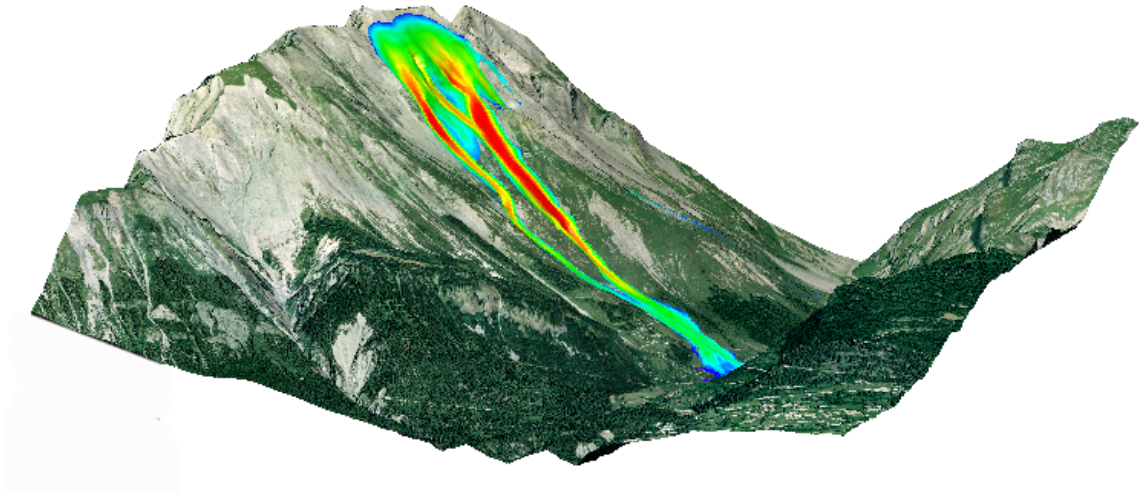


RAMMS 1.3.0

Rapid Mass Movements



A modeling system for snow-avalanches in research and practice

USER MANUAL v1.01



WSL Institute for Snow and
Avalanche Research SLF

March 2010

Written by

Marc Christen, Yves Buehler, Perry Bartelt and Lina Schumacher

Contents

1	Introduction	1
2	Learning by doing	3
3	Setup and First Start	5
3.1	System Requirements	5
3.2	Installation	5
3.2.1	Installation procedure	6
3.3	Licensing Methods	10
3.4	First Start	11
3.4.1	Personal License Request File	11
3.4.2	Personal License Key	12
3.4.3	CYGWIN / GRASS installation	13
3.4.4	Potential problem	17
3.5	Preferences	20
4	Working with RAMMS	23
4.1	General	23
4.1.1	Project / Scenarios	23
4.1.2	Input data	24
4.2	Model input data	24
4.2.1	Topographic data - Digital Elevation Model	25
4.2.2	Forest information	25
4.2.3	Release information	26
4.2.4	Global parameters	26
4.2.5	Friction information	27
4.2.6	Calculation parameters	27
4.3	The Project Wizard	27
4.4	Moving, resizing and changing the view of the model	35
4.5	Global parameters	39
4.6	Release areas	40

4.7	Calculation Domain	45
4.8	Friction parameters	46
4.9	Colorbar	49
4.10	Forest	51
4.11	Project information	53
4.12	Changing maps and remote sensing imagery	55
5	Calculation and Results	57
5.1	Running a calculation	57
5.2	Results	63
5.3	Extras	69
5.3.1	Creating an image and a GIF animation	69
5.3.2	Creating a dam	70
5.3.3	Creating a new DEM with avalanche deposition	72
5.4	How to save input files and program settings	73
5.5	How to open input and output files	74
5.6	About RAMMS	75
6	Program Overview	77
6.1	The Graphical User Interface (GUI)	78
6.1.1	The menu bar	78
6.1.2	Horizontal toolbar	89
6.1.3	Vertical toolbar	91
6.1.4	Main window	91
6.1.5	Dump step slider	91
6.1.6	Left status bar	92
6.1.7	Right status bar	92
6.1.8	Colorbar	92
6.1.9	Panel	92
6.2	File management	95
6.2.1	Software RAMMS	95
6.2.2	Organizing your data	95
	List of Figures	i
	List of Exercises	v

1 Introduction

In the field of natural hazards there is an increasing need for process models to help understand the motion of geophysical mass movements. These models allow engineers to predict the speed and mass of hazardous movements in complex terrain. Such models are especially helpful when proposing mitigation measures, such as avalanche dams or snow sheds. Hazard mapping is an especially important application in Switzerland and other mountainous countries. While well-tested empirical methods are available to determine runout distances, velocities and flow heights, numerical models allow practitioners to predict flow paths in general terrain as well as to model entrainment processes or the breaking effect of forests. Numerical models have the advantage that initial and boundary conditions can be prescribed that more exactly represent the physical problem that needs to be solved.

RAMMS (Rapid Mass MovementS) is a state-of-the-art numerical simulation model to calculate the motion of geophysical mass movements (snow avalanches, rockslides, debris flows, 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 users 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 flows and shallow landslides.

RAMMS was developed by the "Avalanche, Debris Flow and Rockfall" research unit of the WSL Institute for Snow and Avalanche Research, SLF. This manual describes the features of the *RAMMS* program - allowing beginners to get started quickly as well as serving as a reference to expert users.

On the *RAMMS* web page <http://ramms.slf.ch> you find useful features such as a moderated discussion forum, frequently asked questions (FAQ) or recent software updates. Please visit this web page frequently to stay up to date!

2 Learning by doing

This manual provides an overview of *RAMMS*. Exercises exemplify different steps in setting up and running a *RAMMS* simulation especially in **Chapter 4: Working with RAMMS**. However, to get the most from the manual, we suggest reading it through while simultaneously having the *RAMMS* program open, learning by doing. We assume *RAMMS* users to have a basic level of familiarity with windows-based programs, commands and general computer terminology. We do not describe the basics of windows management (such as resizing or minimizing). *RAMMS* windows, click options and input masks are similar to other windows-based programs and can be used, closed, reduced or resized in the same way.

All topographic base maps and aerial images are reproduced ©2010 swisstopo(BA091601).

3 Setup and First Start

3.1 System Requirements

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

- OS: Windows XP (2000) 32 bit
- RAM (memory): 1GB (2 GB recommended)
- CPU: Intel Pentium 1 GHz (dual core recommended)
- Harddisk: ca. 410 MB

Windows Vista and Windows 7 are not supported at the moment.

3.2 Installation

Please download the *RAMMS* setup file *ramms_user_setup.zip* from http://ramms.slf.ch/ramms/downloads/ramms_user_setup.zip.

Please do the following steps before you begin to install:

- Unzip the file *ramms_user_setup.zip* to a temporary directory on your target machine. The unzipped file will be entitled similar to *ramms1.3.0-user_setup.exe*, showing the actual version of *RAMMS* (e.g. 1.3.0).
- You must have **Administrator privileges** on the target machine. If you do not have such privileges, the installer cannot modify the system configuration of the machine and the installation will fail. Note that you do not need Administrator privileges to run *RAMMS* afterwards.
- Read first, install afterwards! Please read the whole installation process once, before you begin the installation!!

3.2.1 Installation procedure

Step 1: Welcome

Start the file *ramms1.3.0_user_setup.exe*. The Welcome dialog introduces you to the English Setup program and will guide you through the installation process. Click "Next" to continue.



Figure 3.1: Installation - Welcome dialog window.

Step 2: Readme

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

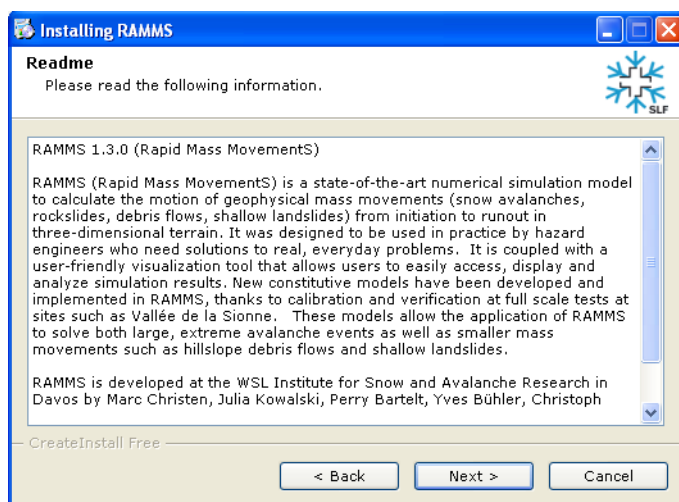


Figure 3.2: Installation - Readme dialog window.

Step 3: Accepting the License Agreement

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

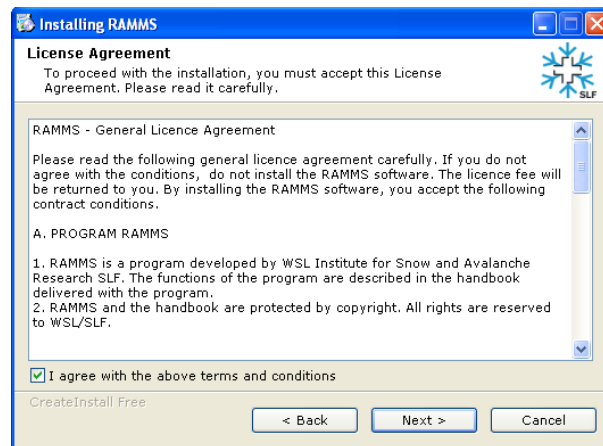


Figure 3.3: Installation - License agreement dialog window.

Step 4: Select Destination Directory

Choose your Destination Directory. Simultaneously this dialog shows the amount of space available on your hard disk and required for the installation. Beware: – Do *NOT* use a blank or special characters within your installation directory path name (e.g.

C:\Program Files\RAMMS is not allowed, use *C:\Programs\RAMMS* or *C:\Programme\RAMMS* instead). Click *Next* to start the installation process.

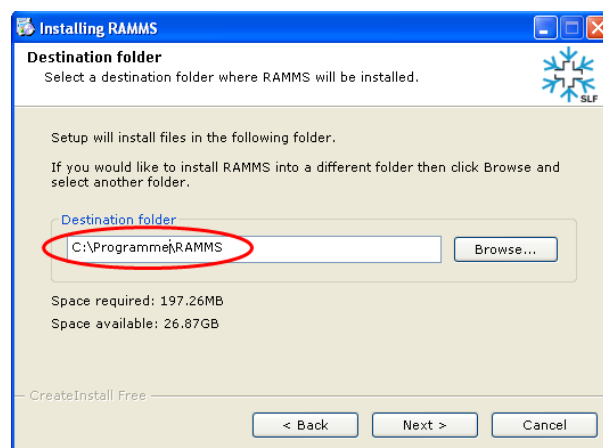


Figure 3.4: Installation - Destination directory dialog window.

Step 5: Installing the files

RAMMS is copying the files to the destination location and showing the installation progress.

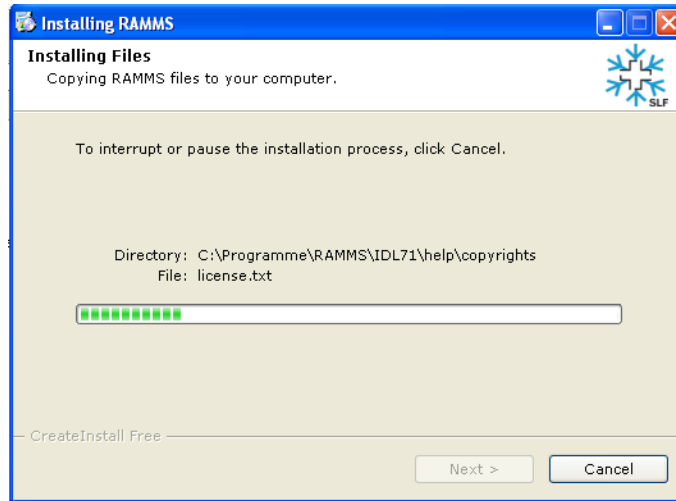


Figure 3.5: Installation - Installing files dialog window.

