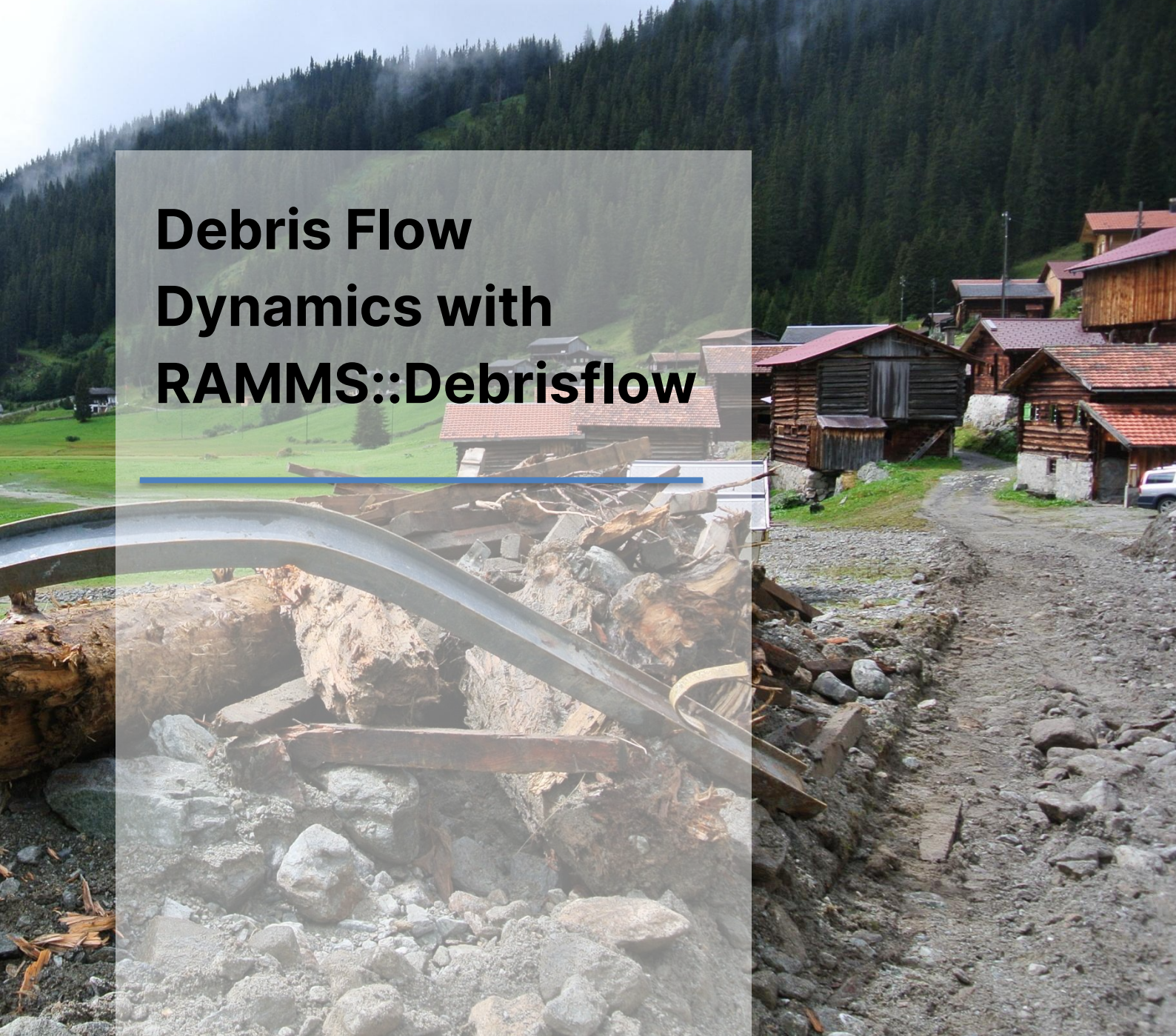


Debris Flow Dynamics with RAMMS::Debrisflow



RAMMS AG

September, 2026

Perry Bartelt & Marc Christen



Title picture: © WSL

Manuscript update

September 2026

Table of Content

1	Introduction	1
1.1	Motivation	2
1.2	RAMMS.....	2
1.3	Learning by doing.....	3
2	Installation and Setup	5
2.1	System requirements	5
2.1.1	GPU acceleration	5
2.2	Installation and Licensing.....	6
2.2.1	Multilingual Installation	6
2.2.2	New Update Procedure	6
2.2.3	Administrative Privileges Requirement.....	7
2.2.4	Licensing.....	7
2.3	First-Time Startup and License Registration.....	7
2.3.1	License Transfer.....	9
2.3.2	Module availability	9
2.3.3	Registering and updating Avalanche and Debrisflow licenses.....	9
3	The RAMMS::Debrisflow GUI.....	11
3.1	Starting the Graphical Interface	11
3.2	Project Creation.....	12
3.2.1	Understanding the New Project Window	13
3.2.2	DEM Files *.tif and *.asc	16
4	Setup and Theory.....	17
4.1	Preparations.....	17
4.1.1	Topographic data - Digital Elevation Model (DEM).....	17
4.1.2	Project and Scenarios.....	19
4.2	Release information.....	19

4.3	Friction information	20
4.3.1	Physical friction model.....	20
4.3.2	Yield stress	21
4.3.3	Curvature	22
4.3.4	Calibration of the friction parameters μ and ξ	22
4.4	Erosion information	23
4.4.1	Interpreting the Default Erosion Thresholds	24
4.4.2	Sensitivity of Results to the Erosion Rate	24
4.4.3	Site Transferability of the Default Coefficients	24
4.4.4	Spatially and Temporally Varying Erosion Parameters.....	25
4.5	Flexible barriers (nets) information	25
4.6	Calculation parameters	26
4.7	Preferences	26
4.8	Working with the RAMMS GUI	28
4.8.1	Open input- and output-files.....	28
4.8.2	Visualizing shapefiles and domain-files	29
4.8.3	Hillshade visualization.....	31
4.8.4	Changing maps and orthophotos (aerial images)	32
4.8.5	Moving, resizing, rotating, viewing	34
4.8.6	Colorbar.....	36
4.8.7	How to save input files and program settings.....	37
4.8.8	About RAMMS	39
4.9	Preparing a simulation	40
4.9.1	Release area	40
4.9.2	Input hydrograph.....	45
4.9.3	Erosion	49
4.9.4	Flexible barriers (nets)	52
4.9.5	Calculation Domain	56
4.10	Running a simulation	59
4.10.1	Run Batch Simulations	67
5	Results.....	69

5.1	Log Files and Project information.....	70
5.1.1	Output Log File.....	70
5.1.2	Input Log File.....	72
5.1.3	Project Region Information.....	72
5.2	Visualization and analysis of the results.....	73
5.2.1	Maximum Results and Dump-Step Results.....	73
5.2.2	Visualize different parameters.....	74
5.2.3	Line profile plot and time plot.....	76
5.2.4	Deposition analysis.....	80
5.2.5	Google Earth Export.....	82
5.2.6	Exporting video, GIF animation or image files.....	86
5.3	Impact Pressures.....	87
5.3.1	Visualization of Impact Pressures.....	87
5.3.2	Impact load per unit width.....	90
5.4	Stopping criteria.....	91
5.4.1	Momentum-based stopping criteria.....	91
5.4.2	Center-of-mass stopping criteria.....	95
5.5	Barrier retention results.....	97
5.6	Modifying the DEM for subsequent simulations.....	97
5.6.1	Adding structures to the DEM.....	97
5.6.2	Adding deposited debrisflow material to the DEM.....	103
5.6.3	Sequential multi-surge simulations using batch mode.....	104
6	Applications.....	108
6.1	Calibration.....	108
6.2	Input hydrograph in comparison with block release.....	112
6.2.1	Comparison between a block release and an input hydrograph....	112
6.2.2	Discussion.....	113
6.2.3	Summary.....	113
6.3	Erosion example.....	114
6.4	Slush Flows.....	117

6.4.1	Definition and Hazard Characteristics.....	117
6.4.2	Release Zone Identification	117
6.4.3	RAMMS Modelling Approach.....	119
6.4.4	Recommended Voellmy Parameters by Return Period	119
6.4.5	Release Area Setup.....	120
6.4.6	Erosion Module Settings.....	121
6.4.7	Digital Elevation Model and Grid Resolution	122
6.4.8	Recommended Simulation Workflow	123
6.4.9	Case Study Summary	124
6.4.10	Limitations and Uncertainties.....	124
7	References.....	126
8	Appendix	130
8.1	Third-Party Software.....	130

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,
- rock-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.

Debris flows pose a major risk to settlements, transport infrastructure and torrent-crossing structures in mountainous and hilly terrain. Physically based 3D simulation tools like RAMMS::Debrisflow allow engineers to quantify runout extent, impact pressure and channel-bed erosion for events where historical records alone are insufficient.

1.2 RAMMS

The RAMMS (RAPid Mass Movement Simulation) software system contains four 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).

CHAPTER 1: INTRODUCTION

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

RAMMS::Debrisflow implements a Voellmy-fluid friction model with depth-averaged mass and momentum conservation, combined with a physically based erosion module for channel-bed entrainment (Frank et al. 2015), enabling simulation of channelized and hillslope debris flows — including volume growth along the flow path — on complex 3D terrain at engineering time scales.

RAMMS::RockIce is an advanced engineering tool designed for use by hazard professionals to analyze both pre-event and post-event hazard scenarios. It is particularly well-suited for assessing situations where avalanche deposits may be re-mobilized into secondary debris flows. A key strength of **RAMMS::RockIce** lies in its ability to predict the water and ice content within avalanche deposits, enabling a detailed understanding of potential remobilization processes. This is made possible by the model's capability to calculate the timing and extent of ice melt within the flowing mass, offering critical insights into the evolving dynamics of complex alpine hazards.

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::Debrisflow. Exercises exemplify different steps in setting up and running a RAMMS simulation especially in Chapter 3 '*Setting up a simulation*'. However, to get the most from the manual, we suggest reading it through while simultaneously having the RAMMS program open, learning by doing.

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::DEBRISFLOW:

- 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 the CPU as before. Nothing needs to be configured for this. To override the

CHAPTER 2: **Installation** and Setup

automatic choice, and force the CPU or GPU solution, use one of 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::DEBRISFLOW setup file "[ramms_avaldbf_setup.exe](https://www.ramms.ch/ramms-debrisflow/ramms_avaldbf_setup.exe)" from <https://www.ramms.ch/ramms-debrisflow/> (Downloads tab 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::Debrisflow 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.

2.2.2 New Update Procedure

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

CHAPTER 2: **Installation** and Setup

- **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::Debrisflow**, **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::Debrisflow** 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.

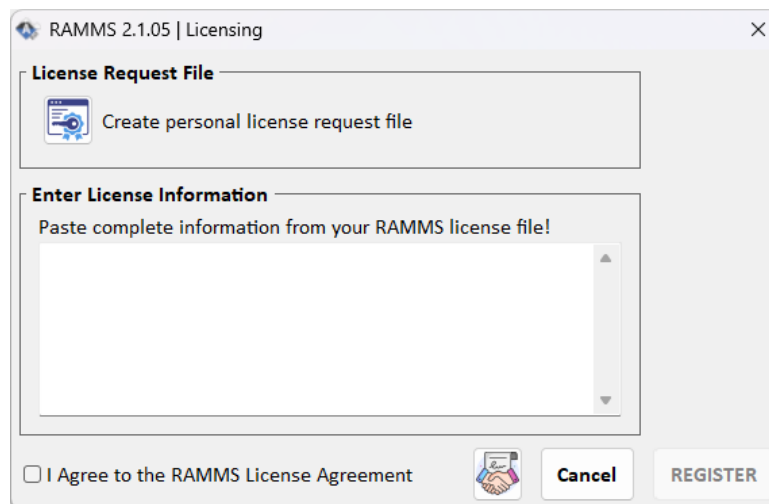
Please note that this splash screen is part of the IDL Virtual Machine startup procedure. RAMMS will not continue to the main interface until the splash screen has been clicked. To continue, click once anywhere on the splash screen image.

CHAPTER 2: **Installation** and Setup



Figure 2-1: IDL Virtual Machine splash screen

RAMMS will then open the main **Graphical User Interface** (GUI), providing a structured and user-friendly workspace. You will then see the **RAMMS | Licensing** window (**Error! Reference source not found.**):



*Figure 2-2: Enter the **full content** of your license file in the **Licensing** 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.

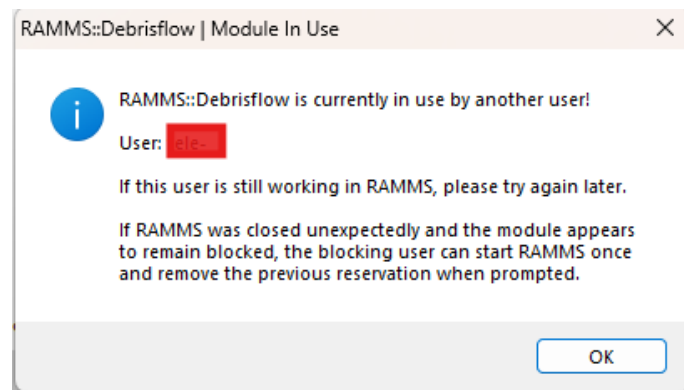


Figure 2-3: RAMMS showing the name of the user currently blocking the module.

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.

CHAPTER 2: **Installation** and Setup

- **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::Debrisflow 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::Debrisflow** icon, the **Graphical User Interface (GUI)** appears,

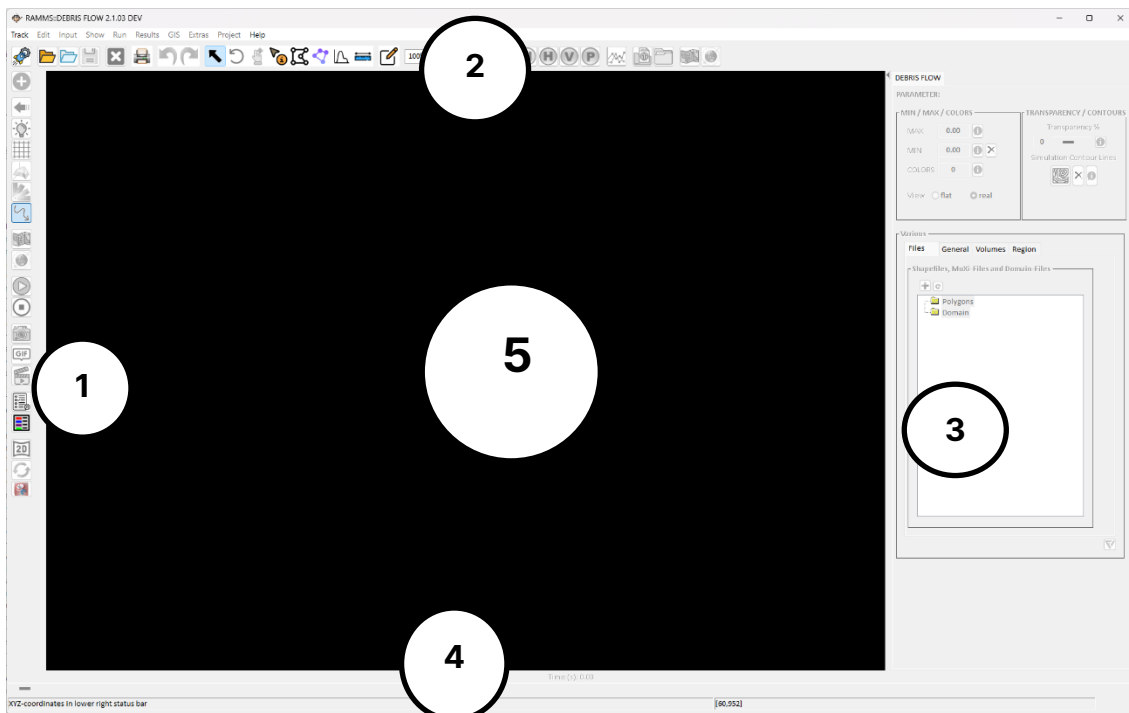


Figure 3-1: RAMMS::Debrisflow GUI at startup. The interface is structured into five main regions.

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.

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::Debrisflow** 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.

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 (see Figure 3-3).

CHAPTER 3: The RAMMS::Debrisflow GUI

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

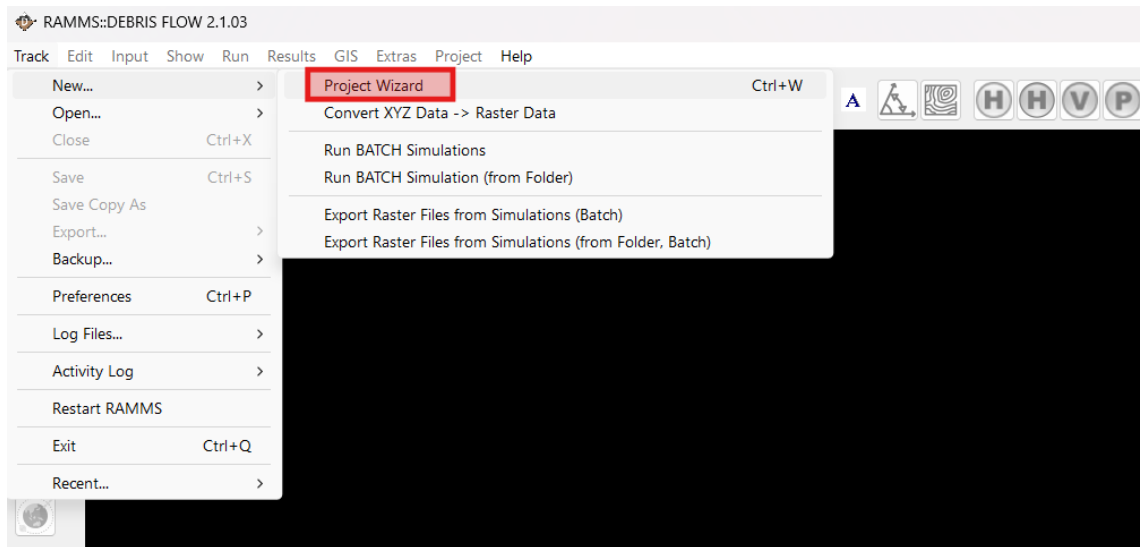


Figure 3-2: Activating the project wizard via pull-down menus.

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. DEM File Selection (Middle Section)

- Here, users import the DEM file, which defines the terrain for the simulation.
- Once selected, **RAMMS::Debrisflow** automatically analyzes and reports the DEM resolution, typically 0.5-5m. 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 debrisflow dynamics simulations**, leading to unnecessary computational complexity.

3. Project Description (Lower Section)

- This section allows users to input descriptive text about the project.

CHAPTER 3: The RAMMS::Debrisflow GUI

- 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.

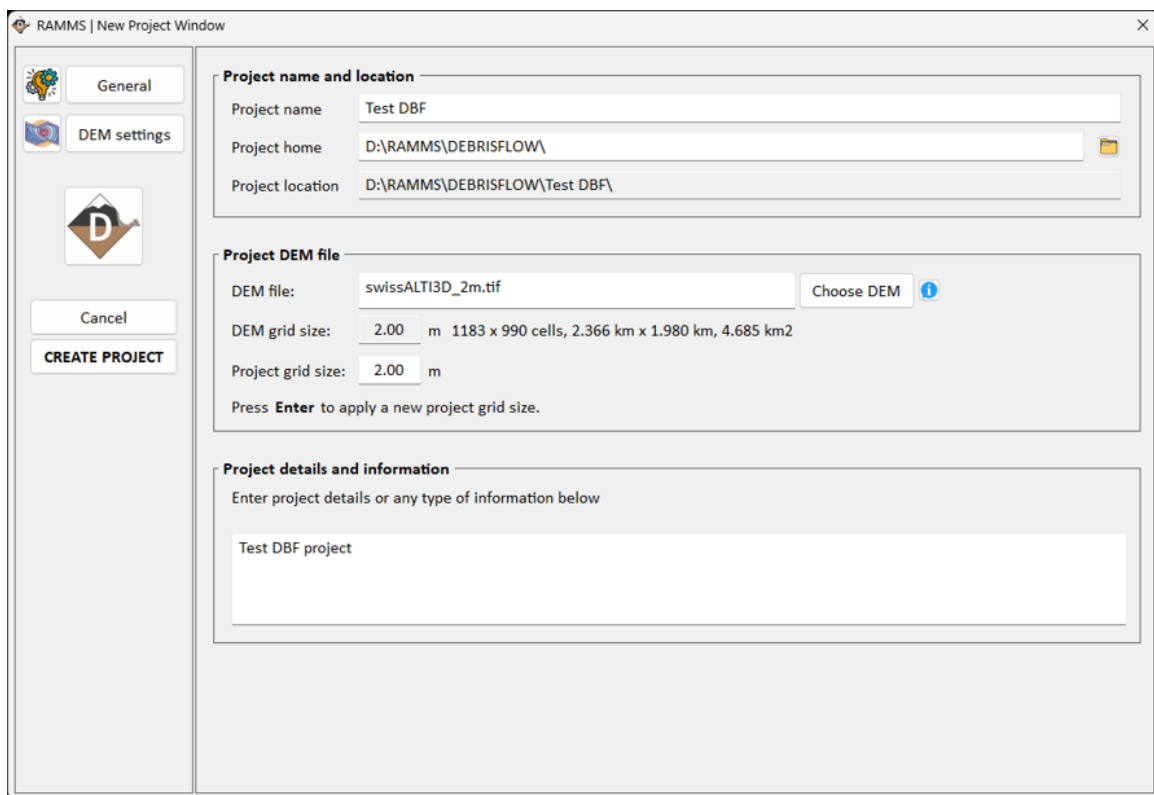


Figure 3-3: The RAMMS::Debrisflow New Project Window. The menu is divided into three sections. In the upper section a project name and home directory is defined. In the middle section the DEM is specified by the user. In the lower section project information can be provided (optional).

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 **0.25m**, it can be resampled to a coarser resolution, such as **1 or 2 meters**, using **bilinear interpolation**. Similarly, if the DEM resolution is too coarse (e.g., **10 meters**), it can be refined to a finer resolution (e.g., **1 or 2 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::Debrisflow**.

The **DEM Settings** menu is accessed from the **Create Project** menu (see Figure 3-3) 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.

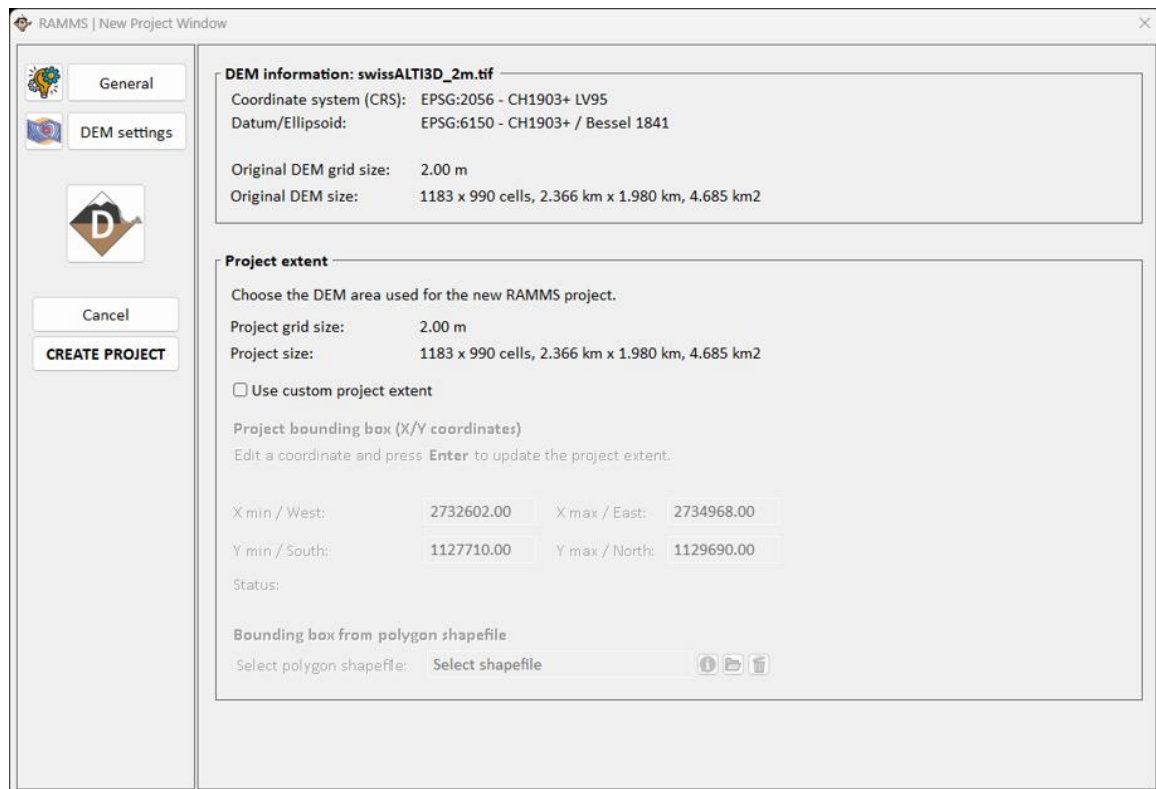


Figure 3-4: The DEM Settings menu. With this menu, the DEM can be reduced by clipping.

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 **domain 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.

3.2.2 DEM Files *.tif and *.asc

RAMMS::Debrisflow 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::Debrisflow setup requires a georeferenced DEM (1-2m recommended), a defined release area with release depth/input hydrograph, friction parameters (μ , ξ), erosion areas (optional but recommended) and a calculation domain. Validate all inputs visually in 2D before running — incorrect release geometry or friction parameters are 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 remote sensing images should be prepared in advance (.tif format and .tfw-file, maps and images are not mandatory, but nice to have). Georeferenced datasets have to be in the same **Cartesian coordinate system** (e.g. Swiss CH1903 LV03) as the DEM. Polar coordinate systems in degree (e.g. WGS84 Longitude Latitude) **are not supported**. For more information about specific national coordinate systems please contact the national topographic agency in your country.

4.1.1 Topographic data - Digital Elevation Model (DEM)

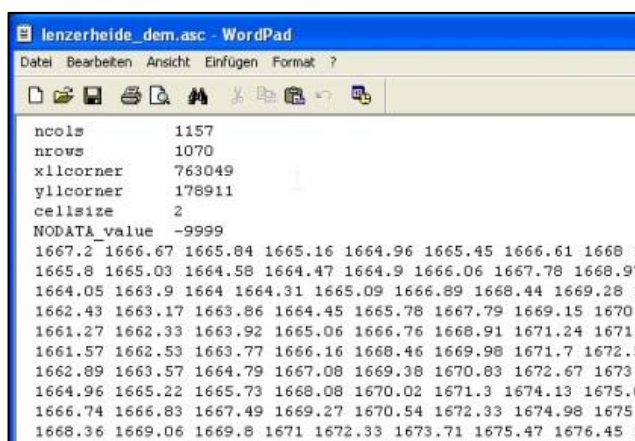
The topographic data is the most important input requirement. The simulation results depend strongly on the resolution and accuracy of the topographic input data. Before you start a simulation, make sure all important terrain features are represented in the input DEM. RAMMS is able to process the following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

- ESRI ASCII grid (Figure 4-1)
- GEOTIFF (georeferenced information embedded within a TIFF file)
- ASCII XYZ regular, single space data (Figure 4-2)

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

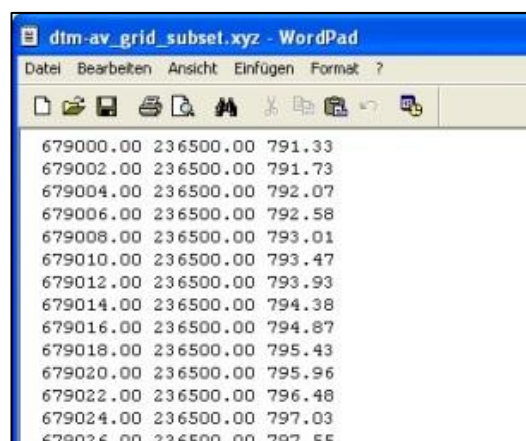
The header of an ESRI ASCII grid must contain the information shown in Figure 4-1.

CHAPTER 4: Setup and Theory



```
lenzerheide_dem.asc
Datei Bearbeiten Ansicht Einfügen Format ?
ncols      1157
nrows      1070
xllcorner  763049
yllcorner  178911
cellsize    2
NODATA_value -9999
1667.2 1666.67 1665.84 1665.16 1664.96 1665.45 1666.61 1668.1
1665.8 1665.03 1664.58 1664.47 1664.9 1666.06 1667.78 1668.97
1664.05 1663.9 1664 1664.31 1665.09 1666.89 1668.44 1669.28 1
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 1672.5
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 1675.0
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 1
```

Figure 4-1 : Example ESRI ASCII grid.



```
dtm-av_grid_subset.xyz
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
679024.00 236500.00 797.03
679026.00 236500.00 797.55
```

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

Choosing an Appropriate DEM Resolution

Grid resolution has a substantial, quantified effect on RAMMS::Debrisflow results, comparable in magnitude to friction-parameter choice and in some documented cases larger. Fan and Galoie (2025), systematically testing cell sizes from 1 m to 20 m against a real, well-documented debris-flow event, found inundation area increased by up to 724% between the 1 m and 20 m simulations of the same event, and recommend re-calibrating friction parameters whenever grid resolution changes rather than reusing a parameter set calibrated at a different cell size. Hohermuth and Graf (2014), comparing an official survey DEM against a 1×1 m LiDAR-derived DEM for a Swiss protection-concept design, found that DEM resolution and quality affected results more than either Voellmy friction parameter; grid resolutions coarser than 5×5 m failed to represent their site's narrow incised channel, causing simulated flow to break out of the channel prematurely. The same study found the official survey DEM contained a structural artifact (a bridge) absent from the LiDAR DEM, which altered the simulated flow path.

The consequences of under-resolving terrain are not confined to methodological studies. Krishnapriya et al. (2024), applying RAMMS::Debris Flow in a rapid post-event assessment of the catastrophic 30 July 2024 Wayanad, India debris flow, found that using a 30 m SRTM DEM (the best readily available pre-event elevation data) produced approximately 85% areal over-prediction relative to the satellite-observed flow footprint, a discrepancy the authors attribute explicitly to DEM coarseness. Graf et al. (2019), summarizing a decade of applied RAMMS practice in Switzerland, note that the Swiss national 2 m DEM is frequently too coarse relative to typical torrent channel widths of 1–10 m, degrading both runout and shear-stress-dependent erosion predictions.

Practical guidance: for channels narrower than approximately 10 m, use a grid resolution of 2 m or finer where possible; resolutions coarser than 5 m have been documented to cause artificial channel breakout in narrow, incised channels. Re-calibrate friction parameters when changing DEM resolution rather than reusing values calibrated at a different cell size. When only a coarse regional DEM is available, treat

CHAPTER 4: Setup and Theory

runout extent as a conservative upper bound and communicate the associated uncertainty explicitly; obtain a high-resolution DEM (LiDAR or UAV photogrammetry) for site-specific design work wherever feasible. Inspect official or national DEM products for structural artifacts (bridges, culverts, buildings) before use.

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.

