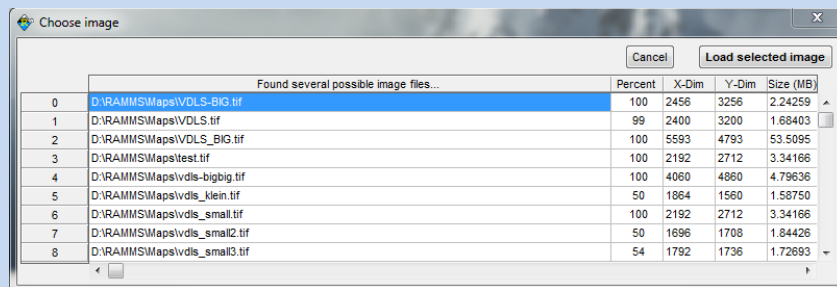


Figure 4-19: Window to choose map image.

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 .

4.7.5 Moving, resizing, rotating, viewing

! Hold **Shift** while dragging with the mouse to rotate the topography directly, without manually switching to the Rotate tool. This makes 3D navigation faster, smoother, and more intuitive.

If you need to rotate the model around a fixed axis, see exercise below.

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.

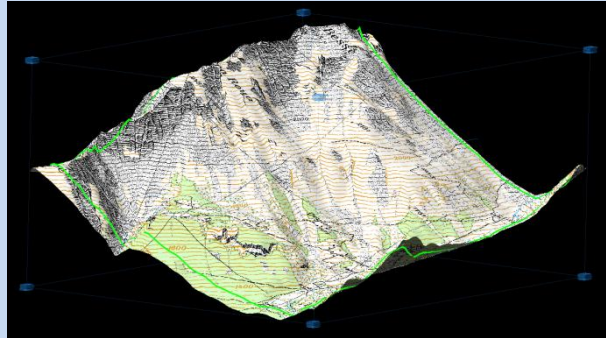


Figure 4-20: Active project with lines and corners for resizing.

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

b. Terrain model has a dimension > 100%:

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

Exercise 3.4d: Rotating the model

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

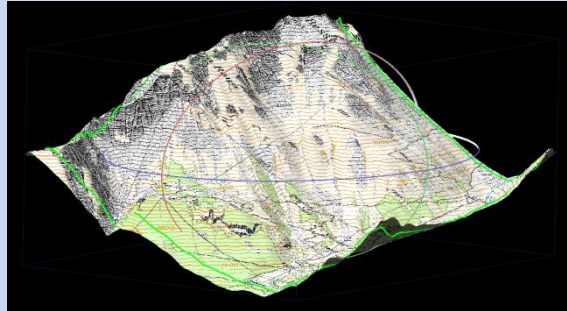


Figure 4-21: Active project with rotation axes.

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

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

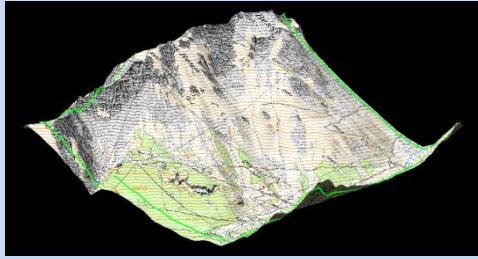


Figure 4-22: 3D view of example model.

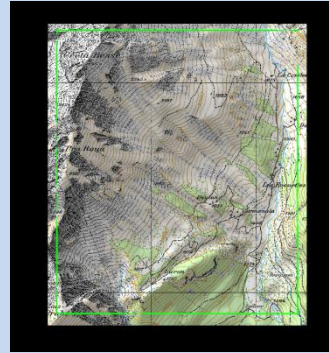


Figure 4-23: 2D view of example model.

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 

4.7.6 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).

CHAPTER 4: RESULTS

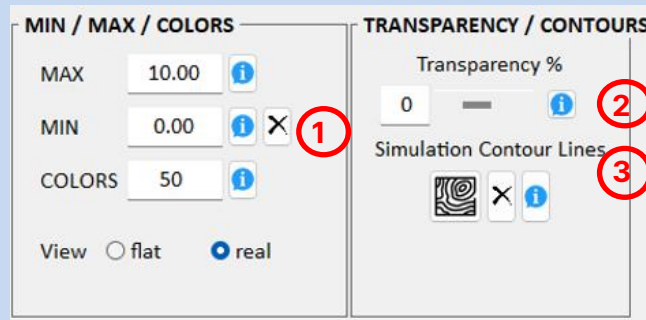


Figure 4-24 Colorbar and visualization properties

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

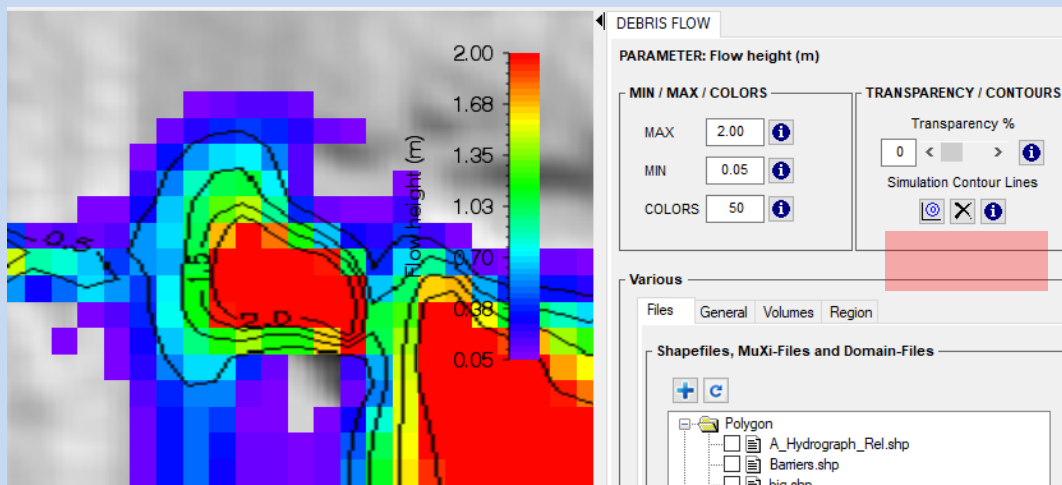


Figure 4-25: Simulation contour lines

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 using **Edit → Colorbar Properties**.
- To change the colorbar properties simply click into the field you want to change, then click **OK**.
- Use **Edit → Toggle Colorbar Color** to toggle the text-color of the colorbar between black and white. This can be useful when changing the background color of your project (**Track → Preferences → Avalanche Tab → Background Color**).

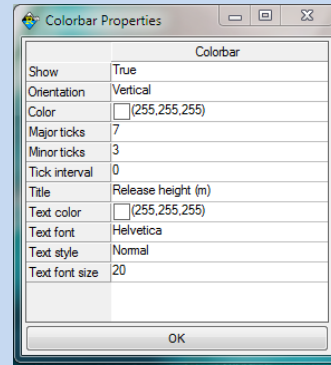


Figure 4-26: The Colorbar Properties window.

4.7.7 How to save input files and program settings

Once a project is created, it is saved under the name and location you entered when creating a new project. 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 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.

CHAPTER 4: RESULTS

4.7.8 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.

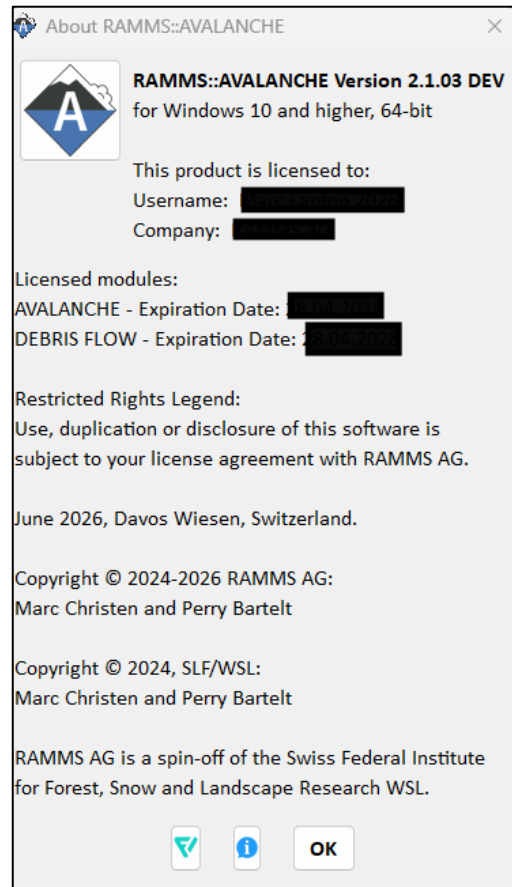


Figure 4-27: About RAMMS

4.8 Preparing a simulation

To run a simulation or a specific scenario within a newly created project (creating a project see section 3.2) 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 4.4.1.

4.8.1 Release area(s)

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

Release Type	Details
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 (<i>Files</i>) 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 in clockwise order.
- To end the release polygon, click the right mouse button. The polygon will be closed automatically.

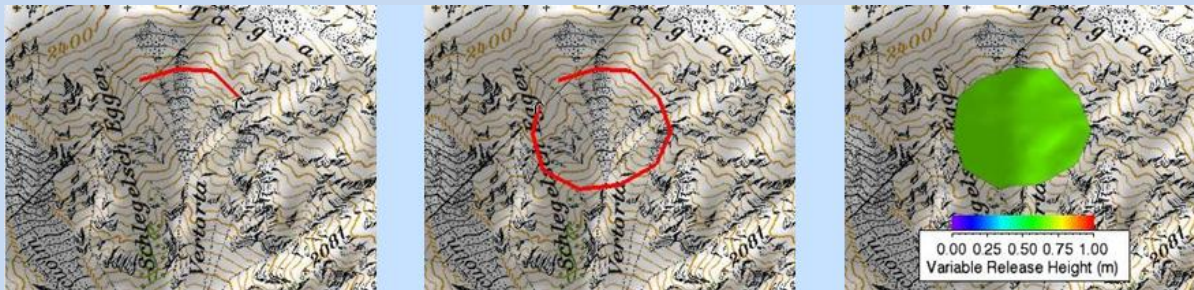


Figure 4-28: Drawing a new release area.

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.

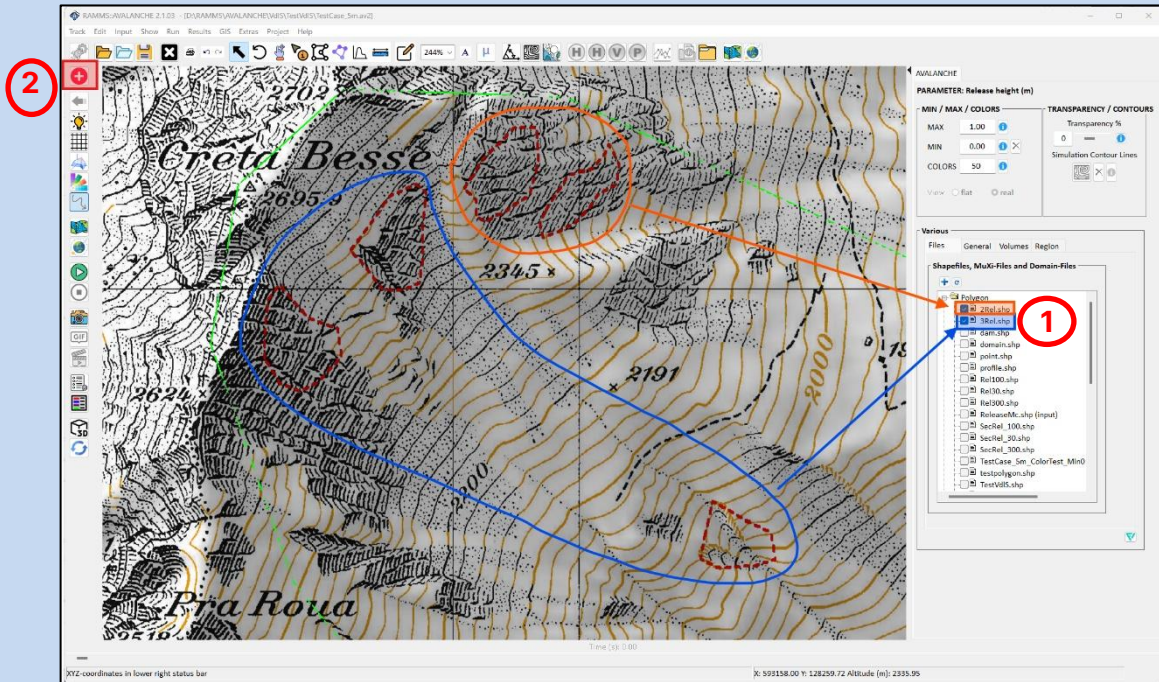


Figure 4-29: How to visualize polygon shapefiles

In Figure 4-29 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 4-30) 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 **2D**.
- 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:

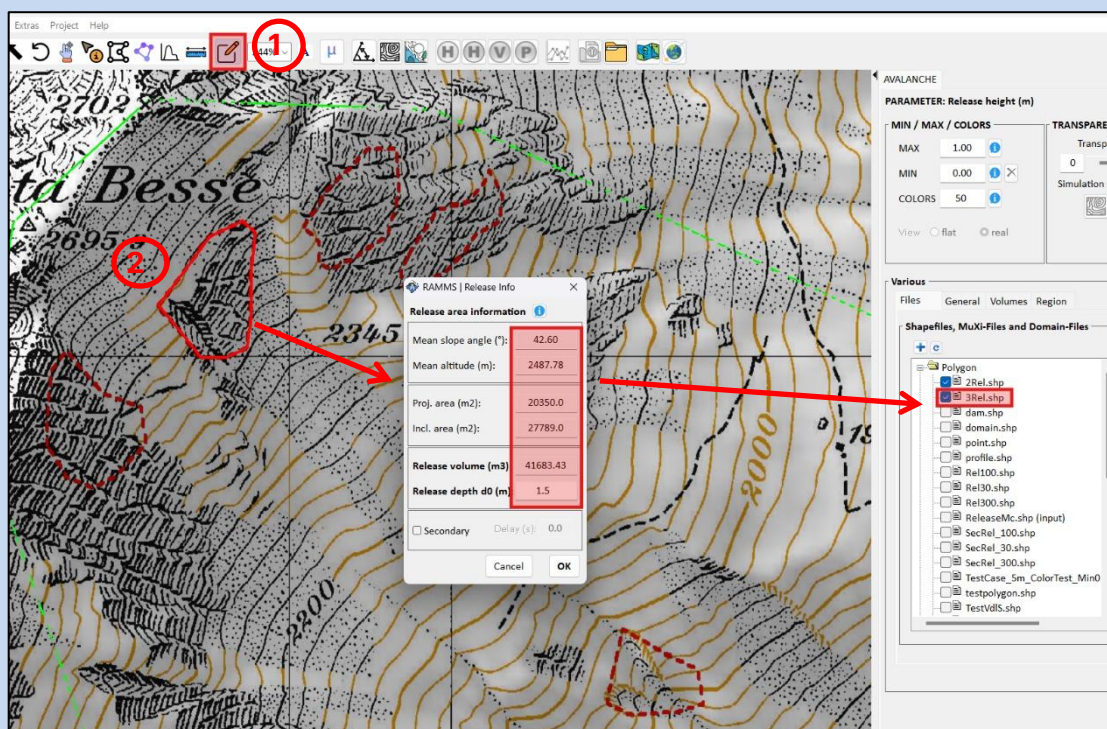


Figure 4-30: **View/Edit release area**

- Enter a release depth in the field *Release depth d0 (m)*. The corresponding release volume is calculated and updated automatically.
- Additionally, the following release area parameters 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.

CHAPTER 4: RESULTS

In Figure 4-30 we selected the northern-most polygon of the shapefile *3Rel.shp* and assigned a release depth of 1.5m. Now let's define a secondary avalanche release for the lowest polygon in *3Rel.shp* with a release delay of 20 seconds.

Therefore, we again click on  (1) and choose the lowest polygon of *3Rel.shp* (2), see below. We then specify a release depth of 0.5m and a delay of 20s (3).

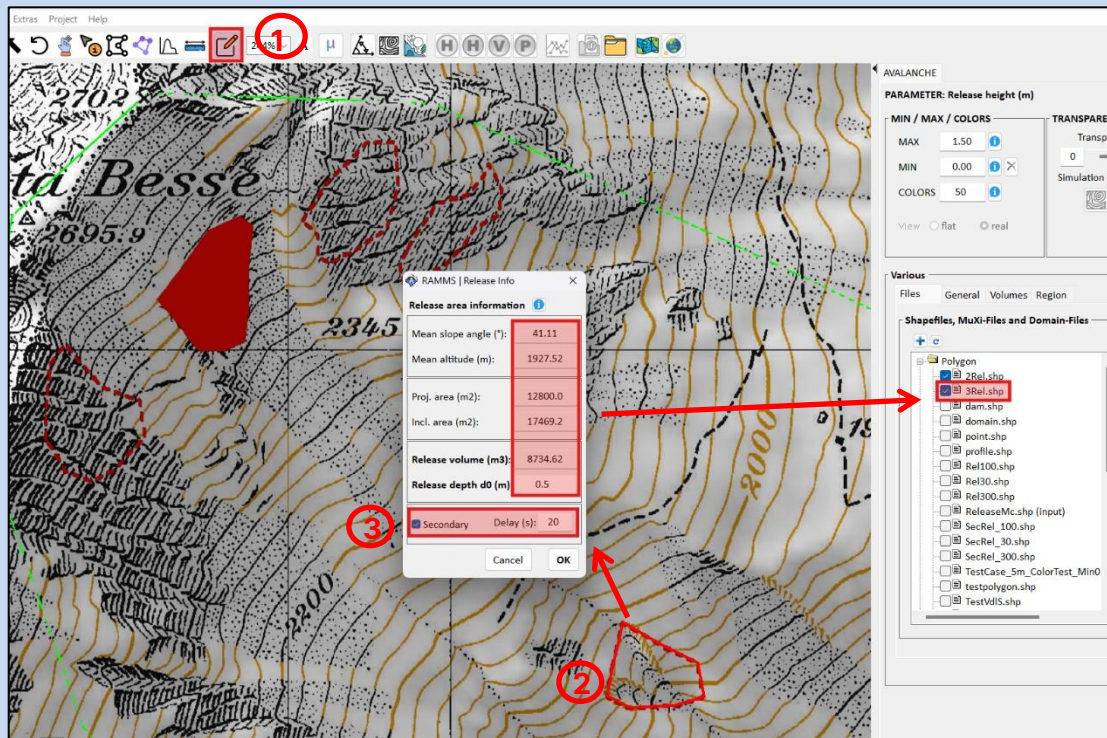
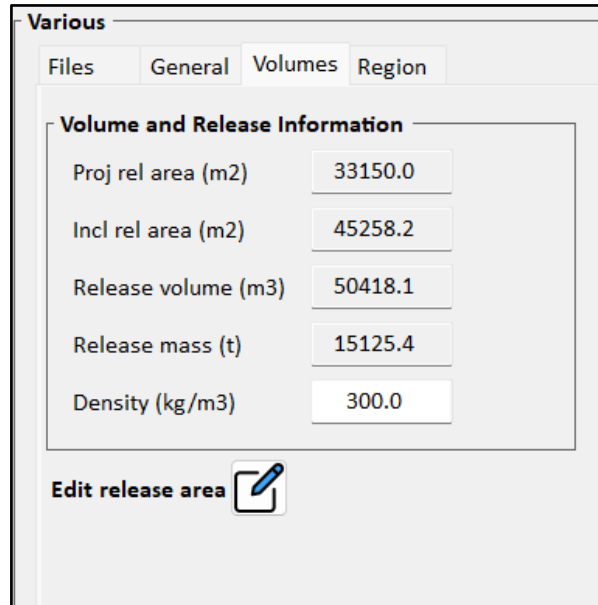


Figure 4-31: Specifying a secondary avalanche release polygon with a time delay.