Step 6: Finished installing the files

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

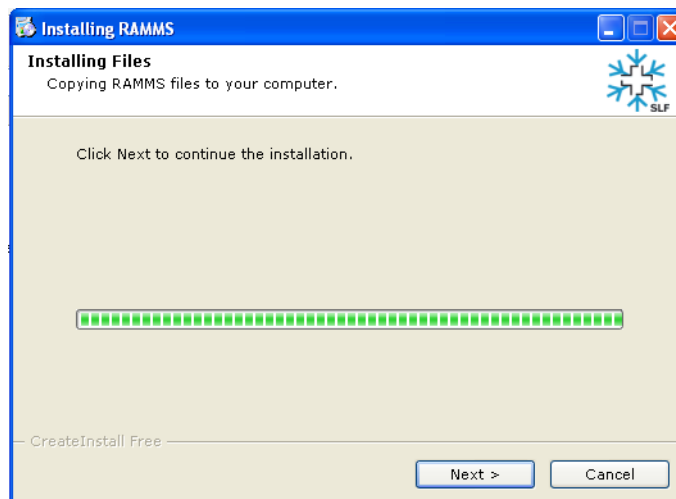


Figure 3.6: Installation - Finished installing files dialog window.

Step 7: RAMMS Installation finished!

RAMMS successfully finished the installation. Click *Finish*.



Figure 3.7: Installation - Finished installation dialog window.

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



Figure 3.8: RAMMS icon.

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

- RAMMS → Run RAMMS
- RAMMS → Uninstall RAMMS



Figure 3.9: RAMMS program group.

3.3 Licensing Methods

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

3.4 First Start

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



Figure 3.10: *RAMMS* start window.

Click on the image. It will disappear and *RAMMS* will start up. The following dialog window appears (Fig. 3.11 *RAMMS* - Licensing):

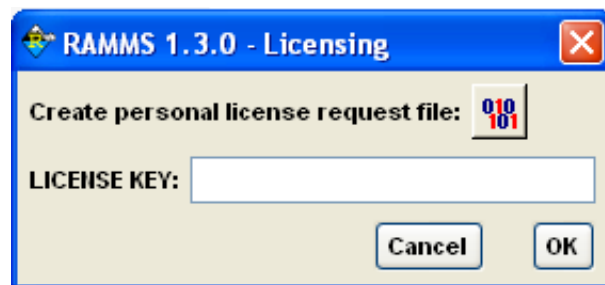


Figure 3.11: *RAMMS* licensing window.

3.4.1 Personal License Request File

Click the button  to create your personal license request file. In Fig. 3.12 enter your full name and the name of your company.

In the next dialogue window, choose the destination directory of your personal license request file and save it to your target machine. Your personal license request file should look similar to Fig. 3.13.

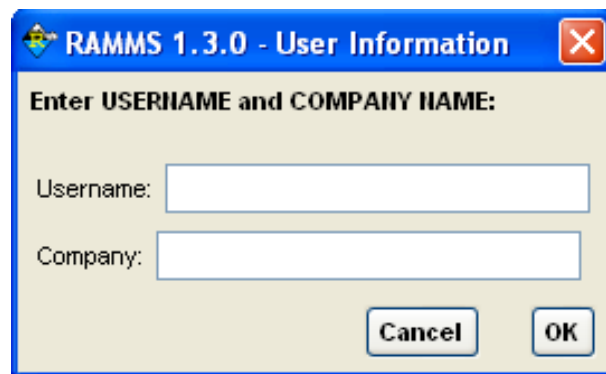


Figure 3.12: Enter user name and company name.

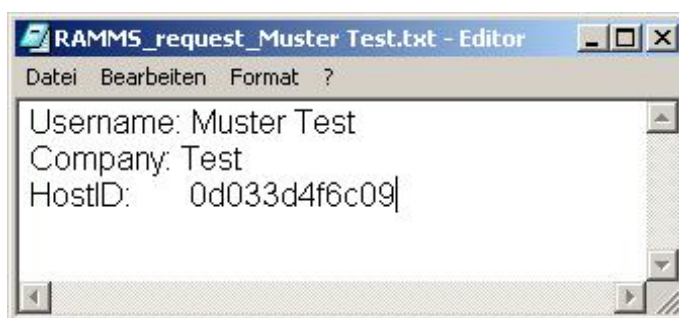


Figure 3.13: Personal license request file *RAMMS_request_Muster Test.txt*

3.4.2 Personal License Key

How to order a Personal License Key

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

An order confirmation email is sent to your Email address. We then process your order and send you an invoice. As soon as we received your payment, we will send you your personal license key. Your personal license key is named similar to *RAMMS_license_Muster Test.txt*. Open the file in a text-editor. It should look similar to Fig. 3.14.

Now, restart RAMMS (as explained before). Again, the pop-up window (Fig. 3.10) and then the dialogue window of Fig. 3.11 appears (*RAMMS* - Licensing).

Copy the license key (in this example: *akck-3ijh-3jtl-2h5h-g340*) and paste it at **LICENSE KEY:** (see Figure 3.11). If *RAMMS* accepts your installation key, you successfully finished

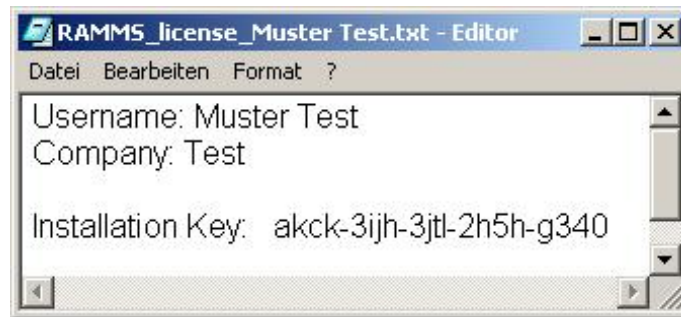


Figure 3.14: Personal license key file *RAMMS_license_Muster Test.txt*

the installation. Please continue with section 3.4.3 and install Cygwin/GRASS (required to use *RAMMS*).

3.4.3 CYGWIN / GRASS installation

RAMMS is coupled to the Open Source GIS Software *GRASS*. Before you can benefit of all the functions of *RAMMS*, you must install *GRASS* on your PC. *RAMMS* works with *GRASS* version 6.2.3. Later versions of *GRASS* are not supported in *RAMMS*. *GRASS* needs *Cygwin*, a Linux-like environment for Windows. Please perform the following steps to install both *GRASS* and *Cygwin*:

- Be aware: You need **Administrator privileges** on the target machine to install *GRASS*!
- Run *Track* → *Install Cygwin/GRASS* to install *GRASS* Version 6.2.3 .

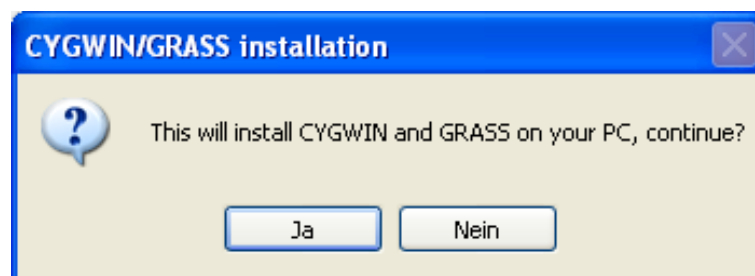


Figure 3.15: Continue Cygwin/GRASS installation?

Click *YES* to begin the installation of *Cygwin* and *GRASS*.

- Choose the file *Cygwin623.zip* in the *Temp*-directory of your *RAMMS*-distribution and click *OK*.

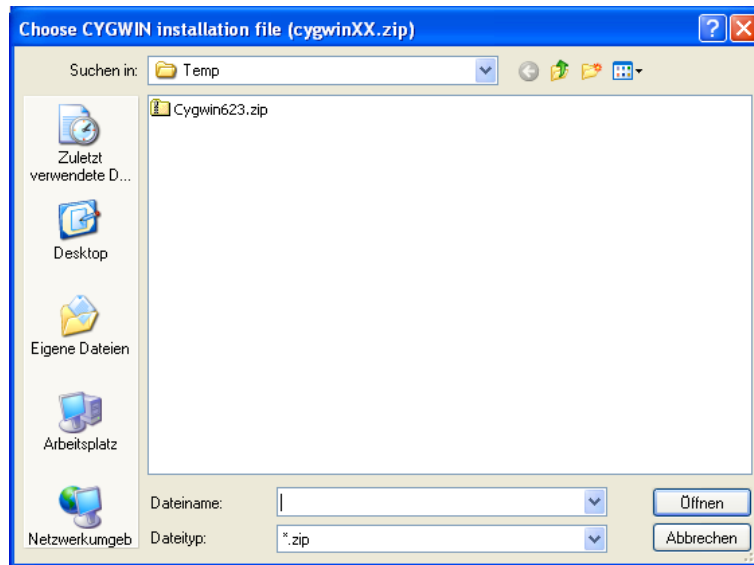


Figure 3.16: Choose Cygwin/GRASS installation zip-file.

- In the next window you are asked to specify the installation directory. Choose e.g. *C:\Programs* without *cygwin*, because *cygwin* will be appended automatically!
VERY IMPORTANT: Do NOT use BLANKS or any special characters in the installation path!!

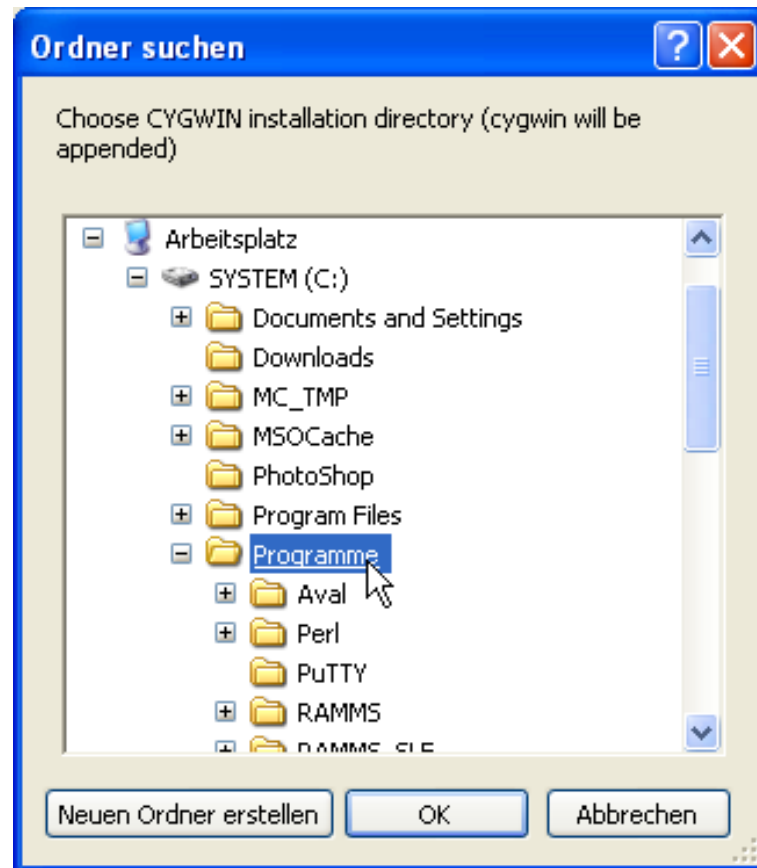


Figure 3.17: Choose Cygwin/GRASS installation directory.