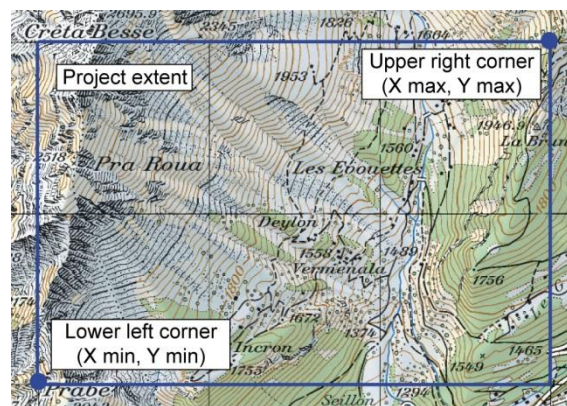


Figure 4-3: The same project extent (area of interest) can be used to calculate different scenarios with different input parameters.

4.2 Release information

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

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

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

For small unchanneled debris flows it is useful to use a release area with a given initial depth, which will be released as a block (block release), while for a channelized debris flow it may be more appropriate to use an input hydrograph. The later

CHAPTER 4: Setup and Theory

requires knowledge of the amount of material that might flow past a certain location in the channel. Chapter 5.6.3 includes several examples for both types of starting conditions.

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

More details on how to create release areas are given in section 4.9.1. Details on how to use an input hydrograph are given in section 4.9.2.

4.3 Friction information

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

The choice of the friction parameters requires *careful calibration* (section 6.1) of the model by using reference information such as field data, photographs of runout zones, estimations or measurements of flow velocities and flow heights as well as estimations of the material composition. This should be done by a person with expertise in debris flow characterization.

4.3.1 Physical friction model

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

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

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 flow 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.

CHAPTER 4: Setup and Theory

Throughout one simulation, the friction coefficients μ and ξ of a calculation domain are constant. However you have the possibility to add up to two polygons within the calculation domain with different μ and ξ friction parameters (see exercise 3.5f "How to run a calculation" on page 59pp.)

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.

4.3.2 Yield stress

The basic Voellmy equation has been modified to include a yield stress (cohesion) (Bartelt et al. 2015). 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

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

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

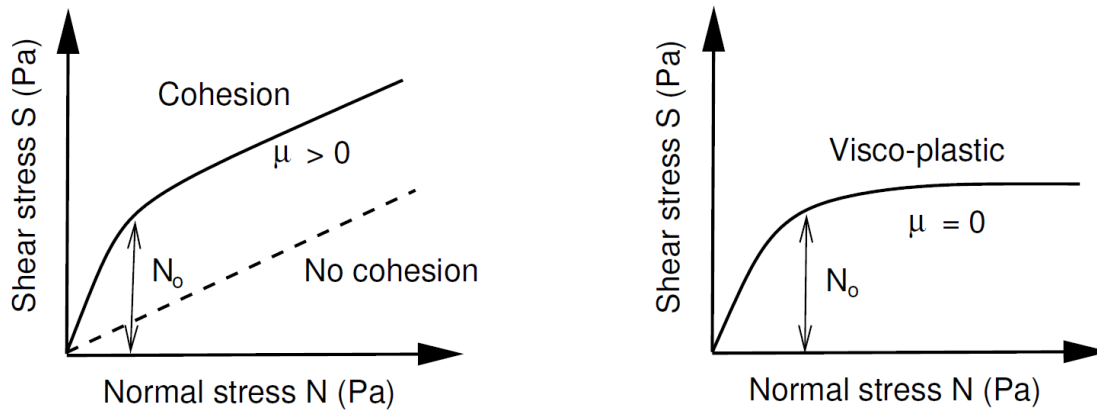


Figure 4-4: Relation between normal and shear stress. Left: Yield stress N_0 serves to increase the shear stress for higher normal pressures. At low normal pressures (small flow heights) the shear stress increases rapidly from $S=0$ to $S=N_0$. The slope of the 'S vs N' relation remains μ , when the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

CHAPTER 4: Setup and Theory

4.3.3 Curvature

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

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

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

4.3.4 Calibration of the friction parameters μ and ξ

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

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

The calibration procedure of the Voellmy model is explained with an example in section 6.1.

Relative Sensitivity of μ and ξ

μ and ξ do not influence results equally. Walser et al. (2014) found that μ has a large, near-linear effect on runout distance — varying μ from 0.1 to 0.2 shifted the simulated runout on an approximately 800 m potential fan by roughly 200 m — while ξ had a negligible effect on affected area and governs mainly in-channel velocity. Fan and Galoie (2025) independently confirm that μ dominates over ξ in their sensitivity tests. Barnhart et al. (2021) further find that, within plausible parameter ranges, initial release

CHAPTER 4: Setup and Theory

volume influences model performance at least as much as either friction parameter. When calibration time is limited, prioritise constraining release volume and μ before fine-tuning ξ .

4.4 Erosion information

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

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

$$e_m = 0 \text{ for } \tau < \tau_c$$
$$e_m = \frac{dz}{d\tau}(\tau - \tau_c) \text{ for } \tau \geq \tau_c$$

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

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

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

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

If the shear stress in any given cell is exceeded after erosion takes place, then the maximum depth of erosion (relative to the initial bed elevation at the start of the simulation) is automatically adjusted and additional erosion can take place until the new value of e_m is reached. The elevation of the channel bed does not change during the RAMMS simulation. Users can incorporate changes in topography due to erosion or deposition, e.g. for further use in hazard modeling scenarios or event analyses, by

CHAPTER 4: Setup and Theory

selecting the post-processing options '*Results → Add Deposition to DEM*' or '*Results → Subtract Erosion from DEM*' or both with '*Results → Add Topographic Changes (Erosion/Deposition) to DEM*'.

4.4.1 Interpreting the Default Erosion Thresholds

The default critical shear stress ($\tau_c = 1$ kPa) marks the point at which erosion becomes statistically detectable in the Illgraben dataset used to derive it (Frank et al., 2015, from a linear regression to the 50th-percentile line of shear-stress-versus-erosion measurements reported in Schürch et al., 2011). It is not the threshold for substantial erosion: Schürch et al. (2011) report, from the same measurement campaign, that substantial channel-bed erosion occurs once basal shear stress exceeds approximately 3–4 kPa. Both figures are field-based and are not in conflict — they describe different points on the same erosion-probability curve — but users should not expect erosion to become significant as soon as the default τ_c is exceeded.

4.4.2 Sensitivity of Results to the Erosion Rate

The default erosion rate ($dz/dt = -0.025$ m/s) was derived from a single well-instrumented event at Illgraben (Berger et al., 2011), where erosion of up to 0.5 m was measured within 15–21 s of front arrival. The same field campaign recorded a maximum instantaneous erosion rate an order of magnitude higher (0.25 m/s) elsewhere in the catchment. Frank et al. (2017), testing the sensitivity of predicted volume growth to this coefficient at two independent sites (Meretschibach, Bondasca), found that halving or doubling the default rate changed peak predicted volume growth by up to approximately 9-fold. For applications where the eroded (bulked) volume materially affects the hazard scenario, a bracketing sensitivity run (e.g. $\pm 50\%$ of the default rate) is recommended rather than relying on the single default value.

4.4.3 Site Transferability of the Default Coefficients

All three default erosion-module coefficients (τ_c , $dz/d\tau$, dz/dt) are derived from the Illgraben test site. Independent tests without re-calibration exist at two other Swiss sites — Spreitgraben (Frank et al., 2015) and the Meretschibach/Bondasca catchments (Frank et al., 2017) — both of which found the defaults broadly plausible but flagged specific conditions (initial volumes below approximately 100 m³, channel gradients below 10°) where the τ_c default "may be necessary or desirable to adjust" (Frank et al., 2015). Schürch and McArdell (2018), applying the module to a non-Illgraben Swiss catchment without an instrumented monitoring station, explicitly note an urgent need for additional erosion-parameter datasets from sites with different slope ranges, channel forms, and lithologies. Users applying the defaults outside similar channelized-torrent settings should treat them as a starting point for sensitivity testing rather than as validated, general-purpose values.

4.4.4 Spatially and Temporally Varying Erosion Parameters

RAMMS::Debrisflow allows multiple erosion polygons, each with independent τ_c , dz/dt and dz/dt values. Two recent studies give quantified reasons to use this capability rather than a single global setting. McArdeell et al. (2025) link the erosion-depth coefficient dz/dt to antecedent channel-bed saturation, demonstrating values of 0.05 (dry), 0.1 (standard/default) and 0.25 (wet) m/kPa that produced an 8-fold difference in final simulated volume for the same initial release. Stammberger et al. (2026) show that erosivity varies systematically with position along the channel (peaking at 40–60% of channel length) and with bed material type (boulder-, gravel- or regolith-dominated beds show fundamentally different spatial erosivity patterns), independent of moisture. Where field evidence supports it, defining erosion polygons with position- or moisture-adjusted coefficients is preferable to a single catchment-wide default.

4.5 Flexible barriers (nets) information

RAMMS::Debrisflow can account for the retention effect of flexible debris-flow and hillslope protection nets ("flexible barriers") placed across the flow path. The retention volume a barrier can hold is calculated according to the method described in "[Practical guide for debris flow and hillslope protection nets](#)", based on:

- barrier height
- width of the barrier
- slope angle between the barrier and the terrain
- slope angle of the terrain
- slope angle of the deposit

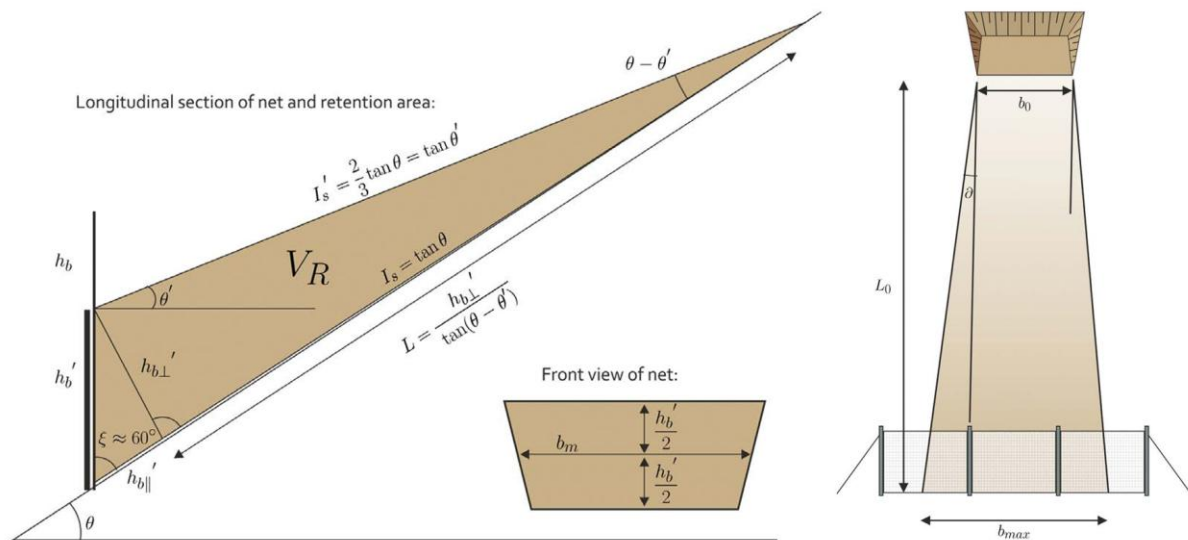


Figure 4-5: Left: Longitudinal section of torrent and front view of the net. Right: Sketch of geometric determination of maximum spread. Figures according to

$$V_R = 0.5(h_b')^2 b_{max} \sin \xi \left(\frac{\sin \xi}{\tan(\theta - \theta')} + \cos \xi \right)$$

with

V_R	=	Retention volume of flexible barrier
h_b'	=	Barrier height
b_{max}	=	Width of barrier
ξ	=	Slope angle between barrier and terrain
θ	=	Slope angle of terrain
θ'	=	Slope angle of deposit

Figure 4-6: Barrier retention volume equation.

Once calculated, RAMMS removes the corresponding retention volume — and its momentum — from the flow at the barrier's location during the simulation.

Flexible barriers must be defined as polygon shapefiles, with a barrier width of at least 2× the calculation grid size.

4.6 Calculation parameters

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

4.7 Preferences

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

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

CHAPTER 4: Setup and Theory

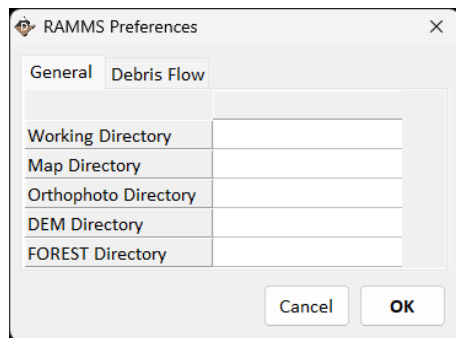


Figure 4-7: General tab of RAMMS preferences.

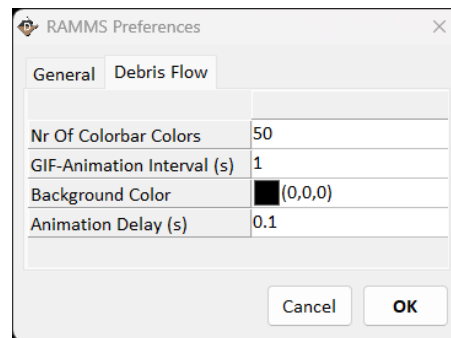


Figure 4-8: Debris Flow tab of RAMMS preferences.

General Tab

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

Debris flow Tab

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

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

Exercise 3.2 : Working directory

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

- 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.

4.8 Working with the RAMMS GUI

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

4.8.1 Open input- and output-files

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

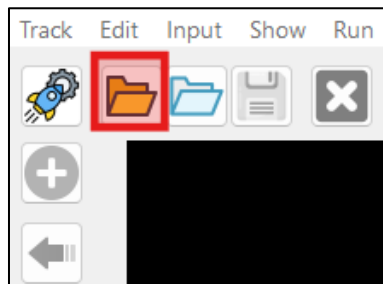
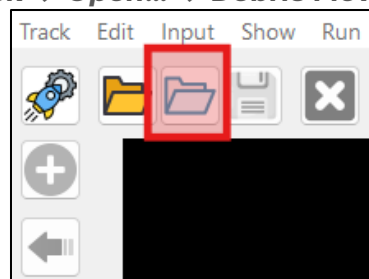


Figure 4-9: Toolbar button to open an input file. You can also use the menu **Track** → **Open...** → **Input File**.

Figure 4-10: Toolbar button to open an output file. You can also use the menu **Track** → **Open...** → **Debris Flow 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.

4.8.2 Visualizing shapefiles and domain-files

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

Exercise 3.4a : Visualizing shapefiles and Domain-files

a. **Files** tab in the right panel:

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

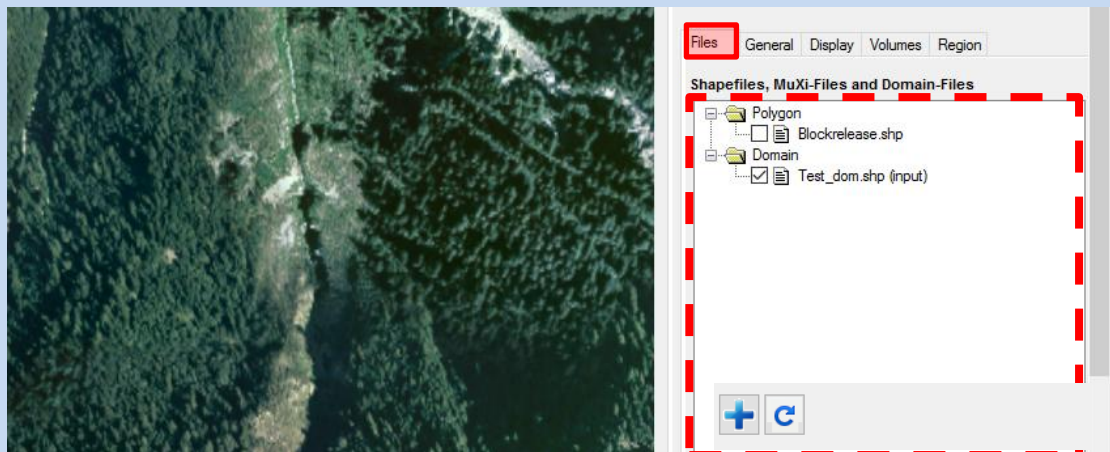


Figure 4-11: Files tab and available project files (file-tree, dashed red). With the **blue + button**, files from external directories can be added to the file-tree. Refresh the tree with the refresh-button.

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

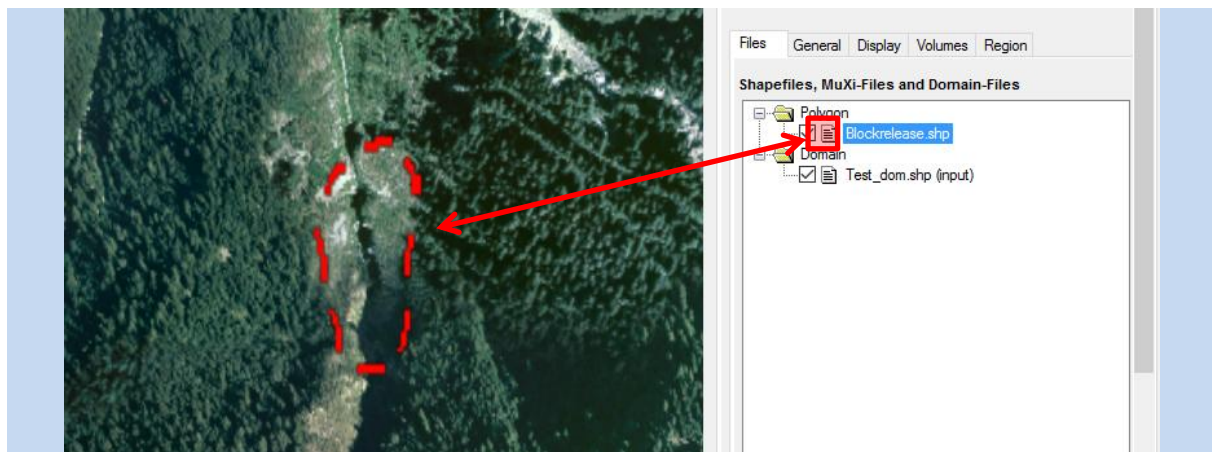


Figure 4-12: Selected file (Blockrelease.shp) on the right is shown in the visualization.

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

Shapefile Properties

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

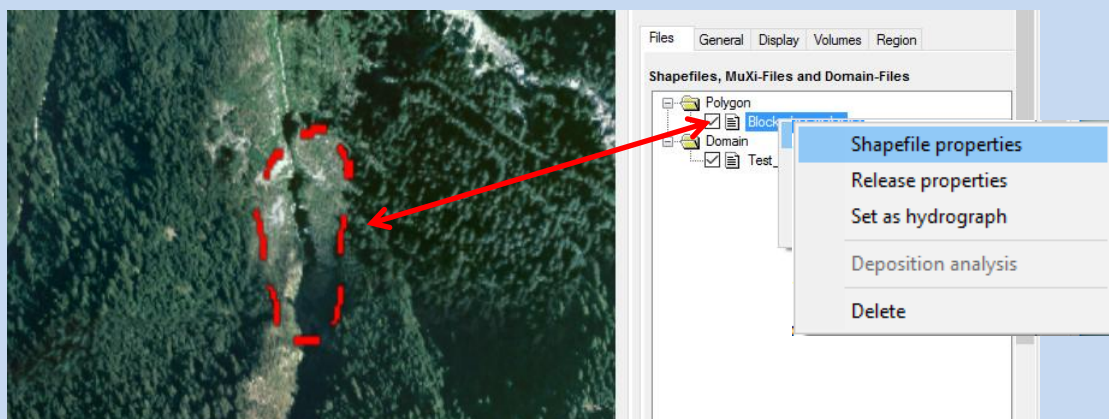


Figure 4-13: Right-click menus **Shapefile properties**, **Release properties**, **Set as hydrograph** and **Delete**.

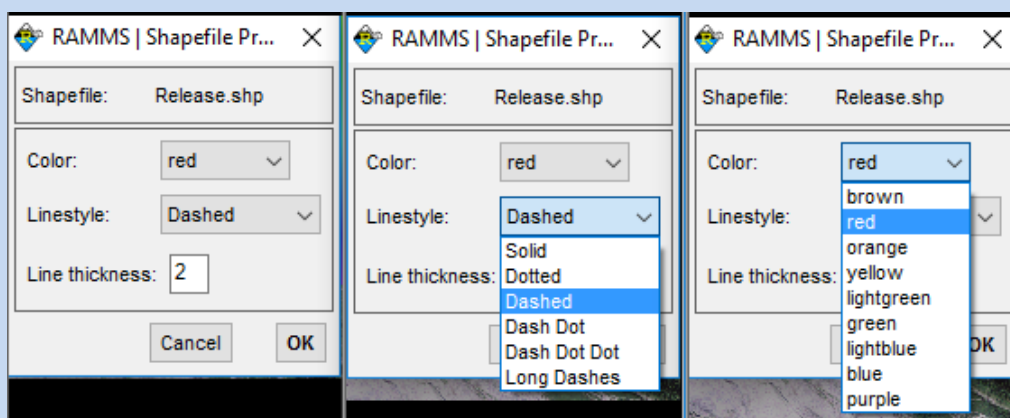


Figure 4-14: Use Shapefile properties to change line thickness, color or linestyle.

Release Properties

- Please see section 4.9.1 on how to specify release area properties.

Set as hydrograph

- With this option you tell RAMMS, that this polygon shapefile is a hydrograph release area. See section 4.9.2 for more about hydrographs.

Deposition analysis

- This function is only available in output mode. 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 (.db2)
- Output files (.out.gz)
- Polygon shapefiles (.shp)
- Domain shapefiles (.shp)

4.8.3 Hillshade visualization

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

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

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

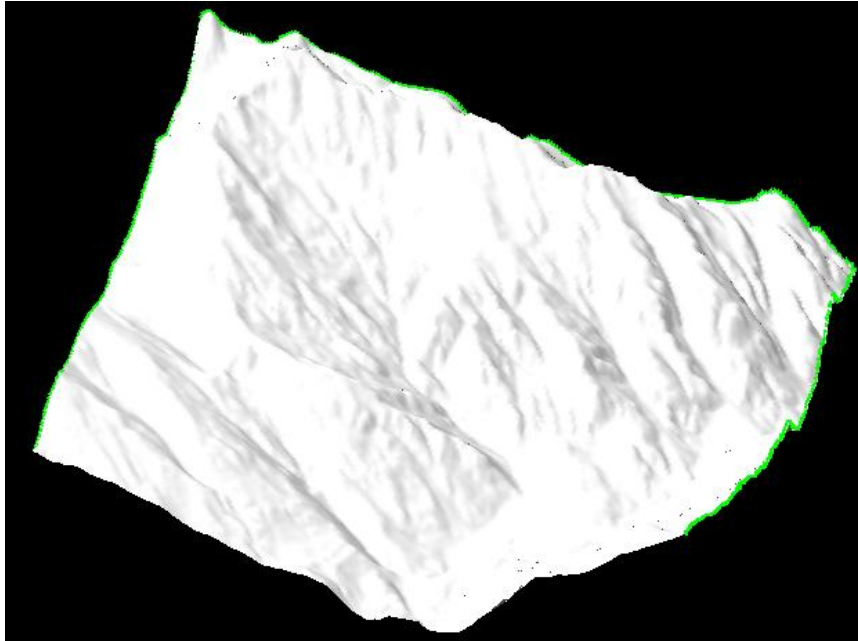


Figure 4-15 DEM surface visualization (with shadows) after creating a new project in RAMMS

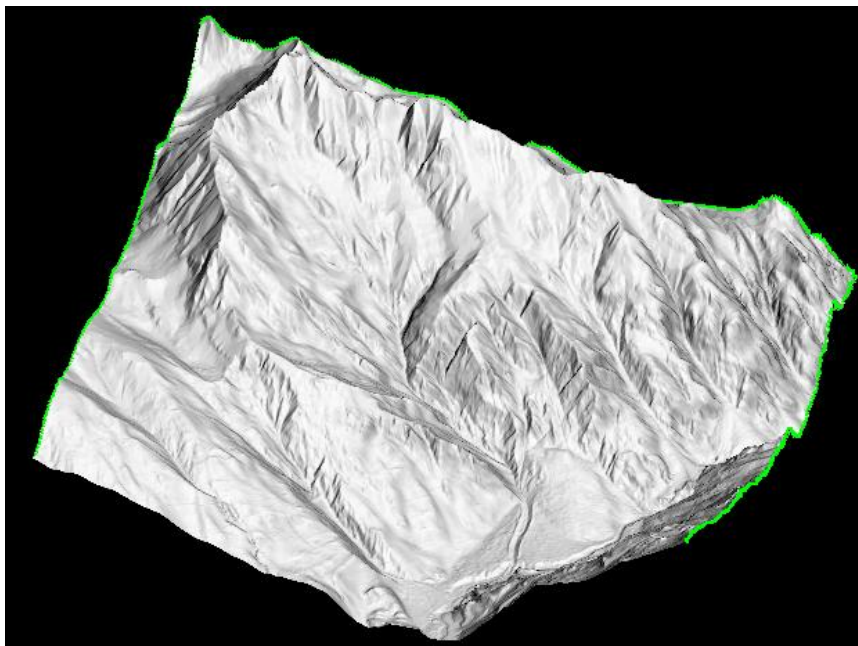


Figure 4-16 Visualization after creating and adding the hillshade image to RAMMS

4.8.4 Changing maps and orthophotos (aerial images)

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

CHAPTER 4: Setup and Theory

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

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

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

d. Add or change a map:

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

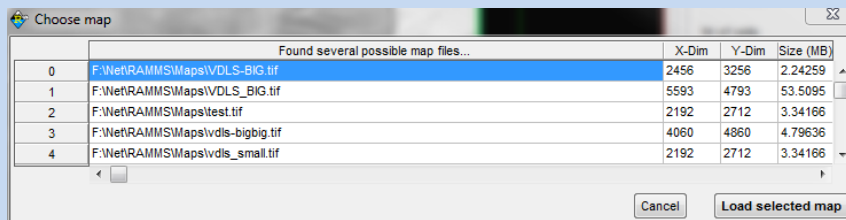


Figure 4-17: Window to choose map image.

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

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

e. Map not found:

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

f. Change remote sensing imagery:

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

4.8.5 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.

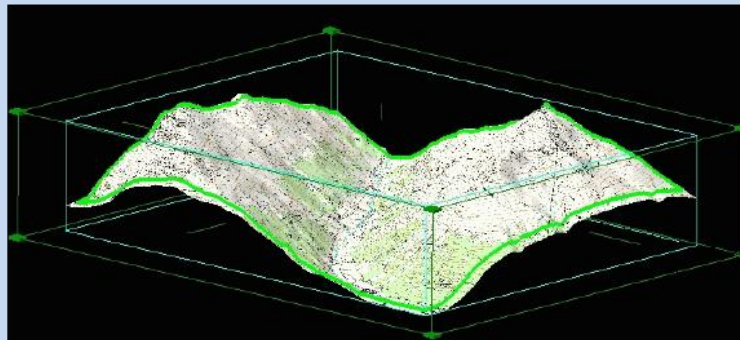


Figure 4-18: Active project with lines and corners for resizing.

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

b. Terrain model has a dimension > 100%:

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

Exercise 3.4d: Rotating the model

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

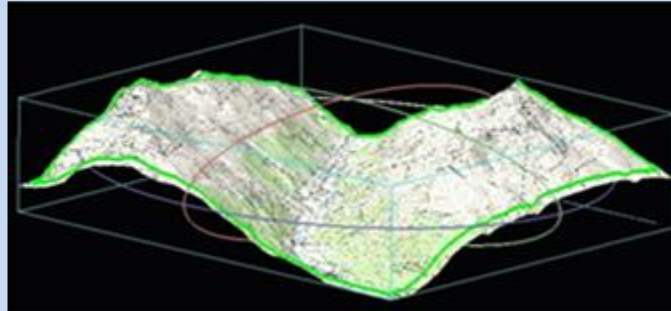


Figure 4-19: Active project with rotation axes.

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.

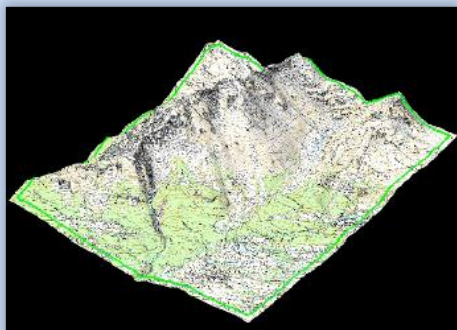


Figure 4-20: 3D view of example model.

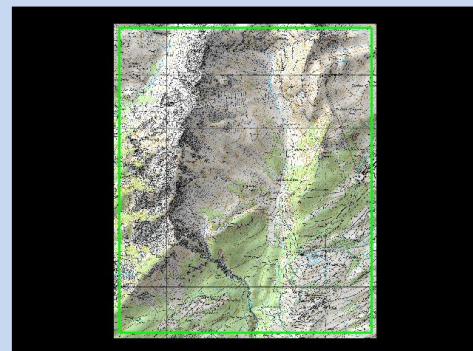


Figure 4-21: 2D view of example model.

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 formation 
- Draw new domain 

OUTPUT:

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

- Measure distance and angle 

4.8.6 Colorbar

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

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

Exercise 3.4f: Editing the colorbar

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

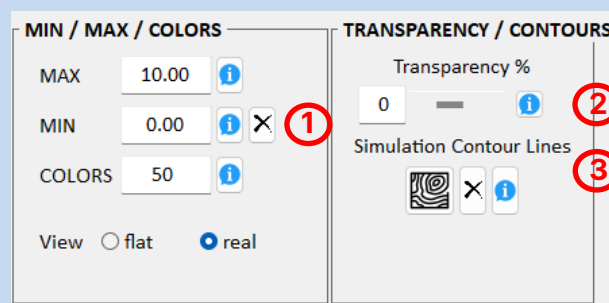


Figure 4-22 Colorbar and visualization properties

- **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)

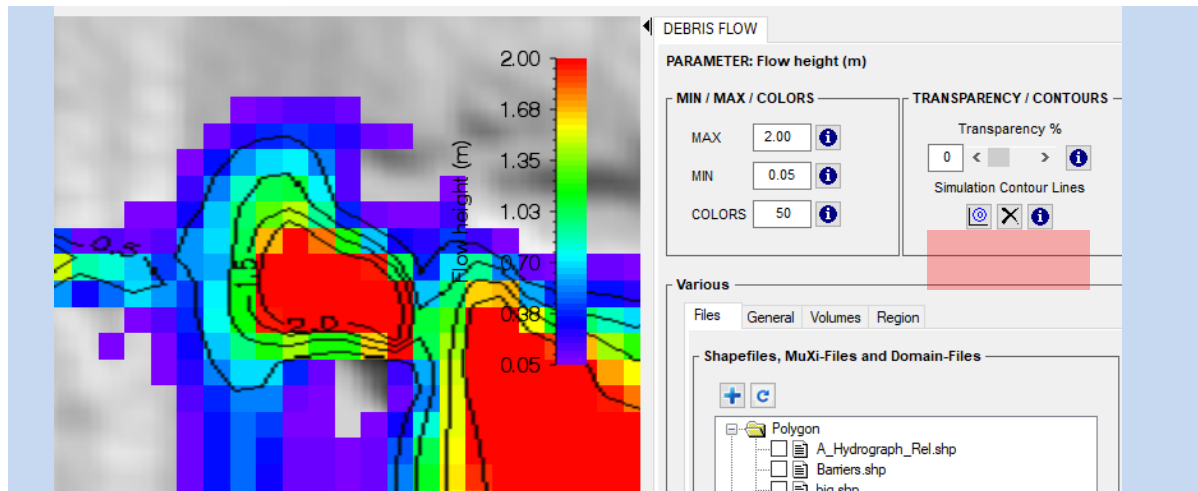


Figure 4-23: Simulation contour lines

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** → **Debris flow Tab** → **Background Color**).