We do not specify a release depth for the last polygon of *3Rel.shp*. In a next step we could now specify a release depth for one of the two polygons of shapefile *2Rel.shp*.



The screenshot shows a software interface window titled 'Various'. It has four tabs: 'Files', 'General', 'Volumes', and 'Region'. The 'Volumes' tab is selected. Inside the 'Volumes' tab, there is a section titled 'Volume and Release Information'. This section contains five rows of data, each with a label and a numerical value in a text box:

Label	Value
Proj rel area (m2)	33150.0
Incl rel area (m2)	45258.2
Release volume (m3)	50418.1
Release mass (t)	15125.4
Density (kg/m3)	300.0

Below the 'Volume and Release Information' section, there is a button labeled 'Edit release area' with a pencil icon next to it.

Figure 4-32: Release area and volume information

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

Import release attributes from a shapefile

When a release area consists of many individual polygons, assigning the release depth manually to each polygon can be time-consuming. RAMMS can therefore import release depths directly from attributes stored in a polygon shapefile. Optionally, release delays can be imported in the same way.

Each polygon in the shapefile must contain an attribute specifying its corresponding release depth d_0 . If delayed release is used, an additional attribute can contain the release delay.

CHAPTER 4: RESULTS

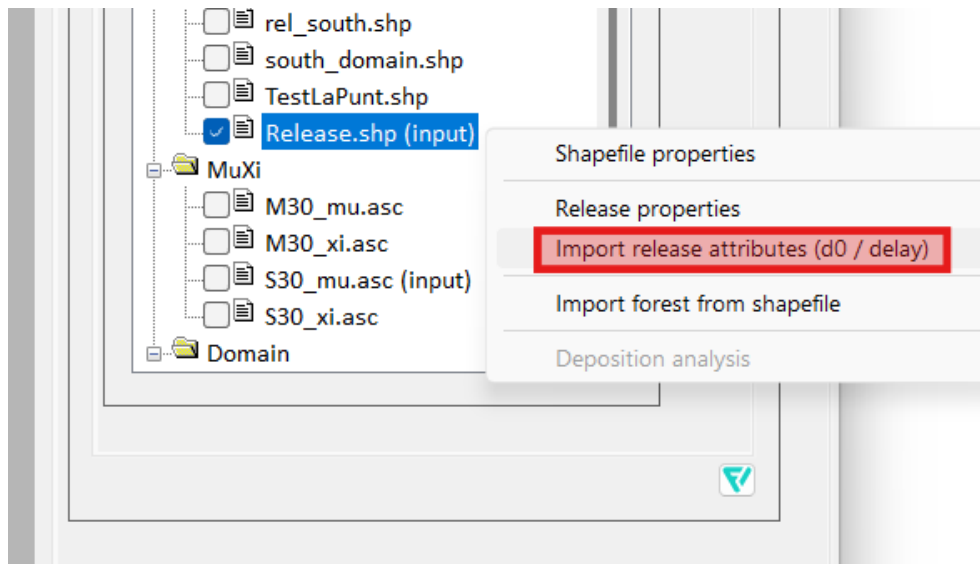


Figure 4-33: Importing release attributes from a shapefile.

Steps to import attributes:

1. Right-click the shapefile in the file tree and select **Import release attributes (d0 / delay)**.
2. In the **Import Release Attributes** dialog, select the shapefile attribute containing the **release depth** d_0 (m).
3. If applicable, select the attribute containing the **release delay** (s). Otherwise, leave the value set to **<none>**.

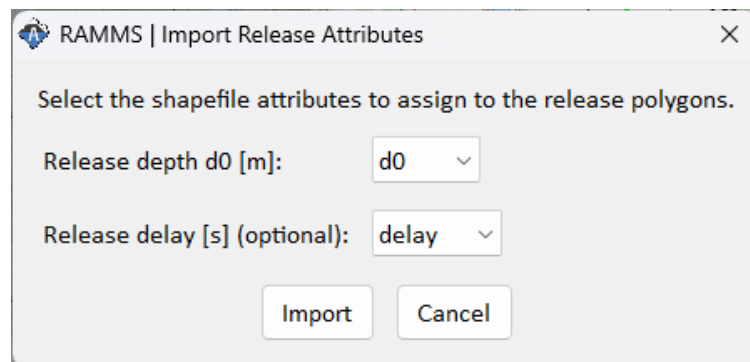


Figure 4-34: Import Release Attributes dialog. Select attributes for release depth and delay (optional).

4. Click **Import**. RAMMS assigns the attribute values to the corresponding release polygons.

Note: The selected attributes must contain valid numerical values for all release polygons. Release depths are specified in meters and release delays in seconds.

CHAPTER 4: RESULTS

4.8.2 Potential Release Areas (PRA)

The PRA visualization implemented in RAMMS (see Figure 4-35) is based on published GIS approaches for automated delineation of potential avalanche release areas. The underlying terrain-based "hard-gate" logic follows the earlier work of Maggioni and related Davos studies, where potential release areas are identified from topographic criteria such as slope angle, forest cover, altitude range, and ridge-related terrain characteristics derived from digital elevation models. This general concept is also consistent with later automated PRA delineation approaches by Bühler and co-authors, who likewise use DEM-based terrain parameters to identify potential release terrain. In addition, the present implementation adopts the multi-scale fuzzy-logic concept introduced by Veitinger, in which terrain properties such as slope and roughness are transformed into continuous suitability values and combined into a graded release propensity map.

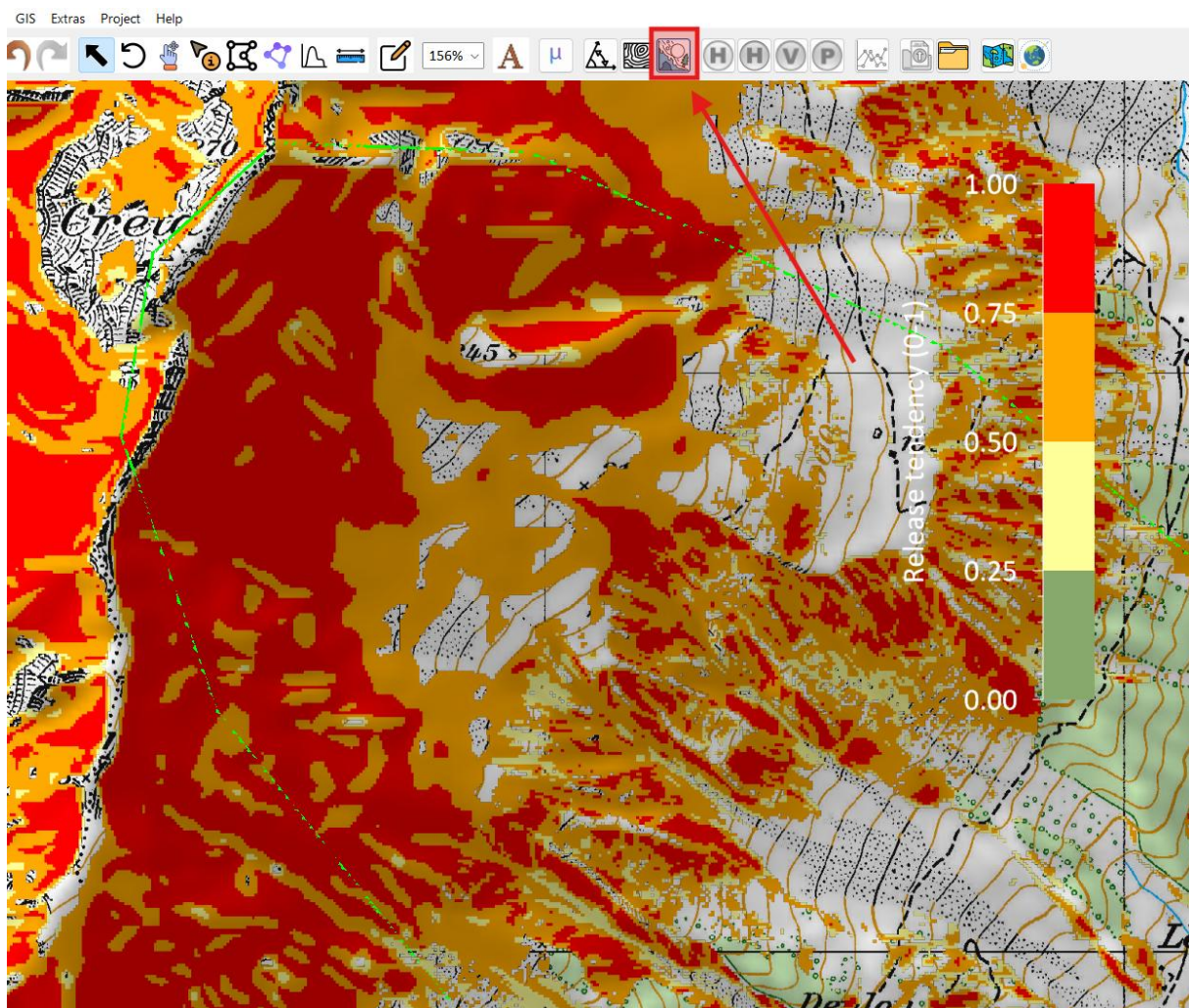


Figure 4-35: Clicking the "Show PRA" button will produce the above "Release tendency" visualization.

Explanation of the approach. The method is designed to identify terrain that is morphologically consistent with slab-avalanche release. In a first step, a binary candidate

CHAPTER 4: RESULTS

mask is generated from hard terrain constraints: only cells within a user-defined slope range are retained; cells inside forest polygons are excluded; optional altitude limits may be applied; and, if activated, main ridges can be removed using DEM-derived ridge indicators. In a second step, the remaining terrain is evaluated with a fuzzy approach. Here, slope is calculated at a characteristic scale and transformed into a slope suitability value, while terrain roughness is estimated from the DEM and transformed into a roughness suitability value. These suitability values are then combined into a final fuzzy score μ between 0 and 1, which expresses how strongly a given terrain cell matches the typical characteristics of a potential avalanche release area.

Meaning of the result. The resulting PRA classes should be interpreted as a relative terrain propensity, not as a calibrated event probability. A class such as 0.75–1.00 does not mean that an avalanche will release there with a 75–100% probability. Instead, it means that, according to the implemented terrain criteria, the location is strongly consistent with known release-area characteristics. Likewise, lower classes indicate terrain that is less consistent with typical release conditions. The output is therefore best understood as a ranking of terrain suitability for release, useful for visualization, plausibility checks, and scenario-based release-area mapping, but not as a direct forecast of avalanche occurrence. A raster of the classification (class_pra.tif) is automatically saved in the project directory.

PRA-parameters such as slope range, altitude limits, ridge exclusion factors as well as forest layer (excluding) can be set with **Input → Release Area... → Set PRA parameters**. Changes can be applied directly within the parameter window.

4.8.3 Calculation Domain

The calculation time of a RAMMS simulation depends strongly on the number of computational grid cells. For simulations with a small grid-cell size, the computational domain should therefore be limited to the area that is relevant for the avalanche.

A convenient way to define an efficient calculation domain is to first perform a coarse preliminary simulation. The resulting flow path can then be used to create a smaller domain for the final, higher-resolution simulation, see exercise below.

Recommendation: Especially for simulations with grid-cell sizes smaller than approximately 5 m, avoid using unnecessarily large calculation domains.

Exercise 3.5d : Finding an optimized calculation domain

- Open the input file and switch to **2D mode**.

CHAPTER 4: RESULTS

- Select **Input** → **Calculation Domain...** → **Draw New Domain** or use the corresponding toolbar button 
- Draw a polygon covering the expected avalanche path and surrounding area. The polygon is created in the same way as a release-area polygon. At this stage, the domain can be relatively generous, since its purpose is only to provide the domain for the preliminary simulation, see Figure below.

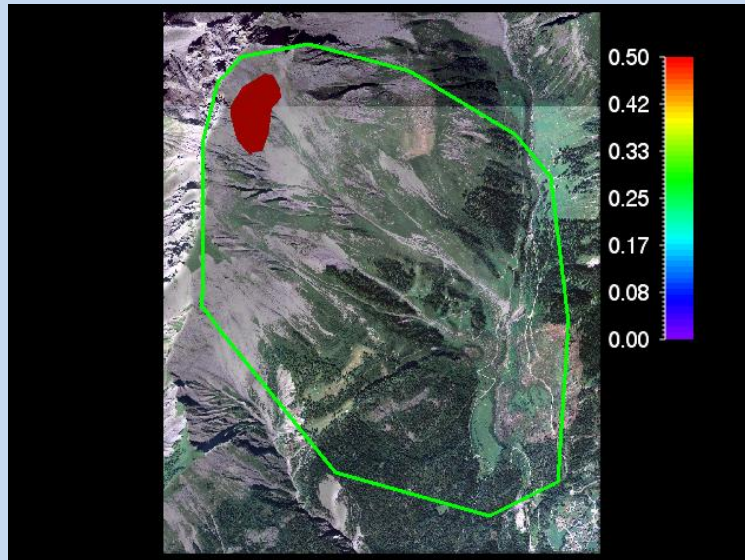


Figure 4-36: Example of an initial calculation domain used for the preliminary simulation.

- Run a preliminary simulation using a coarse simulation resolution, for example **10 m**. For this initial calculation, a simplified friction definition can be used. For example, constant values of $\mu=0.2$ and $\xi=2000$ may be sufficient to obtain an approximate flow path. The purpose of this simulation is not to obtain final results, but to identify the area that is potentially affected by the avalanche.
- After the simulation has finished, display the **Maximum Flow Height** result .

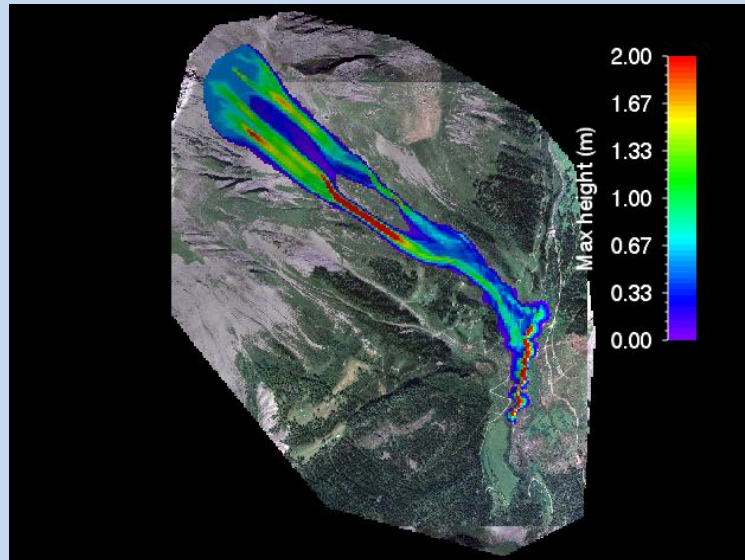


Figure 4-37: Max Flow Height of a 10m simulation with constant μ and ξ

- Create an envelope of the simulated flow path: Select **GIS → Export... → Create Envelope Shapefile**. RAMMS creates a polygon enclosing the displayed simulation results.

Important: The minimum value of the current color scale is used as the lower threshold when creating the envelope. Adjust the minimum value of the color bar before creating the envelope if necessary.

- RAMMS then asks whether a buffer should be added to the envelope.
- Select **Yes** if the envelope will subsequently be used as a calculation domain. The additional buffer provides space around the preliminary flow extent and helps prevent the final simulation from being artificially constrained by the calculation-domain boundary. Choose a filename for the envelope shapefile (a name is proposed).
- Select **No** only if an envelope corresponding closely to the displayed simulation extent is required.
- Choose a filename for the envelope shapefile. RAMMS proposes a default name.
- The created envelope shapefile is shown in the visualization as a dashed red line. The newly created envelope is displayed as a dashed red polygon in the map view.

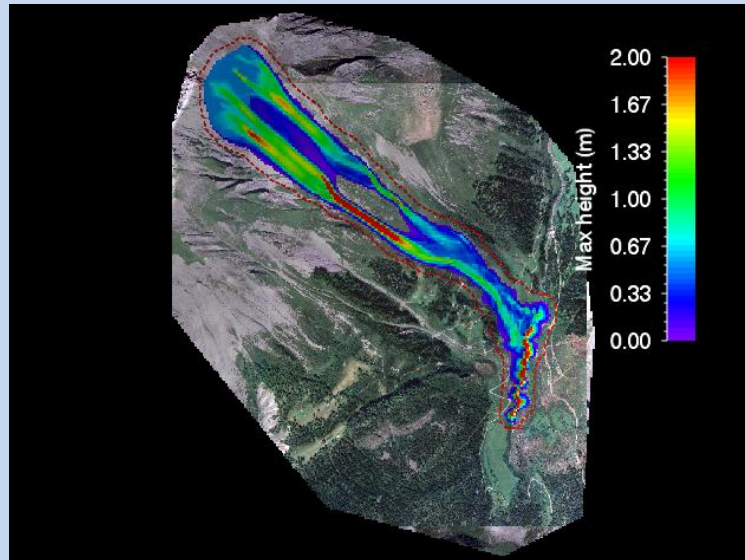


Figure 4-38: Envelope shapefile of Max Flow Height extent

- Return to the input file 
- Select **Input** → **Calculation Domain...** → **Load Existing Domain** to load the buffered envelope shapefile created in the previous step. It now defines the calculation domain for the final simulation, see Figure below.

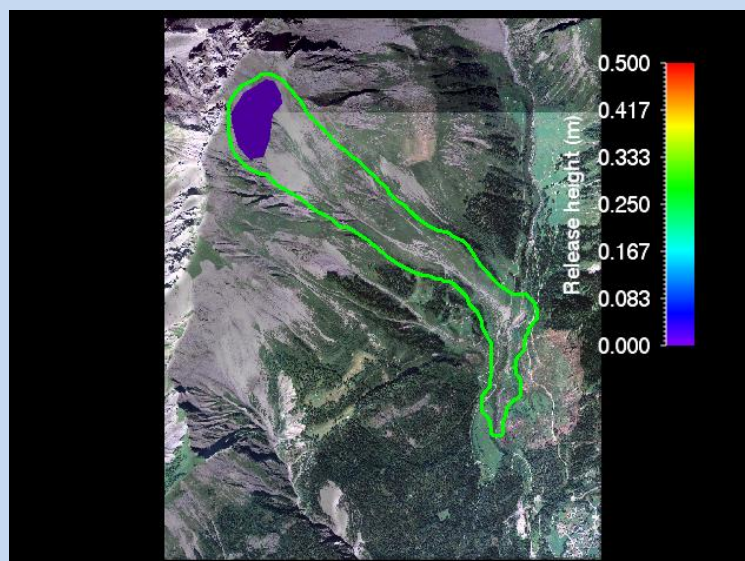


Figure 4-39: Buffered envelope of the preliminary simulation used as the optimized calculation domain.

- The final simulation can now be performed using the desired model settings, for example with a resolution of 5m and a variable MuXi-file.