- The files are now being extracted to the directory you specified before. Please be patient, because this can take a while!

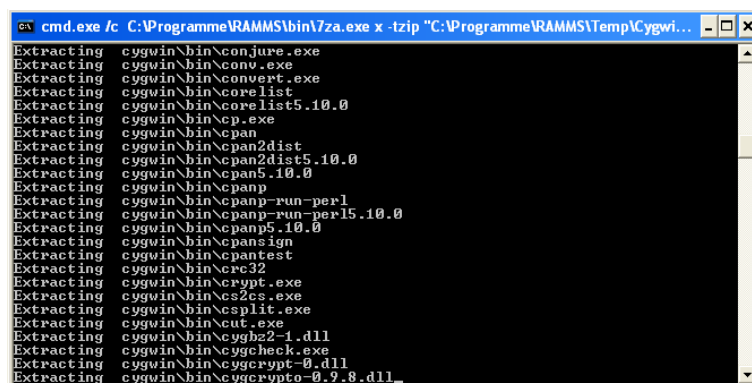


Figure 3.18: Cygwin/GRASS - Extract files.

- To finish our installation of Cygwin/GRASS, we have to update the Windows registry with the Cygwin/GRASS information. Click YES in the window below:

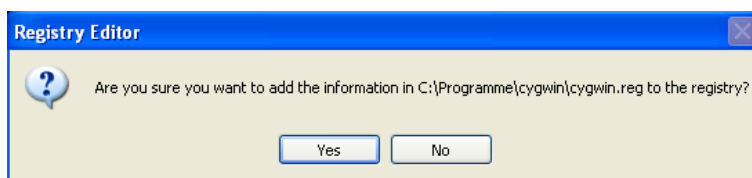


Figure 3.19: Cygwin/GRASS - Add to registry.



Figure 3.20: Cygwin/GRASS - Registered successfully.

- At the end a few Cygwin/GRASS scripts are executed. The installation of Cygwin/GRASS is now completed!

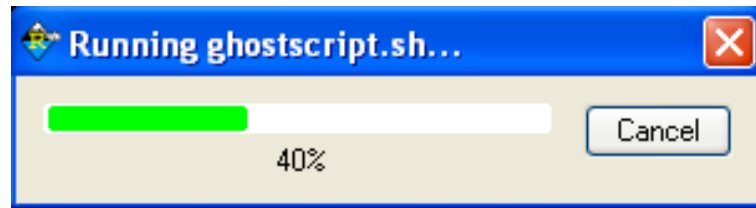


Figure 3.21: Cygwin/GRASS - Automatic scripts.

3.4.4 Potential problem

The following error can occur:

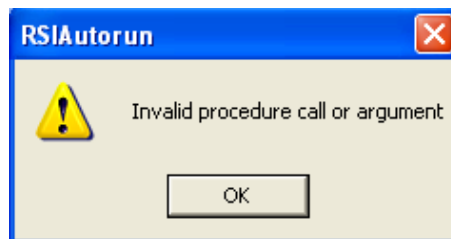


Figure 3.22: Error message

This error occurs, if certain .NET libraries are missing on your target machine. You need to install these missing libraries. Start the installer by double-clicking the file *run_system32_dll.bat* in the *Temp* folder of your *RAMMS* installation directory. Follow the instructions below to install the files:

Step 1: Welcome to IDL Visual Studio Merge Modules

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

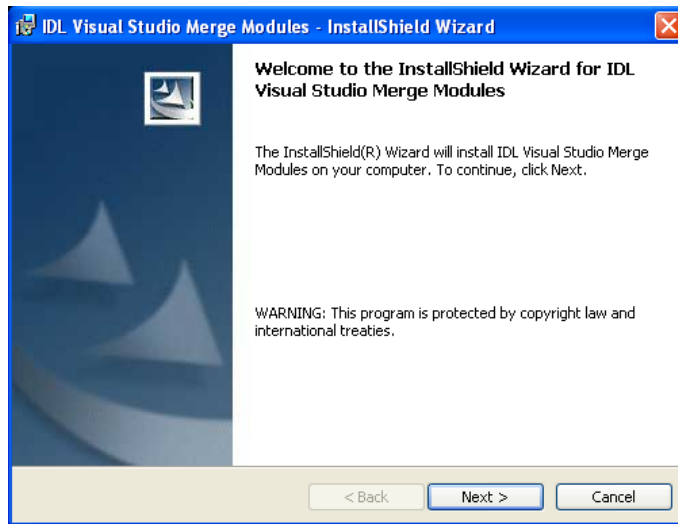


Figure 3.23: IDL Visual Studio Merge Modules - Welcome dialog window.

Step 2: Ready to install the Program

Click "Next" to continue.

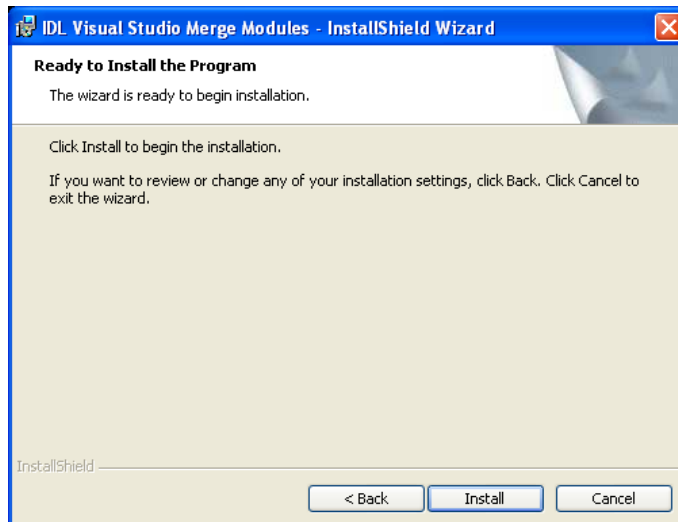


Figure 3.24: IDL Visual Studio Merge Modules - Ready to install the Program.

Step 3: Installing IDL Visual Studio Merge Modules

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

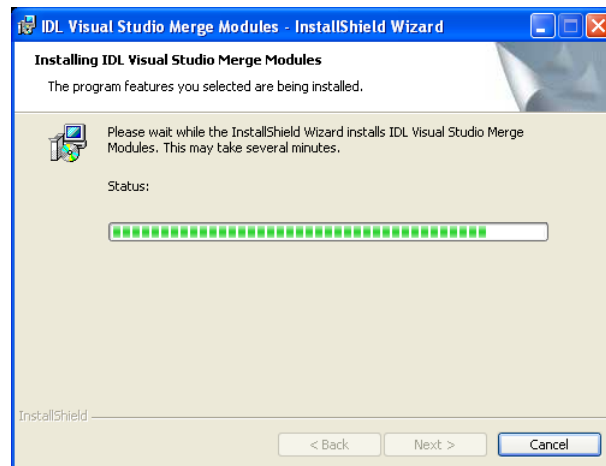


Figure 3.25: IDL Visual Studio Merge Modules - Installing...

Step 4: InstallShield Wizard Completed

The wizard completed the installation. Click *Finish*.

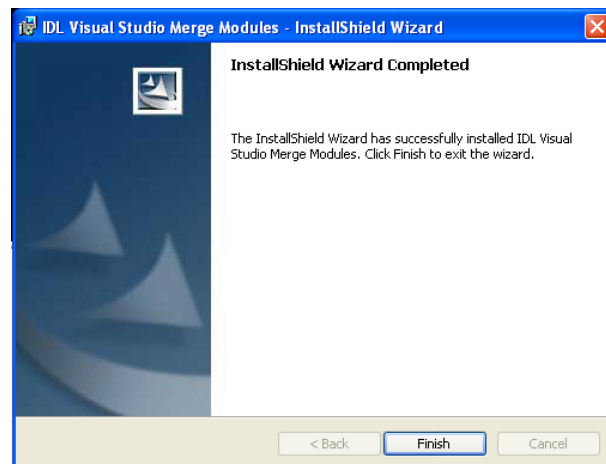


Figure 3.26: Installation - Destination directory dialog window.

You successfully installed the necessary files and *RAMMS* should start now normally. Go back to section 3.4 and start *RAMMS*.

3.5 Preferences

Before starting to work with *RAMMS*, be sure to set your *RAMMS* preferences and place the necessary DEM (Digital Elevation Model) files as well as the FOREST files, MAPS and georeferenced IMAGERY you wish to use in the appropriate folders defined in the preferences, see Figs. 3.27 and 3.28.

Use **Track** → **Preferences** to open the *RAMMS* preferences window or click the button .

- General tab

Setting	Purpose
Cygwin directory	This field allows you to set the <i>Cygwin</i> directory. This should correspond to your installation directory of <i>Cygwin</i> , e.g. <i>C:\Programme\cygwin</i> .
GRASS version	Use this field to specify the <i>GRASS</i> version you use. You can find your current <i>GRASS</i> version here: <i>C:\Programme\cygwin\usr\local\grass-6.X.X</i> (default <i>grass-6.2.3</i>).
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 TIFF-file and a corresponding tfw-file (world-file)).
Image directory	Set the folder where you place your digital georeferenced orthophotos (aerial picture, consists of a TIFF-file and a corresponding tfw-file (world-file)).
DEM directory	Set the folder where you place the D igital E levation M odels (format: ASCII grid, see 4.2.1 on page 25)
FOREST directory	Set the folder where you place your forest-files (formats: ASCII grid or polygon shapefile).

- Avalanche 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.
Background color	Set background color (greyscale between 0:black and 255:white).
Animation delay (s)	Set animation delay to decelerate the animation speed.

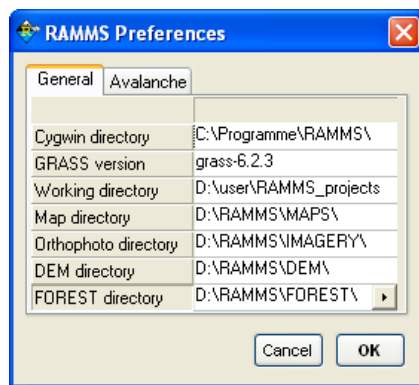


Figure 3.27: General tab of RAMMS preferences.

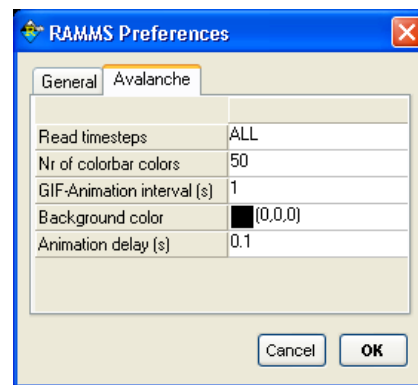


Figure 3.28: Avalanche tab of RAMMS preferences.

See exercise *"Working directory"* on page 22 below on 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 3.5.a: 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**) to open the RAMMS preferences window.
- Click into the field *Working directory*, then the appearing arrow  and **Edit...**. A window pops up where you can choose your new working directory. Click **OK** in both windows.