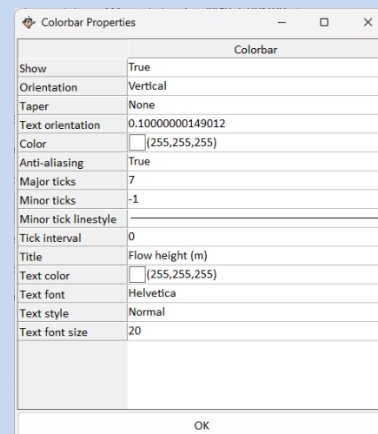


Figure 4-24: The Colorbar Properties window.

4.8.7 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 (see figure 3.9). The created input file has the ending *.db2.

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

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

a. Input file:

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

b. Program settings

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

Exercise 3.4h : How to open an input file

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

Exercise 3.4i : How to visualize a shapefile

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

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

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

4.8.8 About RAMMS

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

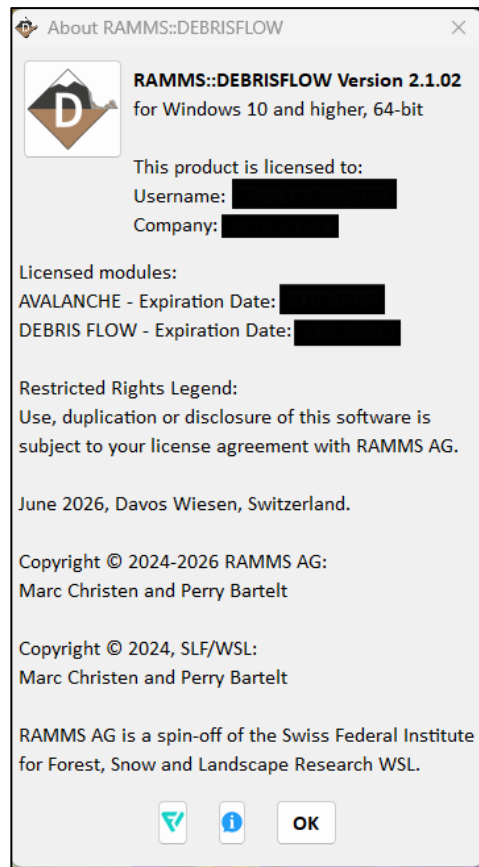


Figure 4-25: About RAMMS::DEBRISFLOW

4.9 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 a block release area or an input hydrograph,
- an erosion area (optional),
- a calculation domain and
- friction parameters μ and ξ .

The definition of a smaller calculation domain is especially useful to keep the number of calculation grid cells as small as possible, that is, it is best to test what the flow path of a simulation will be to limit the calculation domain to this extent. The exercises below show you how to create a release area, how to use an input hydrograph and how to create a calculation domain. A simulation also requires a set of friction parameters. Details on the friction model used in **RAMMS::Debrisflow** are given in section 4.3. How to calibrate the friction values is explained in section 6.1.

4.9.1 Release area

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

Release Type	Details
Create a new release area (exercise 3.5a)	If there is no release area available for your project, or you wish to create a new one, switch to 2D mode and click  (Draw new polygon shapefile)
Open an existing polygon shapefile (exercise 3.5b)	Use the file-tree in the right-hand panel (<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.

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.

CHAPTER 4: Setup and Theory

- To end the release polygon, click the right mouse button. The polygon will be closed automatically.

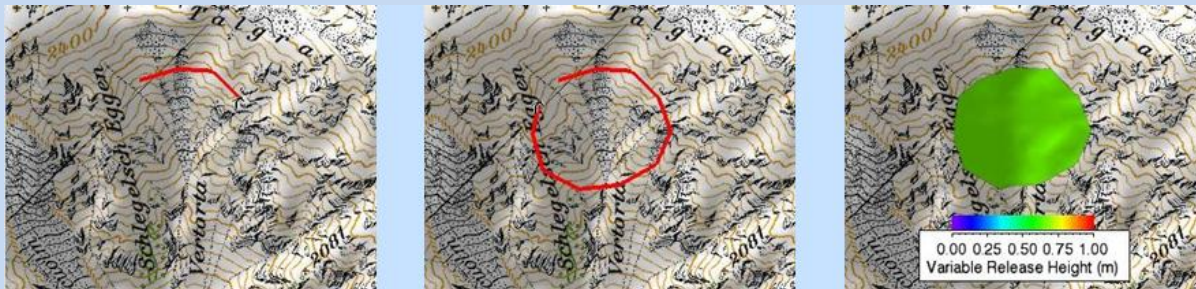


Figure 4-26: Project with emerging release area.

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

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

The polygon shapefile will now be created and opened directly.

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

- Use the file-tree in the right-hand panel (Files) and click the shapefile you want to visualize (see chapter 4.8.1.).
- Alternatively, you can use the Add data button  or menu 'GIS → Add data' to add a shapefile from another source than your project directory to your visualization.

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

Additional release information is found in the Debris Flow panel, tab Volumes (Figure 4-27 and Figure 4-28) below.

DEBRIS FLOW

PARAMETER: Max velocity (m/s)

Files General Display Volumes Region

Volumes and areas

Proj rel area (m2)	904.0
Incl rel area (m2)	1144.6
Block volume (m3)	1144.6
Block mass (t)	2289.3
Density (kg/m3)	2000.0

View/Edit release area

Figure 4-27 : Block release area and volume information.

DEBRIS FLOW

PARAMETER: Max velocity (m/s)

Files General Display Volumes Region

Volumes and areas

Proj rel area (m2)	904.0
Incl rel area (m2)	1144.6
Hydrograph volume (m3)	5000.0
Hydrograph mass (t)	10000.0
Density (kg/m3)	2000.0

View/Edit release area

Figure 4-28: Hydrograph area and volume information.

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

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

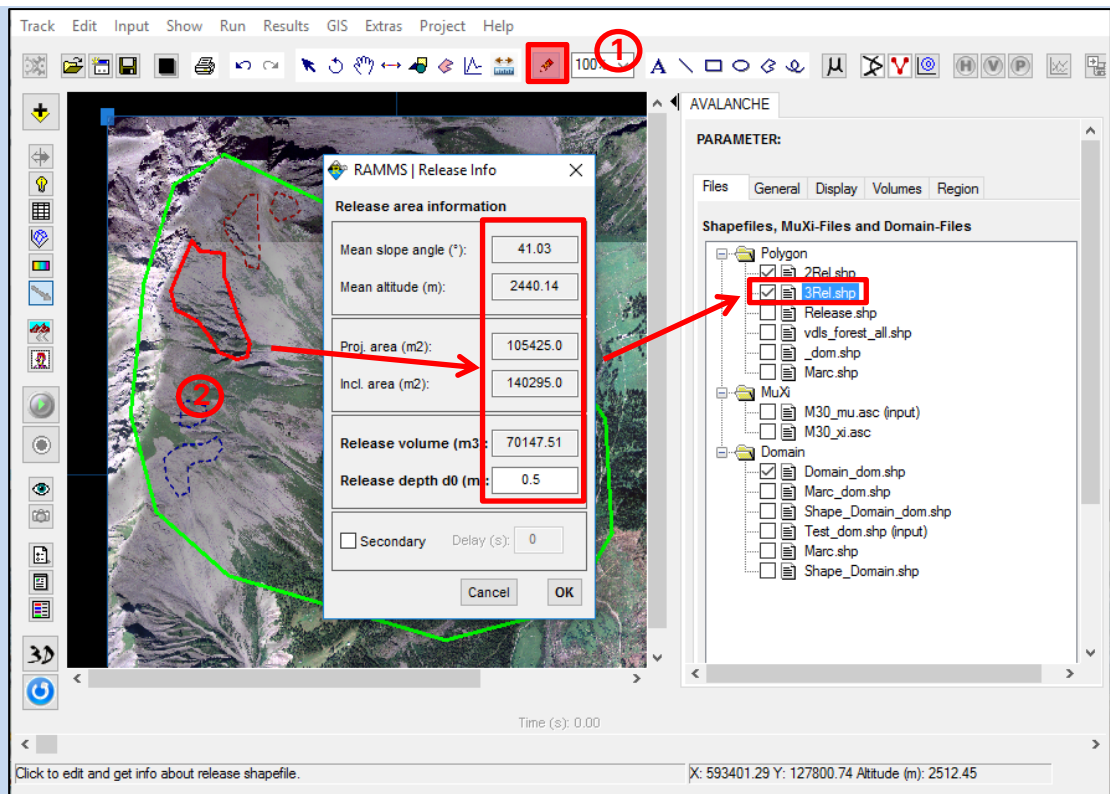


Figure 4-29: View/Edit release area

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

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.

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: Setup and Theory

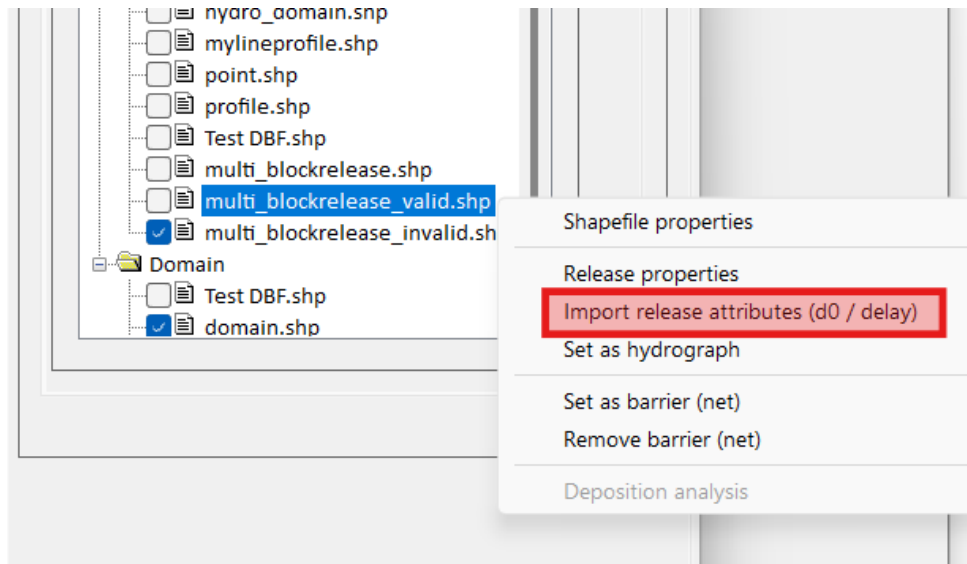


Figure 4-30: Importing release attributes from a shapefile.

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>**.

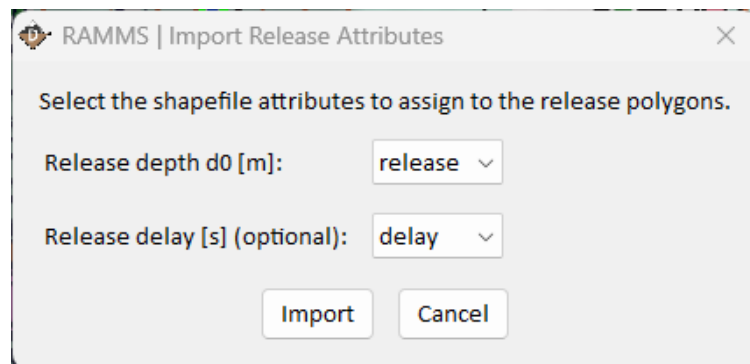


Figure 4-31: Import Release Attributes dialog. Select attributes for release depth and delay (optional).

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.

4.9.2 Input hydrograph

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

Field measurement of debris flow properties is beyond the scope of this manual, however, if debris flow activity in a catchment is frequent, it may be possible to obtain additional information for calibrating RAMMS by installing a monitoring station. Many debris flows, when the monitoring data are simplified, have a relatively simple triangular wave-like shape (Figure 4-32). Assuming a well-estimated total volume (e.g. field data), maximum debris flow discharge (Q) and corresponding time (t_1), as well as the end time (t_2) can be calculated. We call this a three-point hydrograph calculation.

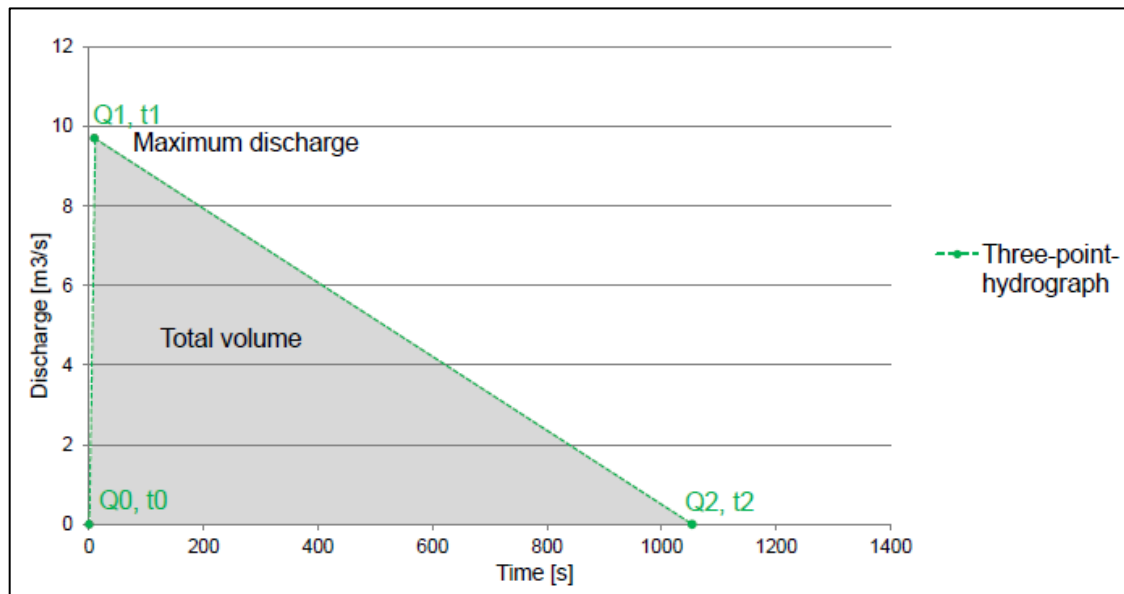


Figure 4-32: Three-point hydrograph with total volume of debris flow.

In nature, discharge hydrographs are generally better described by four points (Figure 4-33 and Figure 4-34). However, a three-point hydrograph results in slightly larger flow heights and flow velocities after the maximum discharge. If the detailed discharge hydrograph is not known it is useful to choose a three-point hydrograph. In critical applications we encourage users to evaluate the sensitivity of the model results to the shape of the input hydrograph, e.g. when preparing different scenarios as a basis for hazard maps.

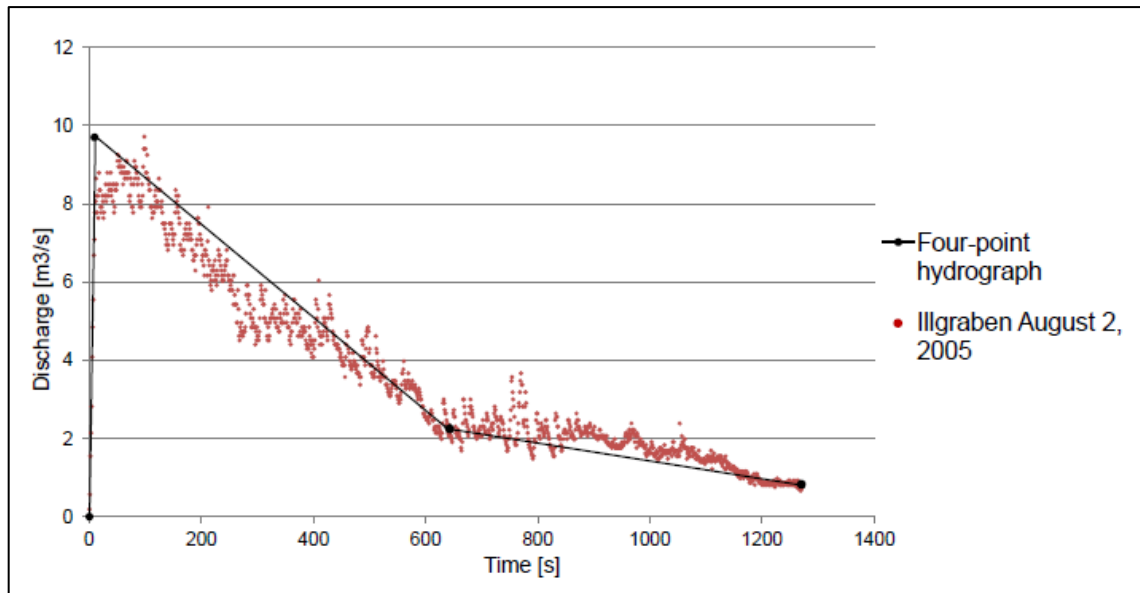


Figure 4-33: Four-point hydrograph for discharge values of an event, August 2, 2005, at Illgraben, Valais, Switzerland.

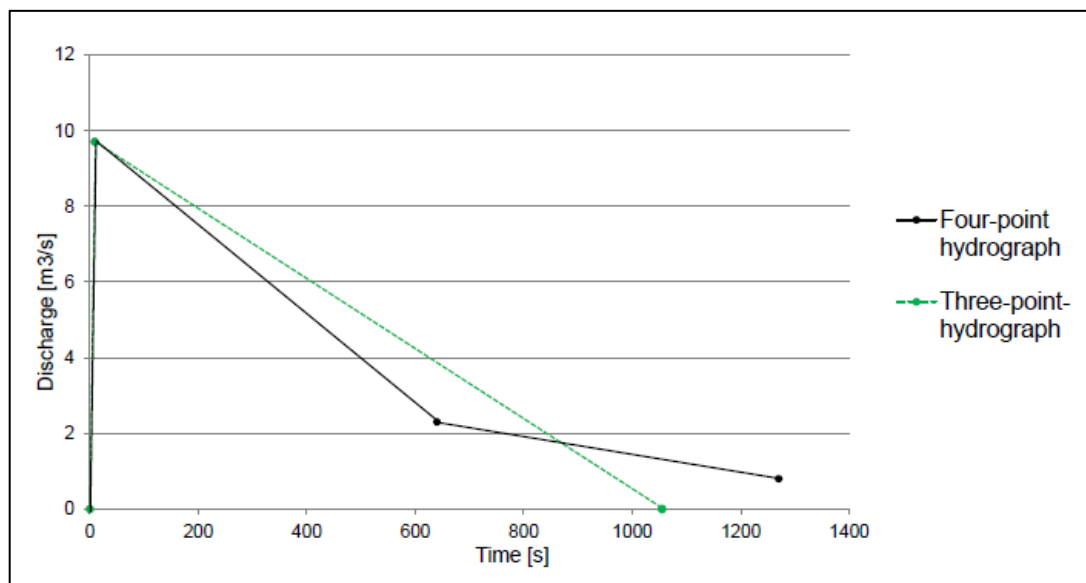


Figure 4-34: Comparison of a three-point with a four-point hydrograph for the given discharge data with the same total volume.

Input data for a discharge hydrograph

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

CHAPTER 4: Setup and Theory

Table 4.1: Data for discharge hydrograph.

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

RELEASE PARAMETERS ☐ Use BLOCK release

Release Information

Hydrograph Block Release

Hydrograph Parameters

Release shapefile(s)
 Hydro.shp

Inflow direction
 Inflow direction (degrees) 300 ☐ Automatic

3-Point Hydrograph

Vol (m3)	Qmax (m3/s)	t1 (s)	v (m/s)	t2 (s)
180.0	7.6	10.0	5.0	47.37

☒ Edit hydrograph table

Hydrograph-Plot Hydrograph-Table

	t (s)	Q (m3/s)	v (m/s)
0	0.000000	0.000000	5.00000
1	10.0000	7.60000	5.00000
2	47.3684	0.000000	5.00000
3	0.000000	0.000000	0.000000
4	0.000000	0.000000	0.000000
5	0.000000	0.000000	0.000000

Figure 4-35: Edit hydrograph table. The checkbox (red rectangle) must be set to be able to edit the hydrograph table.

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

In both cases, the inflow direction has to be defined, which is described as the angle in counterclockwise direction from the x-coordinate of the topographic data (Figure 4-36).

CHAPTER 4: Setup and Theory

Automatic hydrograph inflow direction

By default, the direction in which a hydrograph injects momentum into the flow is set manually via the inflow direction (angle). When **automatic mode** is enabled, RAMMS instead computes this direction itself from the terrain: it determines the local slope-aspect direction (the downslope direction of the DEM) at every cell in the hydrograph release area, averages them into a single representative direction, and uses that for the whole simulation.

This is useful when the natural flow direction at the release location is not known precisely, or when specifying it manually would require estimating an angle from the terrain by hand. Because a single direction is used for the entire hydrograph area, the result stays physically consistent even where the local terrain is uneven or the release area spans a bend or fan — each hydrograph cell injects mass in the same shared direction rather than following its own local terrain angle.

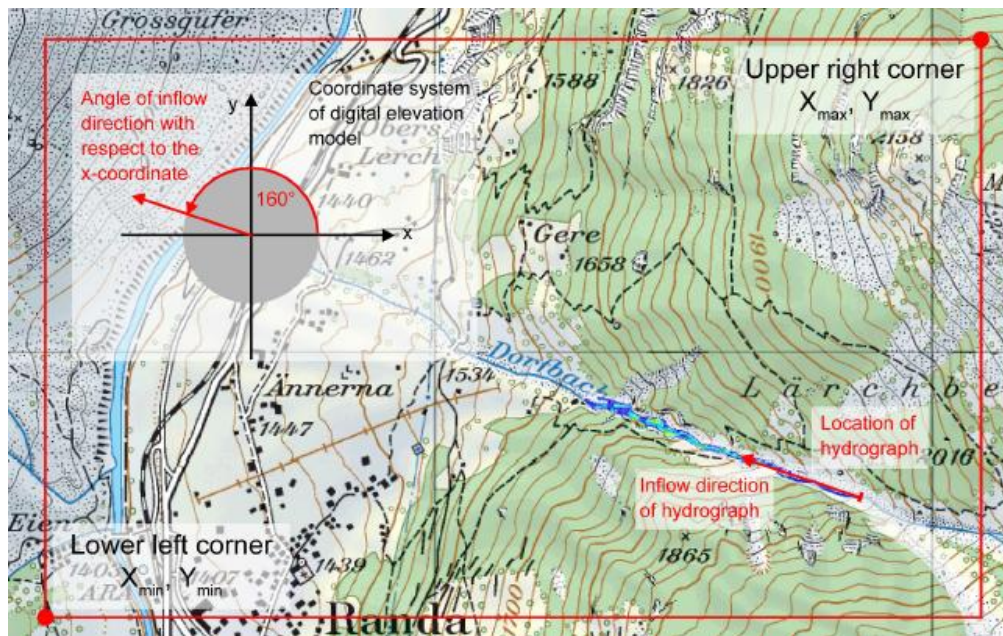


Figure 4-36: Definition of inflow direction of an input hydrograph.

Exercise 3.5d: How to use the input hydrograph

Hydrograph location and calculation domain

- Draw a polygon area which extends over the potential debris flow width with the **'Draw new polygon shapefile'** tool (see section 4.9.1) where you want to place the input hydrograph (Figure 4-56). **Be sure your release area covers several grid cells!!** The input will initially flow downhill perpendicular to the line. Name your release area accordingly.
- Right-click your release area shapefile and choose **Set as hydrograph**.

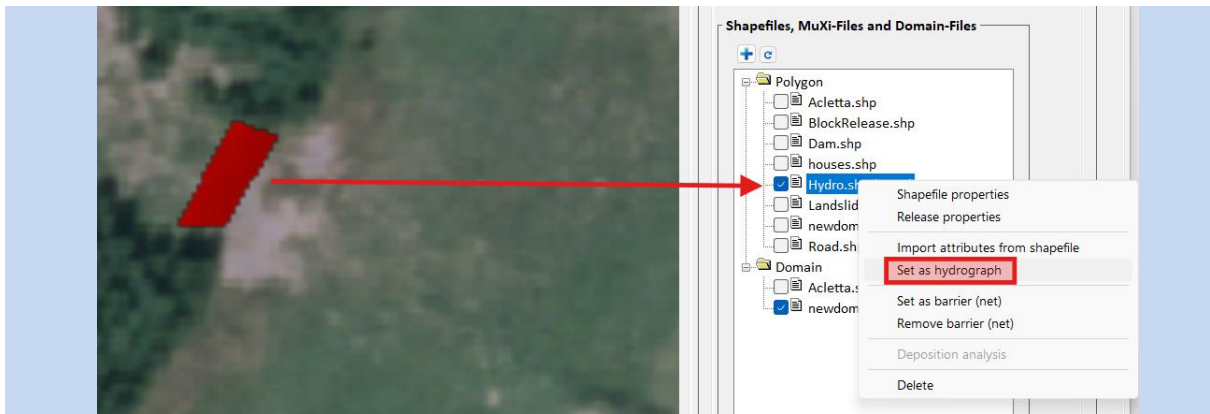


Figure 4-37: Beam shaped polygon area (red rectangle) for the input hydrograph inside the calculation domain (green line).

- Draw a **calculation domain** (see section 4.9.5), such that the area of the input hydrograph is located inside the domain

Enter data of discharge hydrograph

- Once you start to **RUN a calculation** (see section 3.5.4) you can enter the discharge hydrograph information in the corresponding tab **Hydrograph**. For details on how to enter the hydrograph data see beginning of this section 0.

4.9.3 Erosion

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

- Draw polygon shapefiles for varying erosion areas. Use **Input → Polygon shapefile → Draw New Polygon Shapefile** to do this.
 - Open the *Run Simulation* window, change to the *Erosion Tab* and start to enter the erosion characteristics.
1. Select shapefile
 2. Choose/select parameters, see Table 4.2 below for details.
 3. Click the **+ button** to add your selection to the *Erosion Shapefile List*.

CHAPTER 4: Setup and Theory

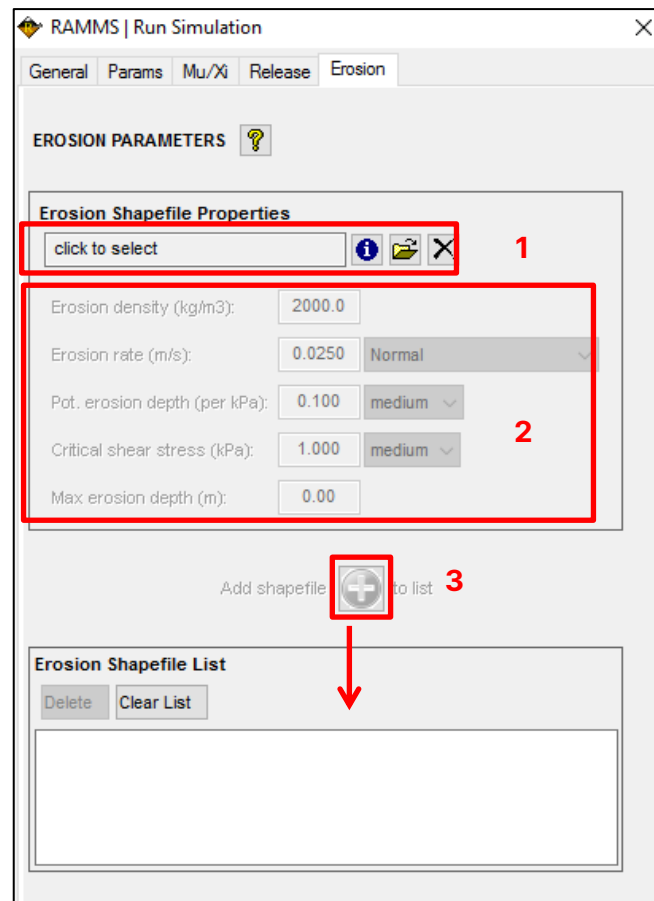


Figure 4-38: Erosion Tab

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

Erosion Parameters

Table 4.2: Parameters for erosion areas.

Parameter	Details
Erosion density: (kg/m³)	We suggest to use the default density (or the density that you selected in the Params tab) unless you have more detailed information.
Erosion rate: (m/s, default 0.025) • Loose, wet sediment (0.05 m/s) • Normal (0.025 m/s) • Densely packed sediment (0.013 m/s)	The rate at which debris flows entrain material from the sediment bed. There are very few values available from the field (see Frank et al.), so we propose using a constant value based on the work published by Berger et al., 2011 (0.025 m/s), for the Illgraben torrent (erosion rates measured using sensors buried in the channel bed). This is a rate that the model will use for entrainment until the predicted erosion depth is reached. Increasing this value will cause sediment to be entrained at a faster rate, potentially resulting in relatively large debris flow snouts.
Pot. erosion depth: (per kPa, default 0.1) • deep (0.2 per kPa) • medium (0.1 per kPa) • shallow (0.05 per kPa)	This parameter controls the maximum potential erosion depth (variable e_m in the paper by Frank et al.) as a function of the maximum shear stress reached in each cell. This value actually is the slope of the line which relates e_m to the maximum shear stress calculated for each cell. By changing the value of this parameter to e.g. 0.2 (menu choice <i>deep</i>) you will double the maximum potential erosion depth.
Critical shear stress: (kPa, default 1.0) • low (0.5 kPa) • medium (1.0 kPa) • high (1.5 kPa)	This is the value at which erosion can start occurring. It might be reasonable to expect that a channel bed consisting of interlocked boulders would have a larger critical shear stress, while a channel bed of saturated sandy gravel would have a lower value. Our value is just a curve fit to the data reported by Schürch et al. (2011).
Max erosion depth: (m, default 0.0)	Here we can set a maximum thickness of the layer of erodible sediment if such data are available (e.g. in some cases we may know that there is only a limited amount of sediment available). The default value is 0.0, meaning that the erosion algorithm is calculating the max erosion depth itself.

4.9.4 Flexible barriers (nets)

Drawing barrier locations. Draw one or more polygons at the net/barrier location(s) — either a single polygon per shapefile, or several polygons within one shapefile if you have multiple barrier locations.