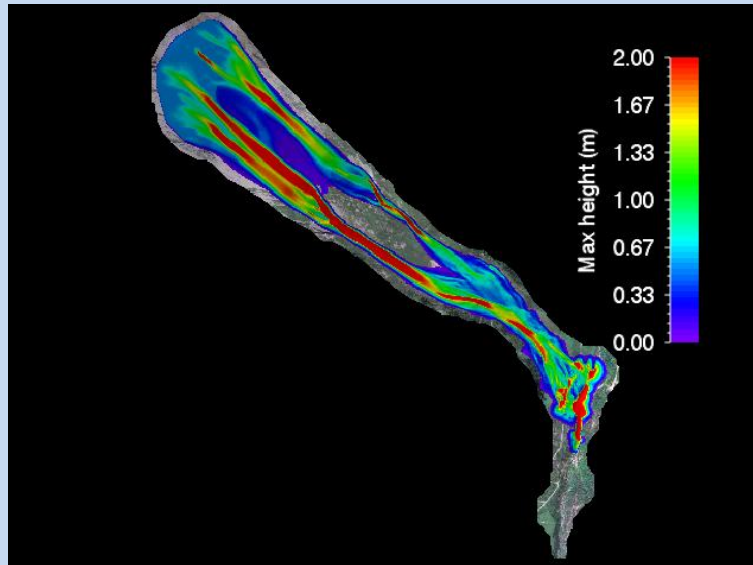


Figure 4-40: Max Flow Height result of a 5m simulation with variable MuXi-file

- Because computational cells outside the relevant flow area are no longer included, the simulation can be considerably faster while retaining the required resolution within the area of interest.
- In the example shown here, reducing the calculation domain decreased the computational time by approximately **70%**.

4.8.4 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 .

CHAPTER 4: RESULTS

- 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 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.

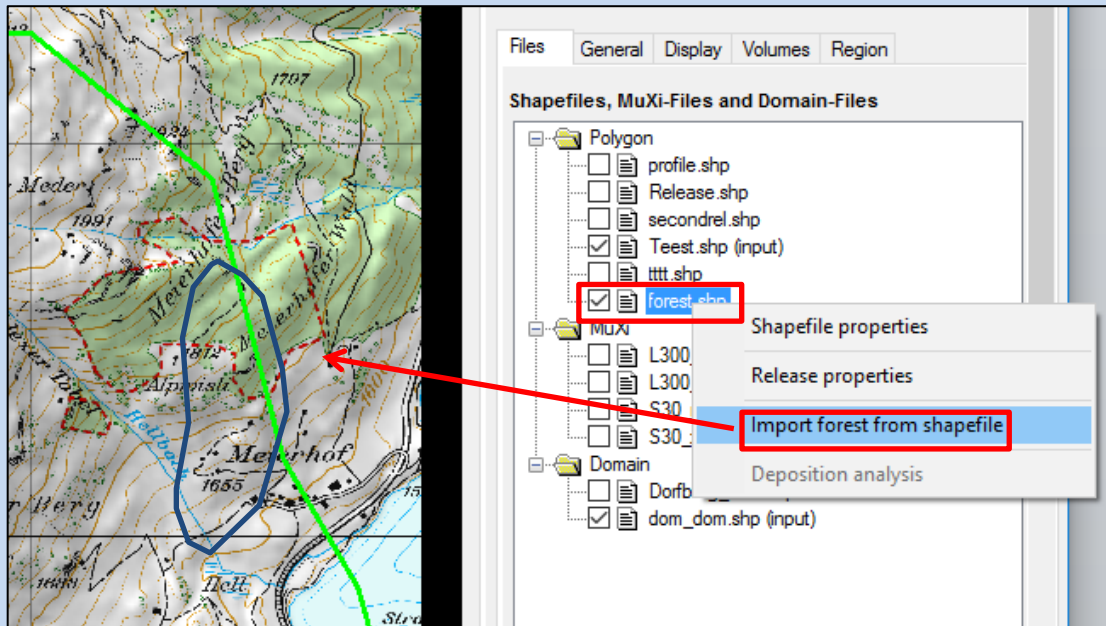


Figure 4-41: Import forest from shapefile

- 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.

CHAPTER 4: RESULTS

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.
- 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.

RAMMS | Automatic MuXi Procedure

Volume = LARGE Return period = 30 [Cancel] [OK]

(Optional: if empty -> default name)
Mu/Xi name:

Altitude Limits (m.a.s.l.) 1: 2:

Above Altitude Limit 1

Open Slope - Mu:	0.170	ξ :	3000
Channelled Slope - Mu:	0.225	ξ :	2000
Gully Slope - Mu:	0.290	ξ :	1500
Flat Area Slope - Mu:	0.155	ξ :	4000
Forested Area (mu=delta, xi=fix) - Mu:	0.020	ξ :	400

Below Altitude Limit 1

Open Slope - Mu:	0.190	ξ :	2500
Channelled Slope - Mu:	0.240	ξ :	1750
Gully Slope - Mu:	0.310	ξ :	1350
Flat Area Slope - Mu:	0.170	ξ :	3500
Forested Area (mu=delta, xi=fix) - Mu:	0.020	ξ :	400

Below Altitude Limit 2

Open Slope - Mu:	0.210	ξ :	2000
Channelled Slope - Mu:	0.260	ξ :	1500
Gully Slope - Mu:	0.330	ξ :	1200
Flat Area Slope - Mu:	0.190	ξ :	3000
Forested Area (mu=delta, xi=fix) - Mu:	0.020	ξ :	400

Figure 4-42: 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.

4.9 Running a simulation

To run a simulation you have to open a created project, load a release area and a calculation domain. 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 simulation

- 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.

CHAPTER 4: RESULTS

- 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
- (8) Center of mass stopping criterion

- 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.

The screenshot shows the 'RAMMS | Run Simulation' dialog box with the 'General' tab selected. The 'General Simulation Information' section includes a 'SCENARIO Name' field (labeled 1) and a checkbox for 'Save Max Values Only' (labeled 2). The 'Additional Information' section contains 'Project name' (labeled 3), a 'Details' text area (labeled 4), 'Calculation domain' (labeled 5), and 'Digital elevation info' (labeled 6). The 'Stop Criteria' section has 'Percentage of total momentum (%)' (labeled 7) and 'Center-of-Mass vel threshold (m/s)' (labeled 8). The 'Remarks' section (labeled 9) contains instructions. At the bottom, there is a 'Run in background' checkbox (labeled 1) and 'Cancel' and 'RUN SIMULATION' buttons.

Figure 4-43 General Information

Continuation of exercise 3.5g: How to run a avalanche simulation

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/m³).

Figure 4-44: 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.

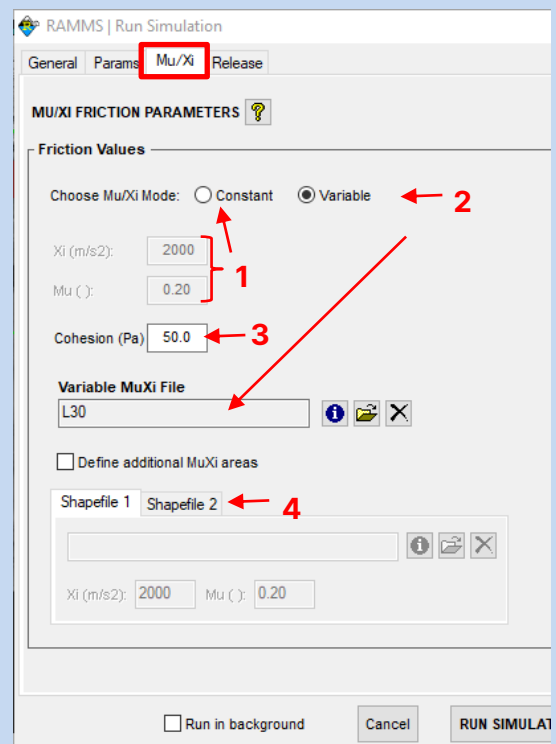
(6) Curvature: Switch *Curvature* On or Off. The normal force *N* includes centrifugal forces arising from the terrain curvature. We use the method proposed by Fischer et al. (2012) which was specifically developed for RAMMS. Typically, this increases the friction, causing the flow to slow down in tortuous and twisted flow paths.

See <https://www.ramms.ch/ramms-avalanche/friction-parameters/> for more information.

Continuation of exercise 3.5g: How to run avalanche simulation

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 in the appendix 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 fine (suggested: 50Pa). For wet avalanches, slightly higher values (up to 200Pa) can be used. See <https://www.ramms.ch/ramms-avalanche/friction-parameters/> 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 4-45: Friction values μ and ξ .

CHAPTER 4: RESULTS

RELEASE PARAMETERS

- (1) **Filename:** List of all the release shapefiles in use.
- (2) **Volume (m^3):** 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 (m^3):** Sum of all the release volumes from above.
- (6) **Run Simulation Button**

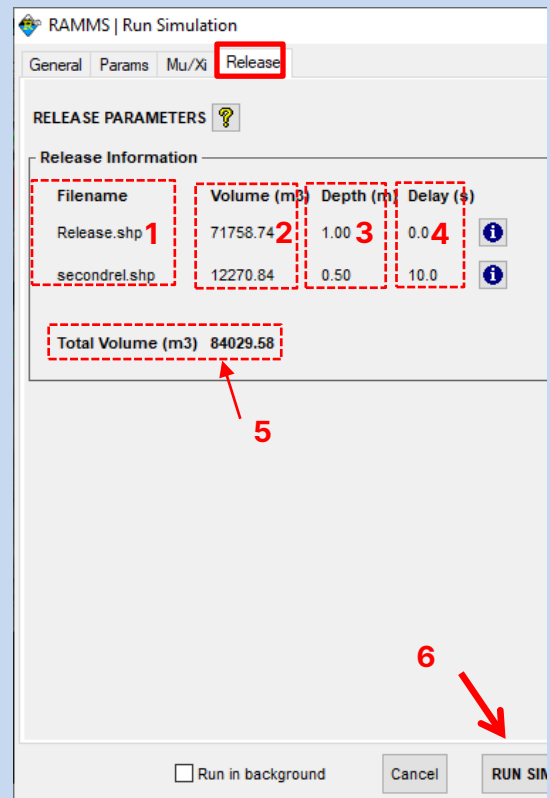


Figure 4-46: Release information.

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

Run Simulation

- Click **RUN SIMULATION**
- When a simulation is started, RAMMS opens a separate calculation window in which the numerical solver is executed. ()

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 (6).

CHAPTER 4: RESULTS

```
cmd.exe /c E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out.bat

*****
Welcome to the RAMMS::AVALANCHE x86 calculation module!

Reading input file...

1 Starting calculation with 8822 nodes, 8402 elements and 16385 edges
Starting time is Mon Nov 27 14:57:39 2017

Initializing avalanche...

INITIAL FLOW VOLUME: 106784 m3

2 Writing output file: E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out...

Writing coordinates...
Writing connectivity list...
Writing initial conditions...

3 Starting explicit time integration...
*. *. *. *. *.

Time 2.3
4 Step 7 dt 0.317321 Cfl 0.498169
Hmax 1.93 m Vmax 13.72 m/s

5 MOVING MOMENTUM: 100.0 percent ( 8196.5 / 8196.5 )

FLOW VOLUME: 106783.7 m3

6 NUMERICAL VOLUME LOSS: 0.01 m3
```

Figure 4-47: RAMMS calculation window showing the progress and status information during a simulation.

```
cmd.exe /c E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out.bat

Finished writing RAMMS output file: E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out...
*****

D:\Programs\RAMMS_USER_64bit_IDL85_1.7.0>"D:\Programs\RAMMS_USER_64bit_IDL85_1.7.0\bin\7za.exe" a -tgzip "E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out.gz" "E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out"
7-Zip (A) 9.20 Copyright (c) 1999-2010 Igor Pavlov 2010-11-18
Scanning
Creating archive E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out.gz
Compressing Test.out
Everything is Ok

D:\Programs\RAMMS_USER_64bit_IDL85_1.7.0>del /F /Q E:\RAMMS\AVALANCHE\Beispiele\Test\Test.out

D:\Programs\RAMMS_USER_64bit_IDL85_1.7.0>echo SIMULATION Test.out.gz FINISHED SUCCESSFULLY!
SIMULATION Test.out.gz FINISHED SUCCESSFULLY!

D:\Programs\RAMMS_USER_64bit_IDL85_1.7.0>pause
Drücken Sie eine beliebige Taste . . .
```

Figure 4-48: RAMMS calculation window showing a **background** simulation. Press any button to close the window once the simulation finished.

CHAPTER 4: RESULTS

4.9.1 Run Batch Simulations

RAMMS can run several simulations automatically in batch mode. This is particularly useful when many scenarios should be calculated unattended, for example overnight.

To start a batch calculation, select **Track → New... → Run Batch Simulations**.

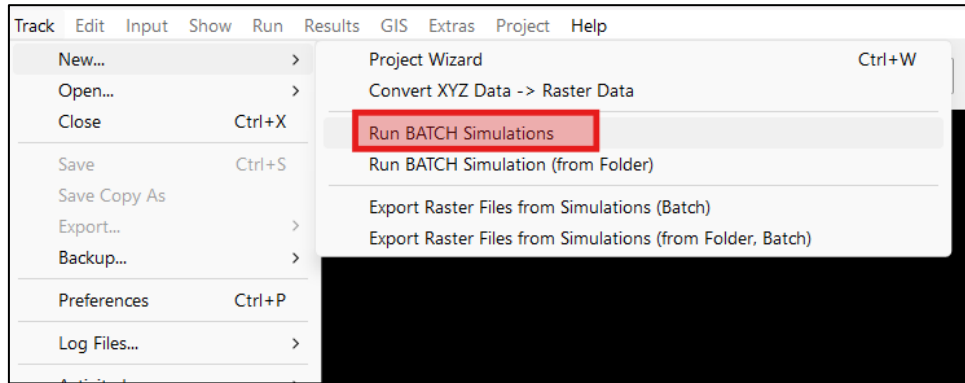


Figure 4-49: Batch-Simulations

In the Batch Simulation window, you can then select any number of prepared RAMMS input files and specify the maximum number of simulations that may run simultaneously.

For example, if you select 20 simulations and specify 4 computational cores, RAMMS starts four simulations in parallel. As soon as one simulation is completed, the next simulation in the batch is started automatically. This continues until all 20 simulations have been calculated.

Input files may also be selected from different directories or projects. Optionally, RAMMS can create a separate directory for each batch simulation.

Preparing Input Files for Batch Simulations

Before starting the batch calculation, prepare and save an input file for each simulation scenario:

1. **Open an existing input file** from your RAMMS project.
2. Click the large green **Run Simulation** button in the left toolbar to open the **Run Simulation** window.
3. Go through all tabs of the **Run Simulation** window and specify all required simulation settings and parameters.
4. When all settings have been defined, click **Cancel** instead of starting the simulation. The settings entered in the Run Simulation window are retained.
5. Select **Track → Save Copy As** or click the corresponding **Save Copy As** button. Save the scenario as a new RAMMS input file (.av2) using a meaningful file name.

CHAPTER 4: RESULTS

6. Repeat this procedure for every scenario that you want to calculate.

Once all input files have been prepared, select **Track → New... → Run Batch Simulations**, add the desired input files, specify the maximum number of simulations that should run in parallel, and start the batch calculation.

Tip: When running many scenarios, use descriptive input file names. This makes it easier to identify the corresponding simulations and results later.

5 Results

Once the simulation has finished, RAMMS automatically opens the calculated result together with the corresponding **Output Log File**. If the simulation was started in **Background Mode** (see Figure 4-48), the RAMMS calculation window remains open after the calculation has finished. Press any key inside this window to close it, then open the completed simulation manually in RAMMS.

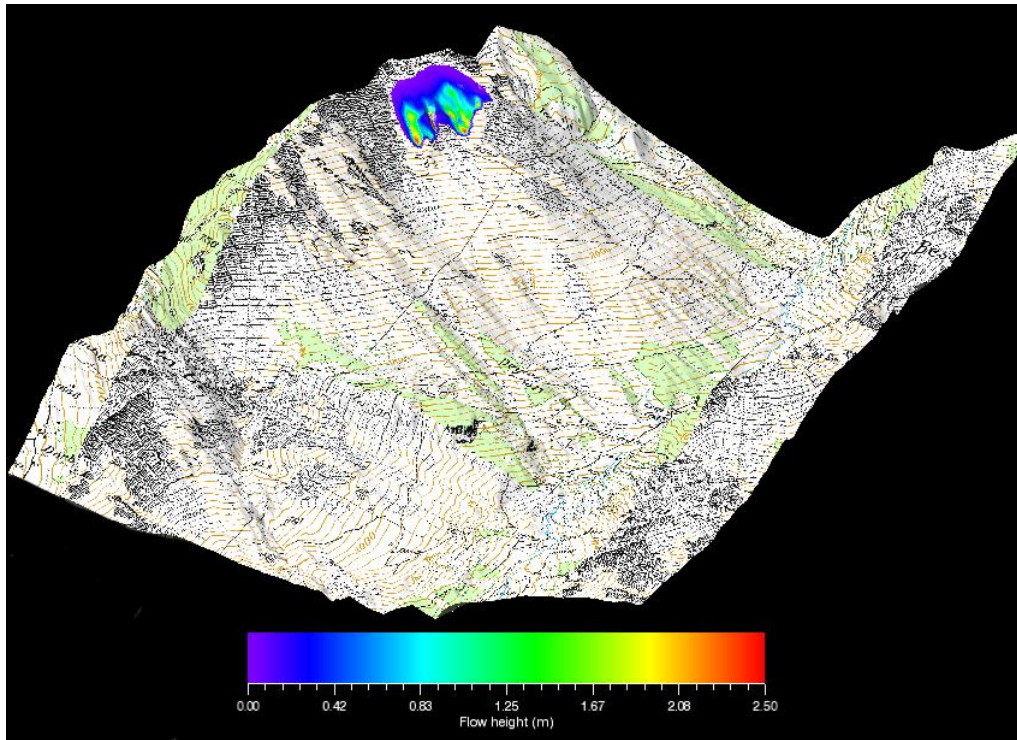


Figure 5-1: Main window in output mode.

If material flows out of the defined calculation domain during the simulation (RAMMS will mention this in the logfile) the calculation domain should generally be enlarged to ensure that the relevant flow process is fully represented (see section 4.8.3).

5.1 Log Files and Project information

5.1.1 Output Log File

After a simulation has finished, RAMMS automatically opens the Output Log File. The logfile can also be opened later in Output Mode by selecting **Project → Output Log File**.

The Output Log File provides a compact summary of the completed simulation and should be one of the first places to check after a calculation. It contains information about the calculation itself, the most important simulation results, and the input parameters used for the run.

At the beginning of the logfile, RAMMS reports which solver was used. Depending on the calculation, this indicates whether the simulation was performed using the CPU or GPU. For GPU calculations, the GPU model and the number of computational cells are also listed. In the example logfile, the simulation was calculated using an NVIDIA RTX 2000 Ada Generation Laptop GPU with 93291 computational cells.

RAMMS also reports why the simulation was stopped. This information is particularly important when interpreting the calculated runout. In the example logfile, the simulation stopped because of low mass flux, which is good, after 136s. The simulation could also stop, because the simulation reached the specified end time. RAMMS would then issues a warning that the runout distance may increase if the simulation is repeated with a longer end time.

The logfile then summarizes important information about the calculation, including:

- calculation time,
- simulation resolution,
- number of computational cells and nodes,
- calculated release or hydrograph volume,
- overall maximum velocity,
- overall maximum flow height,
- overall maximum pressure.
- Solver (GPU if available).