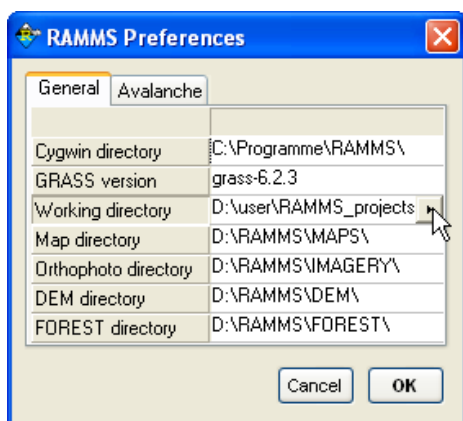


Figure 3.29: RAMMS preferences.

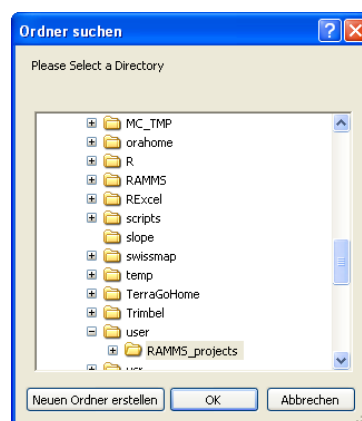


Figure 3.30: Browse for the correct folder.

End of Example 3.5.a

4 Working with RAMMS

4.1 General

To successfully start a new RAMMS project, a few important preparations are necessary. Topographic input data (ascii format), project boundary coordinates and georeferenced maps or remote sensing imagery should be prepared in advance (.tif format and .tfw file, maps and imagery are not mandatory, but nice to have). Georeferenced datasets have to be in a Cartesian coordinate system (e.g. Swiss CH1903+ LV95), polar coordinate systems (e.g. WGS84 Long Lat) are not supported. For more information about specific national coordinate systems please contact the national topographic agency.

4.1.1 Project / 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 parameter files (release and friction files). 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 (like friction parameters, release areas, calculation domain, calculation grid resolution, end time, time step etc.) can be changed for every scenario.

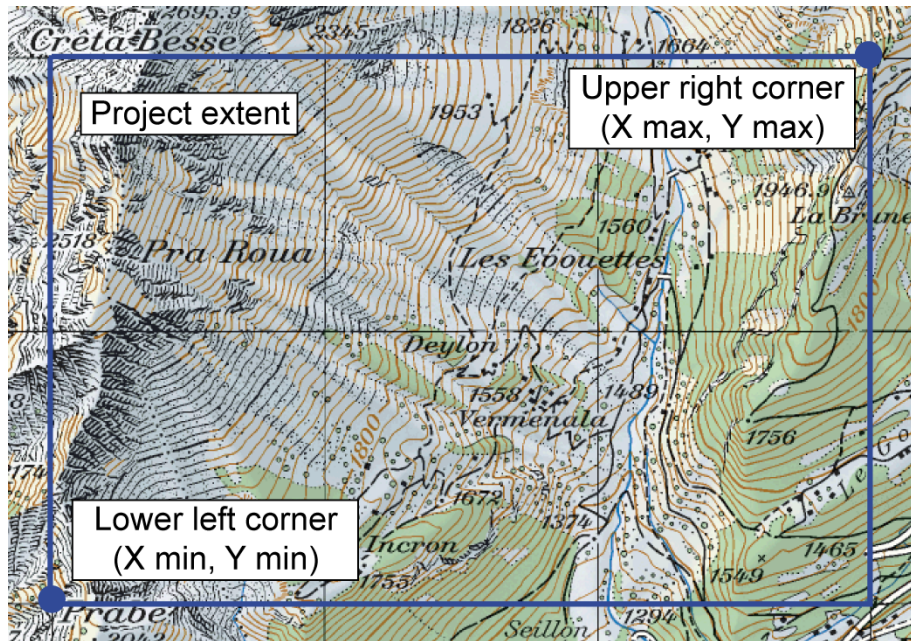


Figure 4.1: Project extent (area of interest) - topographic map ©2010 Swisstopo (BA091601).

4.1.2 Input data

There are different kind of data to be provided to successfully perform a calculation with RAMMS. Topographic data, definition of release area and release mass as well as information about friction parameters are mandatory.

RAMMS is able to process (see Figs. 4.2 and 4.3)

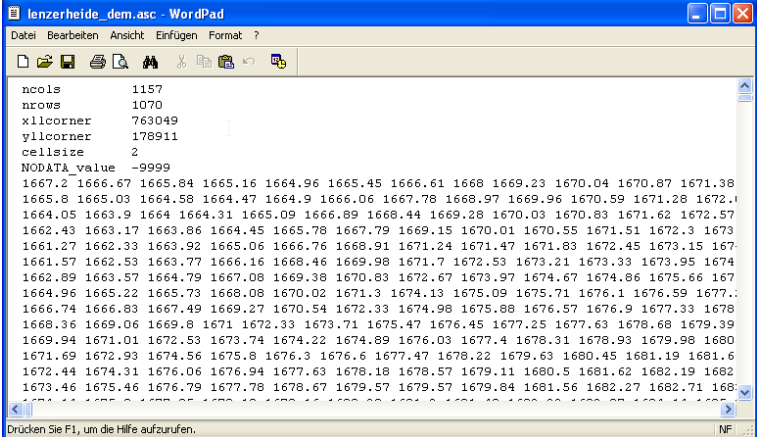
- ESRI ASCII Grid and
- ASCII X,Y,Z single space data.

These data types are also available e.g. from www.swisstopo.ch. Because RAMMS needs the topographic data as an *ESRI ASCII Grid*, *ASCII X,Y,Z* data can be converted within RAMMS into an *ESRI ASCII Grid*.

4.2 Model input data

4.2.1 Topographic data - Digital Elevation Model

The topographic data is the most important input requirement. The simulation results depend strongly on the resolution and accuracy of the topographic input data. The topographic data **MUST** be provided as an *ESRI ASCII Grid*. No other type is allowed at the moment. The user must therefore prepare the topographic data according to this limitation. The header of an *ESRI ASCII Grid* must contain the information shown in Fig. 4.2



```

lenzerheide_dem.asc - WordPad
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 1669.23 1670.04 1670.87 1671.38
1665.8 1665.03 1664.58 1664.47 1664.9 1666.06 1667.78 1668.97 1669.96 1670.59 1671.28 1672.1
1664.05 1663.9 1664 1664.31 1665.09 1666.89 1668.44 1669.28 1670.03 1670.83 1671.62 1672.57
1662.43 1663.17 1663.86 1664.45 1665.78 1667.79 1669.15 1670.01 1670.55 1671.51 1672.3 1673
1661.27 1662.33 1663.92 1665.06 1666.76 1668.91 1671.24 1671.47 1671.83 1672.45 1673.15 167
1661.57 1662.53 1663.77 1666.16 1668.46 1669.98 1671.7 1672.53 1673.21 1673.33 1673.95 1674
1662.89 1663.57 1664.79 1667.08 1669.38 1670.83 1672.67 1673.97 1674.67 1674.86 1675.66 167
1664.96 1665.22 1665.73 1668.08 1670.02 1671.3 1674.13 1675.09 1675.71 1676.1 1676.59 1677.
1666.74 1666.83 1667.49 1669.27 1670.54 1672.33 1674.98 1675.88 1676.57 1676.9 1677.33 1678
1668.36 1669.06 1669.8 1671 1672.33 1673.71 1675.47 1676.45 1677.25 1677.63 1678.68 1679.39
1669.94 1671.01 1672.53 1673.74 1674.22 1674.89 1676.03 1677.4 1678.31 1678.93 1679.98 1680
1671.69 1672.93 1674.56 1675.8 1676.3 1676.6 1677.47 1678.22 1679.63 1680.45 1681.19 1681.6
1672.44 1674.31 1676.06 1676.94 1677.63 1678.18 1678.57 1679.11 1680.5 1681.62 1682.19 1682
1673.46 1675.46 1676.79 1677.78 1678.67 1679.57 1679.57 1679.84 1681.56 1682.27 1682.71 168

```

Figure 4.2: Example ESRI ASCII Grid.

An *ESRI ASCII GRID* can be created in ArcGIS with the function *ArcToolbox*→*Conversion Tools*→*From Raster*→*Raster to ASCII*.

It is possible to import *ASCII X,Y,Z single space* data and convert the data into an *ESRI ASCII Grid* (using *Track*→*New...*→*Convert XYZ to ASCII Grid*).

4.2.2 Forest information

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

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

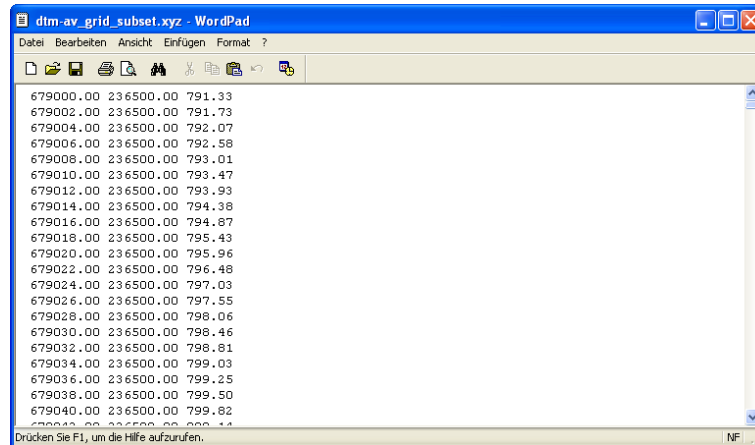


Figure 4.3: Example ASCII X,Y,Z single space data.

If no such files are available, the user can draw a polygon shapefile in RAMMS and import it as forest information (see section 4.10 on page 51).

4.2.3 Release information

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

Users have to draw their own release polygon shapefiles, see section 4.6 on page 40. All release informations are saved as polygon shapefiles and can be easily imported in GIS-Software (e.g. ArcGIS). Shapefiles created in e.g. ArcGIS can be imported into RAMMS by using *GIS/GRASS* → *Convert Shapefile...* → *Polygon Shapefile to RAMMS Release Shapefile*.

4.2.4 Global parameters

The two global parameters *return period* and *avalanche volume category* influence the classification of the friction information.

4.2.5 Friction information

An automatic RAMMS procedure calculates friction values based on topographic data analysis (slope angle, altitude and curvature), forest information and global parameters, see section 4.8 on page 46. All friction informations are saved as polygon shapefiles and can be easily imported in GIS-Software (e.g. ArcGIS).

4.2.6 Calculation parameters

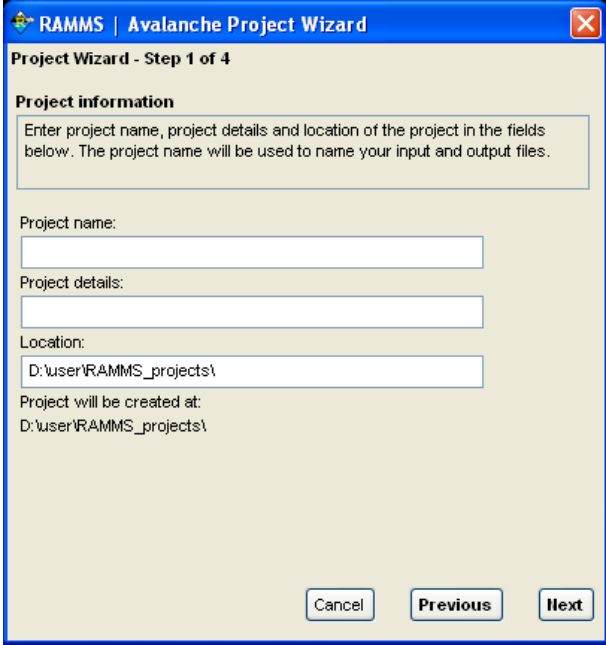
Calculation parameters such as output name, simulation grid resolution, end time, time step etc. can be changed interactively in RAMMS.