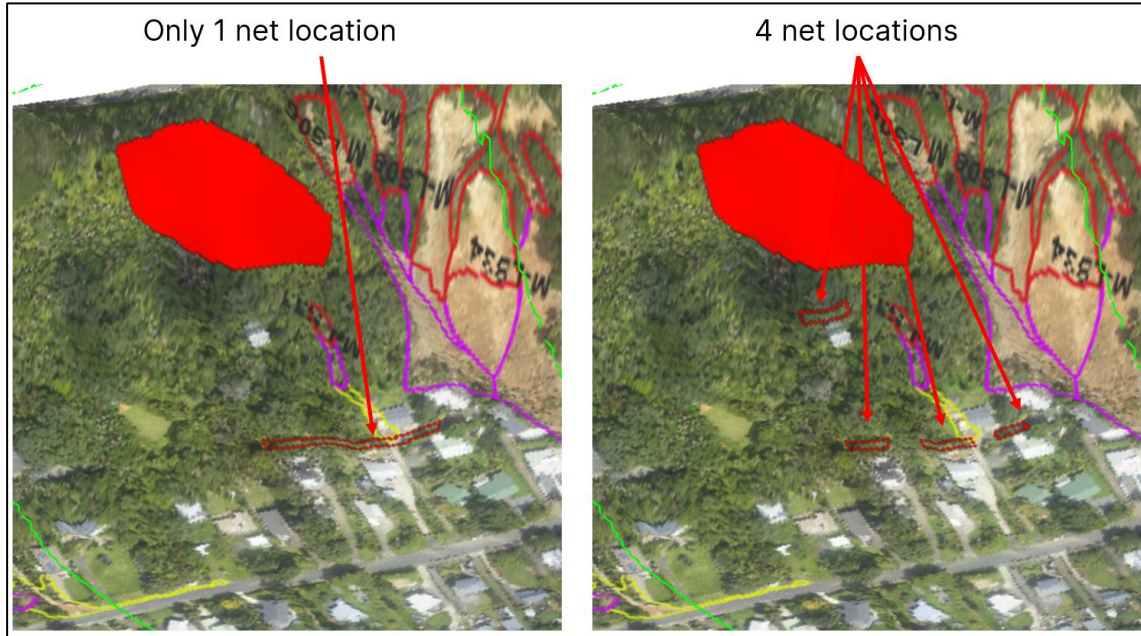


Figure 4-39: Left: aerial orthophoto with a single flexible-barrier polygon (red dashed outline, arrow) drawn across the flow path. Right: the same scenario with four separate barrier polygons drawn at different net locations within one shapefile.

Setting a shapefile as a barrier. Right-click the shapefile in the file list and choose "Set as barrier (net)."

CHAPTER 4: Setup and Theory

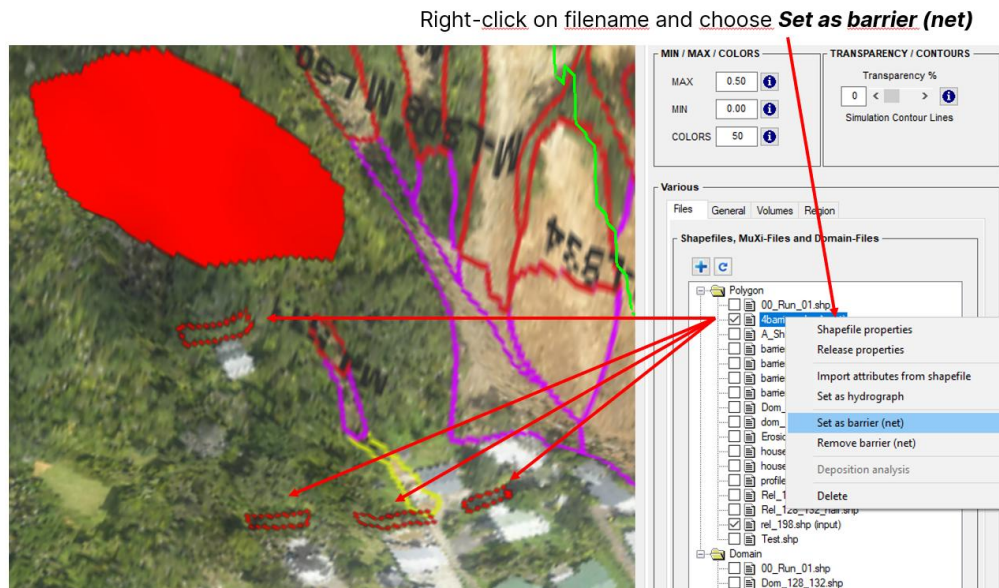


Figure 4-40: Right-click a shapefile in the file list and select "Set as barrier (net)" to activate barrier retention for that polygon.

Setting barrier parameters. For each polygon, set the barrier's width and height (in m). A green arrow indicates which polygon you're currently parameterizing. Clicking Cancel skips that polygon — no retention is applied there.

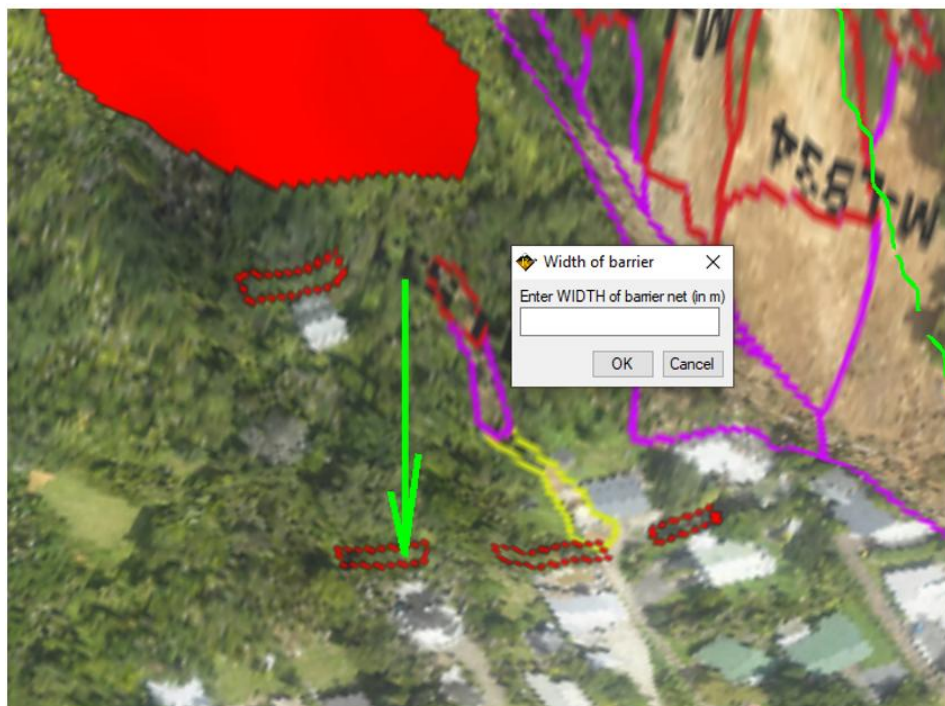


Figure 4-41: The "Width of barrier" dialog, prompting for the barrier width (m). The green arrow indicates which drawn polygon the parameters currently apply to.

CHAPTER 4: Setup and Theory

After entering width and height, RAMMS displays the calculated possible retention volume for that barrier. If this value doesn't match your expectations, click No and enter a custom retention volume (in m³) directly. Repeat for each polygon in the shapefile.

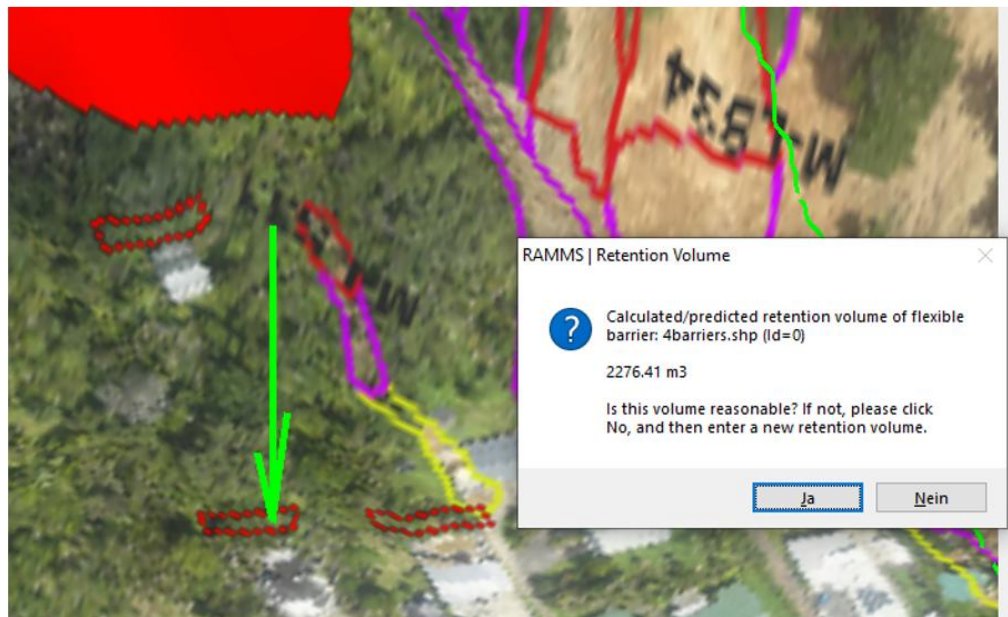


Figure 4-42: RAMMS displays the calculated retention volume for the barrier (here 2276.41 m³) and asks the user to confirm it is reasonable, or reject it to enter a custom value.

Checking and changing parameters. Right-click the shapefile again at any time to re-view the current barrier parameters. To change them, first remove the barrier definition from the shapefile (also via right-click), then redefine it.

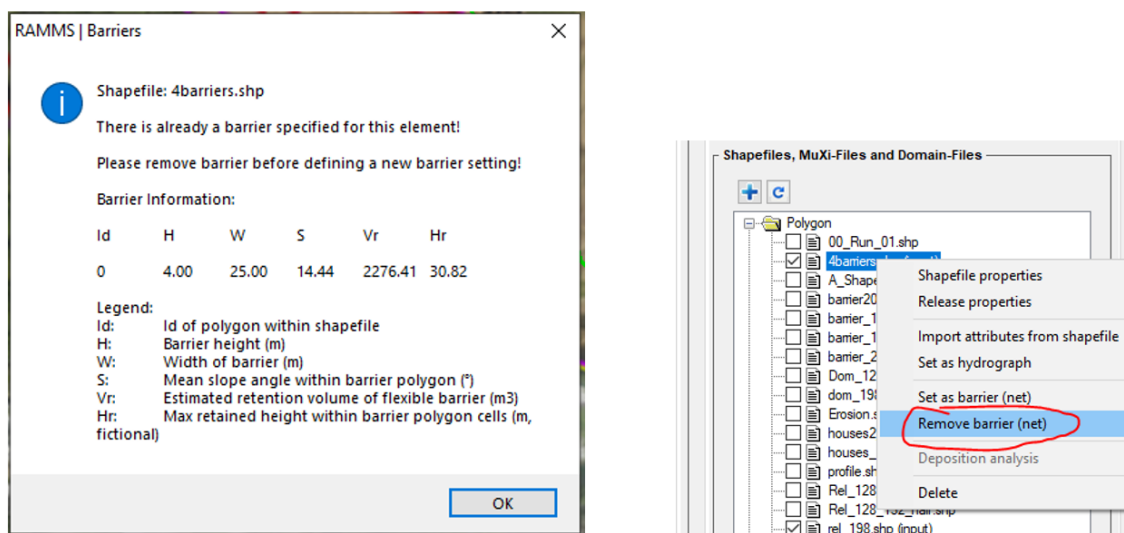


Figure 4-43: Left: the "RAMMS | Barriers" dialog, shown when right-clicking a shapefile that already has barrier parameters defined — lists barrier height (H), width (W), mean slope (S), estimated retention volume (Vr) and maximum retained height (Hr) per polygon. Right: right-click "Remove barrier (net)" to clear the existing definition before redefining it.

CHAPTER 4: Setup and Theory

Barrier parameters are also visible in the Run Simulation window before you start the calculation.

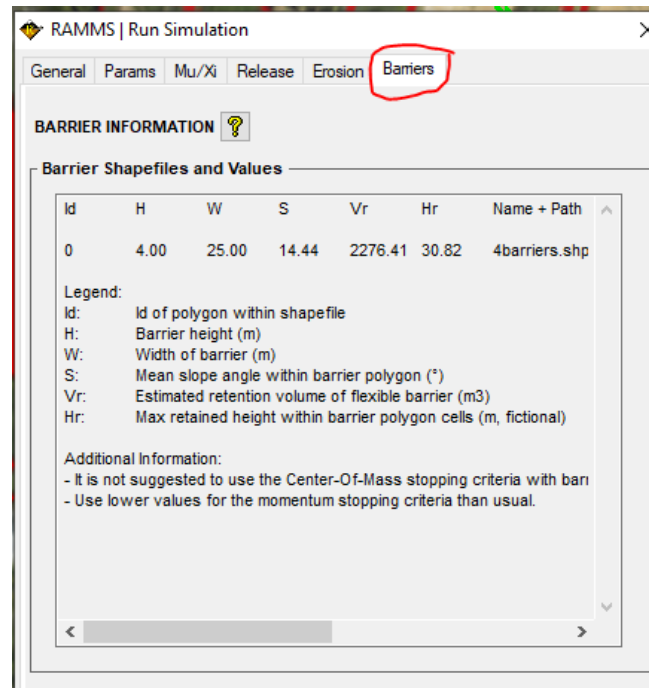


Figure 4-44: The Barriers tab of the Run Simulation window, showing the same barrier parameter table as Figure 4-40 plus reminders to avoid the Center-of-Mass stopping criterion with barriers and to use a lower momentum stopping-criterion value than usual.

Running with barriers. Start the simulation as usual. The console window reports how much volume is being retained as the run progresses.

To add rigid obstacles the flow must not enter (e.g. buildings), use the **Obstacle/Dam File** field on the Params tab of the Run Simulation window — see page 61.

Important: when using flexible barriers, set the (momentum-based) stopping criterion to 1%. Do NOT use the Center-of-Mass stopping criterion together with barriers — see the Stopping criteria section in Chapter 4.

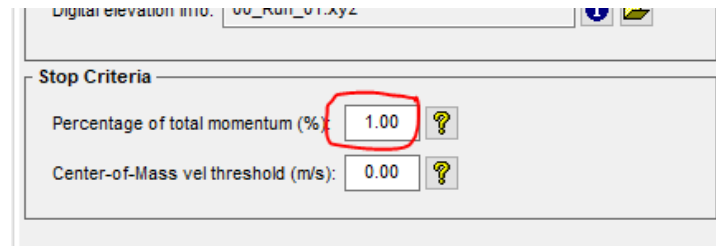


Figure 4-45: The Stop Criteria panel of the Run Simulation window, with the momentum-based stopping criterion set to 1% — the recommended value when using flexible barriers.

4.9.5 Calculation Domain

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

Switch to *2D mode* and choose **Input → Calculation Domain... → Draw New Domain** or click . Now you can draw a polygon containing the area of interest similar to drawing a new polygon area (see exercise "Create release area" on page 40). We strongly recommend using smaller calculation domains especially if you simulate with small cell sizes (e.g. < 5m).

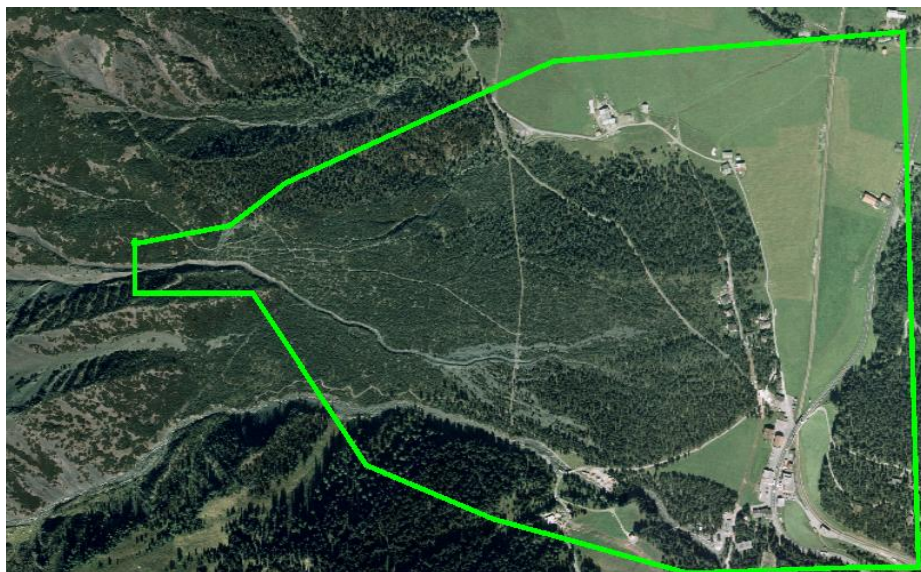


Figure 4-46: Calculation domain in green encloses the area of interest and reduces calculation. time in comparison with the default rectangular domain which is automatically generated

Exercise 3.5e : Finding an optimized calculation domain

- Open your input file.
- Draw a rough calculation domain as explained above, see Figure 4-47 below.

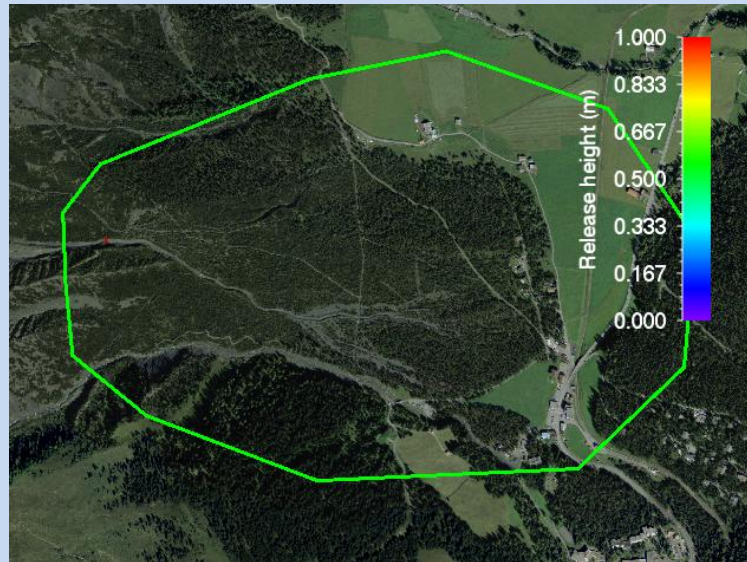


Figure 4-47: Input file with a rough calculation domain

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

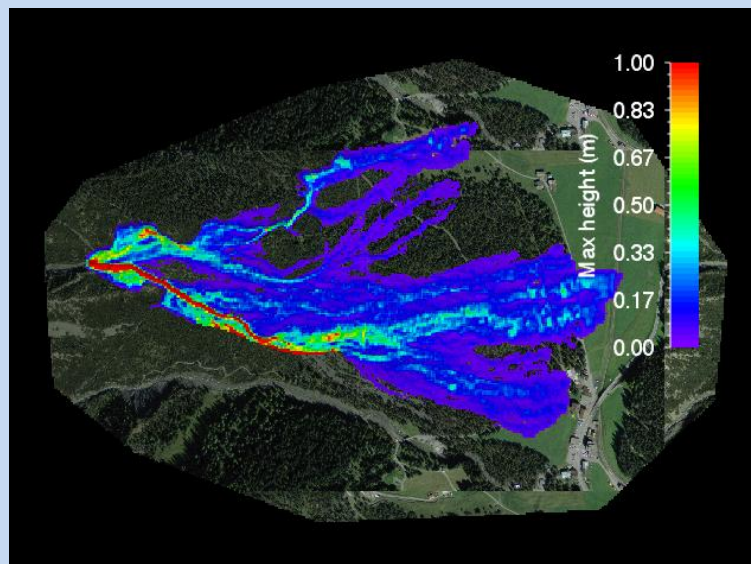


Figure 4-48: Max Flow Height of a 4m simulation

- Click **GIS → Export... → Create Envelope Shapefile**
- Question: *Use buffer?*
- Click **Yes**. We want to use the envelope shapefile as a calculation domain, and therefore we want to buffer it a little bit. Click **No** if you want to have

the exact *Envelope Shapefile* of your output result. Beware: the MIN-value of your colorbar will be used as the lower limit for the *Envelope Shapefile*!

- Choose a filename for the envelope shapefile (a name is proposed, including the MIN-value of your colorbar, if you want an exact envelope).
- The created envelope shapefile is shown in the visualization as a dashed red line.

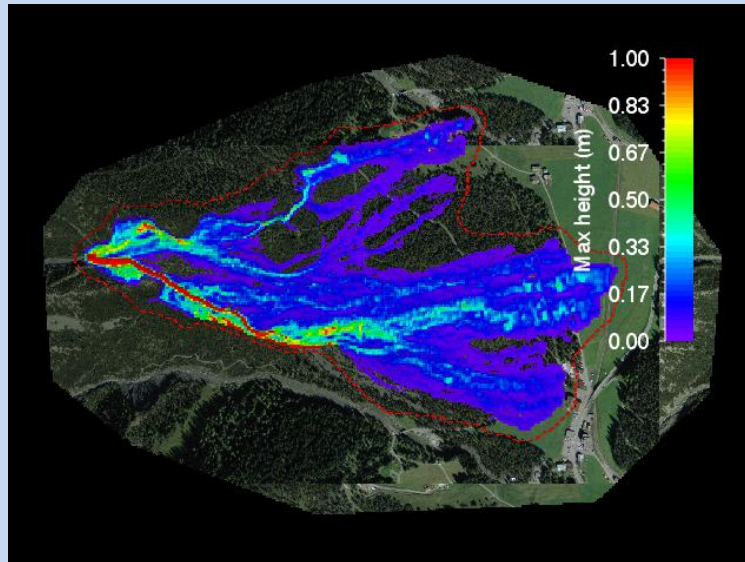


Figure 4-49: Envelope shapefile (dashed red line) of Max Flow Height extent

- Switch back to the input file 
- Use **Input** → **Calculation Domain...** → **Load Existing Domain** to load your envelope shapefile as a new calculation domain, see Figure 4-50 below.

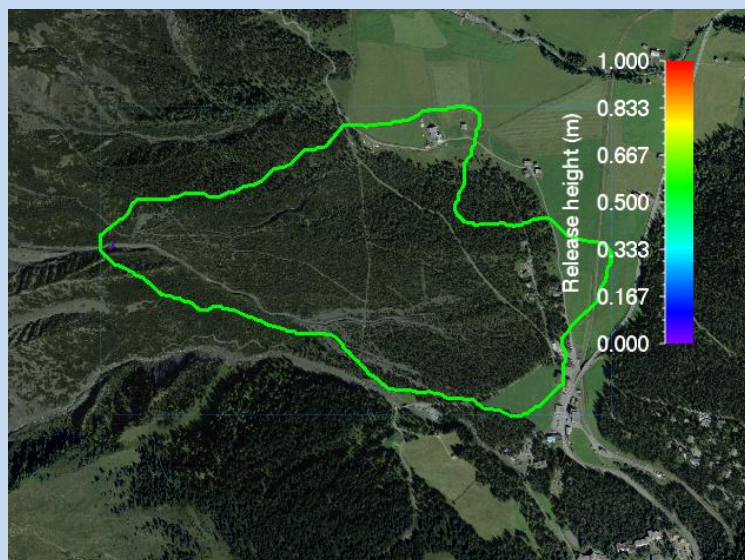


Figure 4-50: Input file with optimized calculation domain (envelope shapefile)

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

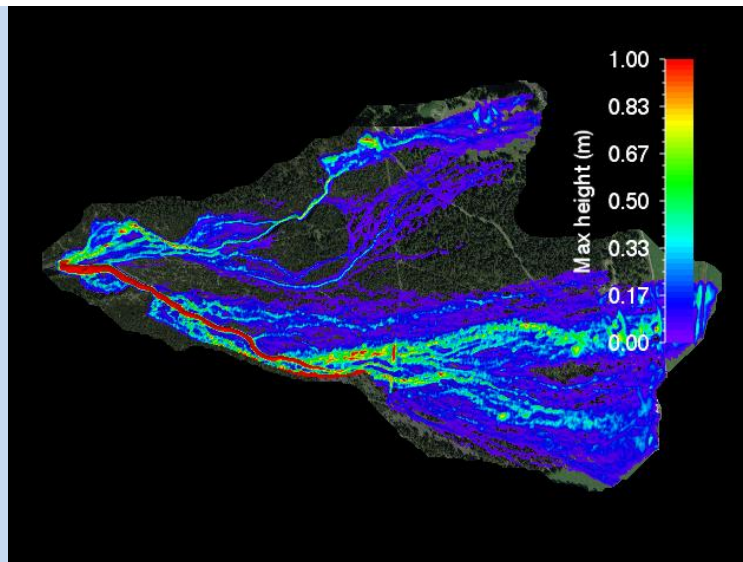


Figure 4-51: Max Flow Height result of a 2m simulation

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

4.10 Running a simulation

The following exercise guides you step by step through the setup and execution of a debris-flow simulation, including block release, hydrographs, erosion, friction parameters, and other relevant model settings.

Exercise 3.5f: How to run a simulation

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

General Tab:

- SCENARIO Name

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

- Additional Information

(3) **Project name:**

(4) **Details:** 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 5.4.

(8) Center of mass stopping criterion, see section 5.4.

- 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.

RAMMS | Run Simulation

General Params Mu/Xi Release Erosion

GENERAL SIMULATION INFORMATION

SCENARIO Name

Manual_Example

☐ Save Max Values Only ("_Max" will be appended to the output name)

Additional Information

Project name: Wildibach

Details: Example simulation for manual

Calculation domain: Wildibach_5m_Max_height_(m)_envelope.d

Digital elevation info: Wildibach.xyz

Stop Criteria

Percentage of total momentum (%): 5.00

Center-of-Mass vel threshold (m/s): 0.50

Remarks

- use "Escape" to close/cancel this window

- use "Ctrl+R" to start a simulation

☐ Run in background

Cancel RUN SIMULATION

Figure 4-52 General Information

Params Tab

(1) Simulation Parameters

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

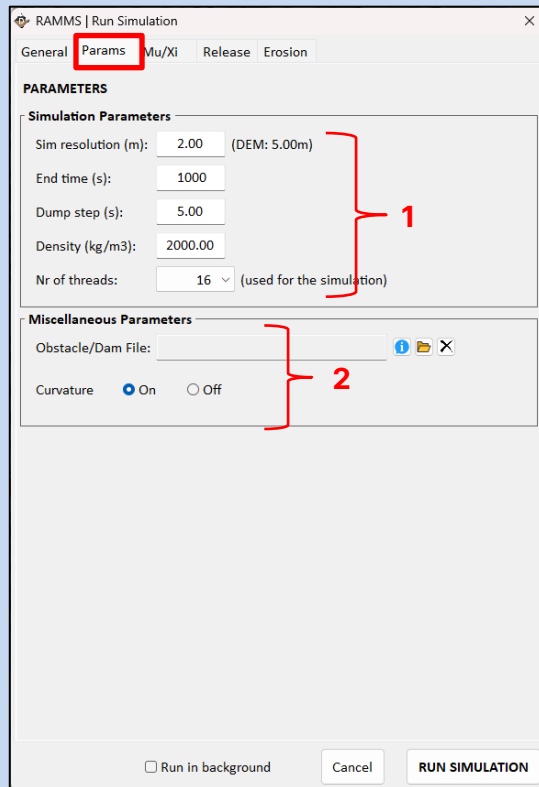


Figure 4-53: Parameter Tab

(2) Miscellaneous Parameters

- Obstacle/Dam File: Draw polygons of areas, where no flow should pass (houses, deflecting dams, obstacles). The flow is deflected. (commonly used together with flexible barriers (nets) — see section 4.9.4.)
- 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-debrisflow/friction-parameters/> for more information.

Continuation of exercise 3.5f: How to run a simulation

MuXi Tab

- (1) Enter the friction parameters μ and ξ . Start first scenario simulation with default values ($\mu = 0.2$, $\xi = 200 \text{ m/s}^2$). More information on frictional resistance is given in sections 4.3 and 6.1.
- (2) Check box **Define Additional MuXi Areas**: if you want to add polygon regions with different μ and ξ values. e.g. to represent significant changes in the channel, heavily forested areas or perhaps smooth roads. Load the corresponding polygon shapefiles and define μ and ξ for up to two additional regions. Be very careful in using this feature!
- (3) **Yield Stress**: Define a polygon area where you want to use an additional yield stress (cohesion). Specify a yield stress in the *Yield stress (Pa)* field. See section 4.3 for more details about yield stress.

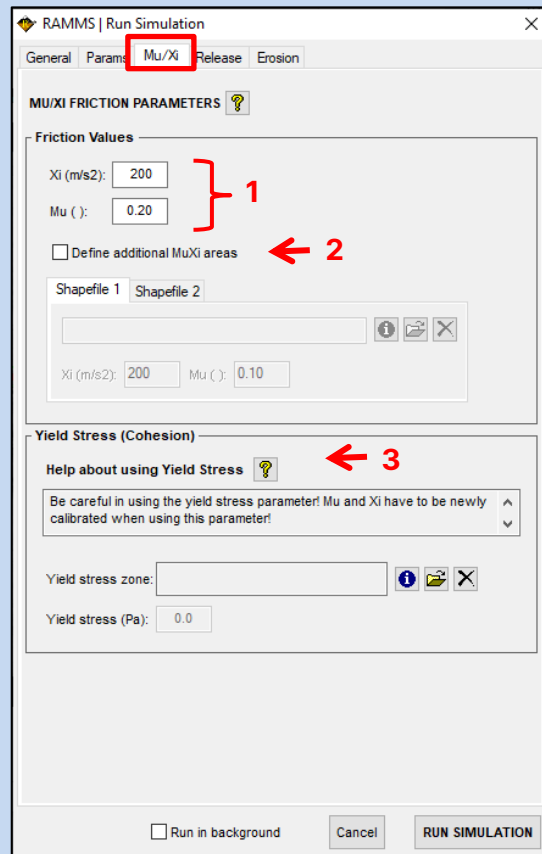


Figure 4-54: MuXi Tab

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

Release - Hydrograph Tab

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

- (1) **Use BLOCK release** (Check box):
As an alternative to a simulation with a hydrograph.
- (2) **Hydrograph Help**
- (3) **Hydrograph/Release Shapefile(s)**: Release shapefile(s). If more than one shapefile is specified, a **+** button is shown. A tooltip will show the additional shapefile names when moving the mouse over the button.
- (4) **Inflow direction (degrees)**: Define the inflow direction of the debris flow at the hydrograph input location. Check **Automatic** to let RAMMS set the inflow direction. Click the info button for a detailed description of inflow direction.
- (5) **3-Point Hydrograph Calculation**: Define a discharge hydrograph by entering total volume V (maximum discharge Q_{max} is calculated automatically according to Rickenmann, 1999), corresponding time at which the maximum occurs (t_{max}) and velocity v . End time (t_{end}) of the input hydrograph is automatically calculated.
- (6) **Edit hydrograph table** (Check box): Activates the table (7) and is an alternative to a 3-Point Hydrograph Calculation (5).

RAMMS | Run Simulation

General Params Mu/Xi **Release** Erosion

RELEASE PARAMETERS ? ☐ Use BLOCK release

Release Information

Hydrograph Block Release

Hydrograph Parameters ?

Release shapefile(s)
No release shapefile specified!