CHAPTER 5: RESULTS

```
RAMMS | Standard Output Log File
File
*****
RAMMS::AVALANCHE RAMMS OUTPUT LOGFILE

Output filename: Test 2.1 GPU.out.gz

>>> USING GPU SOLVER - NVIDIA RTX 2000 Ada Generation Laptop GPU, 93291 cells <<<

Simulation stopped due to LOW FLUX!

Simulation stopped after 136.000s
Calculation time (min.): 0.28

Simulation resolution (m): 5.00

SIMULATION RESULTS

Number of cells: 93291
Number of nodes: 94008

Calculated Release Volume (m3): 22705.9

Overall MAX velocity (m/s): 33.4247
Overall MAX flowheight (m): 7.08889
Overall MAX pressure (kPa): 335.163

*****
RAMMS::AVALANCHE 2.1.03 INPUT LOGFILE

Date: Sat Aug 29 15:54:15 2026
Input filename: Test 2.1 GPU.av2

Project: TestLaPunt
Details:

--- DEM / REGION INFORMATION ---
DEM file: Test.xyz
DEM resolution (m): 5.00
(imported from: dem.tif)

Nr of nodes: 400000
Nr of cells: 398701

Project region extent:
E - W: 2790995.0 / 2787000.0
S - N: 1159000.0 / 1161495.0

--- CALCULATION DOMAIN ---
domain.dom

--- GENERAL SIMULATION PARAMETERS ---
Simulation time (s): 300.000
Dump interval (s): 2.00
Stopping criteria (momentum threshold) (%): 5
Constant density (kg/m3): 300

--- NUMERICS ---
Numerical scheme: SecondOrder
H Cutoff (m): 0.000001
Curvature effects are ON!

--- RELEASE ---
Depth: 1.00 m Vol: 22619.3 m3 Delay: 0.00 s Name: Release.shp
Estimated release volume: 22619.27 m3

FRICTION MUXI:
Altitude_limit_1: 1500 m a.s.l
```

CHAPTER 5: RESULTS

Figure 5-2: Example output log file.

These values provide a quick overview of the simulation and can be used for an initial plausibility check. The reported maximum values are the overall maxima reached anywhere in the calculation domain during the simulation.

The second part of the logfile records the **input specifications used for this particular simulation**. This includes, among other information, the DEM and calculation domain, simulation time and output interval, stopping criteria, density, numerical scheme, release volume, friction parameters, cohesion, and map or orthophoto information.

The logfile therefore provides an important record of both the **simulation setup and the resulting calculation** and is useful when documenting, comparing, or reproducing simulations.

5.1.1 Input Log File

The input information can also be viewed separately by selecting **Project → Input Log File** (in Input-Mode).

The Input Log File contains the project and simulation settings stored in the currently loaded input file, including the DEM, calculation domain, release or hydrograph definition, numerical and physical parameters, friction settings, and other simulation specifications.

5.1.2 Project Region Information

To view information about the spatial extent of the project, select the **Region** tab in the RAMMS panel.

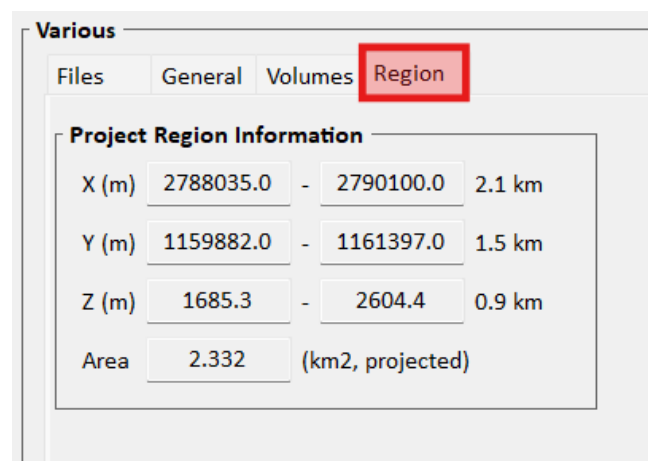


Figure 5-3: Project region information showing the X-, Y- and Z-extent and the total project area.

The Region tab displays the minimum and maximum **X- and Y-coordinates** of the project area, corresponding to its spatial extent in the horizontal plane. The minimum and maximum terrain elevation are shown as the **Z extent**. RAMMS also displays the total projected **project area** in km².

CHAPTER 5: RESULTS

These values refer to the complete project region defined when the project was created and not necessarily to a smaller calculation domain used for an individual simulation.

5.2 Visualization and analysis of results

This section provides a brief overview of the tools available in RAMMS for visualizing and analyzing simulation results. The results should be interpreted by experienced users who are familiar with the local topographic and meteorological conditions of the study area.

As with any numerical model, RAMMS represents a simplification of reality. Simulation results should therefore be critically evaluated and interpreted in the context of the model assumptions, input data, and local conditions. We strongly recommend performing sensitivity analyses to assess the influence of relevant model parameters on the results.

5.2.1 Maximum Results and Dump-Step Results

RAMMS provides two different types of simulation results:

Maximum results show, for each computational cell, the highest value reached during the entire simulation. For example, the **Max Flow Height** map contains the maximum flow height attained at each location at any time during the calculation. The same applies to every **Max-Result**.

These maximum-value maps are therefore **not snapshots of the flow at a particular time**. The maximum at one location may occur early in the simulation, while the maximum at another location may occur much later.

In addition, the maxima of different result quantities do not necessarily occur at the same time. For example, the maximum flow height at a particular cell may occur at a different time than the maximum velocity or maximum pressure at that cell. Maximum maps should therefore generally be interpreted independently.

Dump-step results, in contrast, represent the state of the simulation at a specific simulation time. RAMMS stores these results at the specified **dump interval**. For example, with a dump interval of 2 s, results are stored at approximately 0, 2, 4, 6 s, etc.

Each dump step contains the spatial distribution of quantities such as flow height, velocity and pressure **at that particular time**. By displaying the dump steps consecutively, the temporal evolution of the flow can be visualized as an animation.

Note: Maximum results and dump-step results serve different purposes. Maximum maps are particularly useful for identifying the largest loading or flow intensity reached at each location, whereas dump-step results are used to analyze the temporal development and dynamics of the flow.

CHAPTER 5: RESULTS

Values from different maximum maps should not automatically be combined, because their respective maxima may have occurred at different simulation times.

5.2.2 Visualize different parameters

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

- Flow Height 
- Flow Velocity
- Flow Pressure
- Flow Dynamic Load
- Flow Momentum
- Max values
 - Height 
 - Velocity 
 - Impact Pressure 
 - Line Load  (see section 5.3.3)
 - Momentum,
 - Shear Stress
- DEM Adaption (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (μ , ξ)
- Cell area (m²)

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 flow velocity** 
- **Max impact pressure** 

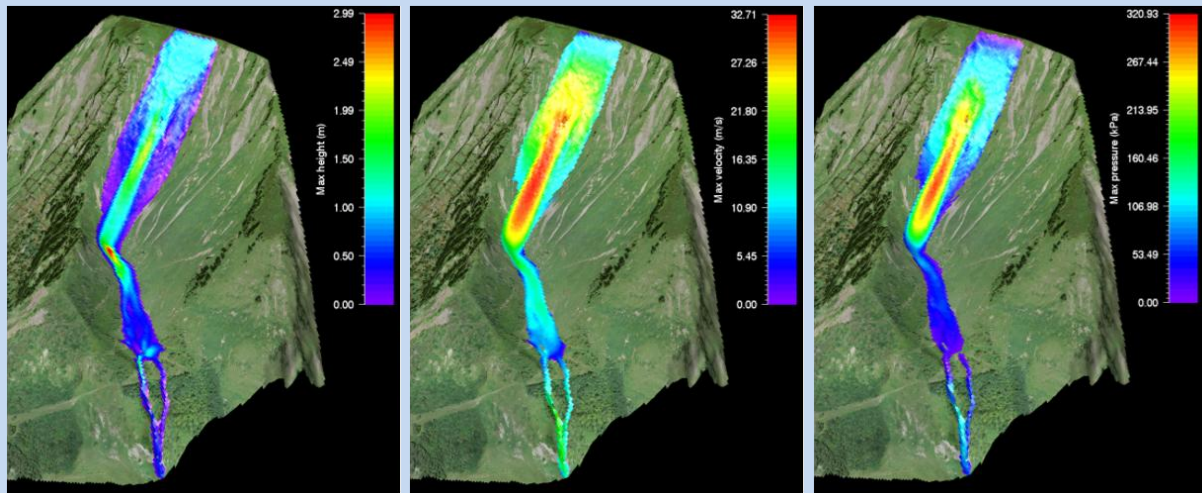


Figure 5-4: Results: Maximum values of flow height (left), velocity (middle) and impact pressure (right).

The flow height can be visualized exaggerated by the flow height (quasi-3D). Use the View radio-buttons to toggle between flat (no exaggeration), and real (quasi-3D), see figures below.

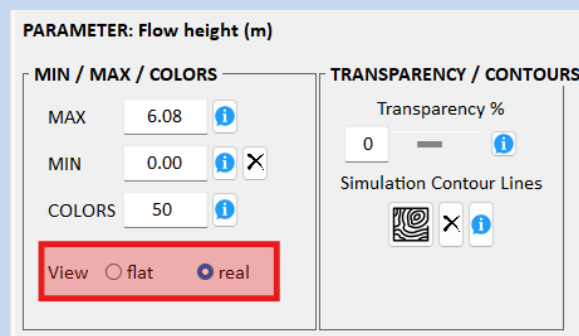
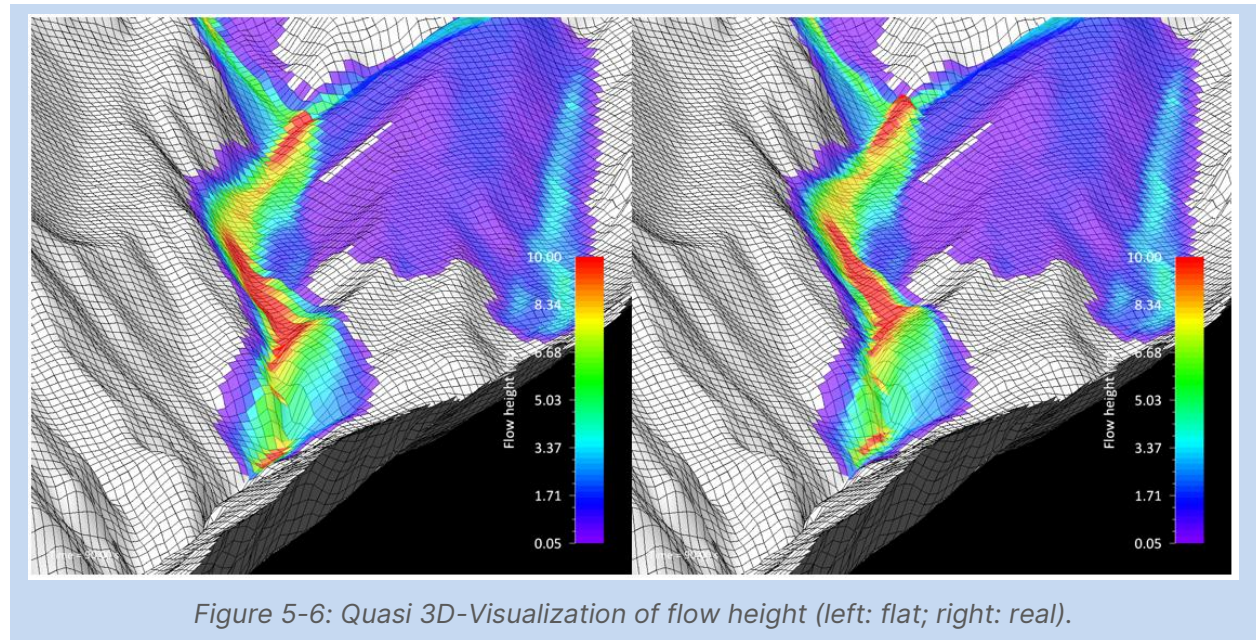


Figure 5-5: Toggle between flat and real view (quasi-3D).



5.2.3 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.

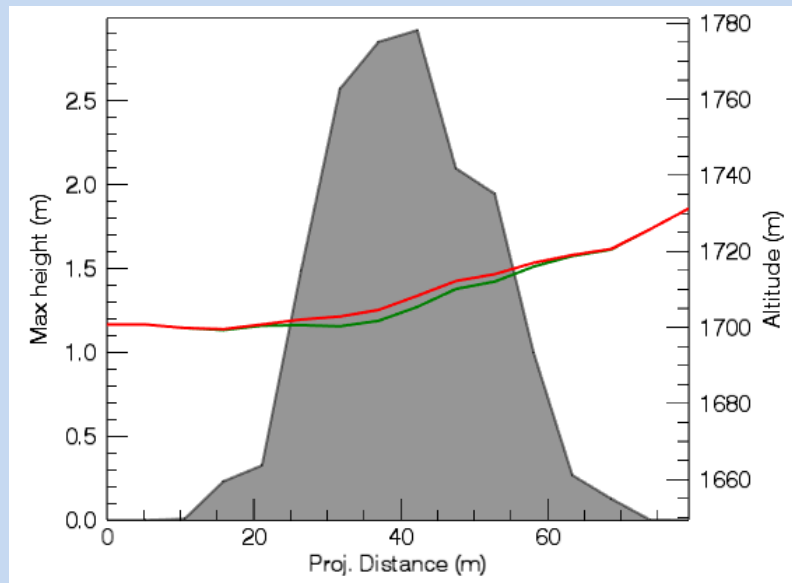


Figure 5-7: Line profile plot.

- 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

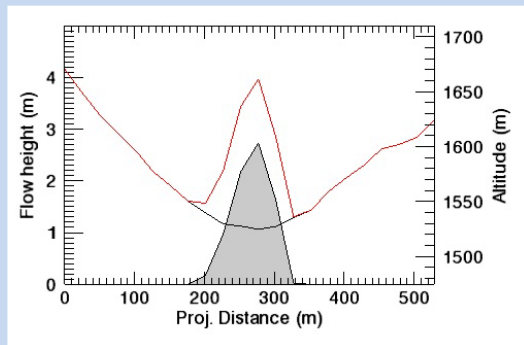
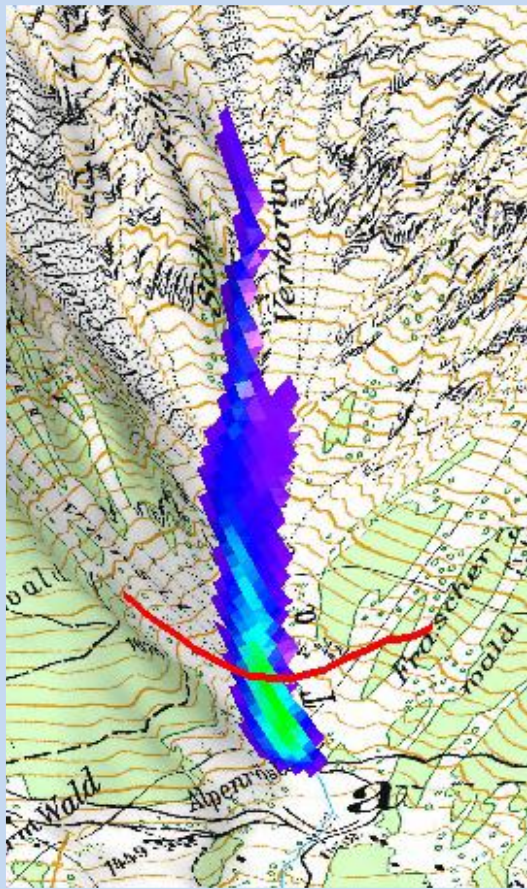


Figure 5-8: Line profile perpendicular to flow direction.

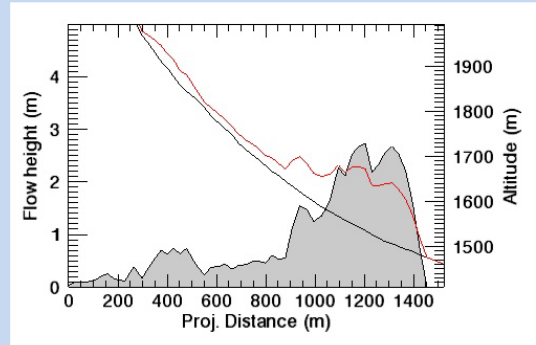
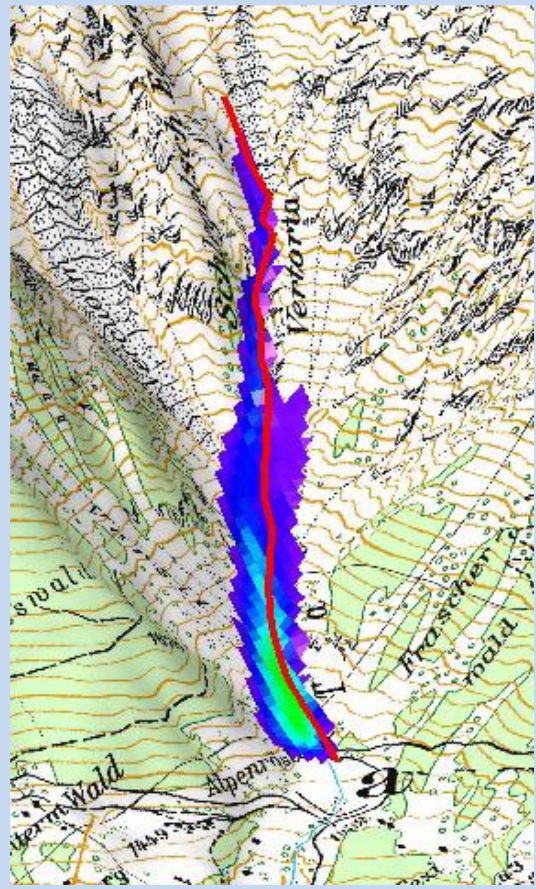


Figure 5-9: Line profile along the flow direction.

- 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.

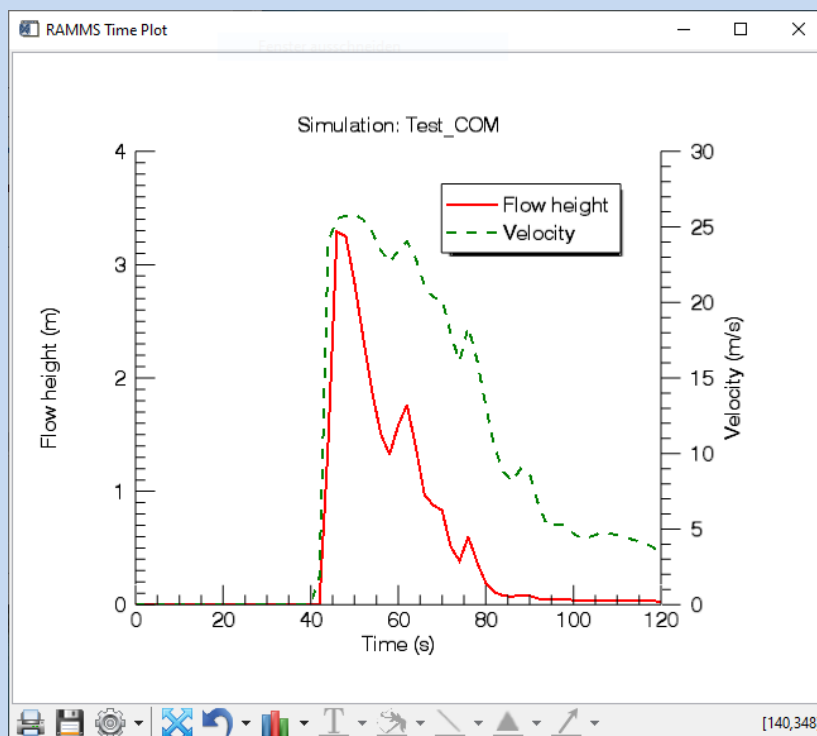


Figure 5-10: Time plot window.