4.3 The Project Wizard

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

Exercise 4.3.a: How to create a new project.

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



RAMMS | Avalanche Project Wizard

Project Wizard - Step 1 of 4

Project information

Enter project name, project details and location of the project in the fields below. The project name will be used to name your input and output files.

Project name:

Project details:

Location:

Project will be created at:

Figure 4.4: RAMMS Avalanche Project Wizard: Step 1 of 4.

Step 1:

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

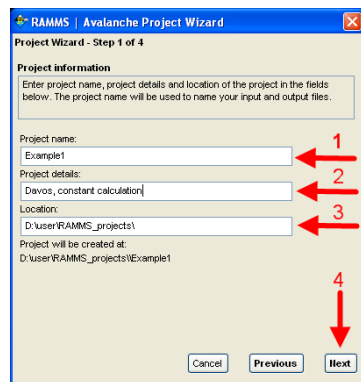


Figure 4.5: Step 1 of the RAMMS project wizard: Project information.

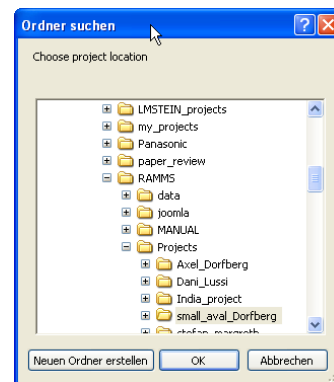


Figure 4.6: Window to browse for a new project location.

Step 2:

- Locate your DEM- and FOREST-file in the folder set in the RAMMS preferences. Click into the corresponding fields to browse for the appropriate files (1).
- If you don't want to use a FOREST-file, select "Do NOT use forest information" (2).
- Click Next (3).

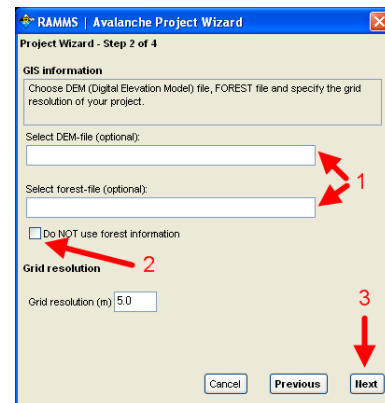


Figure 4.7: Step 2 of the RAMMS project wizard: GIS information.

Step 3:

- Enter the X- and Y-coordinates of the lower left and upper right corner of your project area, using the Swiss Coordinate System CH1903+ LV95 (or another cartesian coordinate system), as it is shown below for the Valle de la Sionne area.

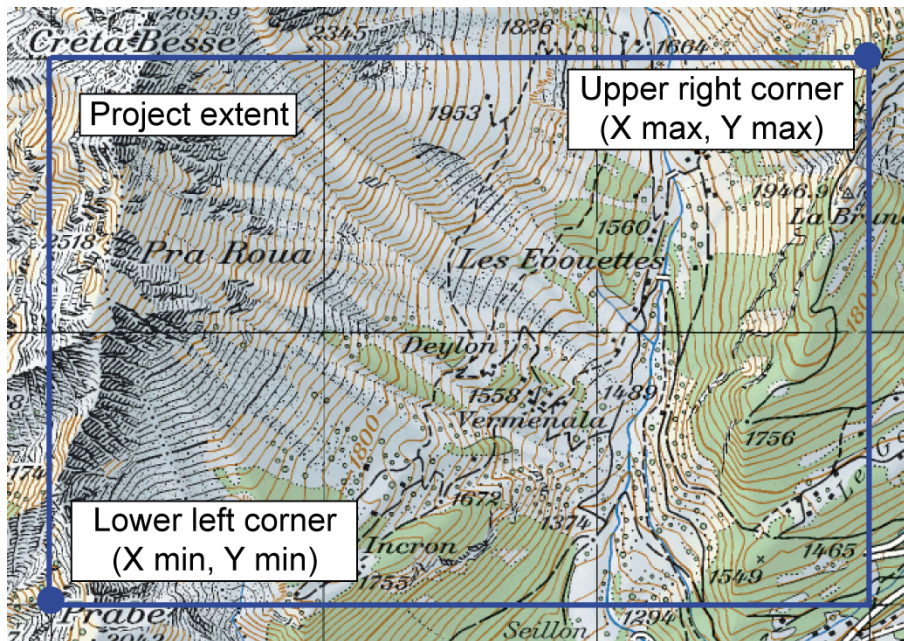


Figure 4.8: Project coordinates: lower left and upper right corner of project area - ©2010 swisstopo (BA091601).

Figure 4.9: Step 3 of the RAMMS project wizard: Project boundary coordinates.

Step 4:

- Check the project summary, especially if a DEM- and FOREST-file was found.
- To make changes click **Previous**, to create the project click **Create Project Example1**.

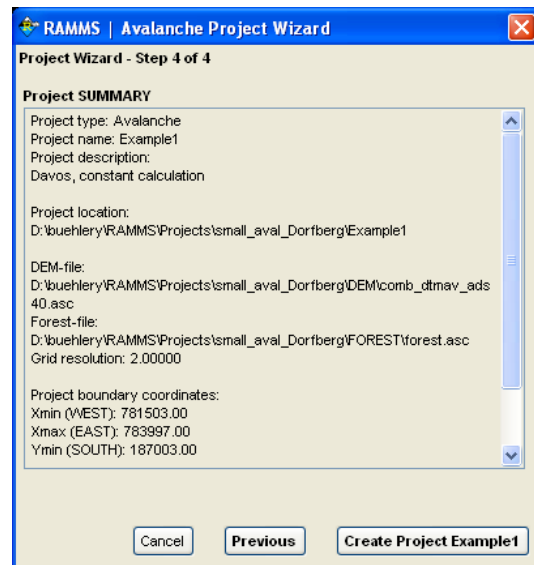
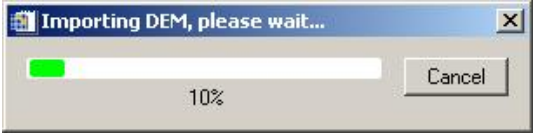
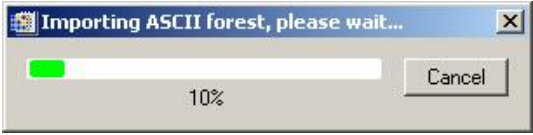
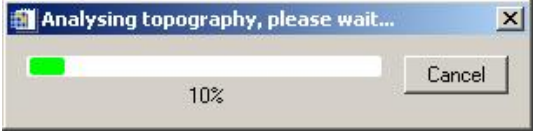
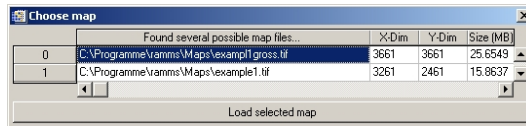


Figure 4.10: Step 4 of the RAMMS project wizard: Project summary.

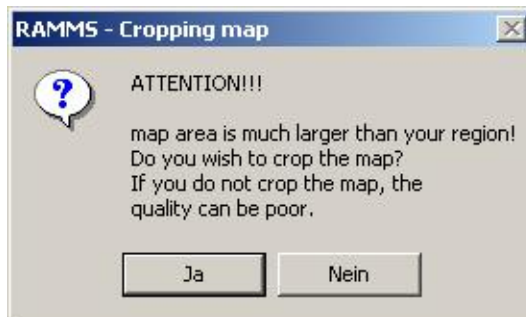
End of exercise 4.3.a

The creation process can take a while. Different status bars will pop up and show the progress of the project creation process, see table below:

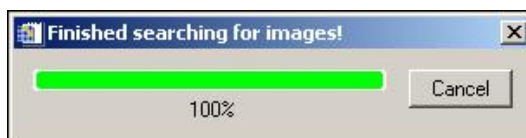
Progress bar	Purpose
 A progress bar window titled "Importing DEM, please wait...". It shows a green progress bar at 10% and a "Cancel" button.	GRASS function: Import the DEM file
 A progress bar window titled "Importing ASCII forest, please wait...". It shows a green progress bar at 10% and a "Cancel" button.	GRASS function: Import the FOREST file and export the file <i>forest5m.asc</i>
 A progress bar window titled "Creating Mesh Data, please wait...". It shows a green progress bar at 30% and a "Cancel" button.	GRASS function: Export the file <i>Example1.xyz</i> with the topographic data
 A progress bar window titled "Reading 19200 Z-coordinates, please wait...". It shows a green progress bar at 20% and a "Cancel" button.	Reading topographic data (x-, y- and z-coordinates).
 A progress bar window titled "Analysing topography, please wait...". It shows a green progress bar at 10% and a "Cancel" button.	Analysing the topography and calculating curvature and slope angles
 A progress bar window titled "Preparing topography, please wait...". It shows a green progress bar at 55% and a "Cancel" button.	Preparing topography for displaying in RAMMS
 A progress bar window titled "Finished searching for maps!". It shows a green progress bar at 100% and a "Cancel" button.	Searching georeferenced maps in the Maps-Folder



If more than one map was found, the user can select a map



If the map is much bigger than the project-region, the user can crop the map to improve the quality of the map-visualization



Searching georeferenced maps in the Orthophoto-Folder



If no image was found, the user can specify another folder or abort the process

The following files will be created in the project-folder

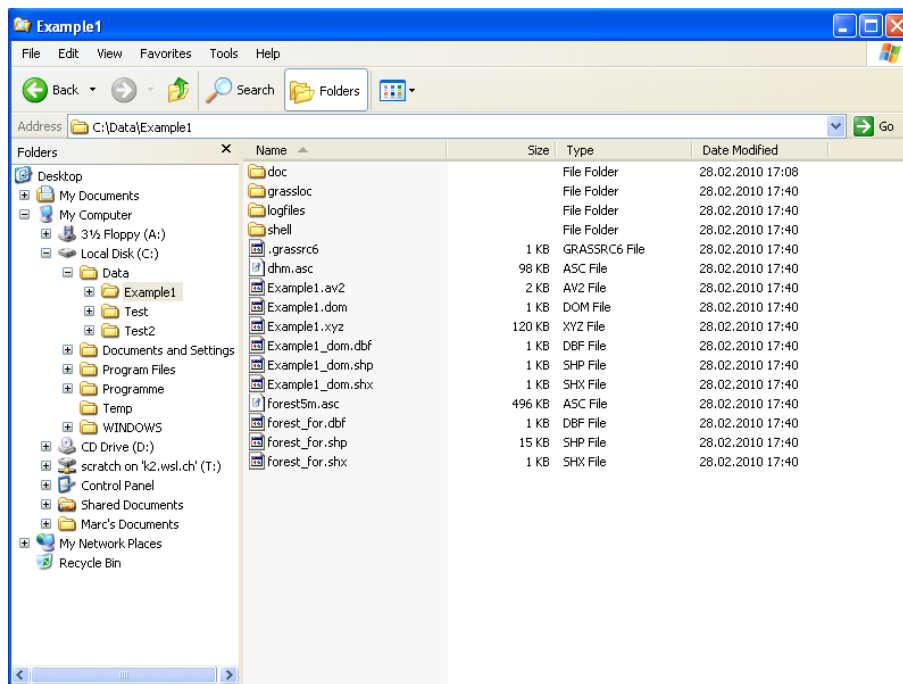


Figure 4.11: Created project files.