Inflow direction

Inflow direction (degrees) 100 ? ☐ Automatic

3-Point Hydrograph

Vol (m3)	Qmax (m3/s)	t1 (s)	v (m/s)	t2 (s)
10000	100	10	5	200.000

☐ Edit hydrograph table

Hydrograph-Plot Hydrograph-Table

Hydrograph-Plot: Q (m³/s) vs Time (s)

Run in background Cancel **RUN SIMULATION**

Figure 4-55: Hydrograph Tab

Release - Block Release Tab

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

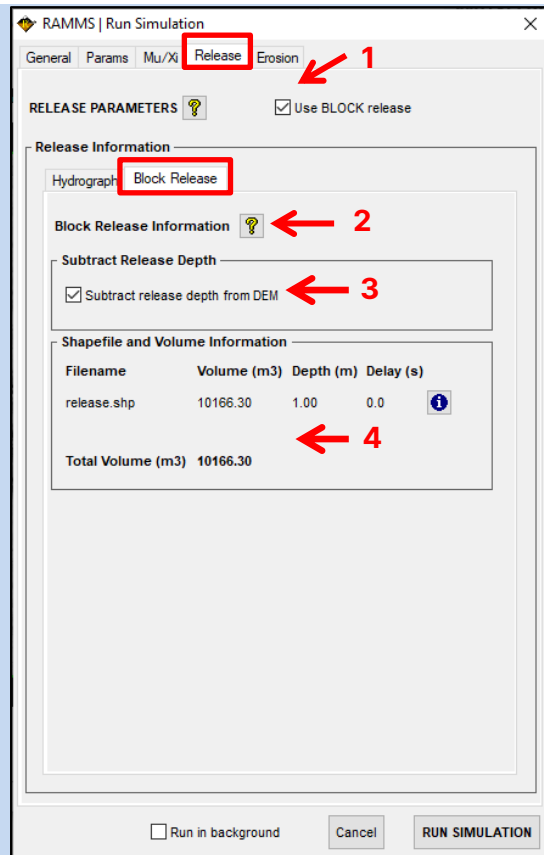


Figure 4-56: Hydrograph Tab

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

Erosion Tab

- (1) **Erosion Shapefile:** Specify your erosion shapefile area.
- (2) **Erosion Parameters:** Specify erosion parameters. Use dropdown menus to select parameter values. See section 4.9.3 for more details about erosion parameters.
- (3) **Add to list button:** Click the button to add your erosion shapefile and parameters to your erosion list.
- (4) **Erosion shapefile list:** List of erosion shapefiles and parameters. You can delete single entries or the whole list with the buttons *Delete* and *Clear List*.
- (5) **Run Simulation** button.

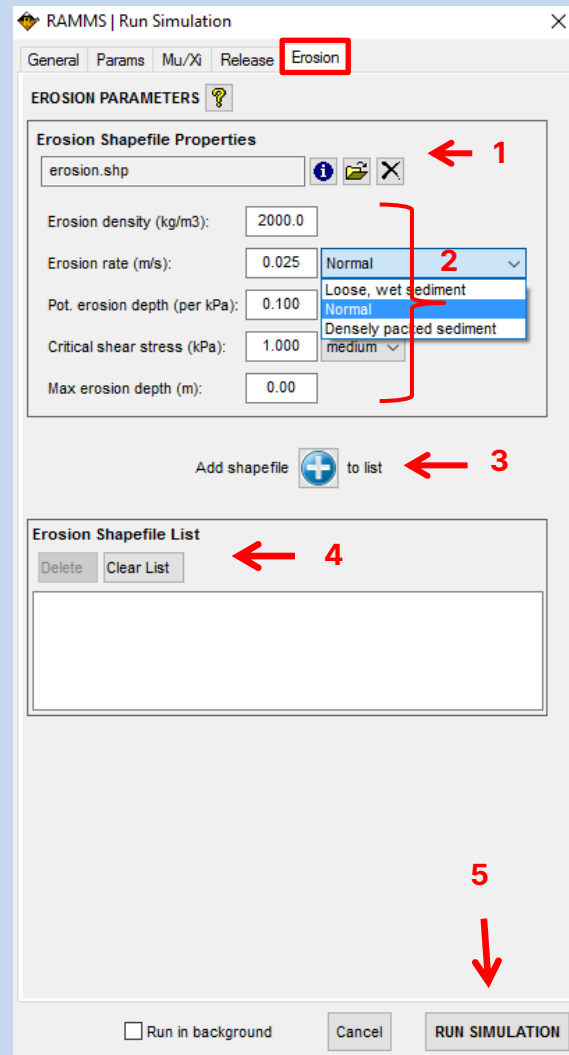


Figure 4-57: Erosion Tab

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

- Click RUN SIMULATION.
- When a simulation is started, RAMMS opens a separate calculation window in which the numerical solver is executed (Figure 4-58).
(1) General information of the simulation, (2) output file, (3) starting the calculation (4) for every time step RAMMS calculates the maximal values (height, velocity and pressure) as well as the outflow mass, the moving momentum and the flow volume.

```

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

Welcome to the RAMMS::DEBRISFLOW x86 calculation module!

Reading input file...
1 starting calculation with 24009 nodes, 23631 elements and 46885 edges
starting time is Mon Nov 27 14:04:02 2017

Initializing debrisflow...
2 Writing output file: E:\RAMMS\DEBRISFLOW\Beispiele\Test\Test.out...

Writing coordinates...
Writing connectivity list...
Writing initial conditions...
3 starting explicit time integration...

INITIAL BLOCK VOLUME: 1196.598101
***.....

4 Time 2.1
Step 12 dt 0.150584 Cfl 0.450232
Umax 1.81 m Vmax 10.29 m/s

MOVING MOMENTUM: 100.0 percent ( 237.7 / 237.7 )

FLOW VOLUME: 1196.60 m3

ERODED VOLUME: 0.00 m3
BLOCK VOLUME: 1196.60 m3
NUMERICAL VOLUME LOSS: 0.00 m3

VOLUME CHECK: 1.0
.....

```

Figure 4-58: RAMMS calculation window showing the progress and status information during a simulation.

```

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

Finished writing RAMMS output file: E:\RAMMS\DEBRISFLOW\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\DEBRISFLOW\Beispiele\Test\Test.out.gz" "E:\RAMMS\DEBRISFLOW\Beispiele\Test\Test.out"

7-Zip (A) 9.20 Copyright (c) 1999-2010 Igor Pavlov 2010-11-18
Scanning

Creating archive E:\RAMMS\DEBRISFLOW\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\DEBRISFLOW\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. 4-59: RAMMS calculation window showing a **background** simulation. Press any button to close the window once the simulation finished.

CHAPTER 4: Setup and Theory

4.10.1 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**.

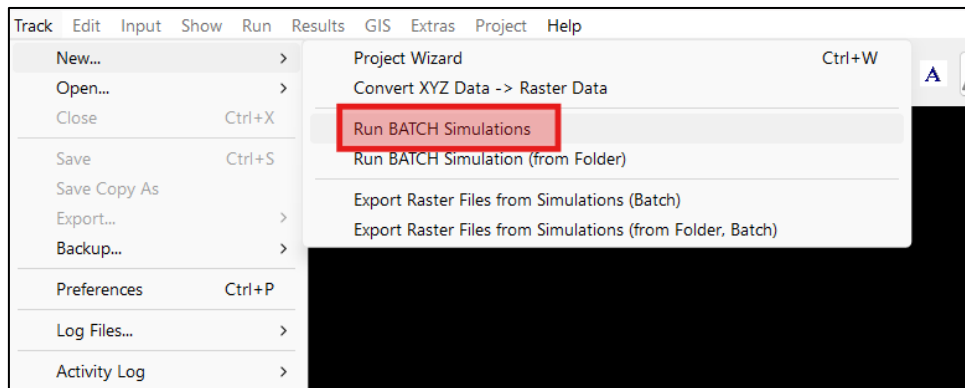


Figure 4-60: Batch-Simulations

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.

CHAPTER 4: **Setup** and Theory

5. Select **Track → Save Copy As** or click the corresponding **Save Copy As** button. Save the scenario as a new RAMMS input file (.db2) using a meaningful file name.
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 is finished, the simulation as well as the output logfile (see next section) are visualized in RAMMS (if you ran the simulation in background mode, see Figure 4. 4-59, click any button inside the DOS window to close the window. Afterwards, open the simulation in RAMMS manually).

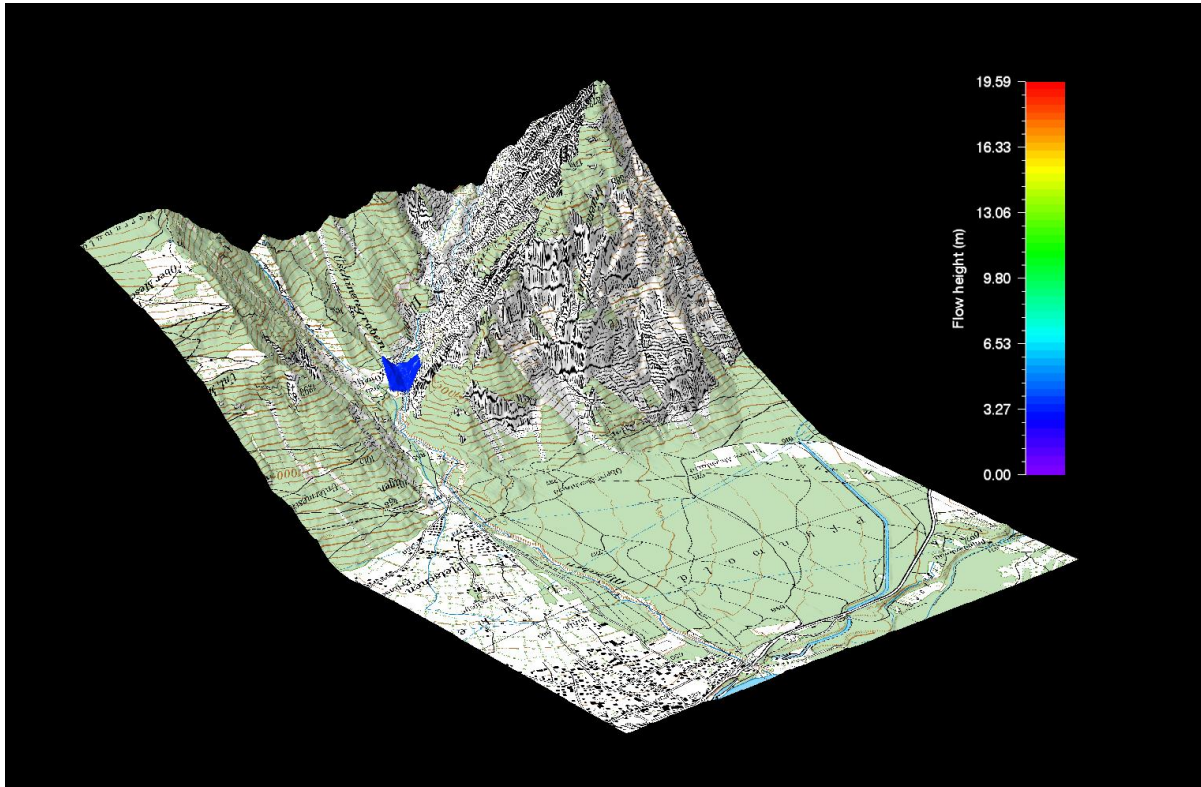


Figure 5-1: Main window in output mode.

If mass flows out of the calculation domain (RAMMS will mention this in the logfile, see example log file below) the calculation domain should generally be enlarged to ensure that the relevant flow process is fully represented (see section 4.9.5).

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.

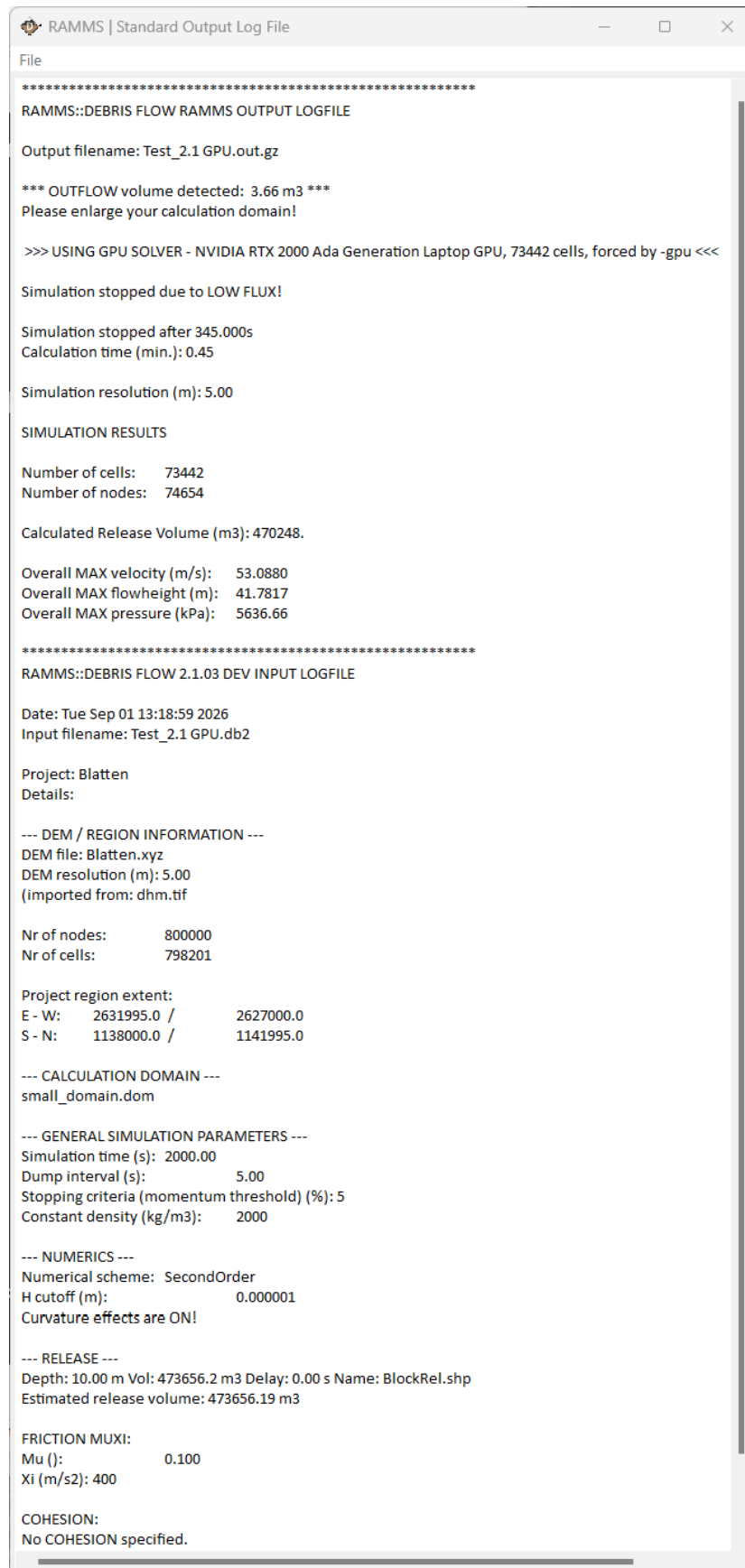
At the beginning of the logfile, RAMMS reports which solver was used. Depending on the calculation, this indicates whether the simulation was performed using the CPU or GPU. For GPU calculations, the GPU model and the number of computational cells are also listed. In the example logfile, the simulation was calculated using an NVIDIA RTX 2000 Ada Generation Laptop GPU with 73442 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 345s. 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::DEBRIS FLOW RAMMS OUTPUT LOGFILE

Output filename: Test_2.1 GPU.out.gz

*** OUTFLOW volume detected: 3.66 m3 ***
Please enlarge your calculation domain!

>>> USING GPU SOLVER - NVIDIA RTX 2000 Ada Generation Laptop GPU, 73442 cells, forced by -gpu <<<

Simulation stopped due to LOW FLUX!

Simulation stopped after 345.000s
Calculation time (min.): 0.45

Simulation resolution (m): 5.00

SIMULATION RESULTS

Number of cells:    73442
Number of nodes:   74654

Calculated Release Volume (m3): 470248.

Overall MAX velocity (m/s):    53.0880
Overall MAX flowheight (m):    41.7817
Overall MAX pressure (kPa):    5636.66

*****
RAMMS::DEBRIS FLOW 2.1.03 DEV INPUT LOGFILE

Date: Tue Sep 01 13:18:59 2026
Input filename: Test_2.1 GPU.db2

Project: Blatten
Details:

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

Nr of nodes:      800000
Nr of cells:      798201

Project region extent:
E - W:  2631995.0 /      2627000.0
S - N:  1138000.0 /      1141995.0

--- CALCULATION DOMAIN ---
small_domain.dom

--- GENERAL SIMULATION PARAMETERS ---
Simulation time (s): 2000.00
Dump interval (s):      5.00
Stopping criteria (momentum threshold) (%): 5
Constant density (kg/m3): 2000

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

--- RELEASE ---
Depth: 10.00 m Vol: 473656.2 m3 Delay: 0.00 s Name: BlockRel.shp
Estimated release volume: 473656.19 m3

FRICTION MUXI:
Mu ():      0.100
Xi (m/s2): 400

COHESION:
No COHESION specified.
```

Figure 5-2: Example output log file.

CHAPTER 5: Results

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 or hydrograph definition, friction parameters, cohesion, and map or orthophoto information. For hydrograph simulations, the logfile also records the individual hydrograph points with time, discharge, and inflow velocity.

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.

5.1.2 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.

5.1.3 Project Region Information

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

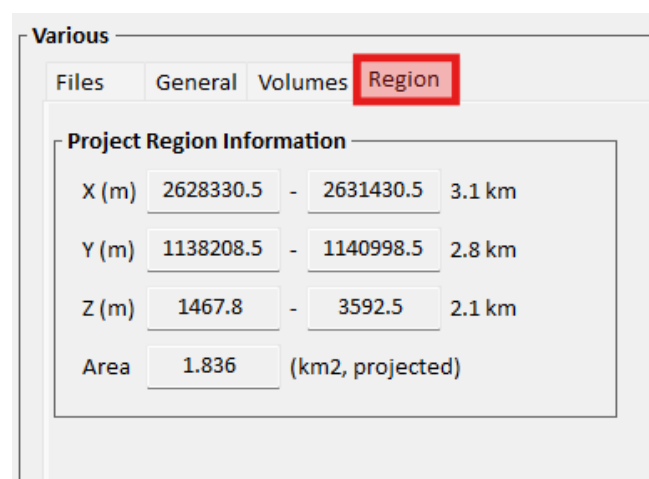


Figure 5-3: Project region information showing the X-, Y- and Z-extent and the total project area.

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

CHAPTER 5: Results

maximum terrain elevation are shown as the **Z extent**. RAMMS also displays the total projected **project area** in km².

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 the 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

CHAPTER 5: Results

at each location, whereas dump-step results are used to analyze the temporal development and dynamics of the flow.

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 Momentum
- Erosion
- Max values
 - Height 
 - Velocity 
 - Impact Pressure 
 - Line Load  (see section 5.3.2)
 - Momentum
 - Shear Stress
 - Erosion
- DEM Adaptions (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (μ , ξ)

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

Exercise 4.2a: Displaying calculation values

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

- Max flow height 
- Max flow velocity 
- Max impact pressure 

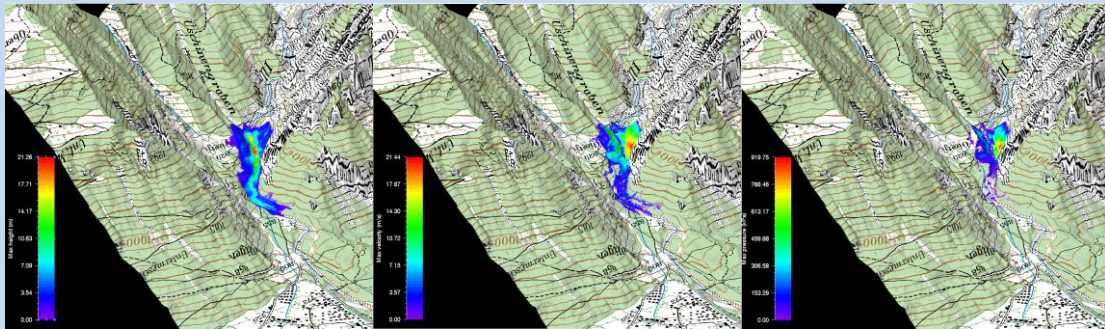


Figure 5-4: Results: Maximum values of flow height (left), velocity (middle) and impact pressure (right)

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.

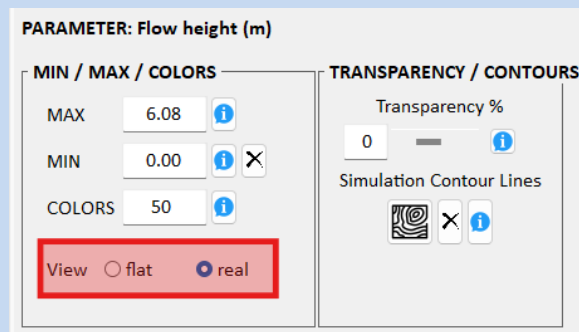


Figure 5-5: Toggle between flat and real view (quasi-3D).

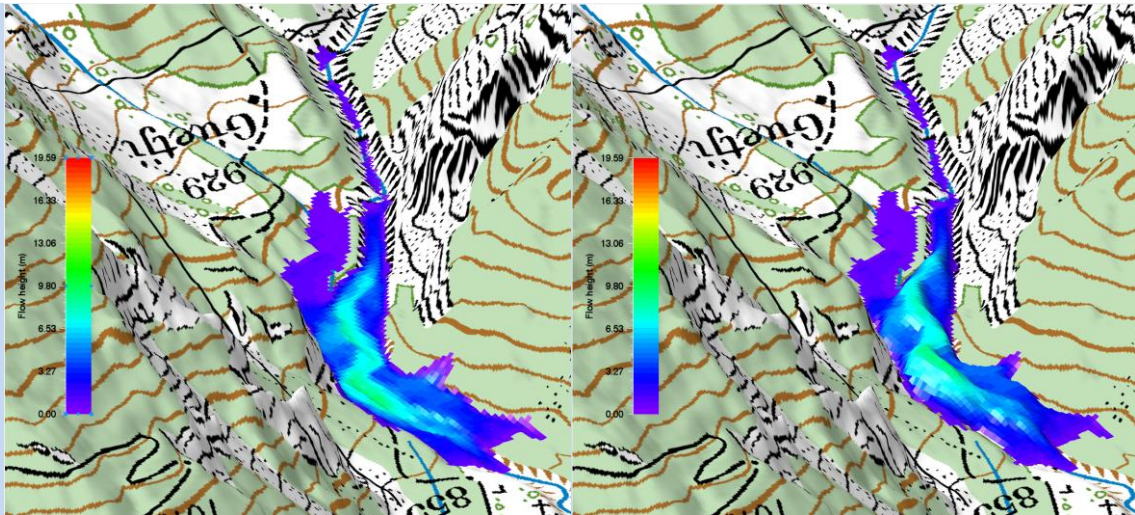


Figure 5-6: Quasi 3D-Visualization of flow height (left: flat; right: real).

5.2.3 Line profile plot and time plot

In the horizontal toolbar you find two further functions:

- Line Profile 
- Time Plot 

Line profile plot

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

Time plot

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

Flow discharge plot

The **Results → Discharge at Line Profile** tool calculates the debris-flow discharge across a user-defined line profile and displays the resulting discharge hydrograph over simulation time. For each grid cell intersected by the profile, the local flow depth and flow velocity are multiplied by the exact length of the profile within that cell. The contributions from all intersected cells are summed to obtain the total discharge in m^3/s . The

CHAPTER 5: Results

profile should therefore be drawn approximately perpendicular to the main flow direction. Results are shown as a function of simulation time in seconds and are also exported to a CSV file in the project folder.

Exercise 4.2b: How to draw a line profile

a) Draw a new line profile:

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

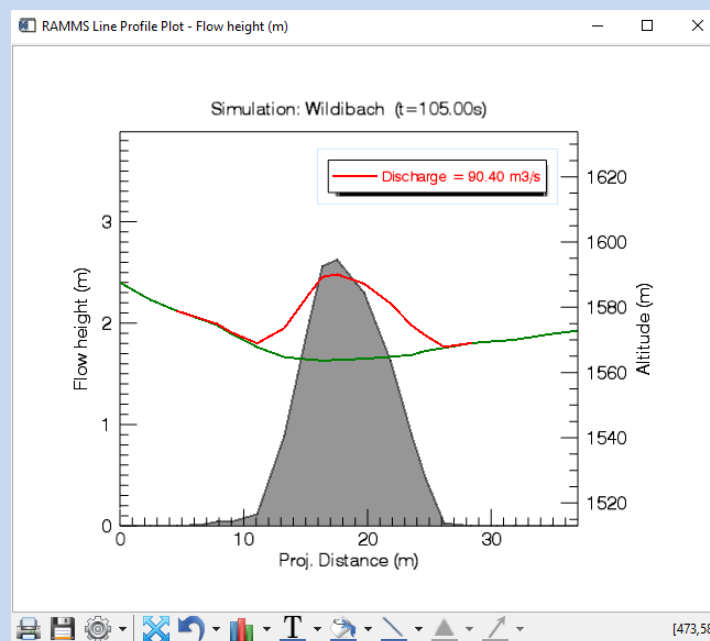


Figure 5-7: Line profile plot, with calculated flow discharge.

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

CHAPTER 5: Results

- 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 profile path is possible.

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

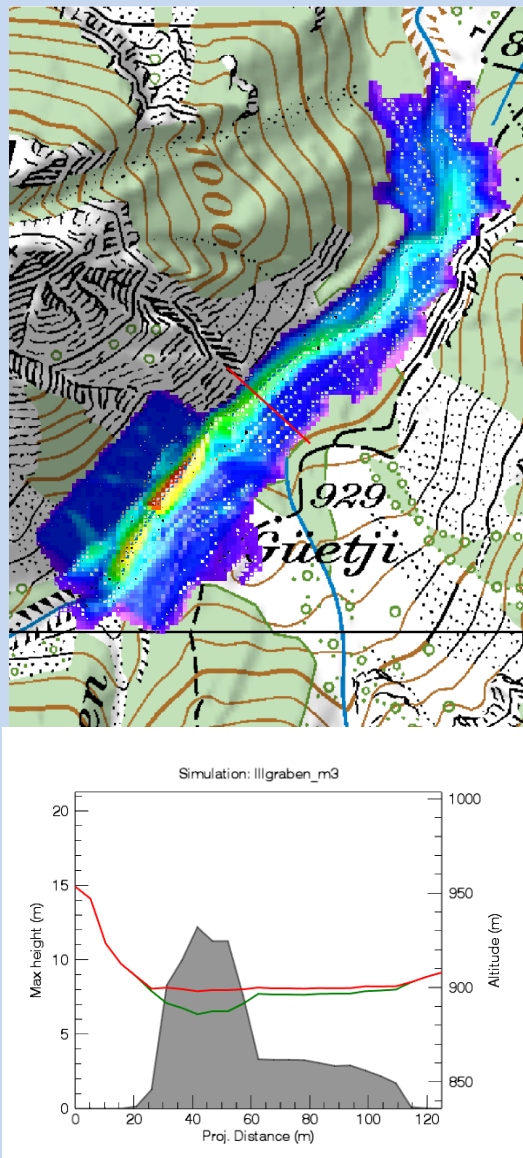


Figure 5-8: Line profile perpendicular to debris flow direction.

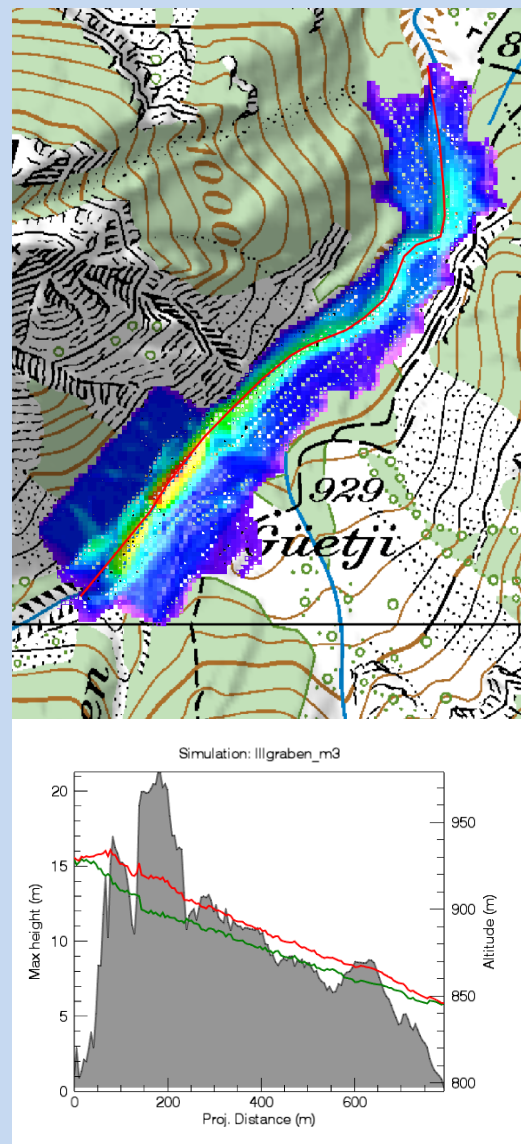


Figure 5-9: Line profile along the debris flow direction.

- To save the coordinates of the points belonging to the line profile, go on **Extras** → **Profile** → **Save Line Profile Points** and enter a file name.

CHAPTER 5: Results

- 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.

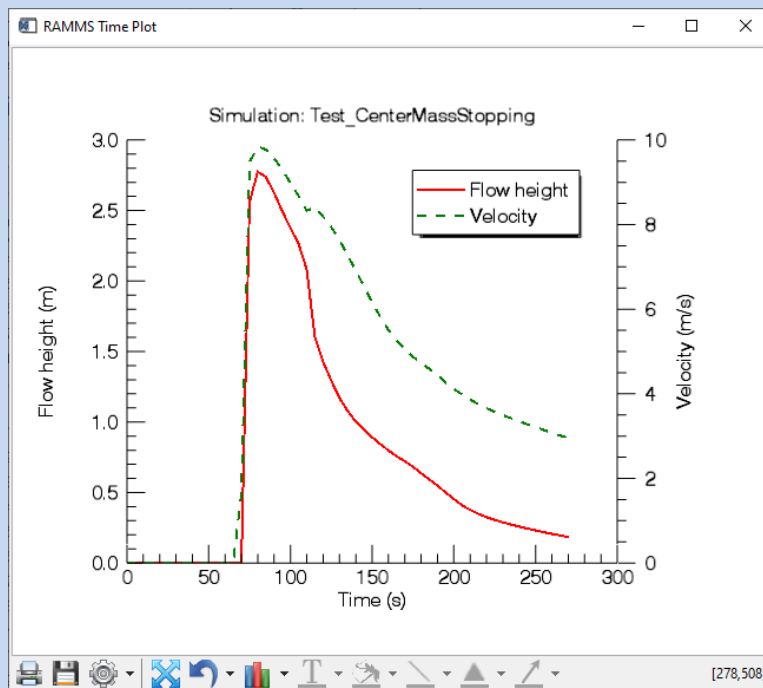


Figure 5-10: Time plot window.