CHAPTER 5: RESULTS

- 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.

5.2.4 Deposition analysis

The Deposition analysis tool can be used to determine the **volume of simulated material** located within a user-defined region of interest (ROI) at a **selected simulation time**. The ROI is defined by a polygon shapefile.

The analysis is based on the flow height results of the currently selected time step. RAMMS identifies the simulation grid cells located within the polygon and calculates the corresponding material volume. The contributions of all grid cells within the ROI are summed to obtain the **total deposition volume**.

To perform a deposition analysis:

- Select the simulation time to be analyzed using the time slider.
- Load or create a polygon shapefile defining the region of interest.
- In the Files tab, right-click the polygon shapefile.

CHAPTER 5: RESULTS

- Select **Deposition analysis**.

The polygon used for the analysis is shown in the map view (**Error! Reference source not found.**).

The resulting analysis window (**Error! Reference source not found.**) reports the **total deposition volume** within the polygon for the selected simulation time. In addition, statistical information about the flow heights within the analyzed region is provided, including the minimum, mean, median, and maximum flow height and the number of data values considered.

The analysis can, for example, be used to estimate the amount of debris accumulated within a deposition area, retention basin, channel section, or other user-defined region.

Note: The calculated volume represents the simulated material located within the selected polygon at the selected simulation time. It should therefore not automatically be interpreted as the total volume that has passed through or entered the region during the entire simulation.

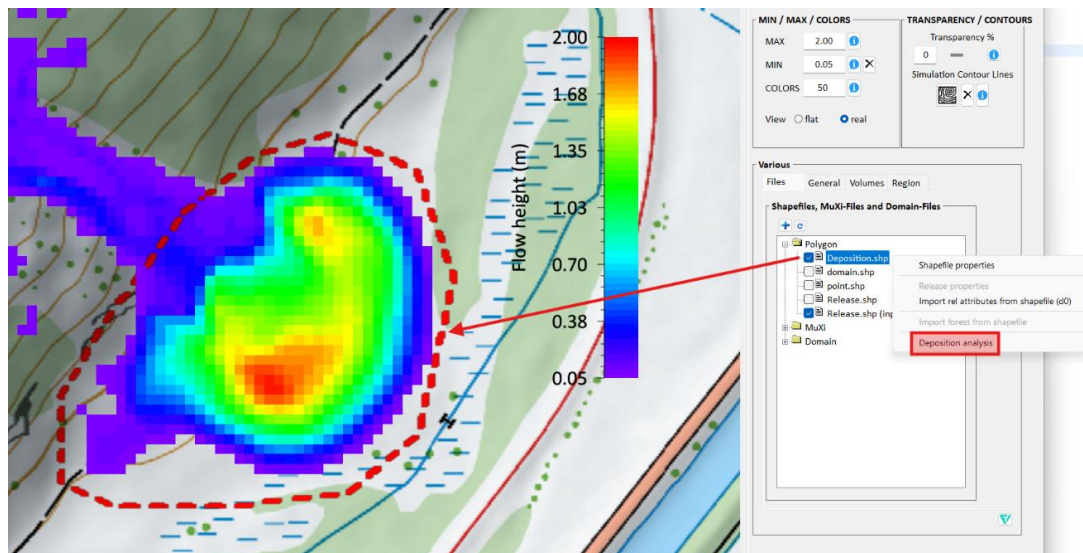


Figure 5-11: Deposition analysis of region of interest.

CHAPTER 5: RESULTS

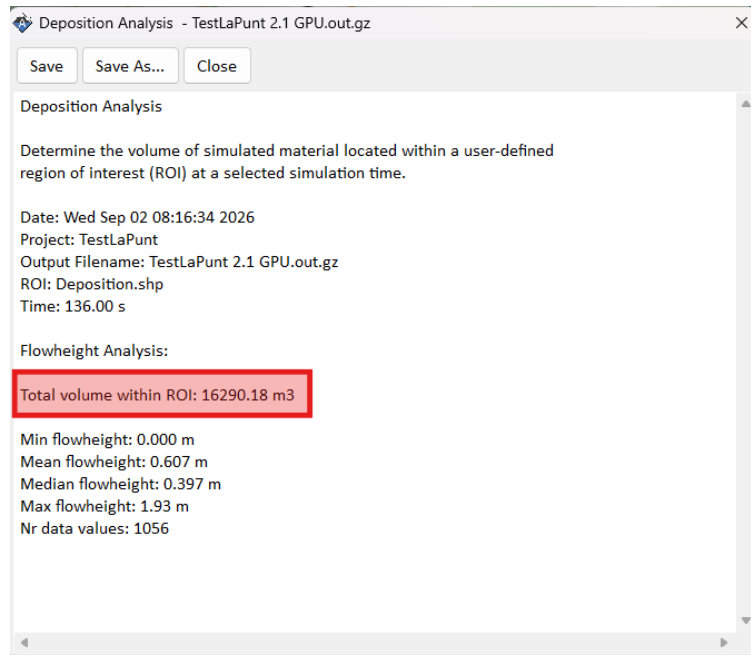


Figure 5-12: Result of a deposition analysis. Total deposition volume (m³) as well as some statistical values are shown (min, mean, max).

CHAPTER 5: RESULTS

5.2.5 Google Earth Export

RAMMS simulation results can be exported for visualization in **Google Earth**. This allows the simulated flow extent and result values to be viewed together with satellite imagery and the three-dimensional terrain representation available in Google Earth.

Google Earth must be installed on the computer to use this feature.

RAMMS exports the currently visualized result as two files:

- a **PNG image** containing the displayed RAMMS result, and
- a **KML file** defining the geographic position and extent of the image in Google Earth.

The PNG image is displayed in Google Earth as a georeferenced ground overlay.

Google Earth Options

Before exporting a result, the **Google Earth Options** should be configured according to the coordinate system of the RAMMS project (Figure 5-13). Correct projection information is required to transform the RAMMS project coordinates to the geographic coordinates used by Google Earth.

For projects in Switzerland, the default settings can normally be used directly.

For projects outside Switzerland, select the appropriate **Projection** and **Spheroid** for the project coordinate system. Depending on the selected projection, additional information such as the UTM zone or projection parameters may be required.

Important: Incorrect projection settings may cause the exported result to appear at the wrong location or to be spatially shifted in Google Earth.

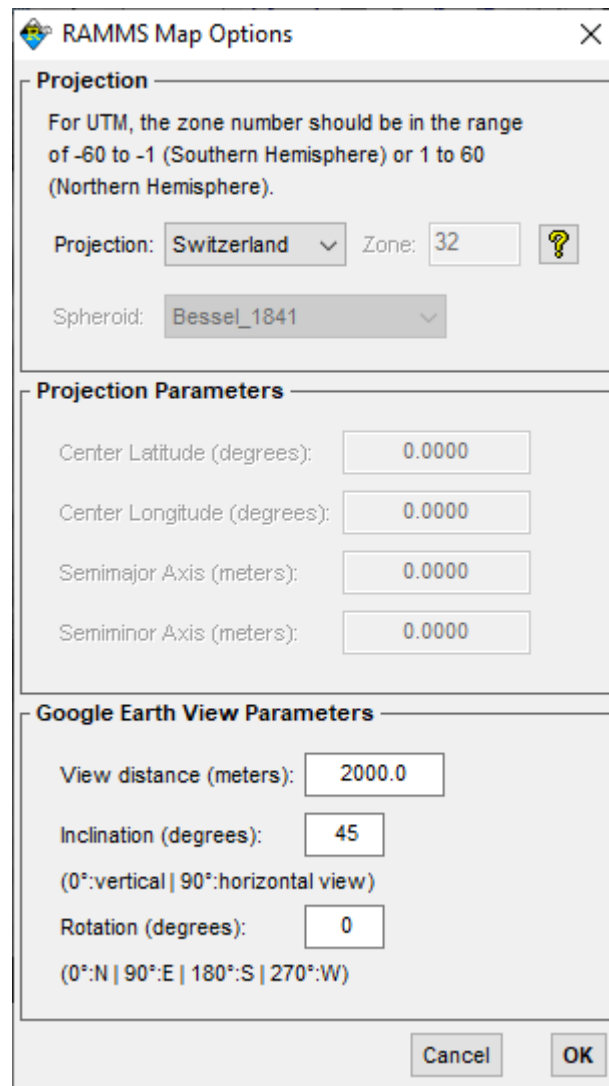


Figure 5-13: Google Earth Options dialog. Specify the coordinate system of the RAMMS project and the initial Google Earth viewing parameters.

Projection settings

The **Projection** field specifies the projection used by the RAMMS project. Available projection types include **Switzerland**, **UTM**, and **State Plane**.

For a UTM projection, specify the corresponding **UTM zone**. Zone numbers from 1 to 60 are used for the Northern Hemisphere and -1 to -60 for the Southern Hemisphere.

The **Spheroid** field specifies the reference ellipsoid associated with the project coordinate system. Select the spheroid corresponding to the coordinate reference system of the project.

Depending on the selected projection, additional values can be entered in the **Projection Parameters** section.

Figure 5-14 shows the available selections for **Projection** and **Spheroid**.

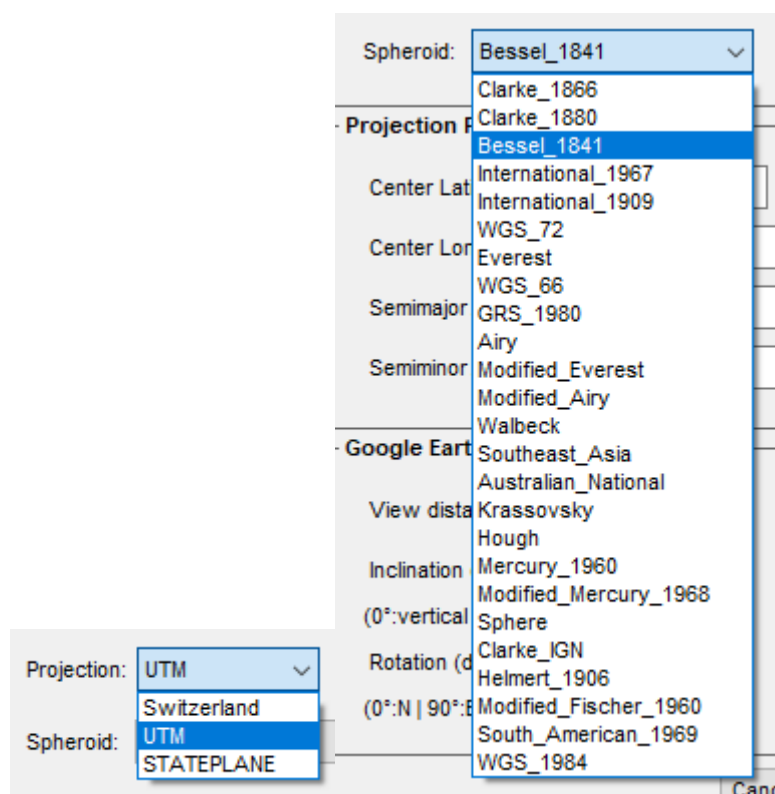


Figure 5-14: Available Projection and Spheroid selections for the Google Earth export.

If you are unsure which settings to use, click the **Help** button  in the Google Earth Options dialog for additional information on the required projection parameters.

Google Earth view parameters

The **Google Earth View Parameters** control the initial viewing position when the exported result is opened in Google Earth. They do not affect the geographic position of the exported result.

The following parameters can be specified:

- **View distance** – initial viewing distance from the result.
- **Inclination** – initial viewing angle, where 0° corresponds to a vertical view and 90° to a horizontal view.
- **Rotation** – initial viewing direction, with 0° = north, 90° = east, 180° = south, and 270° = west.

These parameters only define the initial view and can subsequently be changed freely in Google Earth.

CHAPTER 5: RESULTS

Exporting a result

After the Google Earth settings have been defined, visualize the simulation result and simulation time you want to export.

Then select: **Extras → Google Earth... → Export Result to Google Earth**

RAMMS creates a KML file together with the corresponding PNG image of the currently displayed result. When the KML file is opened in Google Earth, the result image is positioned at the corresponding geographic location and displayed as an overlay on the terrain.

Figure 5-15 shows an example of a simulation result visualized in RAMMS and the corresponding result exported to Google Earth.

Note: The KML file references the corresponding PNG image. Both files are therefore required for the exported result to be displayed correctly in Google Earth.

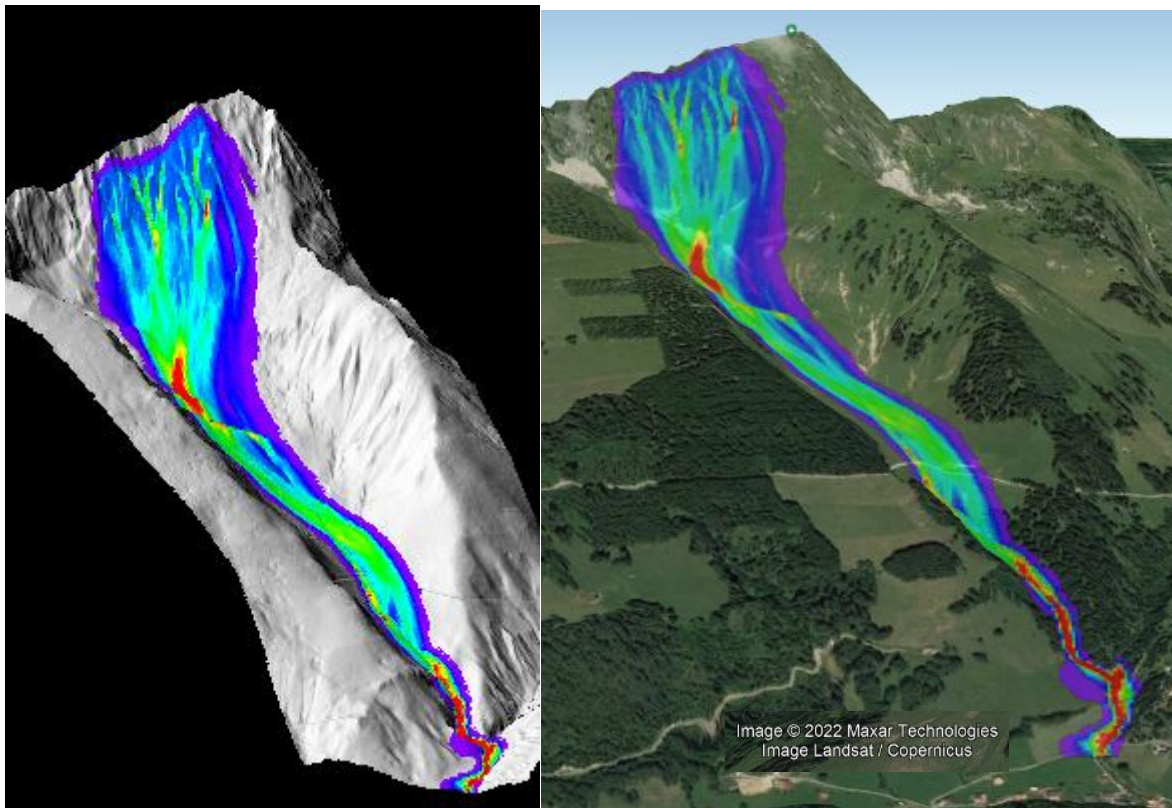


Figure 5-15: Example of a simulation result visualized in RAMMS (left) and exported as a ground overlay to Google Earth (right) (Google Earth imagery © Google, Maxar Technologies, Landsat/Copernicus). s).

CHAPTER 5: RESULTS

5.2.6 Exporting video, GIF animation or image files

Video animation

It is possible to use **FFmpeg** to export videos. FFmpeg is a leading Open-Source multi-media framework. FFmpeg is not bundled with RAMMS due to licensing constraints. You therefore have to download the ffmpeg-binary yourself, by following this procedure once:

1. Download FFmpeg: Visit the official FFmpeg builds page -> <https://www.gyan.dev/ffmpeg/builds/>
2. Choose the correct version: Download the archive named “ffmpeg-release-essentials.7z” or “ffmpeg-release-essentials.zip”
3. Extract and locate *ffmpeg.exe*: After extraction, find *ffmpeg.exe* inside the bin folder.
4. Copy *ffmpeg.exe* to the “bin” folder of your RAMMS installation (You may need administrator privileges to copy files into this location).
5. If you do not have administrator privileges, then copy *ffmpeg.exe* to any location on your computer, and then add the full path to the *ffmpeg.exe* file in the GUI Preferences like this:

FFMPEG <full_path_to_ffmpeg.exe_file> (without <>)
(if the path contains blanks, you need “ ”)

Once added, RAMMS will automatically detect it and enable video export functionality.

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]).

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.

5.3 Avalanche Impact Pressures

5.3.1 Swiss Pressure Guidelines for Hazard Mapping

The approach to calculate **avalanche impact pressures** is closely tied to Switzerland's classification of pressure levels for avalanche hazard mapping. Switzerland employs a scenario-based hazard mapping system that categorizes terrain into distinct hazard zones based on calculated avalanche impact pressures for specific avalanche return periods. These guidelines are essential for land-use planning, infrastructure protection, and engineering design, helping to ensure that buildings and other structures can withstand potential avalanche forces.

Key Pressure Threshold: 30 kPa

A fundamental benchmark in Swiss avalanche hazard zoning is **30 kPa**, which is often used as a threshold for defining high-risk zones. This value is significant because:

- **30 kPa represents the pressure required to destroy an unreinforced building**, making it a practical threshold for determining areas where human life and property are at risk.
- **Structures located in areas with expected impact pressures exceeding 30 kPa require specialized engineering measures or are prohibited from construction.**
- This value has been widely validated through **historical avalanche damage assessments** and structural impact studies.

For example, Switzerland classifies avalanche-prone areas using a **color-coded hazard map system**, which defines different levels of risk based on expected avalanche impact pressures:

Red Zone (High Hazard) – Impact Pressures $\geq$ 30 kPa

- Areas exposed to **high-density, high-velocity avalanches**.
- **Buildings are not permitted, except for avalanche-resistant structures.**
- Infrastructure must be **heavily reinforced**, with strict engineering requirements.

Blue Zone (Moderate Hazard) – Impact Pressures between 3 kPa and 30 kPa

- Avalanche impact pressures **can cause structural damage** but are not necessarily destructive.
- Residential buildings may be permitted but **require reinforced construction** (e.g., impact-resistant walls, deflection structures).

CHAPTER 5: RESULTS

- Infrastructure design must consider **potential dynamic forces and impact duration**.

Yellow Zone (Low Hazard) – Impact Pressures < 3 kPa

- Areas where **light structures may experience damage** but human safety is not significantly threatened.
- Buildings are allowed but may require protective measures such as avalanche fences, reinforced walls, or deflecting berms.

White Zone (No Significant Hazard)

- No avalanche-related building restrictions.