File / Folder	Purpose
doc (folder)	Folder containing input and output LOG files
grassloc (folder)	Folder containing GRASS files: do NOT delete or change them
logfiles (folder)	Log-files for GRASS operations
shell (folder)	Folder containing GRASS shell scripts
.grassrc6	GRASS ini file: do NOT delete or change
dhm.asc	ASCII Grid with altitude values
Example1.av2	Input file
Example1.dom	Calculation domain ASCII file
Example1_dom.shp	Calculation domain shapefile
Example1_dom.shx	Calculation domain shapefile
Example1_dom.dbf	Calculation domain shapefile
Example1.xyz	Topographic data used in RAMMS
forest_shp.shp	Extracted forest shapefile
forest_shp.shx	Extracted forest shapefile
forest_shp.dbf	Extracted forest shapefile

forest5m.asc	ASCII Grid with forest information (0:no forest, 1:forest)
--------------	---

4.4 Moving, resizing and changing the view of the model

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

Exercise 4.4.a: 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.

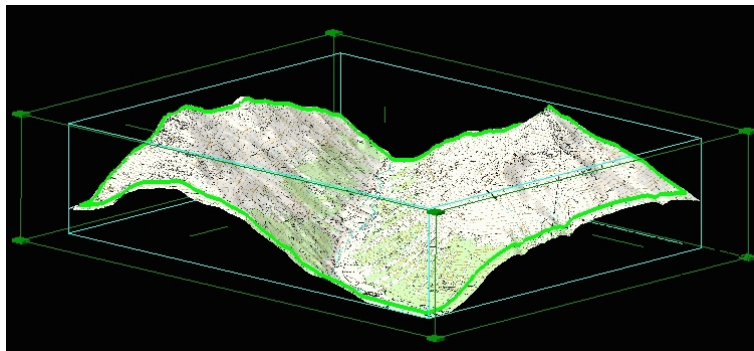


Figure 4.12: "Active" project with lines and corners for resizing, topographic map ©2010 Swisstopo (BA091601)

- To move the model without changing size or aspect ratio, move 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, move to one of the green corners. If the cursor turns to , click and hold the left mouse button. By moving the mouse towards the model, the size will be reduced (move away to enlarge the model). Alternatively you can resize the model by changing the percentage value in the horizontal toolbar .

- To resize the model by changing the aspect ratio, go on one of the green lines. When the mouse cursor changes to , click and hold the left mouse button and change the size by moving the mouse.
- b) Terrain model has a dimension $> 100\%$:
 - All steps explained above are still possible. Especially for the last two steps explained, the view of one of the corners or lines is required.
 - 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.

End of exercise 4.4.a

Exercise 4.4.b: 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.

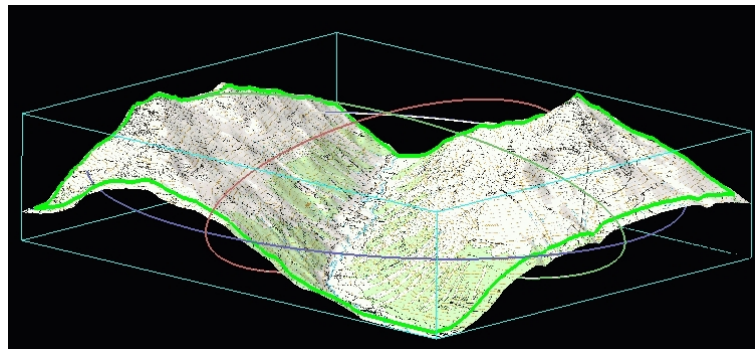


Figure 4.13: "Active" project with rotation axes, topographic map ©2010 swisstopo (BA091601).

End of exercise 4.4.b

Exercise 4.4.c: 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.

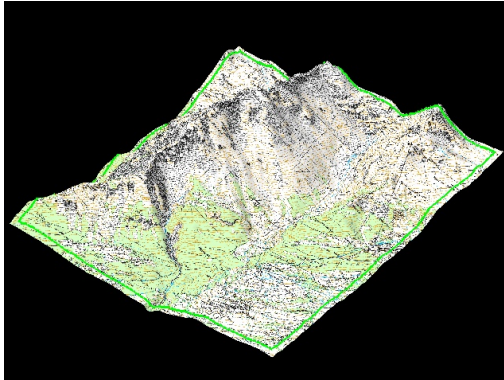


Figure 4.14: 3D view of example model, topographic map ©2010 swisstopo (BA091601).

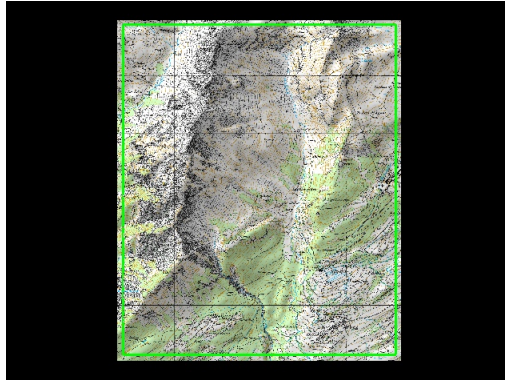


Figure 4.15: 2D view of example model, topographic map ©2010 swisstopo (BA091601).

In 2D-mode you have all possibilities that you know from 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:

- New release area 
- Release area information 
- Edit Release area 
- New domain 

OUTPUT:

- Line profile 

End of exercise 4.4.c

4.5 Global parameters

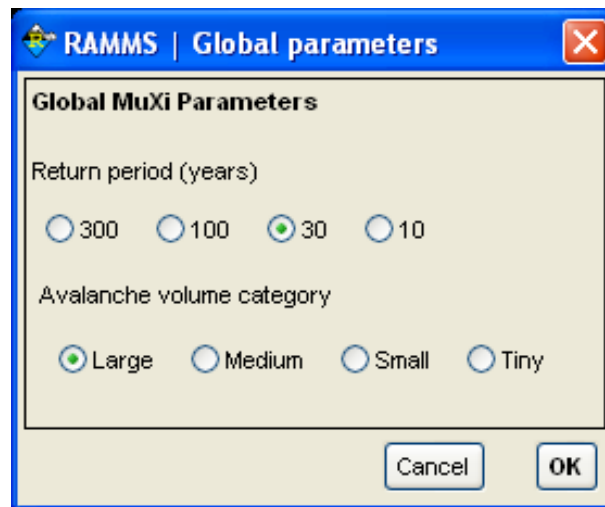


Figure 4.16: RAMMS global parameters.

Prior to creating a new muxi-file, choose the return period and the volume category. The MuXi-file will be calculated based on these values. Changing the return period and/or volume category has no effect on already existing muxi-files. The default volume category is chosen based on the specified release volume.

4.6 Release areas

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

Create a new release area	If there is no release area available for your project, or you wish to create a new one, switch to 2D mode and click  .
Load an existing release area	Load an existing release area with Input $\Rightarrow$ Release area $\Rightarrow$ Load existing release area .
Import a shapefile and convert it to a release area	Draw a release area using a GIS-tool and save it as shapefile (.shp). Then convert the shapefile using GIS/GRASS $\Rightarrow$ Convert Shapefile $\Rightarrow$ Polygon Shapefile to RAMMS Release Shapefile .

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

Exercise 4.6.a: How to create a new release area.

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

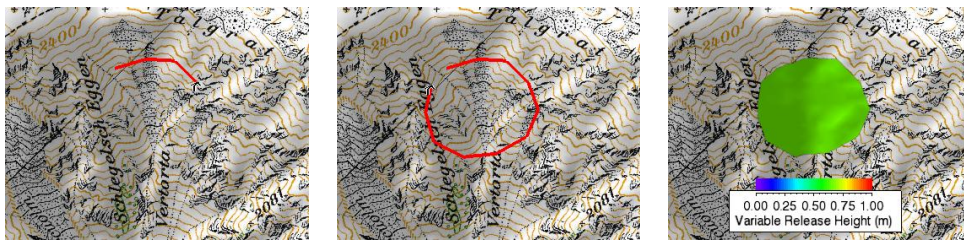


Figure 4.17: Project with emerging release area.

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

- **Add more release 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 release filename:**
Enter a new name for the release area. The ending *.rep.shp is added automatically.

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

End of exercise 4.6.a

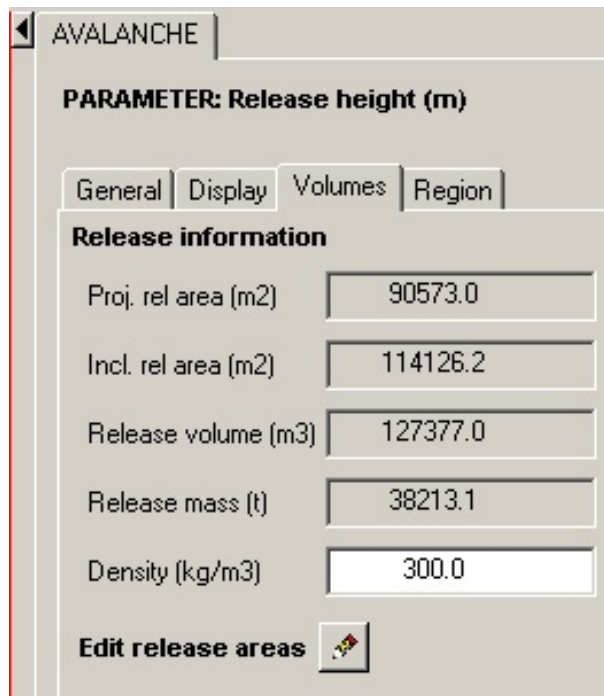
Exercise 4.6.b: How to load an existing release area.

- Choose **Input**→**Release area**→**Load existing release area**.
- Select release file (*rep.shp) and click **open**.
⇒ The release area appears in the project as well as the colorbar for the variable release height (m)

End of exercise 4.6.b

Once a release area is created or loaded, you have to specify the release height. Choose **Input**→**Release Area...**→**View and Edit Release Areas** or click the button  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 height can be entered, see exercise below.

Additional release information is found in the avalanche panel in the volumes-tab, see Figure below.



AVALANCHE

PARAMETER: Release height (m)

General | Display | **Volumes** | Region

Release information

Proj. rel area (m2)	90573.0
Incl. rel area (m2)	114126.2
Release volume (m3)	127377.0
Release mass (t)	38213.1
Density (kg/m3)	300.0

Edit release areas 

Figure 4.18: Release area and volume information.

Exercise 4.6.c: Specify release height and view release information

- Switch into 2D mode by clicking .
- Click on the "View and Edit Release Information" button  (in the horizontal toolbar or in the Volumes-tab in the panel) or choose **Input**→**Release Area**→**View and Edit Release Areas**.
- Then click on the release area you want to get information on. A red polygon is drawn around the selected release area. The following window appears:

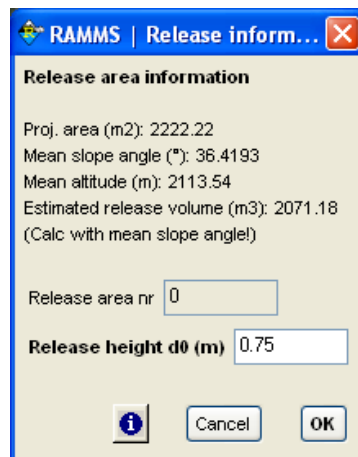


Figure 4.19: Release area information window.

Remark: The estimated release volume shown is estimated, calculated with a mean slope angle for the whole release area.

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

End of exercise 4.6.c

4.7 Calculation Domain

To save calculation time you can specify a **calculation domain**. If you know where the avalanche path is situated (e.g. by running a calculation with a coarse cell size, e.g. 25m), you can exclude the area which is not of interest. Switch to *2D mode* and chose **Input** → **Calculation Domain...** → **Draw new Domain**. Now you can draw a polygon containing the area of interest accordingly to drawing a new release area (see exercise "Create release area" on page 40). We strongly recommend to use *Calculation Domains* especially if you calculate with small cell sizes (e.g. < 5m).