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, use **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.
- Select **Deposition analysis**.

The polygon used for the analysis is shown in the map view (Figure 5-11).

The resulting analysis window (Figure 5-12) 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.

CHAPTER 5: Results

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.

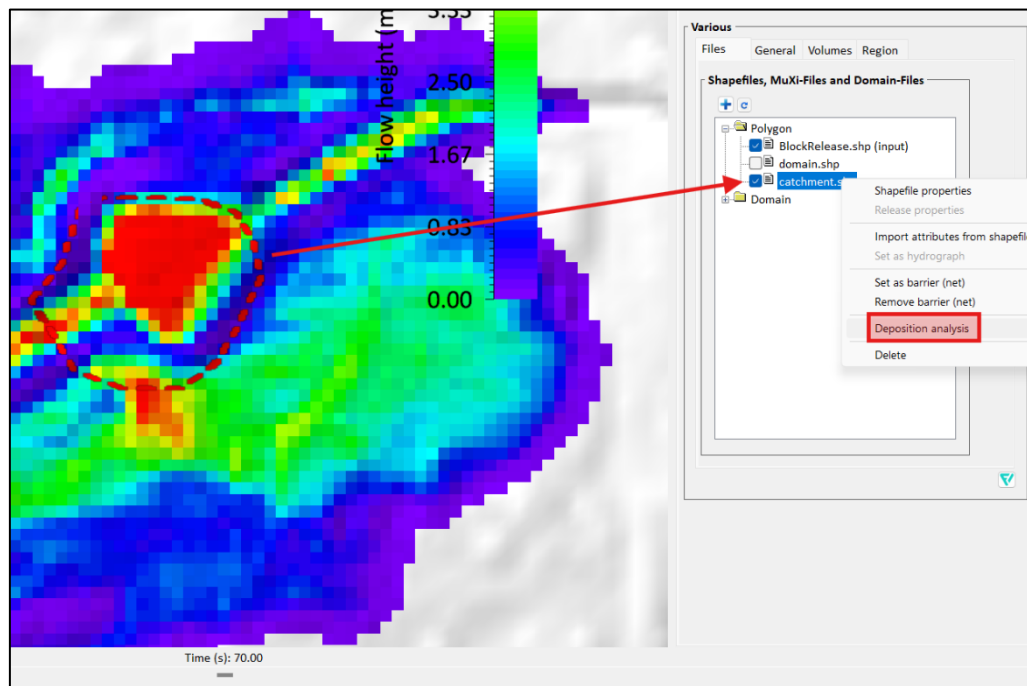


Figure 5-11: Deposition analysis of region of interest.

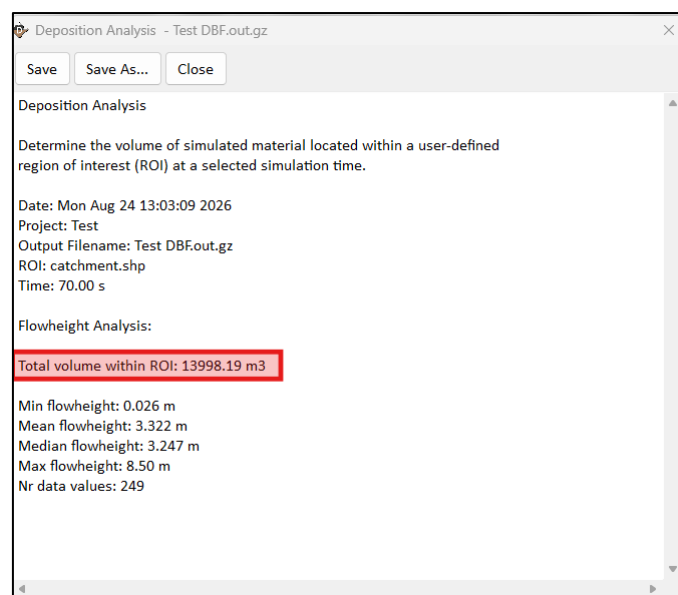


Figure 5-12: Result of a deposition analysis. Total deposition volume (m^3) as well as some statistical values are shown (min, mean, max).

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.

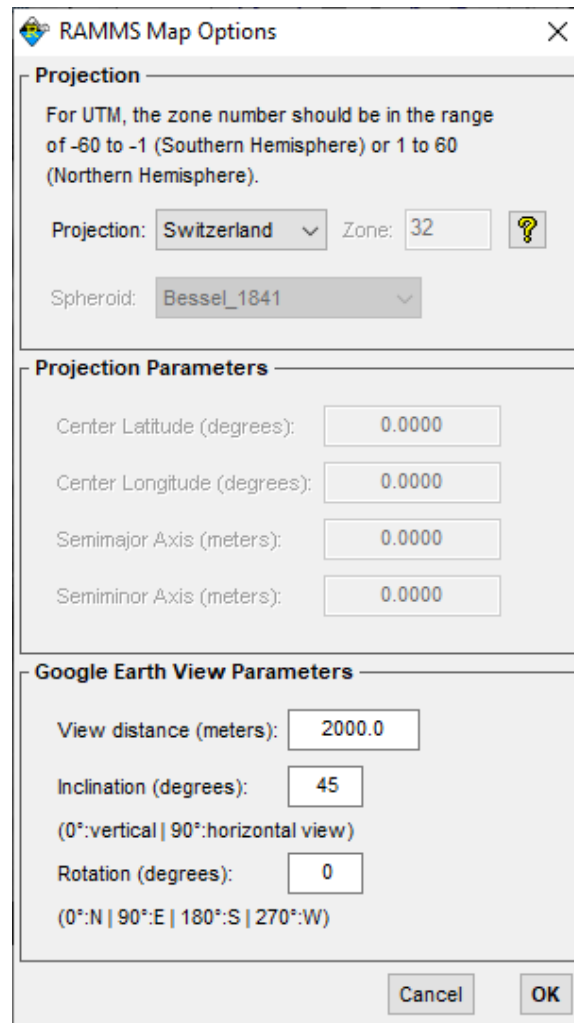


Figure 5-13: Google Earth Options dialog. Specify the coordinate system of the RAMMS project and the initial Google Earth viewing parameters.

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**.

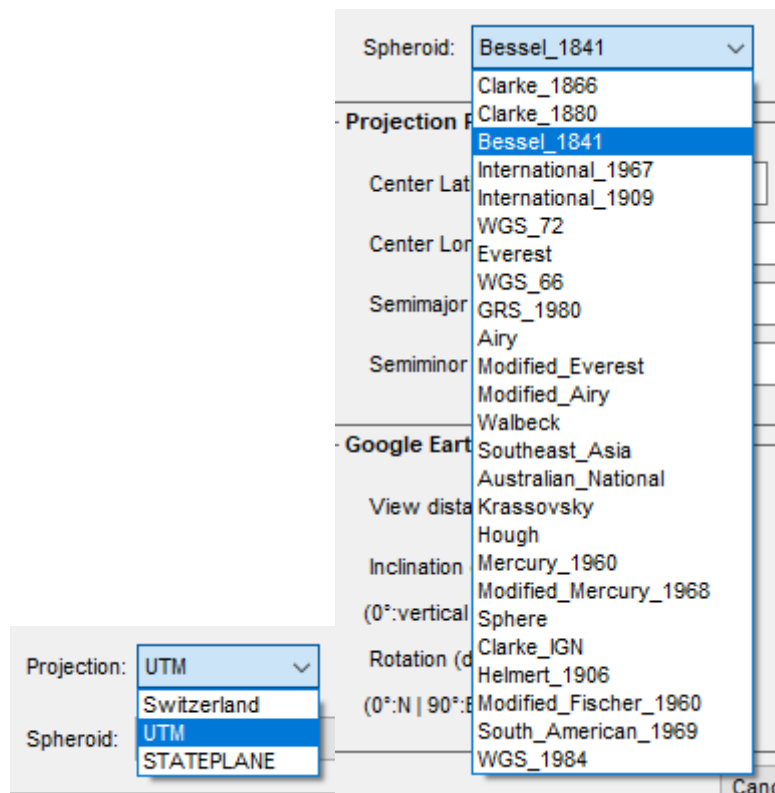


Figure 5-14: Available Projection and Spheroid selections for the Google Earth export.

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.

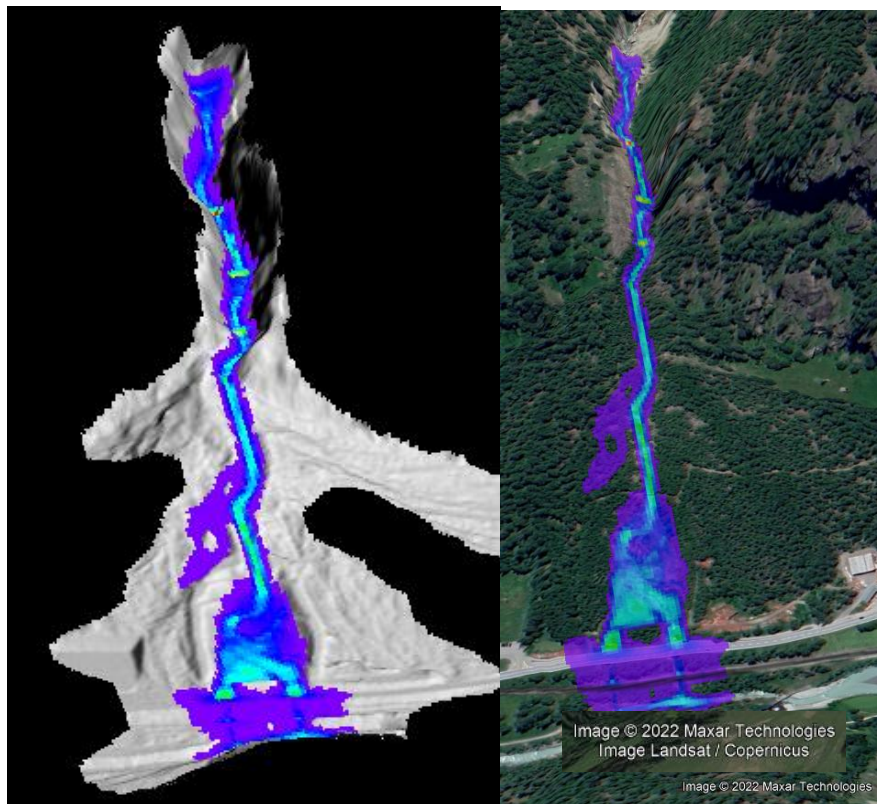


Figure 5-15: Example of a simulation result visualized in RAMMS (left) and exported as a ground overlay to Google Earth (right) (Google Earth imagery © Google, Maxar Technologies, Landsat/Copernicus).

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 *Prefereces (debris flow 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 Impact Pressures

5.3.1 Visualization of Impact Pressures

There are two possibilities to visualize debris flow impact pressures and line loads. The first is to **display maximum pressures over the entire simulation time and domain** (Figure 5-17), the second is to **display the impact pressure at a specific time**.

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 5-16). Maximum pressures are calculated using $C_d = 2$ and therefore represent impact pressures on wide objects (Figure 5-17). 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 5-16). Similar to the maximum pressures, maximum line loads are calculated with $C_d = 2$.

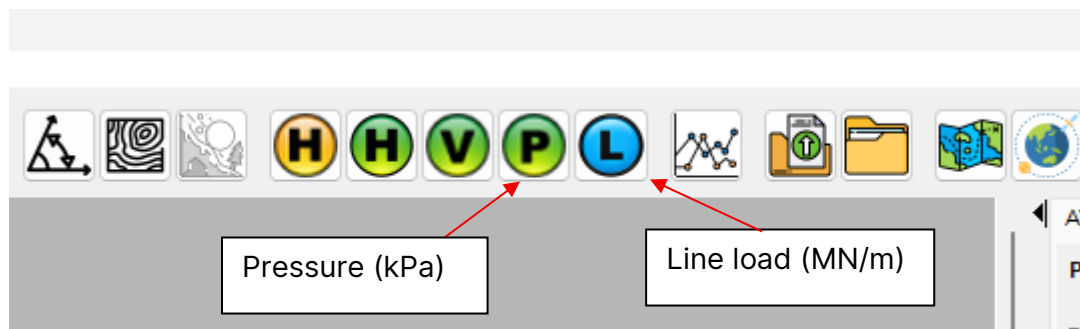


Figure 5-16: Maximum pressures and line loads can be displayed using the green P button ($C_d = 2$) and blue L button ($C_d = 2$).

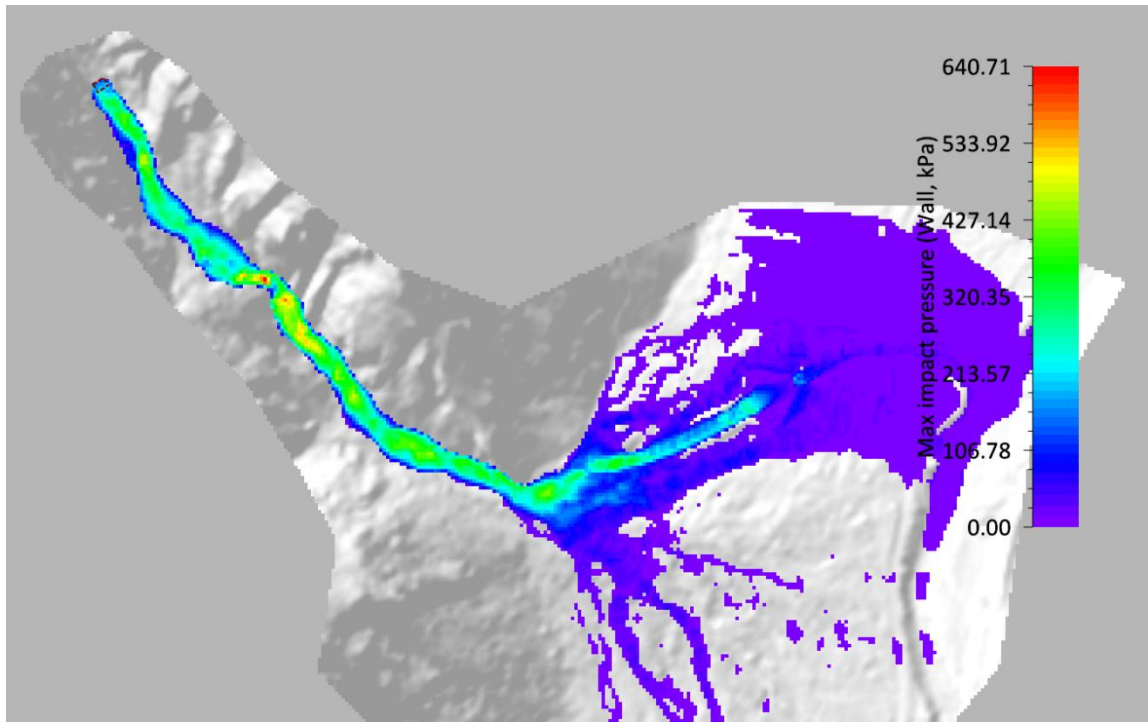


Figure 5-17: Spatial distribution of maximum pressures using the green P icon.

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 5-18).

CHAPTER 5: Results

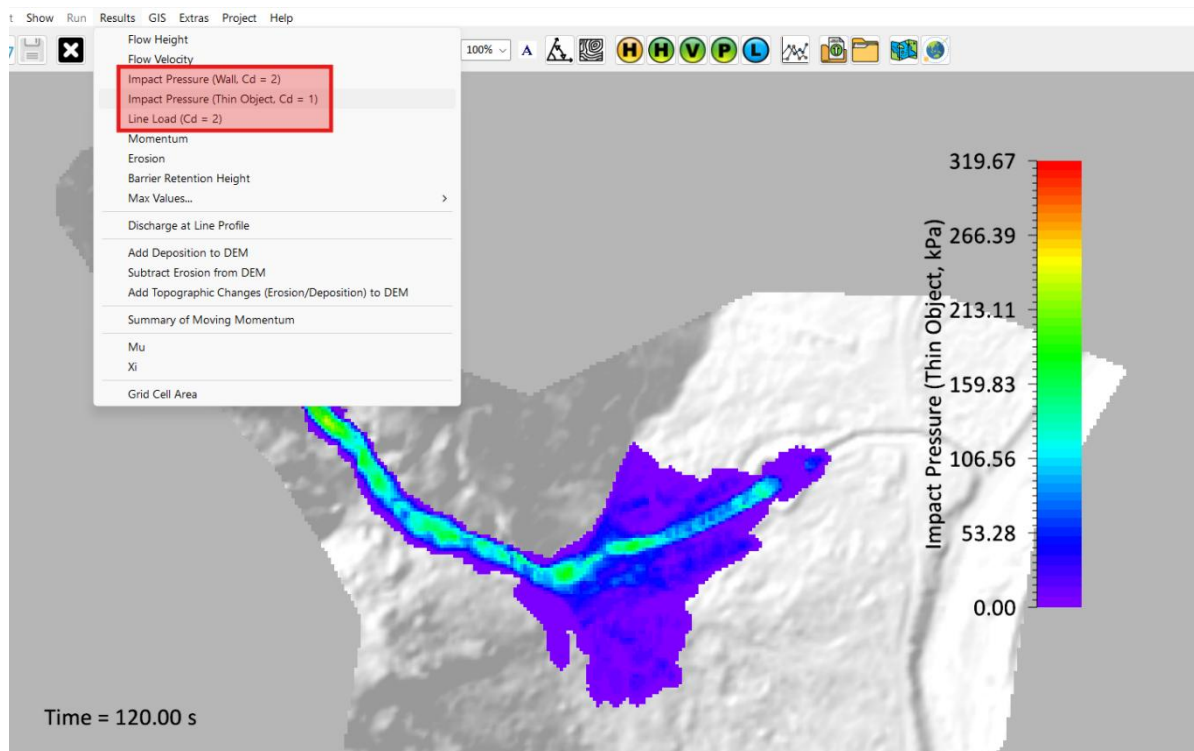


Figure 5-18: Calculated pressures on wide and thin objects at a specific time can be displayed using one of the pressure options in the results menu. The time is set using the time-slider.



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 5-19). Once the plot has been created it is then possible to export the numerical results.

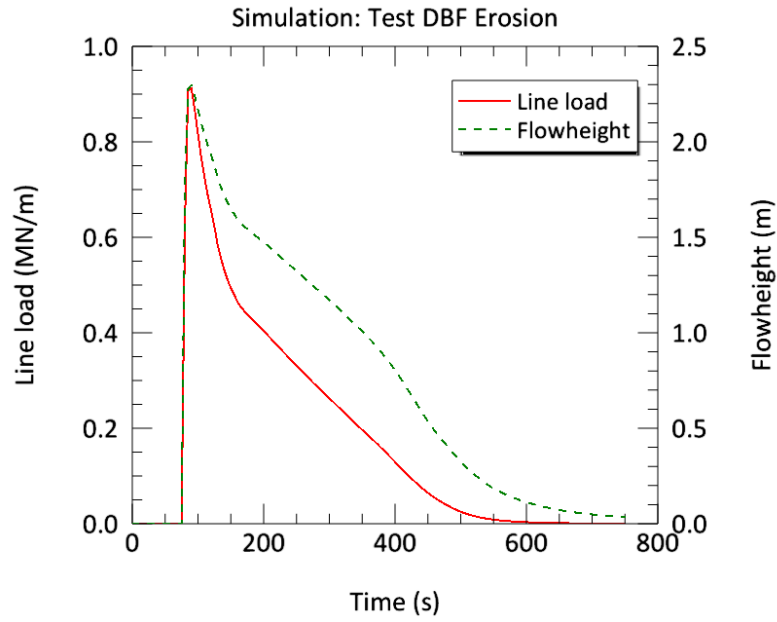


Figure 5-19: Example of a time-history plot of a impact line load. The line loads are depicted with the flow height (dashed line).

5.3.2 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 flow speed but does not distinguish between a very shallow and a very deep flow. Multiplying by the flow depth integrates the impact pressure over the depth of the flow, resulting in a line load in kN/m..

CHAPTER 5: Results

Table 3: Example for $\rho = 2000 \text{ kg/m}^3$. 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	10 m/s	200 kPa	100 kN/m
3.0 m	5 m/s	50 kPa	150 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.

5.4 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 4-52). 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

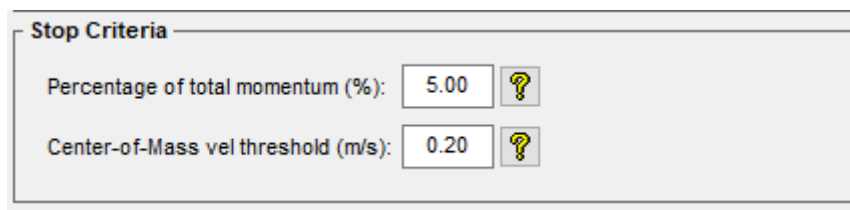


Figure 5-20: Momentum-based and center-of-mass stopping criteria in the Run Simulation window.

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 criteria

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):

CHAPTER 5: Results

$$p=mv$$

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.

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 5-21).

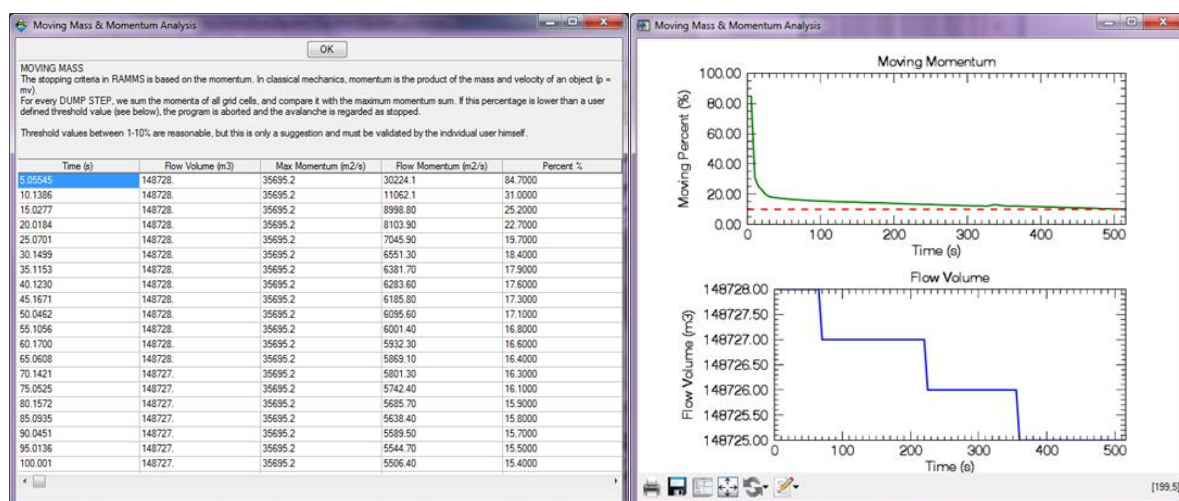


Figure 5-21: Moving Mass & Momentum Analysis used to evaluate the stopping behavior of a simulation.

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

CHAPTER 5: Results

unnecessarily prolong the calculation and allow very small residual velocities, numerical diffusion, or slow creeping of material to continue for a long time.

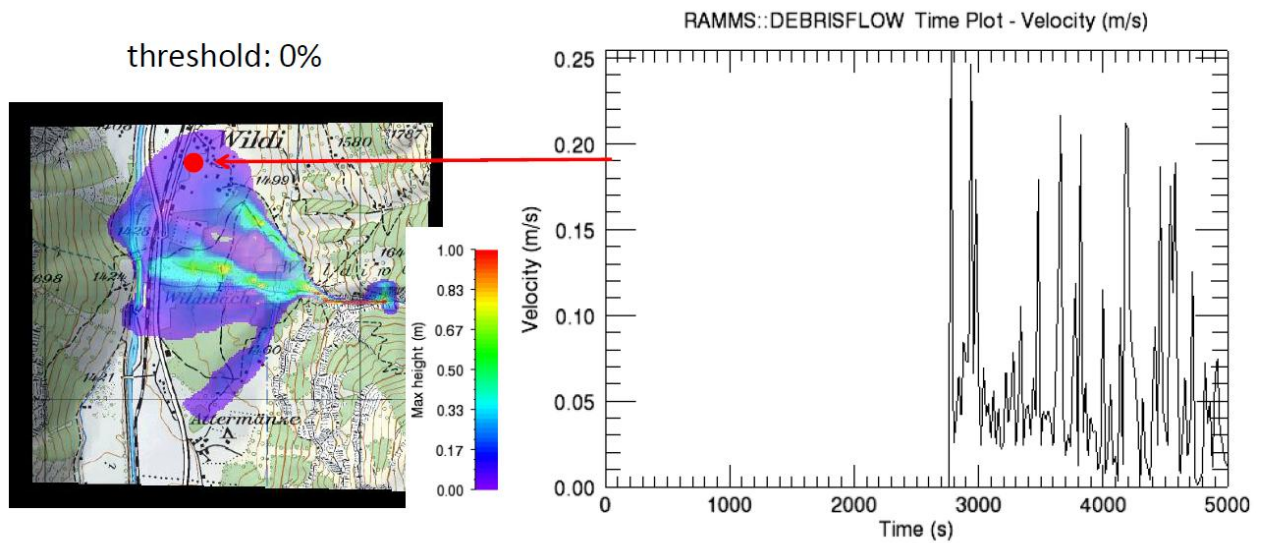


Figure 5-22: Example of residual low-velocity movement when the momentum-based stopping criterion is disabled (threshold = 0%). Very small velocities may persist for a long time even though the main flow mass has effectively stopped.

Because block release simulations (the entire debris flow material starts to flow at the same time) usually show larger values for the total maximum momentum than hydrograph simulations (controlled inflow of debris flow material into calculation domain), the threshold values for block release simulations could be set smaller than the threshold values for hydrograph simulations in some situations (see Figure 5-23 and Figure 5-24).

CHAPTER 5: Results

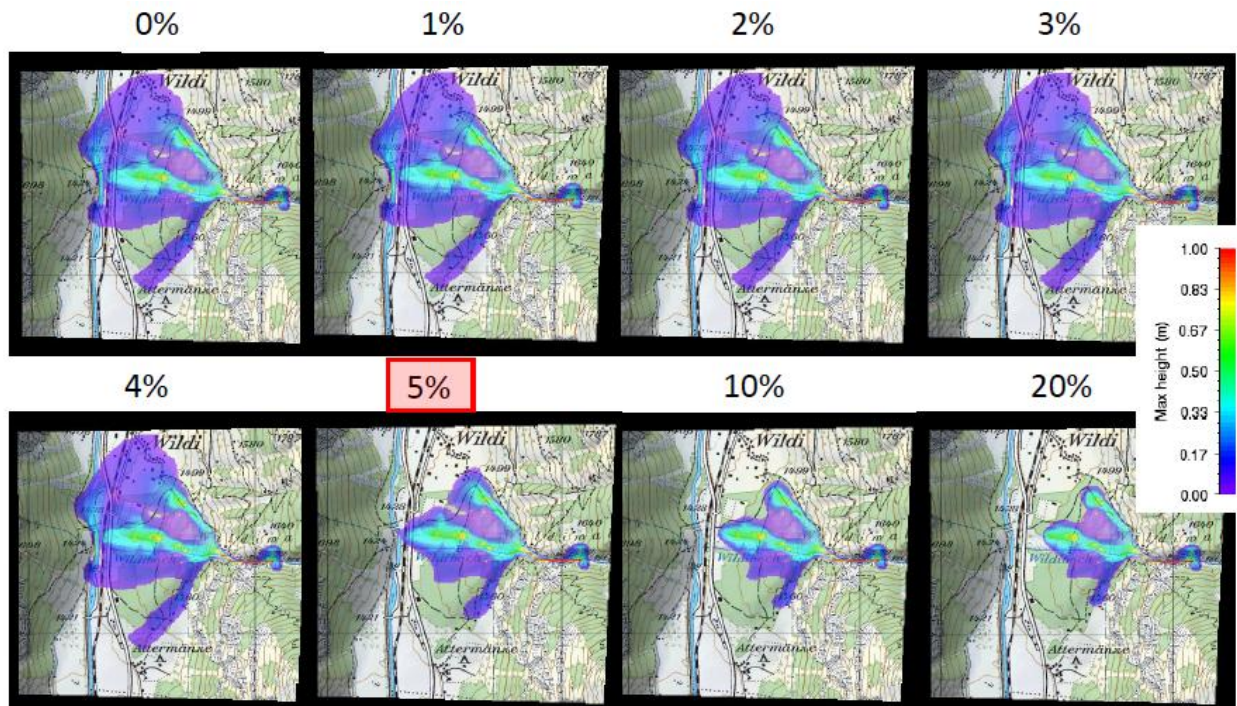


Figure 5-23: Influence of the momentum threshold on a hydrograph simulation. In this example threshold values <5% lead to numerical diffusion of the simulation results. A threshold value of 5% provides a reasonable stopping behavior.

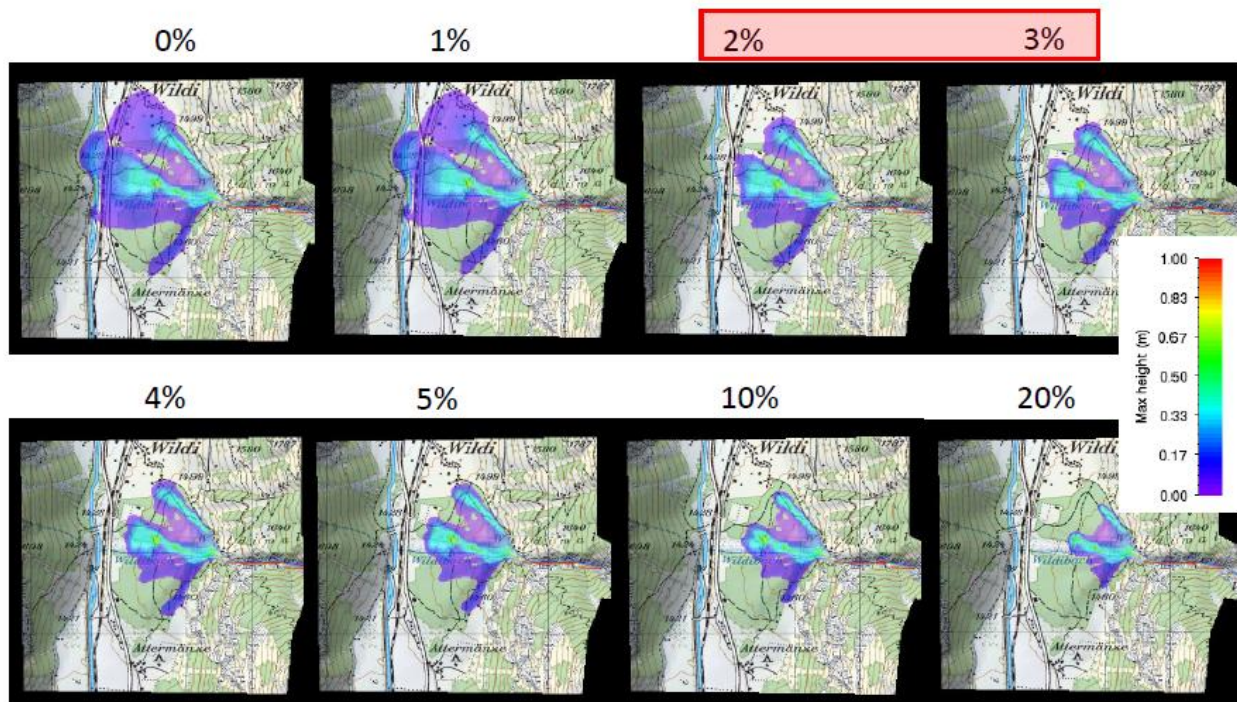


Figure 5-24: Influence of the momentum threshold on a block release simulation. In this example threshold values <2% lead to numerical diffusion of the simulation results. Threshold values of 2-3% provide a reasonable stopping behavior.