It is crucial to recognize that for impact pressures exceeding **30 kPa**, there is minimal benefit in performing detailed force calculations—these high loads typically govern design decisions outright. However, in the intermediate pressure range of **3 kPa to 30 kPa**, precise calculations become **essential**, as the choice of design formulas can significantly influence construction costs and safety margins. Within this critical range, engineering judgment plays a pivotal role in interpreting simulation results and applying them effectively to real-world scenarios.

Most field measurements of impact pressures—such as those conducted at **Ryggfonn** and **Vallée de la Sionne**—have been recorded on steep slopes where avalanches reach extreme velocities and experience substantial gravitational forces. As a result, these measurements often fall outside the **3 kPa – 30 kPa** range, where detailed impact force calculations are most relevant. Furthermore, few research sites have deployed **strain gauges** or **force sensors** capable of measuring the internal structural forces that engineers need to assess load-bearing requirements. Historical measurements, such as those conducted in Ryggfonn during the **1980s**, made some progress in this regard, but even today, there remains a **critical gap** in our ability to translate **external impact pressures** into the **internal structural forces** that dictate a structure's response.

This distinction is vital. Engineers must not only consider the raw magnitude of impact pressures but also how these forces manifest within structural components, whether as **bending moments, shear forces, or foundation loads**. A thorough understanding of these mechanics is essential for designing avalanche-resistant structures that are both **cost-effective and resilient**.

CHAPTER 5: RESULTS

Practical Considerations for Engineers

Obstacle Geometry Determines Impact Forces

- **Wide structures (e.g., walls, dams, buildings)** experience high pile-up forces, requiring consideration of **pile-up pressure models**.
- **Narrow structures (e.g., pylons, bridge supports)** experience lower pile-up **but higher deflection and shear forces**, demanding a different design approach.

Impact Pressure Varies with Flow Properties

- **Faster, denser avalanches exert greater impact forces**, requiring reinforced structures in high-velocity zones.
- **Wet snow avalanches** tend to have **lower impulsive pressures** and longer interaction durations including more pronounced shearing forces.

Elevated Structures Require Powder Cloud Analysis

- **Powder cloud pressures** are usually lower than core pressures but can still damage tall, lightweight structures like transmission towers.
- **Tall buildings** in avalanche-prone areas should consider combined loading from both core and powder cloud forces.
- **RAMMS::Avalanche DOES NOT calculate air-blast pressures.** To calculate air-blast pressure from powder clouds use the RAMMS::Extended module.

Dynamic Effects Can Amplify Stresses

- **Tall, flexible structures** like masts and pylons can **experience dynamic magnification**, where the **natural frequency of the structure** aligns with impact forces, leading to higher-than-expected internal stresses.
- **Anchoring and foundation design** play a crucial role in mitigating dynamic oscillations.

Engineering Design Should Incorporate Multiple Load Scenarios

- A **single force model may not be sufficient**—combined stagnation, deflection, and shearing loads should be considered for comprehensive structural analysis.

CHAPTER 5: RESULTS

5.3.2 Avalanche-Obstacle Interaction Forces

When an avalanche encounters an obstacle, the impact forces arise from a combination of three key mechanisms (Figure 5-16):

Pile-Up Forces

- Occurs when **snow accumulates in front of an obstacle, forming a compacted wedge**.
- Typically observed in wide obstacles such as walls, buildings, and deflection dams.
- Results in high impact pressures as the incoming mass is rapidly decelerated.

Deflection Forces

- Occurs when **avalanche snow moves around an obstacle** rather than stopping.
- Common for **narrow structures** like bridge supports, pylons, or slender buildings.
- The flow exerts **lateral forces on the obstacle**, which can induce bending moments and shear stresses.

Shearing (Edge) Forces - Jamming and Frictional Loading

- Created when **deflected snow interacts with the surrounding avalanche flow**, causing **localized shear stresses** that are taken up by the obstacle.
- Can significantly increase loading on structures with sharp edges or narrow geometries.

RAMMS::Avalanche calculates both **pile-up forces on wide objects** and **deflection forces on narrow structures**. These forces are associated with fast moving avalanches that usually control the design loads on a particular structure.

CHAPTER 5: RESULTS

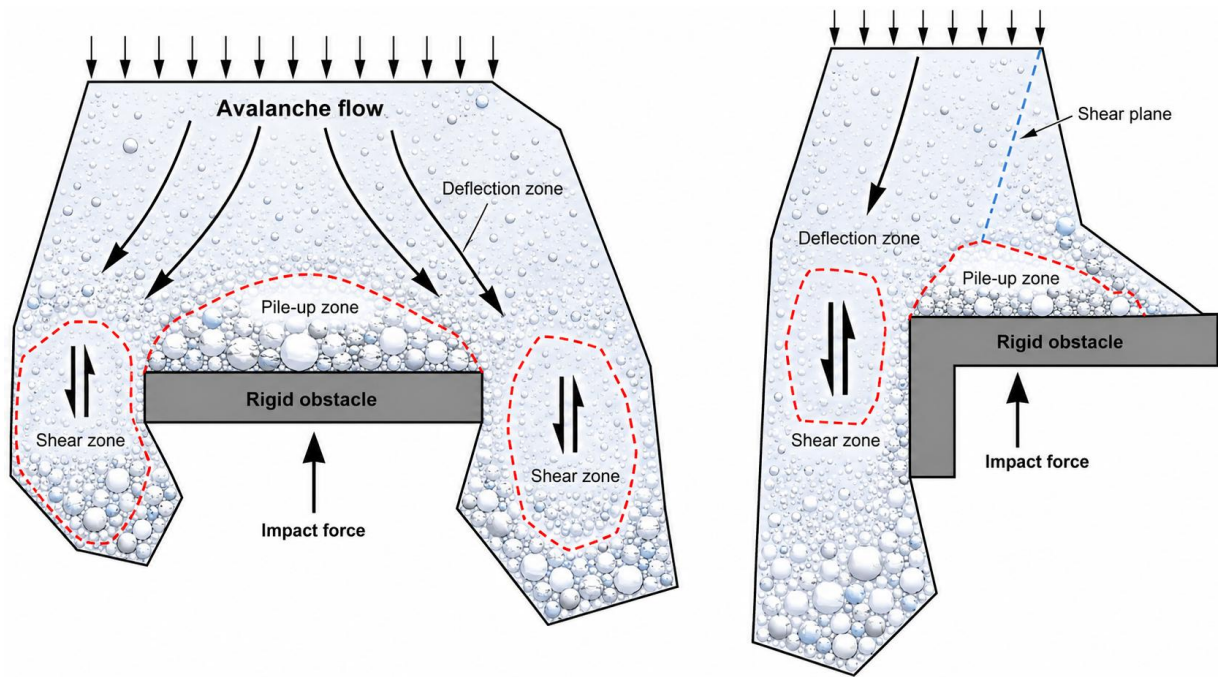


Figure 5-16: When an rigid obstacle is impacted by an avalanche, three different force are created: pile-up forces, deflection forces and shear forces.

RAMMS::Avalanche relies on the well-known **empirical pressure formula** to simplify the complex and dynamic interactions between avalanches and structures. These formulas take the general form:

$$p = \frac{1}{2} \rho C_d \|\vec{u}\|^2$$

where C_d is the pressure coefficient, a key parameter governing avalanche impact forces. This equation establishes a direct relationship between the impact pressure p , the square of the avalanche velocity $\|\vec{u}\|^2$ and the avalanche density ρ . The impact pressure is displayed in terms of kPa, that is a kN force distributed over a unit square m^2 area.

Traditionally, recommended values for C_d are:

1. $C_d = 1$ for thin obstacles, where flow largely deflects around the structure.
2. $C_d = 2$ for wide obstacles, which tend to cause significant pile-up effects.

CHAPTER 5: RESULTS

5.3.3 Impact load per unit width

RAMMS calculates a mean, distributed load according to

$$p' = ph = \frac{1}{2} \rho h C_d \|\vec{u}\|^2,$$

where h is the flow depth measured normal to the terrain, and $\vec{u}$ the depth-averaged flow velocity. As this measure is typically used for long walls, we take $C_d = 2$ and therefore

$$p' = \rho h \|\vec{u}\|^2.$$

The resulting quantity p' has units of force per unit width (kN/m). The values of h and $\vec{u}$ are determined at each simulation time step and therefore the load varies with time $p'(t)$. RAMMS does *NOT* calculate the load from the product of the independently determined maximum pressure and maximum flow depth.

The impact pressure (kPa, kN/m²) includes avalanche speed but does not distinguish between a very shallow and a very deep avalanche. Multiplying by the flow depth integrates the impact pressure over the depth of the flow, resulting in a line load in kN/m.

Table 2: Example for $\rho = 300 \text{ kg/m}^3$. The faster, shallow flow has the higher impact pressure, whereas the slower, deeper flow has the higher depth-integrated impact load.

Flow depth h	Velocity v	$p = \rho v^2$	$h \times p$
0.5 m	20 m/s	120 kPa	60 kN/m
3.0 m	10 m/s	30 kPa	90 kN/m

For a structure oriented perpendicular to the flow, it can be used as a first-order estimate of dynamic line loading. For structures with arbitrary orientation, the velocity component normal to the structure should be used.

The formula for p' assumes that the height of the flow does not change at impact. If at impact, the height increases, then the load can be reduced. The resulting line load can be integrated over the width of the object to find the overall total force acting on the structure.

5.3.4 Visualization of Impact Pressures

There are two possibilities to visualize impact pressures and line loads. The first is to **display maximum pressures over the entire simulation time and domain** (Figure 5-18), the second is to **display the impact pressure at a specific time**.

CHAPTER 5: RESULTS

Maximum pressures and maximum line loads

Maximum pressures (kPa) are displayed using the green P icon found in the upper bar of the graphical interface (see Figure 5-17). Maximum pressures are calculated using $C_d = 2$ and therefore represent impact pressures on wide objects (Figure 5-18). The pressures can be scaled by setting the MAX and MIN settings in the display tab. Maximum pressures on thin objects can be found by dividing the maximum pressures by 2. To display maximum line loads (MN/m) use the blue L icon found next to the pressure icon (Figure 5-17). Similar to the maximum pressures, maximum line loads are calculated with $C_d = 2$.

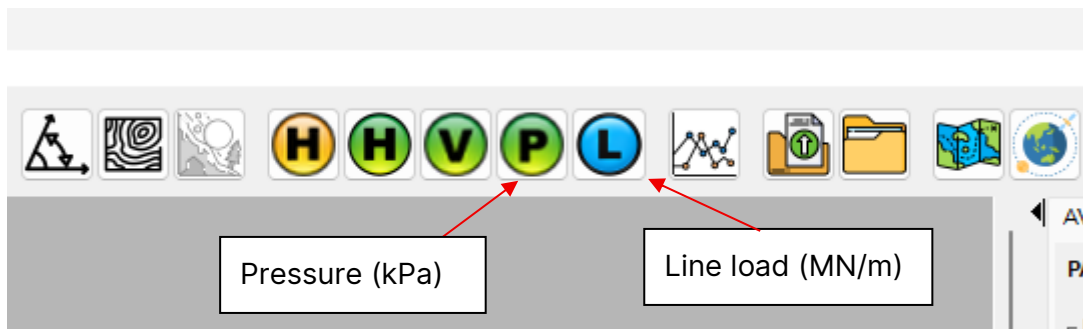


Figure 5-17: Maximum pressures and line loads can be displayed using the green P button ($C_d = 2$) and blue L button ($C_d = 2$).

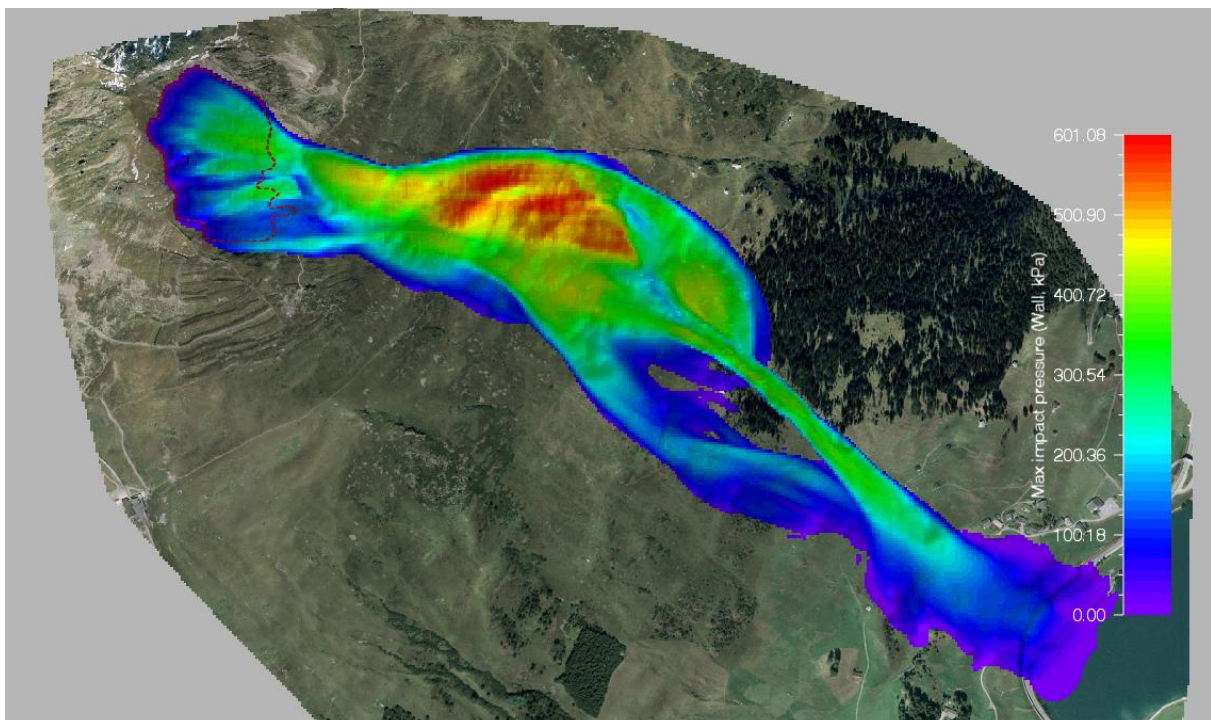


Figure 5-18: Spatial distribution of maximum pressures using the green P icon.

CHAPTER 5: RESULTS

Pressures and Line Loads at a Specific Time and Location

It is also possible to display pressures on wide or narrow objects at a specific time. This is accomplished by selecting **Impact Pressure (Wall, Cd=2)**, **Impact Pressure (Thin Object, Cd=1)** or **Line Load (Cd=2)** in the results menu (Figure 5-19).

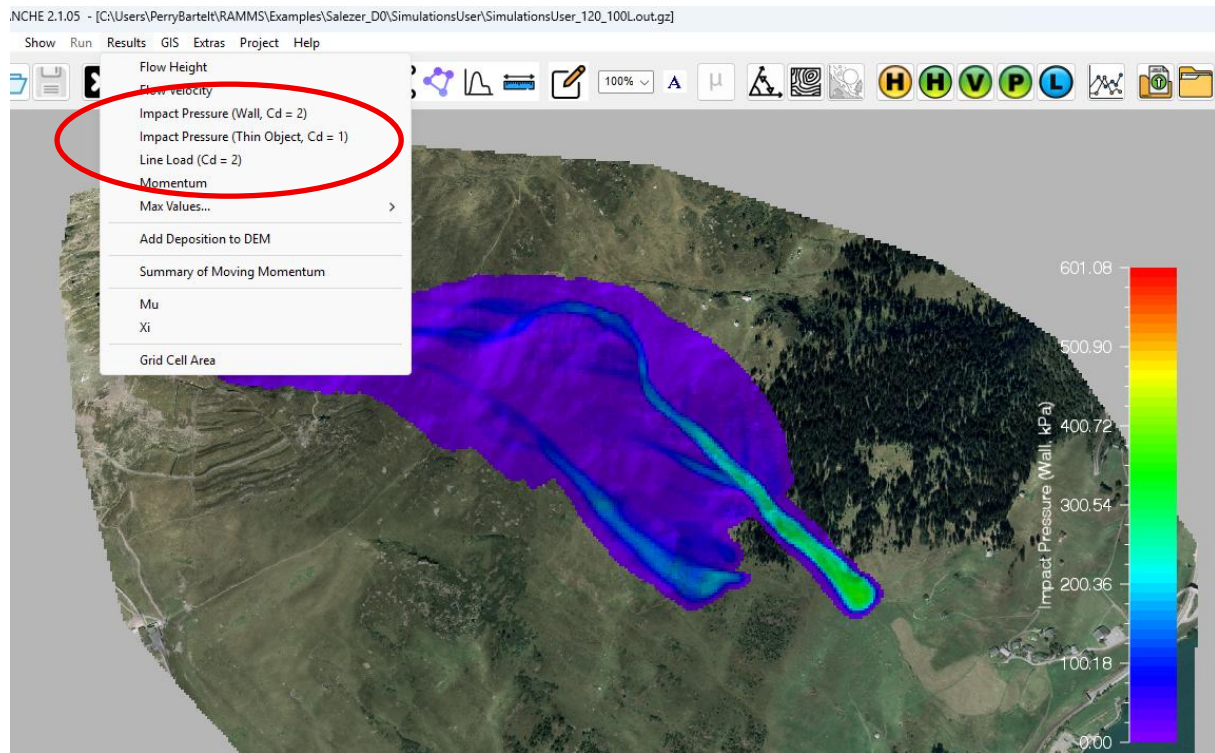


Figure 5-19: Calculated pressures on wide and thin objects at a specific time can be displayed using one of the pressure options in the results menu. The time is set using the time-slider.



The point info button can then be used to select a point in the calculation domain to plot the entire time history of a pressure at that specific location (Figure 5-20). Once the plot has been created it is then possible to export the numerical results.

CHAPTER 5: RESULTS

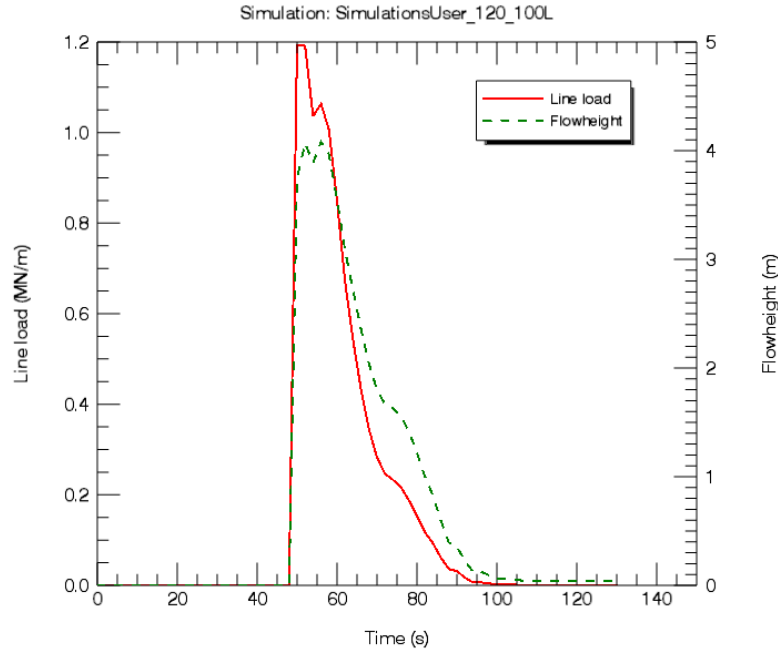


Figure 5-20: Example of a time-history plot of a calculated pressure. The pressure or line load are depicted with the flow height (dashed line).