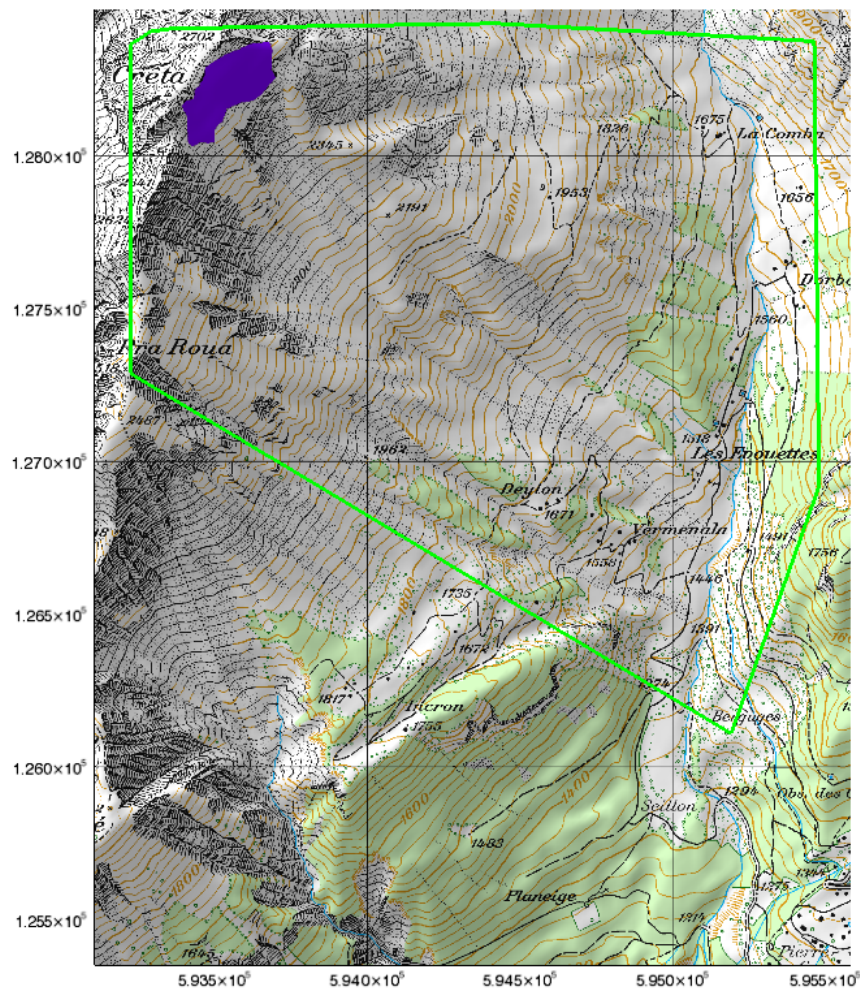


Figure 4.20: Calculation domain in green confines the area of interest and saves calculation time.

4.8 Friction parameters

RAMMS employs a Voellmy-fluid friction model. 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 (coefficient ξ). The frictional resistance S (Pa) is then

$$S = \mu\rho Hg \cos \phi + \frac{\rho g U^2}{\xi}$$

Figure 4.21: RAMMS Voellmy-fluid friction model.

where ρ is the flow density, g gravitational acceleration, ϕ the slope angle, H the flow height and U the flow velocity. This model has found wide application in the simulation of mass movements, especially snow avalanches. The Voellmy model has been in use in Switzerland for a long time and a set of calibrated parameters is available.

RAMMS offers a constant and a variable calculation mode. If a calculation is done with constant friction values, of course, no terrain undulations and forest areas are considered. Therefore we suggest to use the **variable friction values** if possible. μ and ξ depend strongly on the global parameters *return period* and *avalanche volume*. Therefore, you **MUST** define an appropriate return period and check your avalanche volume in the Global Parameters (**Input→Global Parameters**) **BEFORE** creating a new MuXi file!

RAMMS | Automatic MuXi Procedure

Volume = TINY Return period = 30

(Optional: if empty -> default name)

MuXi name:

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

Above Altitude Limit 1

Open Slope - Mu Xi

Channelled Slope - Mu Xi

Gully Slope - Mu Xi

Flat Area Slope - Mu Xi

Forested Area (mu=delta, xi=fix) - Mu Xi

Below Altitude Limit 1

Open Slope - Mu Xi

Channelled Slope - Mu Xi

Gully Slope - Mu Xi

Flat Area Slope - Mu Xi

Forested Area (mu=delta, xi=fix) - Mu Xi

Below Altitude Limit 2

Open Slope - Mu Xi

Channelled Slope - Mu Xi

Gully Slope - Mu Xi

Flat Area Slope - Mu Xi

Forested Area (mu=delta, xi=fix) - Mu Xi

Figure 4.22: RAMMS — Automatic MuXi Procedure.

How to create a new MuXi file is demonstrated in the exercise *"How to create a new MuXi file"* on page 47 below.

Before running a calculation, make sure you chose *variable* friction parameters if you want to take the topography into account (Mu/Xi tab in the Run calculation window).

In case you closed and then reopened a project and did not save it with the created MuXi file, you can simply load the MuXi file again and do not have to create it again (see exercise *"How to load an existing MuXi file"* on page 47).

Exercise 4.8.a: How to create a new MuXi file.

- Choose **Input**→**Friction Values...**→**Create new MuXi file (Automatic Procedure)**.
- Enter a file name.
- Unless you know better, leave the values as they are...
- Click **OK**.
- The MuXi file will be loaded after it is created. If you create the first MuXi file for this project, this procedure can take a while, depending on your computer's performance (CPU and RAM). The creation of following MuXi files will be much faster.
- You can switch between the release area (if already loaded), the Mu and Xi value via the avalanche panel.

End of exercise 4.8.a

Exercise 4.8.b: How to load an existing MuXi file.

- Choose **Input**→**Friction values**→**Load existing MuXi file**.
- A window opens to browse for an existing MuXi file (*MuXi.shp)
- Click **Open** and the file will be loaded.

End of exercise 4.8.b

4.9 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 .

Exercise 4.9.a: Editing the colorbar

Changing the min and max values of the colorbar as well as changing the number of colors used is done in the Avalanche Panel under Display.

- Simply type a new value into the respective field and hit the return key on the keyboard. The display will be refreshed.
- A further, very helpful thing to change is the transparency. This is useful when the topography should be visible through the avalanche.
- ATTENTION: The line **Values < x.xxx are not displayed!!** is important. Depending on your min and max values, check this line to know, which values are not displayed!

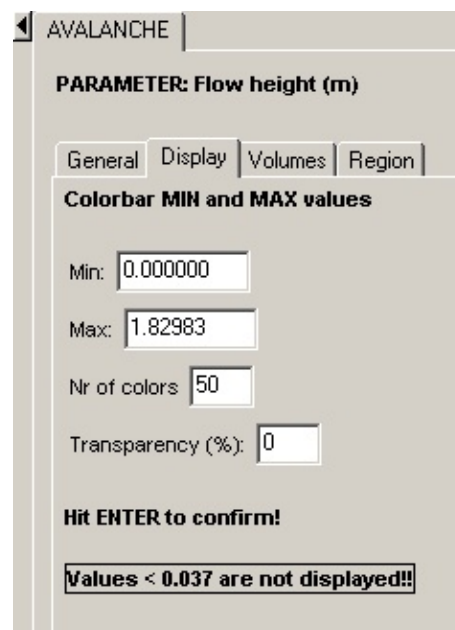


Figure 4.23: The avalanche 'Display' panel.

- Open the editing window by either choosing **Edit→Edit colorbar properties** (only INPUT) or clicking  in the vertical toolbar.
- To change the colorbar properties simply click into the field you want to change, then click OK.

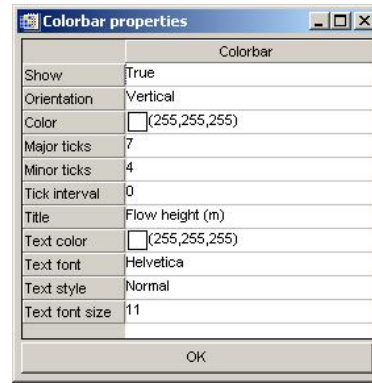


Figure 4.24: The colorbar properties window.

End of exercise 4.9.a

4.10 Forest

It is necessary to take forest areas into account, when running a simulation with variable friction parameters (Mu and Xi).

The easiest way to consider forest, is to have a digital forest file (ASCII grid or forest shapefile) located in the folder set in the RAMMS preferences (FOREST directory). In this case the forest file will already be included while creating the project. Step 2 of the *RAMMS::Avalanche Project Wizard* deals with the DEM- and FOREST-files. Click into the field once, to browse for your FOREST-file.

If you have a forest-file available and located in the FOREST folder but don't want to use it, click "Do NOT use forest information". This is necessary, e.g., if you want to simulate a situation without considering the forest.

Exercise 4.10.a: How to create a FOREST file.

- Switch to 2D mode by clicking .
- Activate project by clicking on the map once.
- Click  or choose **Input→Forest...→Draw New Forest Areas**.
- Trace the forest outline by creating as many FOREST area polygons as necessary (proceed as in exercise *"How to create a new release area"* on page 40) and name your new FOREST file. A new FOREST shapefile is saved.
- You are asked, if you want to import the created FOREST file into your project. Click YES, if you want to use the newly created FOREST (ignore the next point in this case). Otherwise click NO and import the FOREST file later, as explained in the next point.
- Import the new FOREST shapefile: Choose **Input→Forest...→Import Forest Area From SHAPEFILE**, then select your FOREST shapefile.
- This new FOREST information is not automatically taken over in existing MuXi-files. Therefore, recreate existing MuXi-files if needed. If you create a new MuXi file with **Input→Friction Values...→Create new MuXi file (Automatic Procedure)**, the forest will now be considered.

End of exercise 4.10.a

4.11 Project information

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

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

This window provides information about all your project's input specifications, like number of nodes and cells, release areas, forest files, used DEM, the loaded map and ortho images as well as your global simulation parameters.



Figure 4.25: RAMMS Project Input Log Information.

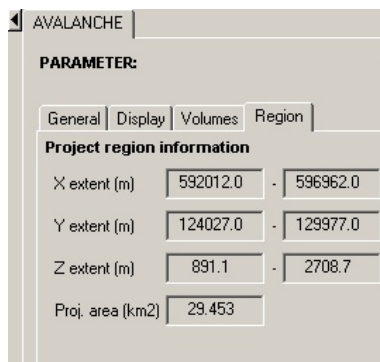


Figure 4.26: Region extent (X-, Y- and Z-Coordinates, total area).

To view the project coordinates, click the region tab in your avalanche panel. The region tab lists X- and Y-Coordinates of the lower left (min values) and upper right (max values) corner (this are the coordinates you entered when creating the project) as well as the global minimum and maximum of altitude (Z value). The X- and Y-Coordinates do not correspond exactly to the clipping values you entered at creation time of the project. This is due to the clipping algorithm in GRASS. Additionally, the total region area is shown (in km^2).

4.12 Changing maps and remote sensing imagery

It is possible to change the map or imagery 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!

Exercise 4.12.a: How to add or change maps.

- Go to **Extras**→**Add/Change Map**.