The **Output Log File** records which stopping condition terminated the simulation. If

CHAPTER 5: Results

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 criteria

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

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 debrisflow path or catchment**.

It should generally **not be used when several independent debrisflows 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** (see Figure below). The same information can be reviewed afterwards by selecting: **Extras → View Simulation Standard Output Log**.

CHAPTER 5: Results

```
CENTER OF MASS: 626517.35 104383.55 --- Travel speed: 2.33 m/s
FLOW VOLUME: 17500.38 m3
ERODED VOLUME: 7479.86 m3
HYDRO VOLUME: 10020.72 m3
NUMERICAL VOLUME LOSS: 0.19 m3
VOLUME CHECK: 1.0
.....
Time 155.0
Step 1286 dt 0.148695 Cfl 0.449627
Hmax 3.18 m Vmax 8.06 m/s
MOVING MOMENTUM: 36.7 percent ( 2350.4 / 6412.8 )
CENTER OF MASS: 626507.54 104385.00 --- Travel speed: 1.98 m/s
FLOW VOLUME: 17500.37 m3
```

Figure 5-25: Center-of-mass position and travel speed reported in the RAMMS calculation window during a simulation.

5.5 Barrier retention results

To review how much volume was retained: select the **Barrier Retention Height** result parameter and animate the simulation, or check the retained-volume Volume Plot. Retained volume is also shown in a dedicated results panel.

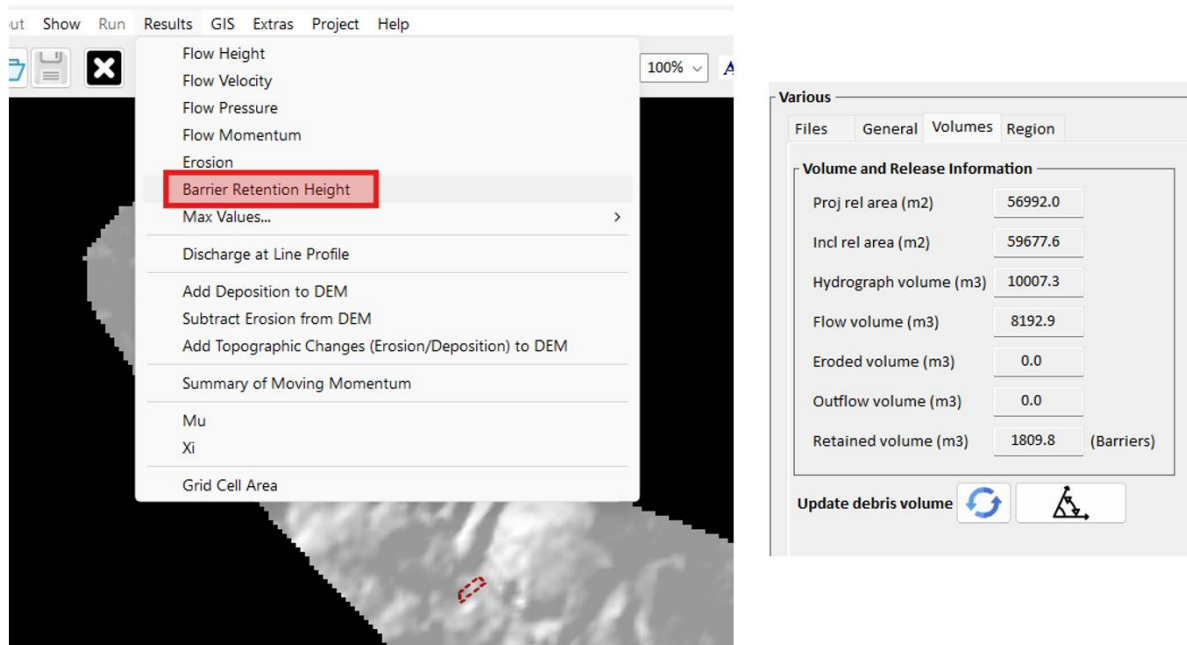


Figure 5-26: Left: select Results → Barrier Retention Height to visualize the retained flow. Right: the Volumes tab reports the total retained volume (here 1809.8 m³, labelled "Barriers").

Barrier information (parameters and final retained volume) is also written to the simulation's output log file.

5.6 Modifying the DEM for subsequent simulations

5.6.1 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 therefore 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), Rudolf-Miklau and Sauermoser (2011), and Rickenmann (1995).

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 modified DEM containing 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.

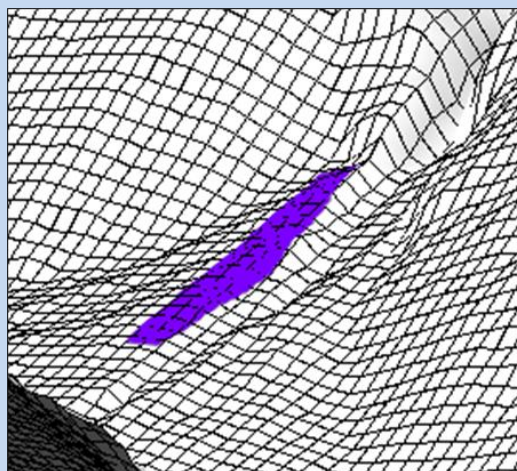


Figure 5-27: Polygon shapefile defining the location and extent of a planned dam.

- 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.

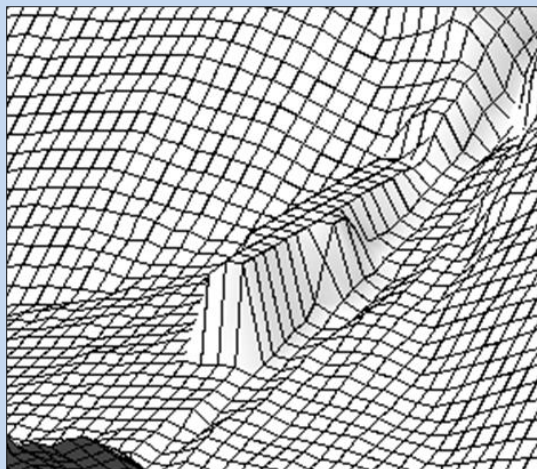


Figure 5-28: Modified DEM after adding the dam geometry defined by the polygon shapefile. The structure is represented by locally increasing the terrain elevation.

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.

CHAPTER 5: Results

General Params Mu/Xi Release Erosion

GENERAL SIMULATION INFORMATION

SCENARIO Name

Manual_Example

☐ Save Max Values Only ("_Max" will be appended to the output name)

Additional Information

Project name: Wildibach

Details: Example simulation for manual

Calculation domain: Wildibach_5m_Max_height_(m)_envelope.c

Digital elevation info: Wildibach.xyz

Stop Criteria

Percentage of total momentum (%): 5.00

Center-of-Mass vel threshold (m/s): 0.50

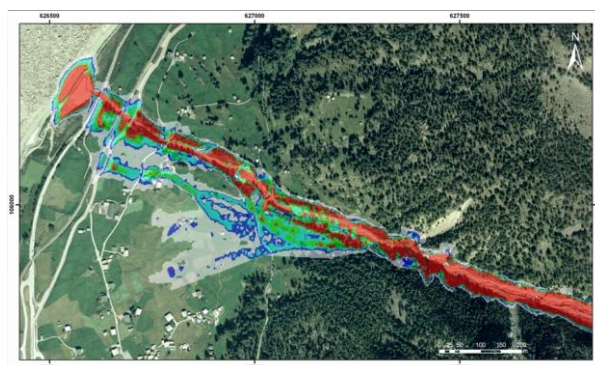
Figure 5-29: Selecting the modified XYZ terrain file containing the dam in the Run Simulation dialog.

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.

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.



CHAPTER 5: Results

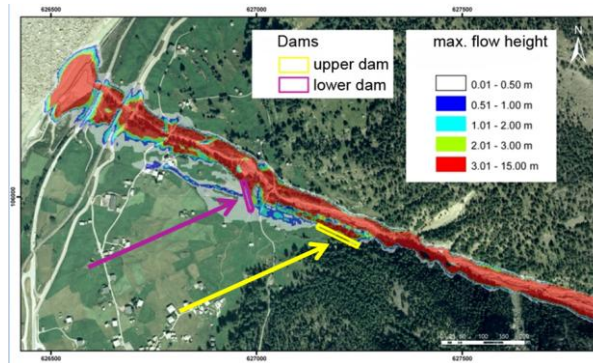


Figure 5-30: Example comparison of simulated maximum flow depth without mitigation structures (top) and with two dams represented in the DEM (bottom). The arrows indicate the locations of the upper and lower dams.

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.

- **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.

CHAPTER 5: Results

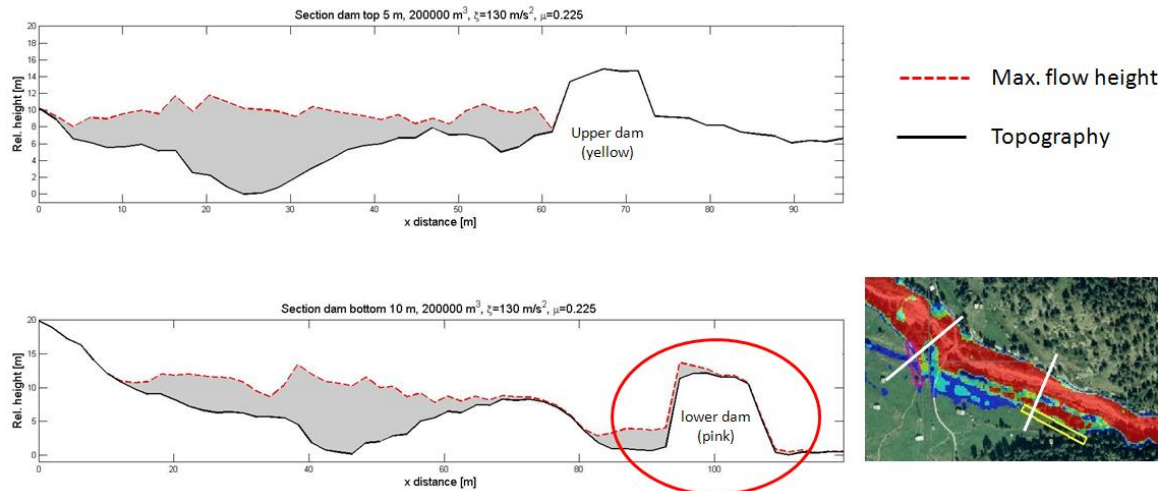


Figure 5-31: Cross-sections through two dams represented in the DEM. The dashed red line shows the simulated maximum flow surface and the black line the modified terrain. The lower dam, which is oriented more directly across the flow path, illustrates the difficulties associated with abrupt topographic barriers.

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.

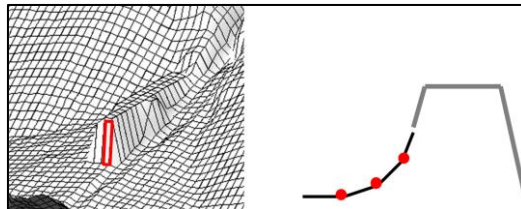


Figure 5-32: Recommended schematic representation of a dam in the DEM using progressively increasing side slopes. Distributing the elevation change over several computational cells avoids an abrupt one-cell topographic step.

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.

CHAPTER 5: Results

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.

5.6.2 Adding deposited debrisflow material to the DEM

Debris-flow events may consist of several individual surges. Deposits produced by an earlier surge can modify the topography and thereby influence the flow path, spreading and deposition of subsequent surges.

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 surges using the debris-flow batch mode**.

Important modeling 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 debris flow deposition to a 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 debris flow 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 Figure 5-29).
The next simulation will then be performed using the terrain modified by the preceding debris-flow deposition.

5.6.3 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 debris-flow 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.

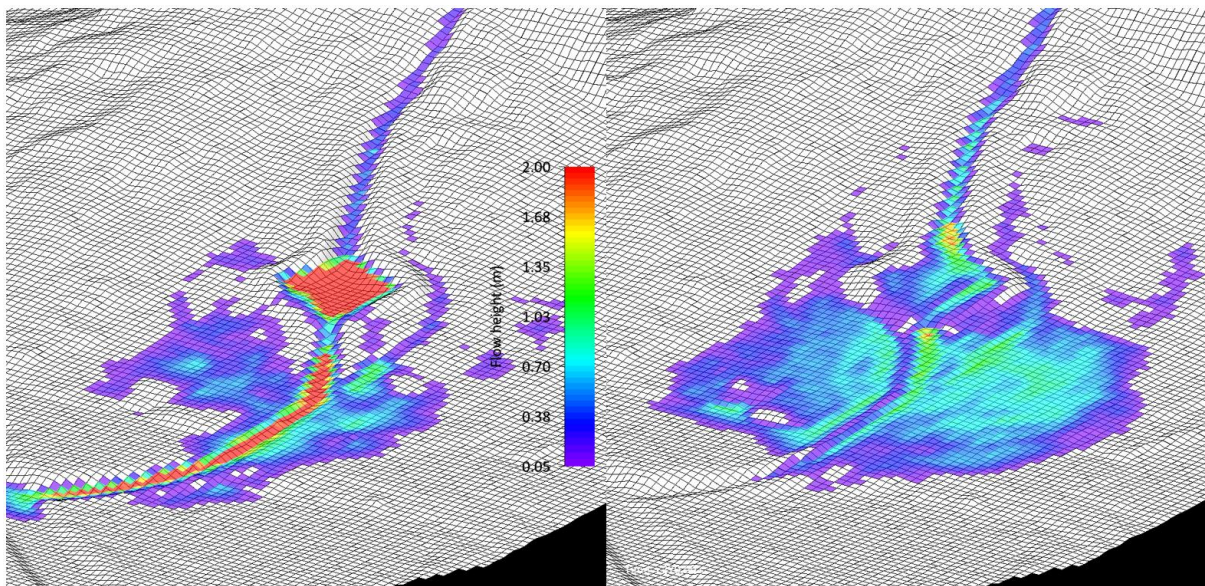


Figure 5-33: First (left) and fourth (right) surge of a sequential multi-surge simulation using identical input parameters. Erosion and deposition from each run are incorporated into the DEM for the subsequent run. Progressive filling of the retention basin leads to increased overtopping and downstream propagation during the fourth surge.

Preparing the base scenario

CHAPTER 5: Results

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

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

- calculation domain,
- release configuration that is not controlled by the CSV file,
- erosion settings,
- 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**.

Defining the surge sequence in a CSV file

The sequence of simulations is defined in a CSV file. Each row represents one debris-flow surge. 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,	Volume,	Qmax,	v,	t1,	Mu,	Xi,	Density,	SimRes,	momentum,	COM,	EndTime,	Comments
Run1,	10000,	200,	10,	10,	0.1,	400,	1800,	4,	10,	0.2,	1000,	Using Base DEM
Run2,	10000,	200,	10,	10,	0.1,	400,	1800,	4,	10,	0.2,	1000,	Using DEM_after_Run1
Run3,	10000,	200,	10,	10,	0.1,	400,	1800,	4,	10,	0.2,	1000,	Using DEM_after_Run2
Run4,	10000,	200,	10,	10,	0.1,	400,	1800,	4,	10,	0.2,	1000,	Using DEM_after_Run3

Figure 5-34: Example CSV file defining four sequential debris-flow simulations. Each data row defines one surge. Column headings are ignored by RAMMS and may be written in any language; parameters are identified by their column position. The simulation resolution (SimRes) specified for the first run is used for the complete batch.

Table 4: CSV parameters used to define the individual runs of a sequential debris-flow batch simulation.

Column	Parameter	Description
1	Run name	Name assigned to the individual simulation.
2	Volume	Total debris-flow volume used to define the input hydrograph.
3	Qmax	Maximum discharge of the input hydrograph.
4	v	Flow velocity specified for the hydrograph.
5	t1	Time at which the maximum discharge $Q_{\max}$ is reached.

CHAPTER 5: Results

6	μ (μ)	Coulomb friction coefficient.
7	ξ (ξ)	Turbulent friction coefficient.
8	Density	Debris-flow density.
9	SimRes	Computational grid resolution. See note below.
10	Momentum	Momentum-based stopping criterion (% of total momentum).
11	COM	Center-of-mass velocity stopping criterion (m/s).
12	EndTime	Maximum simulation time.
13	Comments	Optional description of the individual simulation.

Note: The input hydrograph is defined using **Volume**, **Qmax** and **t1**, where **t1** specifies the time at which the maximum discharge is reached. The end time of the hydrograph is calculated automatically by RAMMS from these values. It does not have to be specified separately in the CSV file.

The **EndTime** column refers to the maximum duration of the **RAMMS simulation**, not to the end of the input hydrograph.

Simulation resolution: The value of **SimRes** in the **first data row** defines the computational grid resolution for the entire sequential batch. The same resolution is used for all subsequent simulations, irrespective of the **SimRes** values specified in later rows. Sequential simulations must use a common computational grid because terrain changes are transferred directly from one simulation to the next.

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**.

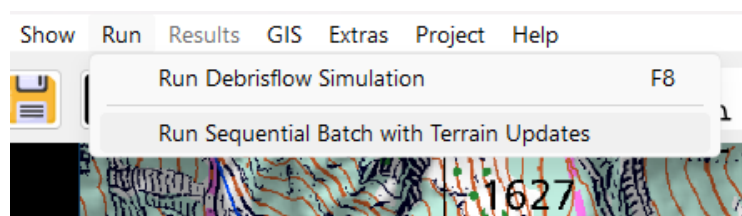


Figure 5-35: Start Run Sequential Batch procedure.

2. Select the CSV file defining the sequence of surges.

CHAPTER 5: Results

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 (.db2),
- the simulation output (.out.gz),
- the modified DEM (.tif),
- the deposition raster (.tif),
- and auxiliary terrain files such as .xyz and .var

Each surge 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

Sequential simulations can be useful for investigating how terrain changes caused by earlier debris-flow surges 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 Applications

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

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

6.1 Calibration

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

Table 6.1: Parameters for calibration for the case study of Dorfbach, Randa (CH).

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

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

Field observations and investigations provided an estimate of a total volume of approximately 10'000 m³. The aim was to calibrate the model by comparing flow velocities and

CHAPTER 6: Applications

flow heights at two locations (Figure 6-1a). The estimated values obtained from the field observations at these locations are:

Table 6.2: Field estimations of height and flow velocity for the case study of Dorfbach, Randa (CH).

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

Find the best-fit Voellmy friction coefficients (procedure)

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

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

Table 6.3: Suggestions for setting the viscous-turbulent friction parameter ξ .

	Granular flow (solid-dominated)	Mud flow (fluid-like)
Viscous-turbulent friction, ξ [m/s²]	100-200 *)	200-1'000 *)
*These values are only suggestions, not fixed definitions.		

Table 6.4: General suggestion for the initial values of the Voellmy friction coefficients used for the calibration procedure.

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

CHAPTER 6: Applications

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

3. Vary μ around the initial definition with steps of ± 1 and ξ with steps of ± 100 m/s². After comparing this initial result with field observations, start with a fine tuning and incrementally change μ and ξ around the initially-found best-fit values. To match the simulation results with the flow heights and velocities from field observations at the given locations we found the following best-fit Voellmy friction coefficients for Dorfbach:

Table 6.5: Best-fit Voellmy friction coefficients for the case study of Dorfbach, Rand (CH).

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

Table 6.6: Resulting maximum flow heights and velocities for total volumes of 10'000 and 5'000 m³ and $\mu=0.225$ and $\xi=130$ m/s²

Volume: 10'000 m ³	Max. height [m]	Max. velocity [m/s]
Location 1	2.7	3.8
Location 2	2.5	2.5
Volume: 5'000 m³		
Location 1	2.2	2.6
Location 2	1.1	2

The best-fit Voellmy friction coefficients are now used to perform several simulations (Figure 6-1). We performed an additional simulation with a small total volume (5'000 m³) to better model the observed flow path. This modification is justified because there are two main locations where deposition was observed during the event of June 7, 2011. One location is out of Figure 6-1 around an elevation of 1'730 m and the second location coincides with location number (3) in the center of the channel. The simulation using a total volume of 10'000 and 5'000 m³ show the following flow heights and velocities at the two locations 1 and 2:

CHAPTER 6: Applications

Both simulations with volumes of 10'000 and 5'000 m³ show locations where the flow leaves the channel (Figure 6-1, locations 1-2), while in reality the debris flow should

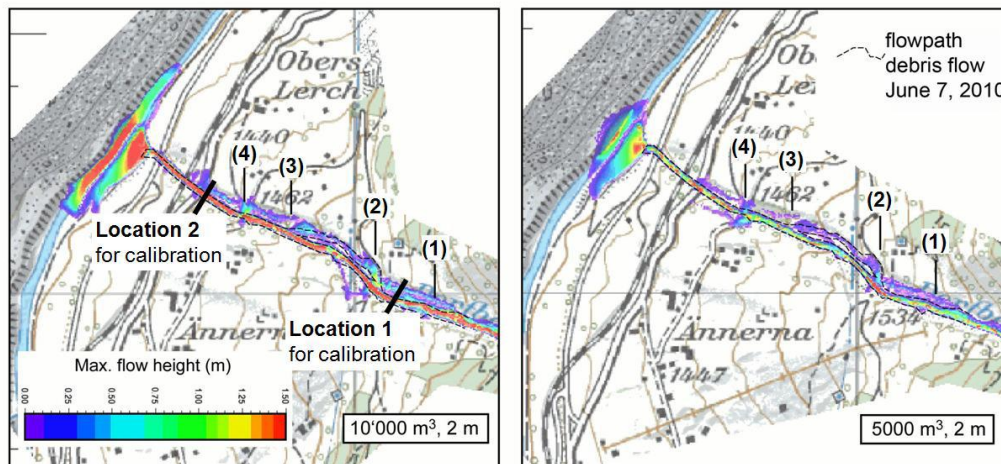


Figure 6-1: Simulations with best-fit parameters $\mu=0.225$ and $\xi=130 \text{ m/s}^2$ for (left) a volume of 10'000 and (right) 5'000 m³ and a DEM resolution of 2m. The dashed line indicates the flow path of the event from June 7, 2010.

simply follow the channel.

Specifically, these are locations indicated by numbers (1) to (4). A breakout, where the flow left the channel, was observed only at location (2) on the orographic right side of the channel.

These differences might be caused for the following reason: For these simulations we use a resolution of the DEM (2010) of 2m. Additional simulations using a DEM-resolution of 1m and a volume of 5'000 m³ could reproduce the flow path at location (1), (3) and (4). At location (2) the outbreak disappeared in the simulation, although it occurred in reality. This result indicates that a resolution of 2m cannot reproduce the exact flow paths because not all vertical curves are visible on a 2m-DEM and therefore material can break out of the channel. The missing outbreak at location (2) for the 2m-DEM can be explained as following: first, it may indicate that at this location the volume was larger than 5'000 m³ and only after location (2) more material was deposited and second, an aspect which will be discussed in the next section (0), the use of a block release rather than an input hydrograph resulted in slightly underestimated flow heights and velocities.

To conclude, the simulations using a volume of 10'000 m³ indicate a slightly too high volume estimation for the lower part of the torrent. However, to calibrate the Voellmy friction coefficients it was still possible to match the flow heights and velocities observed in the field. The small differences in the flow path between the simulation and reality can be explained by no ideally resolved terrain.

6.2 Input hydrograph in comparison with block release

With an input hydrograph it is possible to constrain inflow of the debris material into the calculation domain at a specific location. This is in contrast to a block release simulation where a release area with an initial depth is defined for which the entire block of material is accelerated by gravity given the frictional resistance, e.g. simulating debris flow initiation as an instantaneous landslide failure. Flow depth is a value that can be measured with an observation station or estimated by geomorphic evidence. Assuming that the profile and the velocity are known at the measurement location one can calculate the discharge hydrograph there. Other options are presented in section 0.

6.2.1 Comparison between a block release and an input hydrograph

Simulations with large volumes and a block release usually have larger maximal discharge values (maximum heights) compared to simulations using a hydrograph (Figure 6-2a). The reason is that for a block release the release area is defined and the initial release height has to be adjusted according to the total volume, specified by the user. This block begins moving as soon as the simulation is started. Hence, extremely large and possibly unrealistic initial heights can occur (e.g. we use an initial height of 13.2 m for simulations with a total volume of 200'000 m³ at Dorfbach, see studies by Deubelbeiss and Graf, 2013). In many cases it would be more reasonable to assume a realistic initial release height and adjust the release area to match the corresponding total volume. This effect inverts for small volumes: the inflow velocity of a hydrograph simulation may be larger than what results from a block release simulation at the location of the input hydrograph (Figure 6-2b). The inverse effect is also visible in the flow paths (Figure 6-3). The larger flow heights and resulting velocities of a block release simulation are represented by the larger extent compared to a hydrograph simulation.

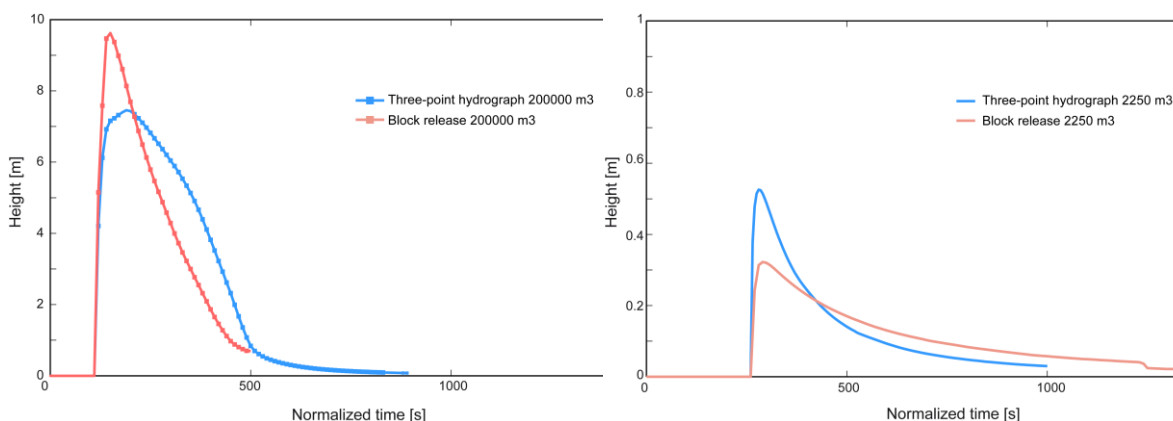


Figure 6-2: Comparison of discharge curves of a simulation with a block release and a hydrograph for (left) a volume of 200'000 m³ and (right) a volume of 3'350 m³. For all simulations $\mu=0.225$ and $\xi=130$ m/s². The discharge curves are measured at the location shown in Figure 6-3.

CHAPTER 6: Applications

6.2.2 Discussion



Figure 6-3: Comparison of the inundated areas of a simulation with a block release and a hydrograph (left) for a volume of 200'000 m³ and (right) a volume of 2'250 m³. For all simulations $\mu=0.225$ and $\xi=130$ m/s². Blue: hydrograph, red: block release. The red points indicate the location of the measured discharge curves (Figure 6-2).

Two general problems appear when comparing the input hydrograph and block release: the friction parameters for a large design event may be different than those for the calibration event, and differences in peak discharge. Large debris flows are often expected to have disproportionally longer runout in comparison with smaller flows (e.g. less friction, e.g. smaller μ values). However, it is difficult to generalize about how much the friction parameters should be changed as a function of event volume. The situation is further complicated by the observation that very large debris flows tend to take place as a series of surges rather than one large surge, and the effective flow topography may change due to deposition (or erosion) during the surge. The problem with the unusually large flow depths and the input hydrograph was described earlier in this section. Another aspect of that problem is that sediment entrainment often takes place along the flow path, a process which is now implemented in RAMMS::Debrisflow. The new erosion module in RAMMS predicts the depth of erosion of sediment caused by debris flows. Using this option, it is possible to predict the increase in volume of a debris flow as it travels along a channel (see sections 4.9.3 and 6.3).

6.2.3 Summary

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

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

CHAPTER 6: Applications

6.3 Erosion example

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

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

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

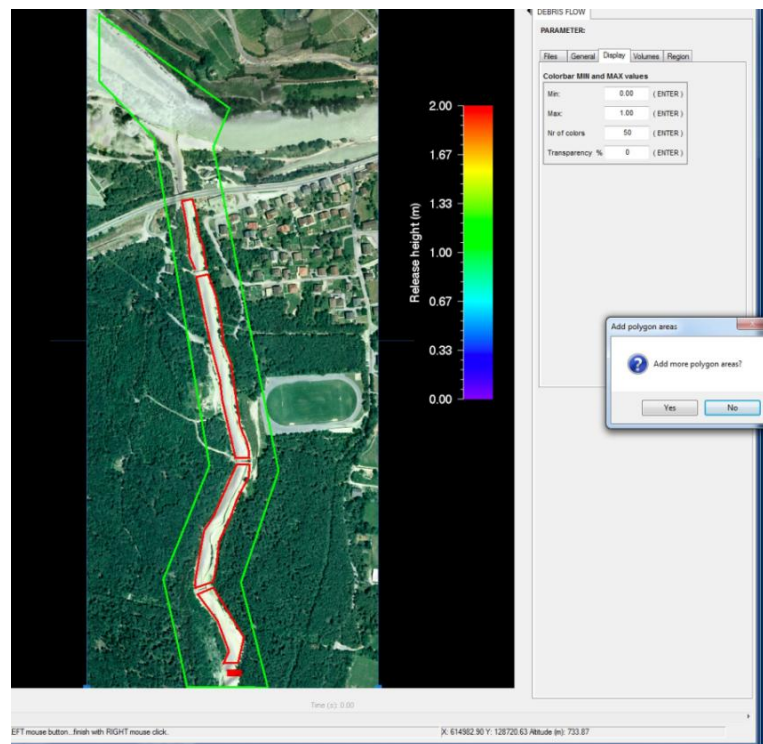


Figure 6-4: Adding erosion polygon areas.

CHAPTER 6: Applications

Step 3: Set erosion parameters. To set the erosion parameters used during the simulation, select **Run** simulation and then click on the **Erosion** tab. Click on the folder icon and select the polygon shapefile (in this case *erodible_bed.shp*, Figure below). After selecting the erosion parameters, click on the button **Add shapefile to list**. The shapefile and values are added to the list. You can delete the whole list, or remove single entries from the list. It's not possible to edit an entry (remove the entry, and create a new entry).