5.3.5 Pressure on Long Walls (Dams)

In avalanche protection design, long walls (wider than the avalanche flow) experience deflective pressures calculated using:

$$p_n = \rho \|\vec{u}\|^2 (\sin \alpha)^2$$

where α is the deflection angle, see Figure 5-21. This formula is recommended by Swiss and other guidelines. The associated pressure coefficient is:

$$C_d = 2(\sin \alpha)^2$$

with $C_d = 2$ at $\alpha = 90^\circ$ (direct impact). For high-velocity, fluidized avalanches, this value is generally sufficient. Shear pressure is given by:

$$p_s = \mu p_n$$

with $\mu = 0.3$ for **snow/snow or snow/concrete** and $\mu = 0.4$ for **snow on rough walls**.

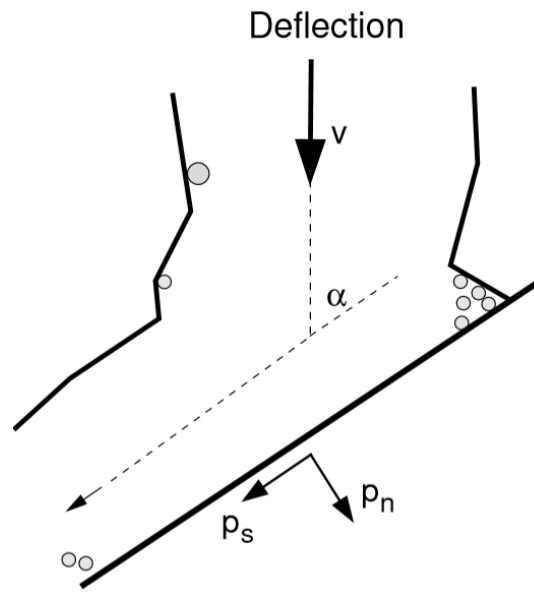


Figure 5-21: Normal p_n and shear p_s pressures on a long wall. The guidelines recommend using $C_d = 2(\sin \alpha)^2$ to calculate the normal pressure. For direct impact $\alpha = 90^\circ$.

5.4 Stopping criteria

RAMMS provides two independent criteria for determining when a simulation has come to a stop:

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

Both criteria can be specified in the **Run Simulation** window (Figure 4-43). If both criteria are active, RAMMS stops the simulation as soon as **either one of the two criteria is fulfilled**. A value of **0** can be used to disable the corresponding stopping criterion.

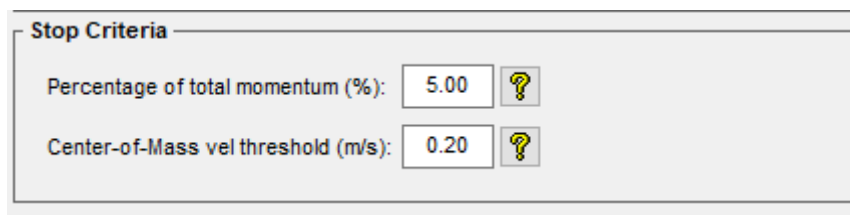


Figure 5-22: Momentum-based and center-of-mass stopping criteria in the Run Simulation window.

The appropriate stopping criterion depends on the simulated process and should always be checked critically. In particular, the stopping behavior may depend on terrain, release volume, friction parameters, and the geometry of the runout area.

5.4.1 Momentum-based Stopping Criterion

The standard RAMMS stopping criterion is based on the **total moving momentum** of the simulated mass.

In classical mechanics, momentum p is defined as the product of mass and velocity (SI unit kgm/s, or, equivalently, Ns):

$$p = mv$$

During the simulation, RAMMS determines the moving momentum of all computational cells at every output step and calculates their total. The largest total momentum reached during the simulation is used as the reference value.

The current moving momentum is then expressed as a percentage of this maximum:

$$\text{Moving momentum (\%)} = 100 * M(t) / M_{\max}$$

where $M(t)$ is the total moving momentum at the current time and $M_{\max}$ is the maximum total momentum reached during the simulation.

When this percentage falls below the user-defined **Percentage of total momentum (%)** threshold, RAMMS considers the flow to have stopped and terminates the simulation.

CHAPTER 5: RESULTS

The development of the moving momentum can be inspected after the simulation by selecting: **Results → Summary of Moving Mass**

The **Moving Mass & Momentum Analysis** window shows the temporal development of the moving momentum and flow volume and allows the selected stopping threshold to be evaluated (Figure 5-23).

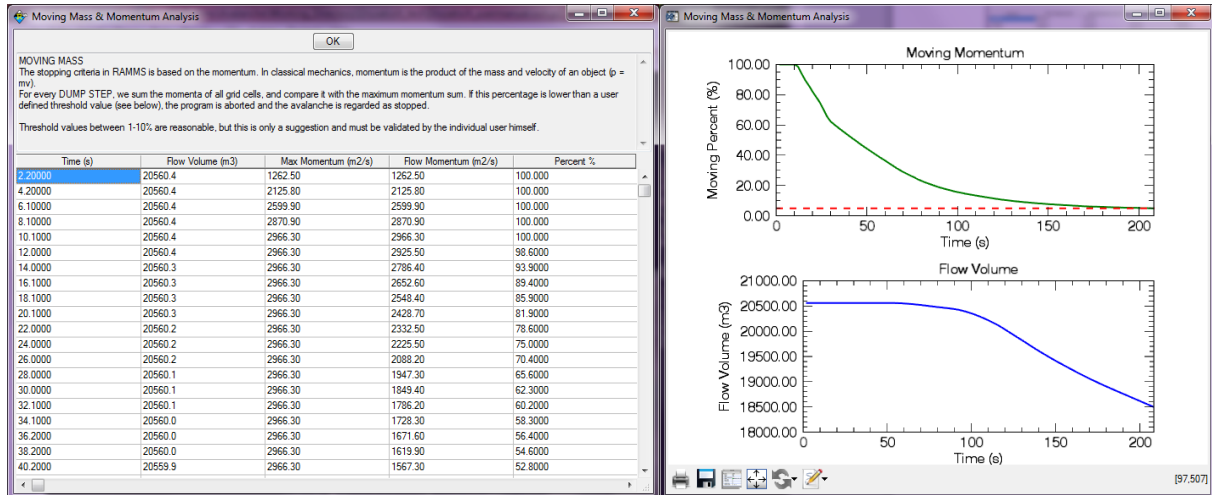
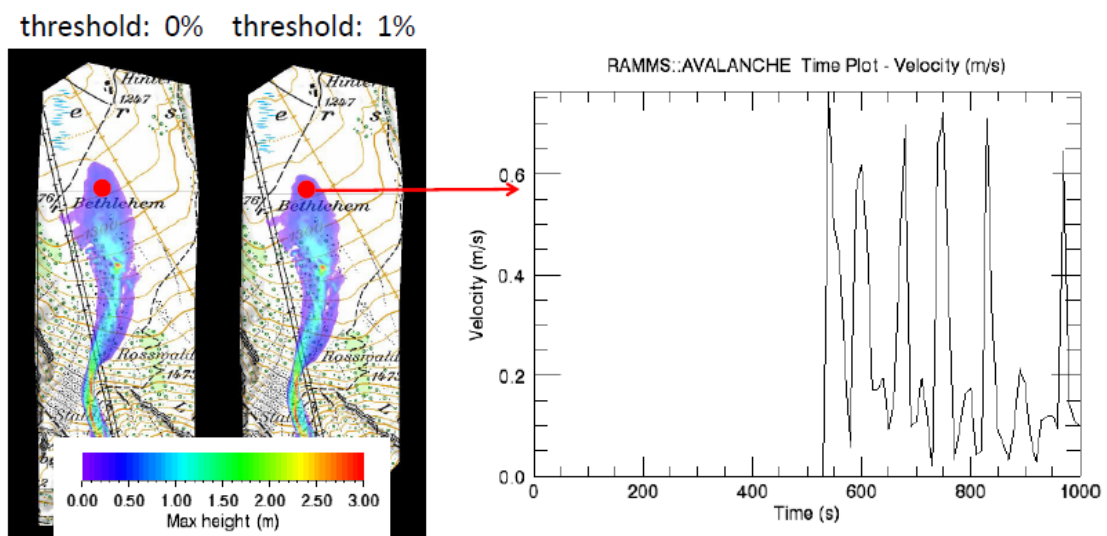


Figure 5-23: Moving Mass & Momentum Analysis used to evaluate the stopping behavior of a simulation.

Threshold values of approximately **1–10%** may provide reasonable starting values. However, there is no universally valid value, and the stopping behavior should be checked for the individual simulation.

A threshold that is **too large** may terminate the simulation prematurely while relevant movement is still occurring. A threshold that is **too small**, on the other hand, may unnecessarily prolong the calculation and allow very small residual velocities, numerical diffusion, or slow creeping of material to continue for a long time.



CHAPTER 5: RESULTS

Figure 5-24: Example of residual low-velocity movement when the momentum-based stopping criterion is disabled (threshold = 0%). Very small velocities may persist for a long time even though the main avalanche mass has effectively stopped.

Figure 5-24 and Figure 5-25 illustrate why the stopping threshold should be selected carefully. In the example shown, very small threshold values allow low-velocity residual movement to continue for a long time. Increasing the threshold reduces this behaviour, while excessively large values may shorten the calculated runout.

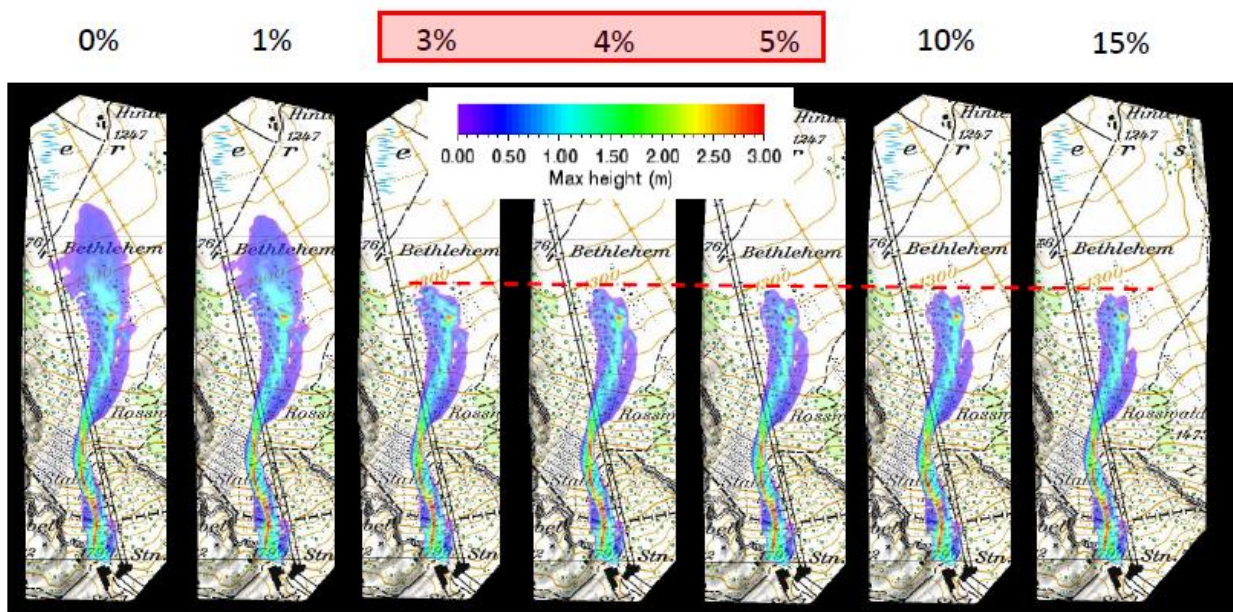


Figure 5-25: Influence of the momentum threshold on the simulated runout. In this example, very small thresholds (<2%) allow residual movement to continue, whereas thresholds of approximately 3–5% provide a reasonable stopping behavior.

As a starting point, a simulation can be performed with a relatively small threshold, for example **1%**, and the resulting moving-momentum curve, flow heights, velocities, and runout should then be inspected. Based on this analysis, the threshold can be adjusted if necessary.

The **Output Log File** records which stopping condition terminated the simulation. If instead the prescribed simulation end time is reached, verify that the calculation time was sufficiently long. If necessary, increase the simulation end time and repeat the calculation.

5.4.2 Center-of-Mass Stopping Criterion

The center-of-mass stopping criterion provides an alternative way of identifying when the bulk of the simulated mass has come to rest.

CHAPTER 5: RESULTS

During the simulation, RAMMS periodically calculates the position of the **center of mass (COM)** of the moving material. From the distance traveled between successive center-of-mass positions, RAMMS determines the corresponding center-of-mass travel speed.

The user specifies a threshold using: **Center-of-Mass vel threshold (m/s)**

When the center-of-mass travel speed falls below this threshold, RAMMS considers the flow to have stopped and terminates the simulation.

This center of mass is weighted by each cell's momentum (height × velocity), not by flow height alone. This keeps the calculation focused on material that is actually still moving: a cell that has deposited and stopped contributes essentially nothing to the center of mass, regardless of how deep the deposit is, so a large stationary deposit cannot mask a smaller, faster-moving flow front that is still advancing

This approach can be useful because it represents the movement of the **bulk of the flow** rather than being controlled by small amounts of material that may continue to move slowly at the margins or at the flow front. It can therefore help prevent simulations from continuing solely because of numerical diffusion or very slow residual creeping.

The center-of-mass criterion is most appropriate for simulations representing **one coherent avalanche path or catchment**.

It should generally **not be used when several independent avalanches or release areas in different catchments are simulated simultaneously**. In such a case, a single global center of mass does not meaningfully represent the stopping behavior of the individual flows. Set the center-of-mass velocity threshold to **0 m/s** to disable this criterion.

For important applications, it can be useful to compare simulations with and without the center-of-mass stopping criterion and verify that the selected threshold does not significantly influence the relevant runout or intensity results.

During the calculation, RAMMS also writes the current center-of-mass coordinates and travel speed to the **RAMMS calculation window** (Figure 5-26). The same information can be reviewed afterwards by selecting: **Extras → View Simulation Standard Output Log**

CHAPTER 5: RESULTS

```
Time 24.0
Step 384 dt 0.0608243 Cfl 0.449928
Hmax 3.78 m Vmax 38.89 m/s

MOVING MOMENTUM: 78.0 percent ( 58243.0 / 74697.3 )
CENTER OF MASS: 2586978.95 1163524.93 --- Travel speed: 22.55 m/s
FLOW VOLUME: 69416.83 m3
NUMERICAL VOLUME LOSS: 0.37 m3
VOLUME CHECK: 1.0
.....

Time 26.0
Step 418 dt 0.057276 Cfl 0.450795
Hmax 3.29 m Vmax 38.55 m/s

MOVING MOMENTUM: 79.3 percent ( 59257.7 / 74697.3 )
CENTER OF MASS: 2587012.00 1163493.13 --- Travel speed: 22.93 m/s
FLOW VOLUME: 69416.78 m3
```

Figure 5-26: Center-of-mass position and travel speed reported in the RAMMS calculation window during a simulation.

5.1 Modifying the DEM for subsequent simulations

5.1.1 Adding structures to the DEM

RAMMS allows the user to modify the digital elevation model (DEM) by locally increasing the terrain elevation. This can be used to represent simplified mitigation structures such as deflecting or catching dams.

The structure is represented only as a modification of the terrain topography. RAMMS does not explicitly model the interaction between the flowing mass and the structure. Consequently, processes such as impact, energy dissipation at the structure, deposition or accumulation in front of the structure, overtopping, and possible failure of the structure are not represented in a physically complete manner.

Important limitations

The **Add DAM to DEM** functionality must be used with great care.

RAMMS should **not be used to dimension a deflecting or catching dam solely on the basis of a simulation containing the dam**, nor should such simulations be used directly to quantify the residual hazard downstream of a mitigation structure.

Important aspects of dam design, including structure geometry, impact loading, energy dissipation, deposition in front of the structure, freeboard, overtopping, and structural stability, must be evaluated using appropriate engineering methods and guidelines. Examples include Johannesson et al. (2009) and Rudolf-Miklau and Sauermoser (2011).

RAMMS is, however, well suited to determine important input parameters for mitigation design, such as **flow depth, velocity, discharge and impact pressure upstream of the planned structure**. A modified DEM can additionally be used to investigate qualitatively how a proposed structure may redirect the simulated flow or alter potential flow paths.

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

A dam can be introduced into RAMMS by creating a polygon shapefile defining its location and then increasing the DEM elevation within this area.

- Create a polygon shapefile at the location of the planned dam.
For a simple dam, one polygon is sufficient. To represent a dam with an intermediate elevation or a more gradual transition between the surrounding terrain and the dam crest, a second polygon can be defined.

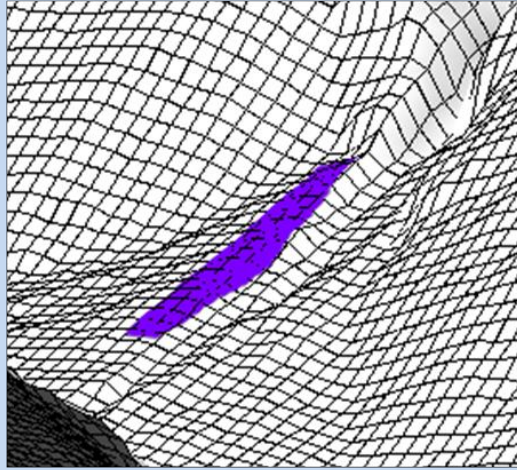


Figure 5-27: Polygon shapefile defining the location and extent of a planned dam.

- Select **GIS → Add DAM to DEM...**

RAMMS provides two options:

- **Enter Relative Dam Height** - raises the DEM by a specified height relative to the existing terrain
- **Enter Dam Elevation** - assigns a specified absolute elevation to the dam

You will first be asked to select the dam shapefile (*.shp). Select the polygon defining the structure. RAMMS then asks whether a second dam shapefile should be used:

- Select **No** if the dam is defined by a single polygon.
- Select **Yes** to load a second, inner polygon and create a two-level representation of the dam.

Depending on the selected option, enter the required dam height or dam elevation. If two polygons are used, also specify the intermediate elevation or height associated with the outer polygon.

- Finally, enter a name for the new XYZ file. RAMMS creates a new terrain file in the project directory containing the modified topography. The original DEM remains unchanged.

CHAPTER 5: RESULTS

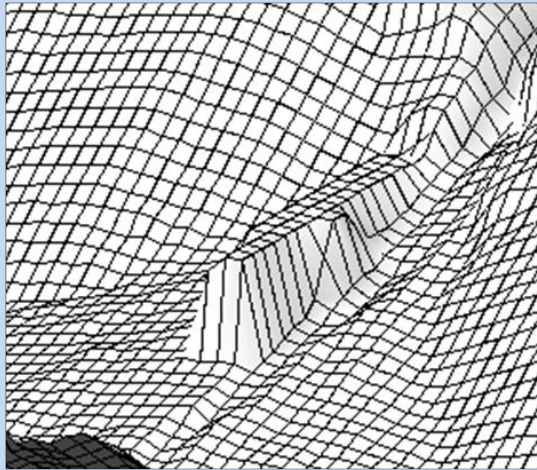


Figure 5-28: Modified DEM after adding the dam geometry defined by the polygon shapefile. The structure is represented by locally increasing the terrain elevation.

Run the simulation with the modified DEM

To use the new terrain in a simulation, open the **Run Simulation** dialog and select the newly created XYZ file under Digital elevation info.