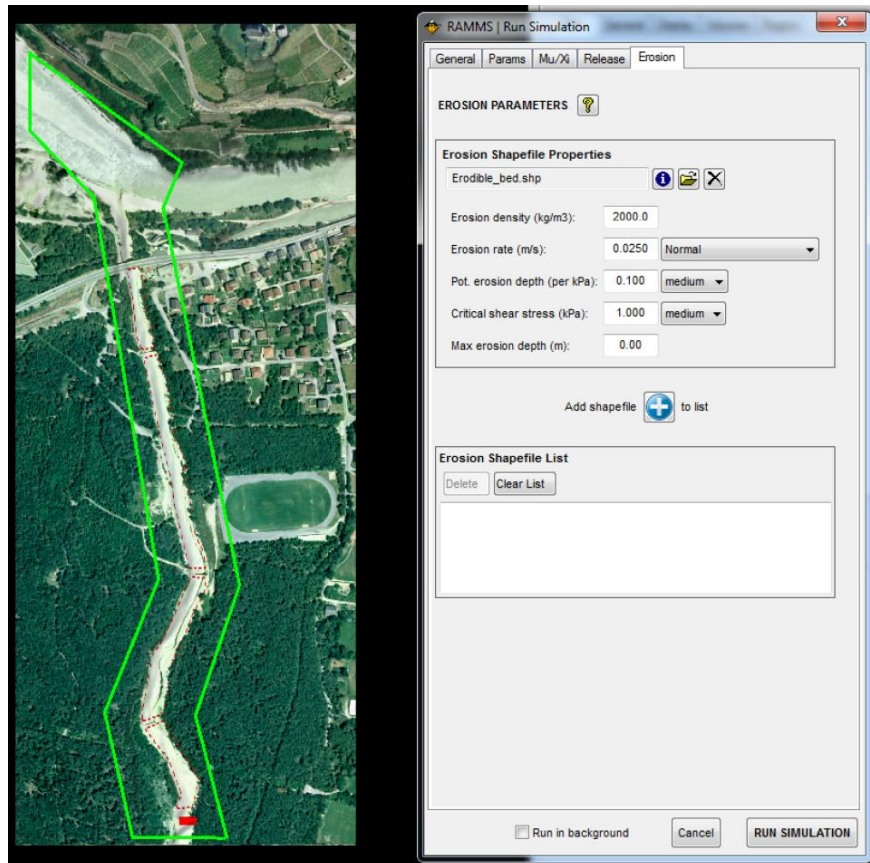


Figure 6-5: Erosion parameters.

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

CHAPTER 6: Applications

The predicted channel-bed erosion for this case is illustrated in Figure 6-6. Here one can clearly see that erosion has taken place only within the user-specified polygons within the erosion shape file, and not on the check dams separating the individual polygons.

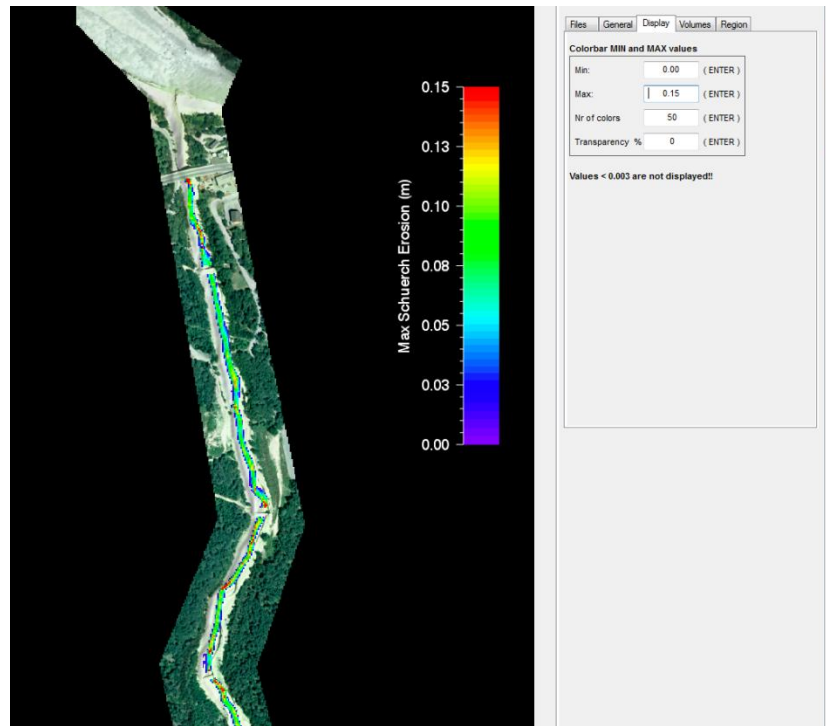


Figure 6-6: Simulated channel-bed erosion.

In this case the model was then re-calibrated with small adjustments to the turbulent friction parameter, and calibrated (for flow speed) to be within 1 second of observed travel time.

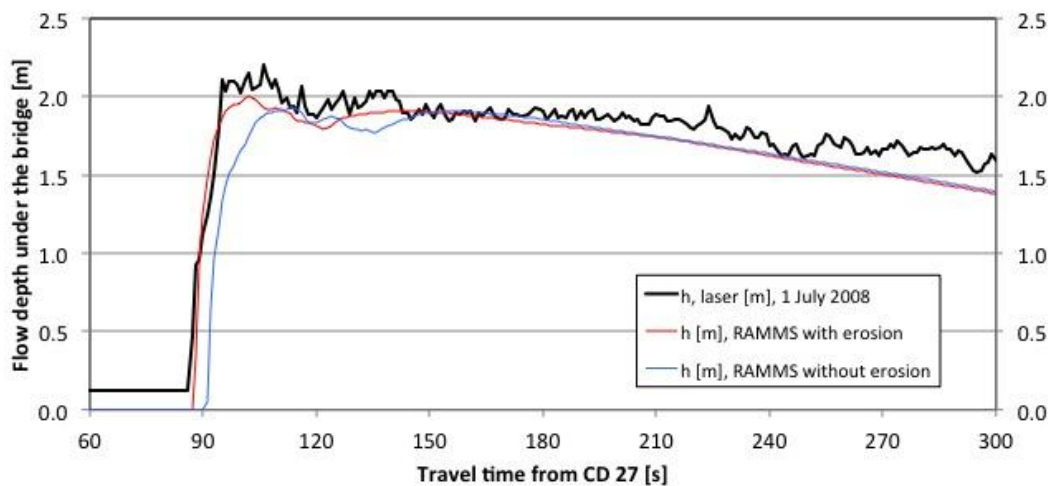


Figure 6-7: Front steepness comparison; laser data, RAMMS simulation with erosion and RAMMS simulation without erosion

CHAPTER 6: Applications

A comparison of the hydrographs (output from a point placed under the bridge, in comparison with the flow height measured in the field) illustrates that the steepness of the front, when including debris-flow entrainment, is somewhat more realistic. However, the case where entrainment is not considered is somewhat slower in travel speed, for the same friction parameters.

6.4 Slush Flows

This chapter provides guidance on the modelling of slush flows (*sørpeskred*) using the **RAMMS::Debris Flow** (RDF) and **RAMMS::Avalanche** software modules. Both modules share the same Voellmy–Salm rheological framework and are equally applicable to slush-flow simulation, provided that module-specific friction parameters are calibrated for the slush-flow regime. The recommendations in this chapter are based on two reports produced for the Norwegian Water Resources and Energy Directorate (NVE): Jaedicke and Sandersen (2021) on slush-flow release zone classification, and Kronholm (2021) on RAMMS::Debris Flow parameter calibration for slush flows.

6.4.1 Definition and Hazard Characteristics

A slush flow is a rapid gravity-driven flow of a snow–water mixture containing a minimum of approximately 15 % water by volume, triggered when the rate of water input to a snowpack — from rainfall, snowmelt, or groundwater emergence — exceeds its drainage capacity (Hestnes, 1985).

Slush flows are distinguished from dry-snow avalanches and debris flows by high bulk density (typically close to 1000 kg/m^3), high flow velocities relative to that density, long runout on very gentle gradients such as valley floors and road embankments, and a transition from slush-flow to debris-flow rheology in the lower track through progressive incorporation of soil and rock (Sletten et al., 2003, cited in Kronholm, 2021) — together giving them high damage potential for infrastructure and buildings.

Slush flows frequently occur during early spring or rain-on-snow events in Norway, Iceland, Svalbard, and other high-latitude or maritime-montane environments. In Norway, the most hazard-prone periods are January–March, when deep snowpacks coincide with rainfall or rapid warming (Jaedicke and Sandersen, 2021).

6.4.2 Release Zone Identification

Based on analysis of 150 Norwegian slush-flow events, Jaedicke and Sandersen (2021) identified five terrain types from which slush flows initiate. Correct identification of the release area is a prerequisite for realistic RAMMS simulation, because the release geometry controls the initial volume and the direction of flow into the track.

CHAPTER 6: Applications

Classification of Release Zone Types

Table 6.7: Classification of slush-flow release zone types after Jaedicke and Sandersen (2021). Frequencies are based on 150 Norwegian events.

Type	Frequency	Key Characteristics	Mapping / Modelling Notes
Stream channels (bekkeløp)	~50 %	Defined channels that concentrate surface and sub-surface water. Often difficult to identify from topographic data alone.	Highest priority for hazard mapping. Well-defined channel geometry aids delineation of computational release area. Consider multiple sub-catchments.
Shallow depressions (for-senkninger)	~22 %	Small concave terrain features that collect water. May not be identifiable at DEM resolutions coarser than 1 m.	High-resolution DEM (LiDAR ≤ 1 m) required for identification. Release area may need to cover the full catchment feeding the depression.
Open slopes (åpne skråninger)	~20 %	Planar or slightly convex slopes without a well-defined channel. Water reaches the snowpack diffusely.	Release geometry is less well constrained. Use sensitivity analysis on release volume. Slope aspect and gradient are the primary spatial predictors.
Bogs and mires (myrområder)	~4 %	Peat-covered areas with poor drainage. Water saturation of near-surface material precedes mobilisation.	Terrain gradient is typically very low; computational release areas must be placed in downstream steeper terrain (see Section 5).
Other / unclassified	~4 %	Road fills, artificially modified slopes, or events where origin could not be determined from documentation.	Site-specific assessment required. Standard parameter recommendations may not apply without additional calibration.

Mapping of release zones should be based on a LiDAR DEM at 0.5–1 m resolution combined with field inspection and historical records. The classification provides a framework for prioritising areas for detailed hazard assessment, but all five release-zone types can produce damaging slush flows.

CHAPTER 6: Applications

6.4.3 RAMMS Modelling Approach

Governing Equations

Both RAMMS::Avalanche and RAMMS::Debris Flow use the same **Voellmy–Salm rheological model**, described in the *Physical friction model* section of Chapter 4.

The governing equations are physically identical in both RAMMS modules. The critical difference is in the **default parameter values**: the standard RAMMS::Debrisflow parameters ($\mu = 0.20$, $\xi = 200 \text{ m s}^{-2}$) are calibrated for typical debris flows and produce velocities and runout distances that are 30–50 % too low for slush flows. Slush-flow calibrated parameters are therefore required regardless of which RAMMS module is used (Kronholm, 2021).

Module equivalence: RAMMS::Avalanche users may apply the slush-flow parameter recommendations in this chapter without modification. The Voellmy–Salm implementation is the same in both modules. Where the two modules differ (e.g., fracture slab options, specific erosion implementations), RAMMS::Debrisflow features are described; the RAMMS::Avalanche equivalent should be used when running simulations in that module.

Physical Basis for Voellmy Parameters in Slush Flows

Kronholm (2021) tested the full space of μ and ξ values against five Norwegian events with documented velocities and runout distances. The key finding is that only combinations with $\mu \leq 0.08$ and $\xi \geq 2000 \text{ m s}^{-2}$ produce acceptable agreement. These values are consistent with the low Coulomb resistance expected for a near-liquid mixture and the low turbulent drag of a high-fluidity flow. For comparison, large, dry-slab avalanches use $\mu \approx 0.15\text{--}0.30$ and $\xi \approx 1000\text{--}4000 \text{ m s}^{-2}$, while typical debris flows use $\mu \approx 0.15\text{--}0.25$ and $\xi \approx 100\text{--}400 \text{ m s}^{-2}$.

Slush flows therefore occupy a distinct parameter space: **low μ (resembling a fluid) and high ξ (resembling a dry avalanche)**. This combination reflects their high water content, low basal friction, and ability to maintain velocity over gentle gradients.

6.4.4 Recommended Voellmy Parameters by Return Period

Table 6.8 gives the recommended RAMMS parameters for slush-flow hazard mapping at three return periods (Kronholm, 2021, Table 14). Separate values are provided for simulations with and without the erosion module active. Because slush flows can scour and entrain significant volumes of material in the track, the erosion scenario generally represents the more conservative (longer runout) estimate.

CHAPTER 6: Applications

Table 6.8: Recommended RAMMS Voellmy parameters for slush-flow hazard mapping by return period (Kronholm, 2021, Table 14). Values apply to both RAMMS::Debris Flow and RAMMS::Avalanche when used with slush-flow release conditions.

Return Period	Without Erosion		With Erosion Module	
	μ (Mu)	ξ [m s^{-2}] (Xi)	μ (Mu)	ξ [m s^{-2}] (Xi)
1 / 100 years	0.08	2000	0.08	3000
1 / 1000 years	0.05	3000	0.05	4000
1 / 5000 years	0.04	4000	0.04	5000

Interpretation of ξ : The turbulent friction coefficient ξ primarily controls peak velocity in the track. Higher ξ (longer return period, erosion scenario) results in higher simulated velocities and longer runout. The recommended values were derived from inverse modelling of five Norwegian events and should be used as a starting point; local calibration against documented events is recommended where data are available.

For preliminary hazard assessments where no historical events have been documented in the specific catchment, use the **1/100 year parameters with erosion ($\mu = 0.08$, $\xi = 3000 \text{ m s}^{-2}$)** as a conservative baseline. Escalate to the 1/1000 or 1/5000 year parameters if the site is in a zone of high potential consequence or if the catchment morphology is similar to a documented high-magnitude event.

6.4.5 Release Area Setup

Computational Release Areas

RAMMS requires the initial flow to have sufficient gravitational driving force to enter the track. The solver will not produce meaningful flow if the entire release polygon lies in terrain shallower than approximately 10° . For slush flows originating in bogs, mires, or shallow depressions (slope $< 10^\circ$), a **computational release area** must be defined in steeper terrain downstream of the actual initiation zone but within the same channel or flow path. The computational release area should be sized to deliver a representative initial volume into the steep track reach.

CHAPTER 6: Applications

The following approach is recommended:

- Identify the actual topographic release zone using the classification in Section 2.
- If the release zone gradient is $< 10^\circ$, shift the release polygon downstream to a slope reach of at least 15° within the same channel.
- Size the computational release area so that the initial **release volume V_0** is consistent with the expected event magnitude for the return period (see Section 6 for erosion-based volume growth).
- Do not enlarge the computational release area to simulate large events if this would cause unrealistic lateral spreading at the point of injection.

Note on release volumes: For slush flows where the primary mass entrainment mechanism is progressive erosion of the snowpack and underlying loose material along the track, the initial release volume is less critical than for events where most of the final volume derives from the release zone. Err on the side of a moderate release volume combined with realistic erosion settings rather than placing an artificially large volume in the release polygon.

Bulk Density

A bulk density of $\rho = 1000 \text{ kg m}^{-3}$ is recommended throughout, consistent with a near-fully saturated snow–water mixture. This value is used for pressure calculations (dynamic pressure $p = \rho u^2$) but does not directly affect simulated flow velocities or runout distances in the Voellmy model. If the flow is expected to transition to a debris flow in the lower track, a higher density (e.g., $1200\text{--}1500 \text{ kg m}^{-3}$) may be more appropriate for pressure calculations in that reach; however, RAMMS uses a single density value per simulation.

6.4.6 Erosion Module Settings

The erosion module simulates the progressive entrainment of erodible material along the flow path. For slush flows, enabling erosion is recommended because progressive incorporation of snowpack and loose surface material is a dominant mechanism of volume growth and is better represented through the erosion module than through artificially large release volumes. The following settings are based on Kronholm (2021) and reflect conditions of loose, wet, near-surface material:

CHAPTER 6: Applications

Table 6.9: Recommended RAMMS erosion module settings for slush-flow simulations (Kronholm, 2021).

Parameter	Recommended Value	Rationale
Erodible material density	1000 kg m ⁻³	Consistent with water-saturated snow. Reflects the slush-flow regime rather than soil or rock erosion.
Erosion rate	0.050 m s ⁻¹	Appropriate for loose, wet sediment/snow. This is the upper end of the RAMMS erosion rate range and is justified by the high fluidity of the erodible material.
Potential erosion depth	0.20 m kPa ⁻¹	Set to 'deep' in the RAMMS interface. Allows substantial erosion where basal shear stress is high.
Critical shear stress	0.5 kPa	Low value consistent with unconsolidated, saturated material. Erosion begins at low flow velocities.
Maximum erosion depth	1.0 m	Limits total erosion to 1 m of material to avoid physically unrealistic entrainment. May need site-specific adjustment if deep snowpack or thick unconsolidated layer is present.

These settings are appropriate for the 'base case' loose snowpack and near-surface material scenario. In areas with deep, compact snowpacks or where the flow path crosses consolidated moraine or bedrock, the erosion rate and maximum erosion depth should be reduced. In areas with very deep unconsolidated material (e.g., thick peat or alluvial fans), the maximum erosion depth may need to be increased with corresponding justification.

6.4.7 Digital Elevation Model and Grid Resolution

DEM acquisition: A LiDAR-derived DEM at 0.5–1 m point spacing is recommended for slush-flow simulation. At this resolution, narrow stream channels, shallow depressions, and small bogs (which account for ~76 % of release zones) can be correctly identified and their geometry captured. Photogrammetric DEMs from drone surveys are an acceptable alternative provided point density and accuracy are comparable to LiDAR.

CHAPTER 6: Applications

Simulation grid: Kronholm (2021) tested both 2 m and 5 m computational grids and found minimal difference in simulated runout and peak velocities for the five case studies. A **2 m simulation grid** is recommended as the standard because it resolves narrow channels and confining topography more faithfully. For large catchments ($> 5 \text{ km}^2$) or screening-level assessments, a 5 m grid is acceptable.

DEM pre-processing: Before importing into RAMMS, the DEM should be hydrologically conditioned (sinks filled, flow directions verified) in the area of the expected flow path. Ensure that the DEM correctly represents stream channels and any road embankments, culverts, or other infrastructure that may redirect or confine flow. Bridges and culverts often require manual DEM editing to represent barrier or pass-through conditions.

6.4.8 Recommended Simulation Workflow

The following workflow is recommended for slush-flow hazard mapping with RAMMS:

- **Step 1 — Terrain analysis.** Obtain LiDAR DEM at $\leq 1 \text{ m}$ resolution. Identify potential release zones using the classification in Section 2. Confirm with field inspection and historical records.
- **Step 2 — DEM preparation.** Prepare a hydrologically conditioned DEM at 2 m grid resolution covering the release zone, track, and runout area with at least 200 m margin around the expected flow extent.
- **Step 3 — Release area definition.** Define release polygons based on mapped release zones. For zones with slopes $< 10^\circ$, use a computational release area in steeper downstream terrain as described in Section 5.
- **Step 4 — Parameter selection.** Select Voellmy parameters from Table 6.8 for the relevant return period and erosion scenario. Start with the 1/100 year case without erosion to verify model setup, then progress to erosion-enabled and higher return period scenarios.
- **Step 5 — Erosion module configuration.** If using the erosion module, apply the settings in Table 6.9. Define the erodible layer extent to cover the full expected flow path. Use a conservative (high) estimate of snowpack depth as the initial erodible layer thickness.
- **Step 6 — Simulation and result extraction.** Run the simulation and extract maximum flow height, velocity, and dynamic pressure grids. Also record total simulated volume at the runout terminus to check for physically unrealistic mass growth.
- **Step 7 — Sensitivity analysis.** Run at a minimum: (a) no erosion vs. erosion scenarios at the same return period; (b) $\mu \pm 0.01$ and $\xi \pm 500 \text{ m s}^{-2}$ to assess parameter sensitivity. Document sensitivity results in the hazard assessment report.

CHAPTER 6: Applications

- **Step 8 — Validation (if data are available).** If a documented slush-flow event exists in or near the study area, back-calculate parameters against observed velocity, flow depth, and runout. Deviations > 20 % from Table 6.8 values should be flagged and justified.

6.4.9 Case Study Summary

Kronholm (2021) calibrated RAMMS::Debris Flow against five documented Norwegian slush-flow events. Table 6.10 summarises the key characteristics of each case and the final calibrated parameters.

Table 6.10: Summary of RAMMS calibration case studies for Norwegian slush flows (Kronholm, 2021). Runout distances are approximate and refer to the distance from the head of the steep track reach to the final deposition zone.

Location	Year	Month	Runout (m)	μ	ξ	Notes
Oppdal / Land-laupet	2018	Mar	~600	0.08	2000	Reference event for 1/100 yr parameters.
Vik / Juvik	2019	Feb	~900	0.06	2500	Longer runout; informed 1/1000 yr guidance.
Balestrand / Ese	2011	Feb	~1200	0.05	3000	Large event; slope transition from steep to gentle.
Tinn / Stavsro	2019	Mar	~400	0.08	2000	Shorter runout event; confirms 1/100 yr values.
Hattfjelldal / Skardmodalen	2010	Apr	~1100	0.05	3000	Transition to debris flow in lower track noted.

6.4.10 Limitations and Uncertainties

The following limitations and uncertainties apply to all RAMMS-based slush-flow simulations and must be documented in the hazard assessment report:

CHAPTER 6: Applications

- **Limited calibration dataset.** The parameter recommendations in Table 6.8 are based on five Norwegian events. The transferability to other climatic or geological settings (Iceland, Svalbard, Alps, Pacific Northwest) is unconfirmed. Additional calibration studies are needed, particularly for events in maritime versus continental snowpack regimes.
- **Single-phase Voellmy model and slush-to-debris-flow transition.** The Voellmy model does not represent the two-phase (water + solid) nature of slush flows — pore-water pressure effects and phase separation are not modelled — and it cannot represent the transition many observed Norwegian slush flows undergo from slush-flow to debris-flow rheology in the lower track. Because RAMMS uses a single constant density ($\rho = 1000 \text{ kg m}^{-3}$) throughout, pressure calculations may underestimate impact forces in the lower track where significant soil and rock incorporation has occurred; consider a higher density for pressure calculations in that reach, or running a separate debris-flow scenario.
- **Release volume estimation.** No validated method currently exists for estimating slush-flow release volumes from topographic and snowpack data alone. The release volume is a significant source of uncertainty when the erosion module is not active.

7 References

Ayotte D. and Hungr O. 2000: Calibration of a runout prediction model for debris flow and Naeser, Rotterdam, 505-514.

Badoux A., Graf C., Rhyner J., Kuntner R. and McArdell B.W. 2009: A debris-flow alarm system for the Alpine Illgraben catchment: design and performance. In: Natural Hazards, 49(3), 517-539.

Bartelt, P., Salm, B. and Gruber, U. (1999). Calculating dense-snow avalanche runout using a Voellmy-fluid model with active/passive longitudinal straining. Journal of Glaciology, 45(150), 242-254.

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

Bartelt, P., McArdell, B., Bühler, Y. and Graf, C. 2024: Zweiphasenmodellierung von Murgängen für die Überprüfung von Schutzmassnahmen. WSL Berichte 155, 49-58.

Barnhart, K.R., Jones, R.P., George, D.L., McArdell, B.W., Rengers, F.K., Staley, D.M. and Kean, J.W. 2021: Multi-Model Comparison of Computed Debris Flow Runout for the 9 January 2018 Montecito, California Post-Wildfire Event. Journal of Geophysical Research: Earth Surface 126, e2021JF006245.

Berger C. 2010: Debris flow entrainment and sediment transfer processes at the Illgraben catchment, Switzerland. Ph.D. thesis, University of Bern, Inst. of Geological Sciences, Bern, Switzerland.

Berger C., McArdell B. W. and Schlunegger F. 2011: Direct measurement of channel erosion by debris flows, Illgraben, Switzerland, J. Geophys. Res., 116, F01002, doi: 10.1029/2010JF001722.

Bovis M.J. and Jakob M. 1999: The role of debris supply conditions in predicting debris flow activity. In: Earth Surface Processes and Landforms, 24, 1039-1054.

D'Agostino V. and Cesca M. 2008: Ricostruzione di alcuni dei principali eventi torrentizi avvenuti in Provincia di Trento. Technical report, Trento Autonomous Province.

Deubelbeiss Y. and Graf C. 2013: Two different starting conditions in numerical debris flow models - Case study at Dorfbach, Randa (Valais, Switzerland). In: Graf, C. (Red.) 2013: Mattertal - ein Tal in Bewegung. Publikation zur Jahrestagung der Schweizerischen Geomorphologischen Gesellschaft 29. Juni - 1. Juli 2011, St. Niklaus. Birmensdorf, Eidg. Forschungsanstalt WSL, 125-138.

Fan, J. and Galoie, M. 2025: Assessment of physical parameters impacts on debris flow modeling with RAMMS. Scientific Reports 15, 36393.

CHAPTER 7: References

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.

Frank, T., McArdell, B.W., Huggel, C. and Vieli, A. 2015: The importance of entrainment and bulking on debris flow runout modeling: examples from the Swiss Alps. Natural Hazards and Earth System Sciences 15, 2569–2583.

Frank, T., McArdell, B.W., Oggier, N., Baer, P., Christen, M. and Vieli, A. 2017: Debris-flow modeling at Meretschibach and Bondasca catchments, Switzerland: sensitivity testing of field-data-based entrainment model. Natural Hazards and Earth System Sciences 17, 801–815.

Graf, C., Christen, M., McArdell, B.W. and Bartelt, P. 2019: An overview of a decade of applied debris-flow runout modeling in Switzerland: challenges and recommendations. Proceedings of the 7th International Conference on Debris-Flow Hazards Mitigation, 685–692.

Hestnes, E. (1985). A contribution to the prediction of slush avalanches. Annals of Glaciology, 6, 1–4.

Hohermuth, B. and Graf, C. 2014: Einsatz numerischer Murgangsimulationen am Beispiel des integralen Schutzkonzepts Plattenbach Vitznau. Wasser Energie Luft 106(3), 195–201.

Hussin, H.Y., Quan Luna, B., van Westen, C.J., Christen, M., Malet, J.-P. and van Asch, T.W.J. 2012: Parameterization of a numerical 2-D debris flow model with entrainment: a case study of the Faucon catchment, Southern French Alps. Natural Hazards and Earth System Sciences 12, 3075–3090.

Hürlimann M., Rickenmann D. and Graf C. 2003: Field and monitoring data of debrisflow events in the Swiss Alps. In: Canadian Geotechnical Journal, 2003, 40(1), 161–175.

Jaedicke, C. and Sandersen, F. (2021). Klassifisering av løsneområder for sørpeskred / Classification of slush-flow release areas. NVE Eksternrapport nr. 8/2021. Norwegian Water Resources and Energy Directorate (NVE), Oslo, Norway. 36 pp.

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

Krishnapriya, V.K. et al. 2024: A rapid run-out assessment methodology for the 2024 Wayanad debris flow. npj Natural Hazards 1, 41.

CHAPTER 7: References

- Kronholm, K. / Skred AS (2021).** Bruk av RAMMS::Debrisflow for modellering av sørpeskred / Use of RAMMS::Debris Flow for modelling of slush flows. NVE Eksternrapport nr. 9/2021. Norwegian Water Resources and Energy Directorate (NVE), Oslo, Norway. 82 pp.
- Li, X., Zhao, J. and Soga, K. 2020:** A new physically based impact model for debris flow. *Géotechnique*.
- McArdell, B.W., Hirschberg, J., Bartelt, P. and Molnar, P. 2025:** TASK 4.3 - Framework for process cascade modelling of rapid mass movements. WSL Berichte 181, 129-134.
- Meyrat, G., McArdell, B., Ivanova, K., Müller, C. and Bartelt, P. 2021:** A dilatant, two-layer debris flow model validated by flow density measurements at the Swiss Illgraben test site. *Landslides* 19, 265-276.
- Meyrat, G., McArdell, B., Müller, C.R., Munch, J. and Bartelt, P. 2023:** Voellmy-type mixture rheologies for dilatant, two-layer debris flow models. *Landslides* 20, 2405-2420.
- Mizuyama T., Kobashi S. and Ou G. 1992:** Prediction of debris-flow peak discharge. Proceedings of the international Symposium Interpraevent, Bern, Switzerland, 4, 99-108.
- Rickenmann D. 1995:** Beurteilung von Murgängen. In: Schweiz. Ingenieur und Architekt 113(48), 1104-1108.
- Rickenmann D. 1999:** Empirical relationships for debris flows. In: *Natural Hazards*, 19, 47-77.
- Rudolf-Miklau F. and Sauermoser S. 2011:** Handbuch Technischer Lawinenschutz. Ernst & Sohn GmbH&Co.
- Salm B., Burkard A. and Gubler H. 1990:** Berechnung von Fließlawinen: eine Anleitung für Praktiker mit Beispielen. Mitteilung 47, Eidg. Institut für Schnee- und Lawinenforschung SLF.
- Salm B. 1993:** Flow, flow transition and runout distances of flowing avalanches. In: *Annals of Glaciology* 18, 221-226.
- Schraml, K., Thomschitz, B., McArdell, B.W., Graf, C. and Kaitna, R. 2015:** Modeling debris-flow runout patterns on two alpine fans with different dynamic simulation models. *Natural Hazards and Earth System Sciences* 15, 1483-1492.
- Schürch, P., Densmore, A.L., Rosser, N.J. and McArdell, B.W. 2011:** Dynamic controls on erosion and deposition on debris-flow fans. *Geology* 39(9), 827-830.
- Schürch, P., Jud, C. and McArdell, B.W. 2018:** Ist die Erosionsabschätzung mittels RAMMS::DEBRISFLOW schon praxistauglich? Agenda FAN 1/2018, 16-19.

CHAPTER 7: References

Sletten, K., Blikra, L. H., Ballantyne, C. K., Nesje, A. and Dahl, S. O. (2003). Holocene debris flows recognized in a lacustrine sedimentary succession: sedimentology, chronostratigraphy and cause of triggering. *The Holocene*, 13(1), 1–15.

Varnes D.J. 1978: Slope movement types and processes. In: *Land-slides Analysis and Control*, edited by Schuster, R.L. and Krizec, R.J., Spec. Per. Natl. Res. Counc. Transp. Res. Board, 176, Natl. Acad. of Sci., Washington D.C., 11-33.

Walser, M., Huggel, C., McArdell, B.W. and Graf, C. 2014: Gefahrenkartierung von Murgängen mit numerischen Simulationsmodellen. *Wasser Energie Luft* 106(3), 187-194.

Zhuang, Y., McArdell, B.W. and Bartelt, P. 2025: Comparative analysis of $\mu(I)$ and Voellmy-type grain flow rheologies in geophysical mass flows. *Natural Hazards and Earth System Sciences* 25, 1901-1912.

Zimmermann, F., McArdell, B.W., Rickli, C. and Scheidl, C. 2020: 2D Runout Modeling of Hillslope Debris Flows, Based on Well-Documented Events in Switzerland. *Geosciences* 10(2), 70.

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::Debrisflow and its applications can be found on the module page at <https://www.ramms.ch/ramms-debrisflow/>. Check out the *Publication*-list at the bottom of the page.

8 Appendix

8.1 Third-Party Software

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

7-zip:

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

Mtee:

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

CHAPTER 8: Appendix

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.
*****;
```