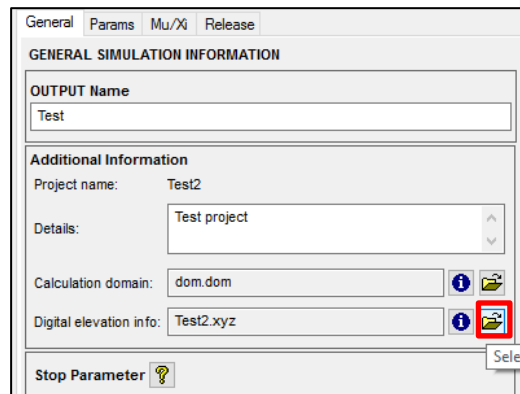


Figure 5-29: Selecting the modified XYZ terrain file containing the dam in the Run Simulation dialog.

Compare simulations with and without the structure

It is generally advisable to run both:

- a reference simulation using the original DEM, and
- a simulation using the modified DEM.

CHAPTER 5: RESULTS

Comparing the two simulations makes it possible to investigate how the topographic modification affects the simulated flow path, runout and spatial distribution of flow depths and velocities.

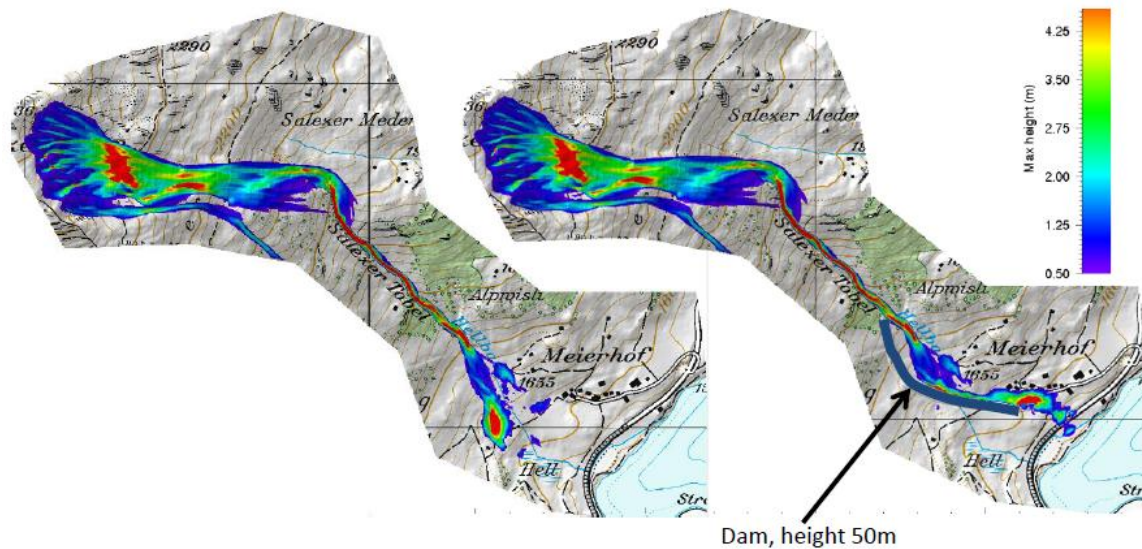


Figure 5-30: Example comparison of simulated maximum flow depth without mitigation structures (left) and a deflecting dam represented in the DEM (right). The arrow indicates the location of the dam.

Numerical limitations when representing dams

The reliability of a modified-DEM simulation depends strongly on the orientation and geometry of the structure.

RAMMS can generally reproduce the **guiding or deflecting effect of structures oriented approximately parallel or oblique to the main flow direction** reasonably well. Greater caution is required for structures extending **across the main flow direction**, particularly catching dams or nearly perpendicular barriers.

There are two main reasons.

- **Missing impact and energy-dissipation processes**

A dam introduced using this method is treated as terrain. The model does not explicitly describe the collision of the flowing mass with the structure or the associated energy dissipation.

Consequently, unrealistically large flow depths or velocities may occur immediately upstream of steep or nearly vertical topographic steps. Such local values should therefore not be interpreted directly as physically meaningful loads on the structure.

CHAPTER 5: RESULTS

- **Influence of numerical resolution**

The numerical solver uses information from neighbouring computational cells. Very narrow structures or abrupt elevation changes represented by only one or very few grid cells may therefore not be adequately resolved.

The representation of a dam should consequently be consistent with the DEM and computational grid resolution. Extremely steep, one-cell-wide structures should be avoided whenever possible.

Recommended representation of steep structures

If numerical problems occur, consider constructing the modified DEM externally in a GIS and representing the structure with progressively increasing side slopes rather than an abrupt elevation step (see Figure below).

A smoother transition distributes the topographic change over several computational cells and generally provides a numerically more robust representation of the structure.

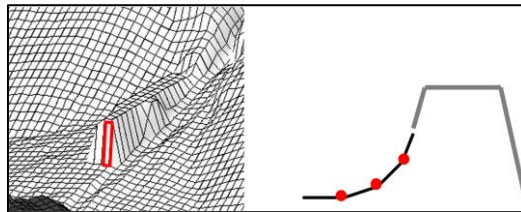


Figure 5-31: Recommended schematic representation of a dam in the DEM using progressively increasing side slopes. Distributing the elevation change over several computational cells avoids an abrupt one-cell topographic step.

Interpretation of results

Simulations containing mitigation structures must be interpreted by experienced users and should always be evaluated together with established engineering design methods.

The modified-DEM approach is primarily intended to investigate the **potential influence of a structure on flow routing**. It should not be used on its own to determine the required dam height, structural loads, overtopping probability or residual hazard downstream of the structure.

For design purposes, RAMMS results from the **undisturbed or appropriately defined design scenario** should be used to derive relevant flow parameters, while the actual mitigation structure should be dimensioned and verified using suitable engineering procedures.

5.1.1 Adding deposited avalanche material to the DEM

Deposits produced by an earlier avalanche can modify the topography and thereby influence the flow path, spreading and deposition of subsequent avalanches.

CHAPTER 5: RESULTS

RAMMS allows the terrain to be updated using the deposition from a simulation. The resulting DEM can then be used as the terrain for a subsequent simulation. This procedure can either be performed **manually for individual simulations or automatically for a sequence of avalanches using the avalanche sequential batch mode**.

Important modelling assumption

The approach assumes that the terrain modification produced by the preceding simulation remains present during the subsequent simulation. In particular, deposition added to the DEM is treated as part of the terrain for the next run.

This should be kept in mind when interpreting multi-surge simulations. In reality, previously deposited material may be remobilized, eroded or otherwise modified by later surges. The suitability of the approach therefore depends on the process being investigated and should be evaluated by an experienced user.

Manually adding deposition to the DEM

The manual procedure is useful if only one or a few successive simulations are required, or if the user wants to inspect the results before defining the terrain for the next simulation.

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

- **Select the relevant simulation time**
Display the simulation results at the time step representing the deposition that should be transferred to the terrain.
In most cases, this will be the final dump step, when the avalanche has largely come to rest. An earlier dump step may be selected if there is a specific reason to use the flow configuration at an earlier time.
- **Add the deposition to the DEM**
Select **Results → DEM Adaptations → Add Deposition to DEM**
RAMMS adds the simulated deposition to the terrain and creates a modified DEM.
- **Save the new terrain**
Enter a name for the new XYZ file. The modified terrain file is stored in the project directory.
- **Use the modified DEM for the next simulation**
Open the **Run Simulation** dialog and select the newly created XYZ file under **Digital elevation info** (see **Error! Reference source not found.**).
The next simulation will then be performed using the terrain modified by the preceding avalanche deposition.

CHAPTER 5: RESULTS

5.1.2 Sequential multi-surge simulations using batch mode

For events consisting of several successive surges, repeatedly updating the DEM and starting a new simulation manually can become cumbersome. RAMMS therefore provides a **batch mode for sequential simulations**.

In batch mode, a series of simulations is defined in a CSV file. After each simulation, the resulting terrain modification is transferred to the next run. The subsequent surge is therefore simulated on the terrain resulting from the preceding surge.

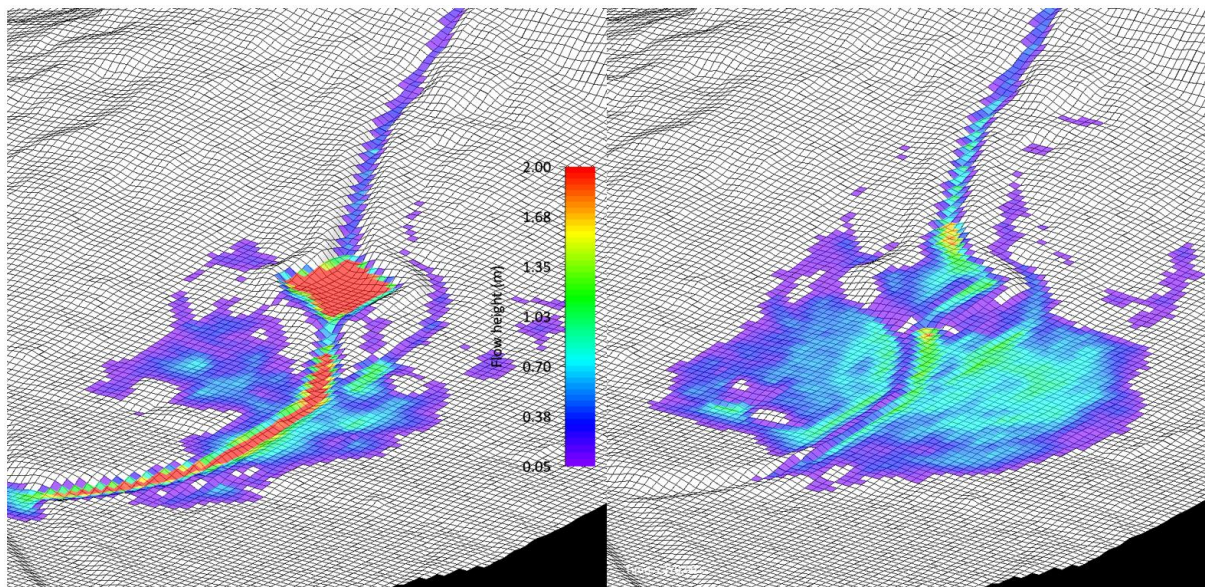


Figure 5-32: First (left) and fourth (right) surge of a sequential multi-surge simulation using identical input parameters. Depositions from each run are incorporated into the DEM for the subsequent run. Progressive filling of the retention basin leads to increased overtopping and downstream propagation during the fourth surge.

Preparing the base scenario

Before starting a batch calculation, create and configure a normal RAMMS avalanche scenario.

Define all settings that should remain common to the batch simulations, including for example:

- calculation domain,
- grid resolution,
- friction and other model parameters not specified individually in the CSV file,
- numerical settings.

Open the **Run Simulation** dialog and configure the base scenario as required, but **do not start the simulation**. Close the dialog using **Cancel**.

CHAPTER 5: RESULTS

Defining the surge sequence in a CSV file

The sequence of simulations is defined in a CSV file. Each row represents one avalanche simulation. The first row is treated as a header and is not evaluated by RAMMS.

Note: The column names in the header are arbitrary and may be written in any language. RAMMS identifies the parameters by their column position, not by the header text. The order of the columns must therefore not be changed.

For example:

RunName	RelFile	RelDepth	SecondRelFile	SecRelDepth	SecRelDelay	MuXi	Comment
Run1	ReleaseMc.shp	1.0		0	15	T10	Using Base DEM
Run2	ReleaseMc.shp	1.0		0	10	T10	Using DEM_after_Run1
Run3	ReleaseMc.shp	1.0		0	12	T10	Using DEM_after_Run2

Figure 5-33: Example CSV file defining three sequential avalanche simulations. Each data row defines one simulation. Column headings are ignored by RAMMS and may be written in any language; parameters are identified by their column position.

Table 3: CSV parameters used to define the individual runs of a sequential avalanche batch simulation.

Column	Parameter	Description
1	Run name	Name assigned to the individual simulation.
2	RelFile	Main release shapefile (absolute or relative path).
3	RelDepth	Release depth (m).
4	SecondRelFile	Secondary release shapefile (absolute or relative path, optional).
5	SecRelDepth	Release depth (m) of secondary release shapefile.
6	SecRelDelay	Release delay (s) of secondary release shapefile.
7	MuXi	Variable friction muxi file. Constant mu and xi values are not supported.
8	Comments	Optional description of the individual simulation.

CHAPTER 5: RESULTS

The CSV filename is also used to create a dedicated batch directory within the RAMMS project. The input files, simulation results and modified terrain files generated during the sequence are stored in this directory.

Note: Only parameters provided in the CSV file are changed between runs. All other parameters are inherited from the prepared base scenario

Starting the batch simulation

After preparing the base scenario:

1. Select **Run → Run Sequential Batch with Terrain Updates**.

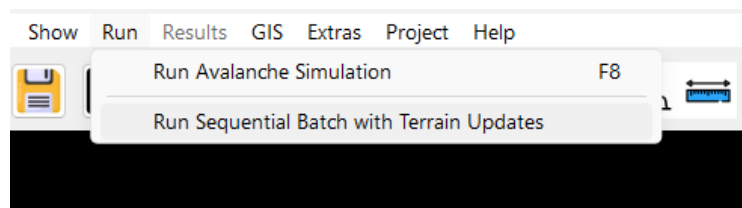


Figure 5-34: Start Run Sequential Batch procedure.

2. Select the CSV file defining the sequence of surges.
3. RAMMS reads the simulations row by row and executes them sequentially.
4. After each run, the terrain modification is used to generate the DEM for the following run.
5. Progress is displayed while the batch sequence is running.

Batch output

For each run, RAMMS stores the files required to reproduce and analyse the individual simulation. The batch documentation lists, among others:

- the RAMMS input file (.av2),
- the simulation output (.out.gz),
- the modified DEM (.tif),
- the deposition raster (.tif),
- and auxiliary terrain files such as .xyz and .var

Each simulation remains available as an individual simulation, and the intermediate DEMs allow the evolution of the terrain throughout the surge sequence to be inspected.

Interpretation of sequential surge simulations

CHAPTER 5: RESULTS

Sequential simulations can be useful for investigating how terrain changes caused by earlier avalanches influence subsequent flow paths and deposition patterns.

The method should not, however, be interpreted as a complete physical model of the interaction between successive surges. Material added to the DEM becomes part of the terrain representation used by the next simulation. Whether this approximation is appropriate depends on the expected stability, remobilization and erosion of the preceding deposits.

Results should therefore be evaluated together with field observations, event chronology and appropriate engineering judgement.

6 References

- [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.
- [7] Voellmy, A., 1955: Über die Zerstörungskraft von Lawinen. *Schweizerische Bauzeitung* 73, 212-285. [Original description of the Voellmy friction law - primary theoretical basis of the RAMMS::Avalanche friction model.]
- [8] Christen, M., Kowalski, J. and Bartelt, P., 2008: Calculation of dense snow avalanches in three-dimensional terrain with the numerical simulation program RAMMS. In: *Proceedings of the International Snow Science Workshop (ISSW)*, Whistler, Canada. [Primary RAMMS development paper - should be cited in technical reports using RAMMS::Avalanche.]
- [9] Christen, M., Kowalski, J. and Bartelt, P., 2010: RAMMS: Numerical simulation of dense snow avalanches in three-dimensional terrain. *Cold Regions Science and Technology* 63, 1-14. [Primary peer-reviewed RAMMS validation paper - key reference for professional use.]
- [10] Christen, M., Bartelt, P. and Kowalski, J., 2010: Back calculation of the In den Arelen avalanche with RAMMS: interpretation of model results. *Annals of Glaciology* 51, 161-168. [Calibration and validation example for a Swiss avalanche event.]
- [11] Casteller, A., Stoeckli, V., Villalba, R. and Mayer, A.C., 2008: An evaluation of dendrochronological indicators of snow avalanche activity in the Patagonian Andes. *Cold Regions Science and Technology* 51, 110-122. Also: Casteller, A. et al. (2008) Validating numerical simulations of snow avalanches using dendrochronology: the Cerro Ventana event in Patagonia, Argentina. *Natural Hazards and Earth System Sciences* 8, 433-443. [Back-calculation methodology for μ and ξ from 10 documented Pyrenean avalanche events.]

CHAPTER 6: REFERENCES

- [12] Buehler, Y., Kumar, S., Veitinger, J., Christen, M., Stoffel, A. and Snehmani, 2013: Automated identification of potential snow avalanche release areas based on digital elevation models. *Natural Hazards and Earth System Sciences* 13, 1321-1335. Related: Buehler, Y. et al. (2011) sensitivity of RAMMS to DEM resolution. [Automated release area identification and DEM resolution sensitivity for RAMMS::Avalanche applications.]
- [13] Feistl, T., Bebi, P., Christen, M., Margreth, S., Diefenbach, L. and Bartelt, P., 2015a: Forest braking mechanisms in RAMMS avalanche simulations. *Natural Hazards and Earth System Sciences* 15, 2349-2365. [Quantification of forest braking in RAMMS.]
- [14] Sovilla, B., Margreth, S. and Bartelt, P., 2007: On snow entrainment in avalanche dynamics calculations. *Cold Regions Science and Technology* 47, 69-79. Also: Sovilla, B. et al. (2008) Impact pressures and flow regimes in dense snow avalanches observed at the Ryggfjonn test site. *J. Geophys. Res.* 113. [Experimental basis for avalanche impact pressure design values - relevant to pressure output interpretation.]
- [15] Sartoris, G. and Bartelt, P., 2000: Upwinded finite difference schemes for dense snow avalanche modelling. *International Journal for Numerical Methods in Biomedical Engineering* 16, 817-837. [Numerical scheme used in RAMMS::Avalanche solver.]

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::Avalanche and its applications can be found on the module page at <https://www.ramms.ch/ramms-avalanche/>. Check out the *Publication*-list 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 m ³)		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 m ³)		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 ($\mu=\delta$, $\xi=\text{fix}$)		0.02	400	0.02	400	0.02	400	0.02	400

Small avalanche (5 - 25'000 m ³)		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 m ³)		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 ($\mu=\delta$, $\xi=\text{fix}$)		0.02	400	0.02	400	0.02	400	0.02	400

7.2 Third-Party Software

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

7-zip:

- RAMMS uses the 7-Zip command-line tool (7za.exe) developed by Igor Pavlov. Licensed under the GNU LGPL v2.1.
- 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/>.

Fanning Software Consulting:

- Some utility routines by Fanning Software Consulting (Scale_Vector() or FPUFIX() among others) are used in RAMMS.

```
*****;
;
; Copyright (c) 1998-2013, by Fanning Software Consulting, Inc. All rights reserved. ;
; Redistribution and use in source and binary forms, with or without ;
; modification, are permitted provided that the following conditions are met:
;
; * Redistributions of source code must retain the above copyright ;
; notice, this list of conditions and the following disclaimer. ;
; * Redistributions in binary form must reproduce the above copyright ;
; notice, this list of conditions and the following disclaimer in the ;
; documentation and/or other materials provided with the distribution. ;
; * Neither the name of Fanning Software Consulting, Inc. nor the names of its ;
; contributors may be used to endorse or promote products derived from this ;
; software without specific prior written permission. ;
;
; THIS SOFTWARE IS PROVIDED BY FANNING SOFTWARE CONSULTING, INC. "AS IS"
AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE
IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PUR-
POSE ARE DISCLAIMED. IN NO EVENT SHALL FANNING SOFTWARE CONSULTING, INC.
BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CON-
SEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUB-
STITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; LOSS OF USE, DATA,
OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY
OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEG-
LIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE,
EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.
*****;
```