If more than one map is found, the following window pops up, listing the maps found:

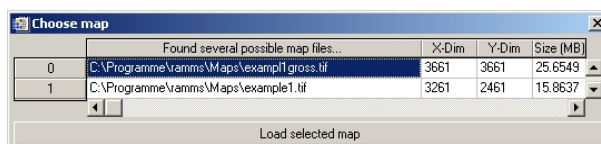


Figure 4.27: Window to choose map image.

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

- If the question "Map area is much larger than your region! Do you wish to crop the map?" appears, as it says, your map is much larger than your project area.
- Click **No** to load the map without cropping. It may lead to a bad resolution.
- We suggest to click **Yes**. The map will be cropped to the same size as the project region. A window opens to enter a name for the new map. The default saving location is the one set in the RAMMS preferences. You can change it if necessary.

End of exercise 4.12.a

Exercise 4.12.b: How to add or change remote sensing imagery.

- Go to **Extras→Add/Change image**.

See exercise "*How to add and change maps*" on page 55 above.

End of exercise 4.12.b

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

5 Calculation and Results

5.1 Running a calculation

To run a calculation one has to have created a project, loaded a release area and if a calculation with variable friction parameters is desired, a muxi-file must have been created as well. Below you find two short examples, one for running a constant calculation (constant release height and constant friction parameters μ and ξ) and one for using variable release height and friction parameters.

Exercise 5.1.a: How to run a constant avalanche calculation.

- To run a calculation choose **Run→Run Avalanche Calculation** or click .
- The RAMMS::Input Parameters window opens.
- Before clicking **RUN CALCULATION**, you should check the input parameters.

- **General:**

- (1) Project name
- (2) Project infos. You can change them by simply typing into the field.
- (3) Additional information: Calculation domain file and DEM file.
- (4) Select an output filename.

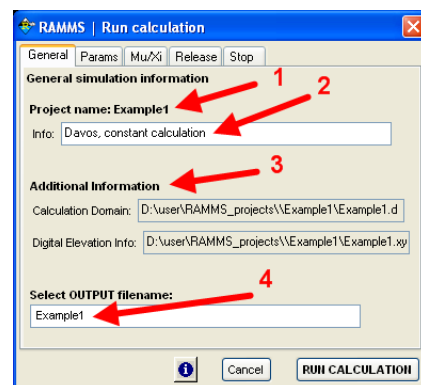


Figure 5.1: General information.

- **Params:**

- (1) Change grid resolution, if necessary.
- (2) Choose end time of simulation.
- (3) Choose dump step interval.
- (4) Keep the default value for density.
- (5) Change numerical solver, 1st or 2nd order scheme (we recommend 2nd order).

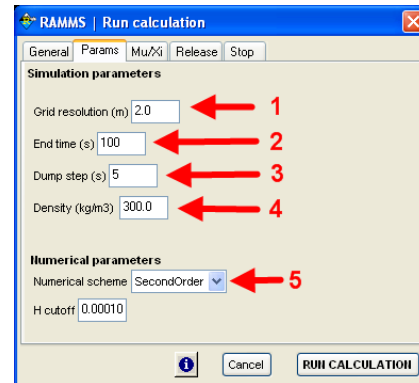


Figure 5.2: Calculation parameters

- **Mu/Xi:**

- (1) For a calculation with constant MuXi-values, click **constant**.
- (2) Enter Mu and Xi values. Choose **Help→RAMMS Manuals...→Friction Parameter Table (PDF)** or see friction value table in the appendix for an idea of μ and ξ !

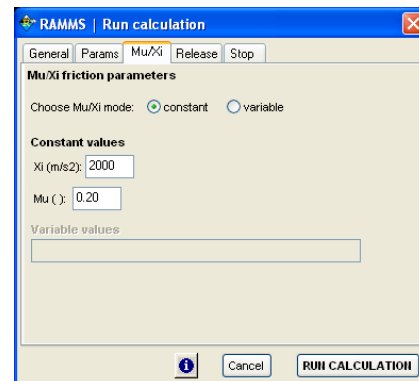


Figure 5.3: Friction values MuXi.

- **Release:**

- (1) The text field should indicate your release shapefile.
- (2) The estimated release volume is stated in the second text field.

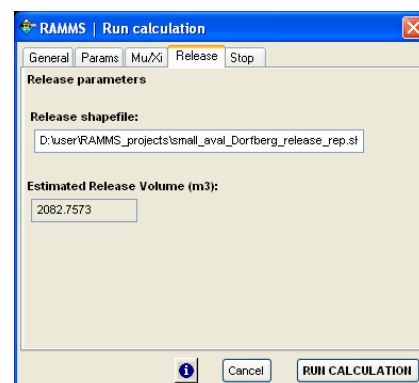


Figure 5.4: Release information.

- **Stop:**

- (1) The stopping criteria in RAMMS is based on the momentum. In classical mechanics, momentum (SI unit kgm/s, or, equivalently, Ns) is the product of the mass and velocity of an object ($p = mv$). For every DUMP STEP, we sum the momenta of all grid cells, and compare it with the maximum momentum sum. If this percentage is lower than a user defined threshold value (see below), the program is interrupted and the avalanche is regarded as stopped. Threshold values between 1-10% are reasonable, but this is only a suggestion and must be validated by the individual user himself.

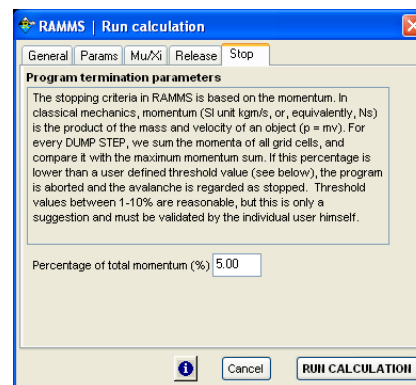


Figure 5.5: Stop criteria.

- Click **RUN CALCULATION** (fig. 5.4).
- The following window appears, showing the status of the calculation.

```

cmd.exe /c E:\net\IDL\Workspace70\ramms1.2\bin\ramms_aval.exe E:\RAMMS_Beispiele\vdls_506...
*****
Welcome to the RAMMS::Avalanche calculation module!
Starting calculation with 78750 nodes, 78186 elements and 155809 edges
Writing output file: E:\RAMMS_Beispiele\vdls_506\Test.out...
Writing coordinates...
Writing connectivity list...
Writing initial conditions...
Starting explicit time integration.....
Time 5
Step 33 DT 0.0823273 Cfl 0.342693
Hmax 1.20618 Umax 32.8201 Umax 24.7106 HEmax 0
MOVING MASS: 98.268442 percent of total mass
Volume 78387.401474 m3
HE_Total 0.000000 GJ
-----
Time 10
Step 99 DT 0.00382922 Cfl 0.365933
Hmax 2.21239 Umax 34.5582 Umax 24.7106 HEmax 0
MOVING MASS: 96.304083 percent of total mass

```

Figure 5.6: Status window of calculation.

- Once it's finished, the simulation is opened in RAMMS.

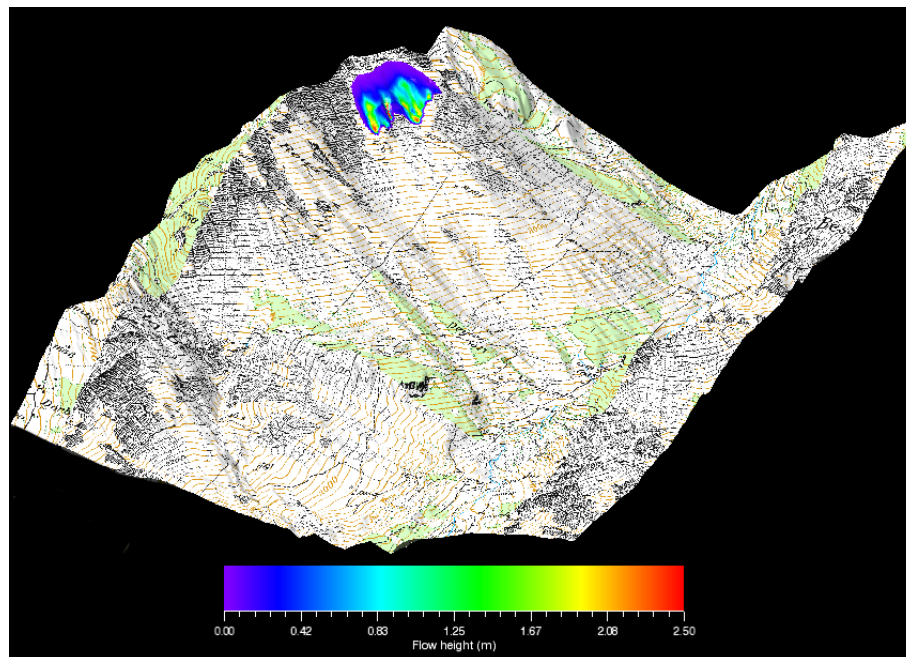


Figure 5.7: Main window in output mode, topographic map ©2010 Swisstopo (BA091601).

End of exercise 5.1.a

Exercise 5.1.b: How to run a variable avalanche calculation.

The process for a variable calculation is almost the same than for a constant calculation! Therefor this exercise only shows the difference to the exercise above (exercise *"How to run a constant avalanche calculation"* on page 57).

- **Mu/Xi:**

- (1) For a calculation with variable MuXi-values, click **variable**.
- (2) Check, if the correct MuXi-file is used.

If a different file should be used, click into the field and browse for the desired file.

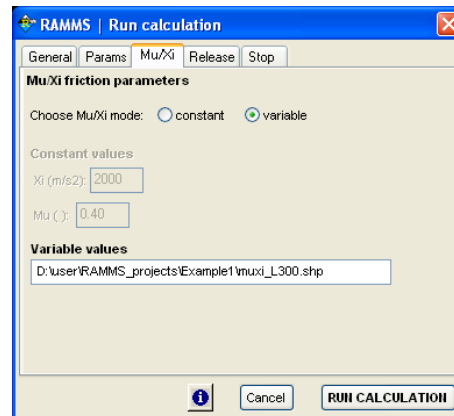


Figure 5.8: MuXi settings for a variable calculation.

End of exercise 5.1.b

5.2 Results

This section just gives a short overview on what is possible to do with the simulation results. The interpretation of the results has to be done by an avalanche expert who is familiar with the topographic and meteorological situation of the investigation area. The RAMMS simulation results should not be used without questioning them! Perform sensitivity studies!

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

- Flow Height
- Flow Velocity
- Flow Pressure
- Flow Momentum
- Max values (Height, Velocity, Pressure, Momentum)
- DEM Adaptations (Add Deposition to DEM)
- Flow Analysis (Summary of Moving Mass)
- Friction Values (Mu, Xi)

These results are all visualized by a colorplot in the topography. See exercise *"Displaying max values"* on page 64.

In the horizontal toolbar you find two further functions:

- Line Profile 
- Time plot 

Line profile

A line profile is a good alternative to the color plot, if the avalanche snow height, velocity or pressure should be known at a specific location. The graph shows the currently active parameter. Every line profile is saved in the file **profile.txt** 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"* on page 64.

Time plot

This function provides a time plot at a single point. This is helpful, when knowing the position of an obstacle (building, dam, tree...) and the (max) values at this location over time are a relevant information. Every point is saved in the file **point.txt** 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 timeplot"* on page 64. The point-info file can be visualized with **Extras**→**Point...**→**View Point Info File**.

Exercise 5.2.a: Displaying max values of flow height, flow velocity and pressure.

The maximum values of snow height, velocity and pressure give a good view on the dimension of the avalanche. You find them under

Results ⇒ **Max values...**

⇒ **Max flow height**

⇒ **Max velocity**

⇒ **Max pressure**

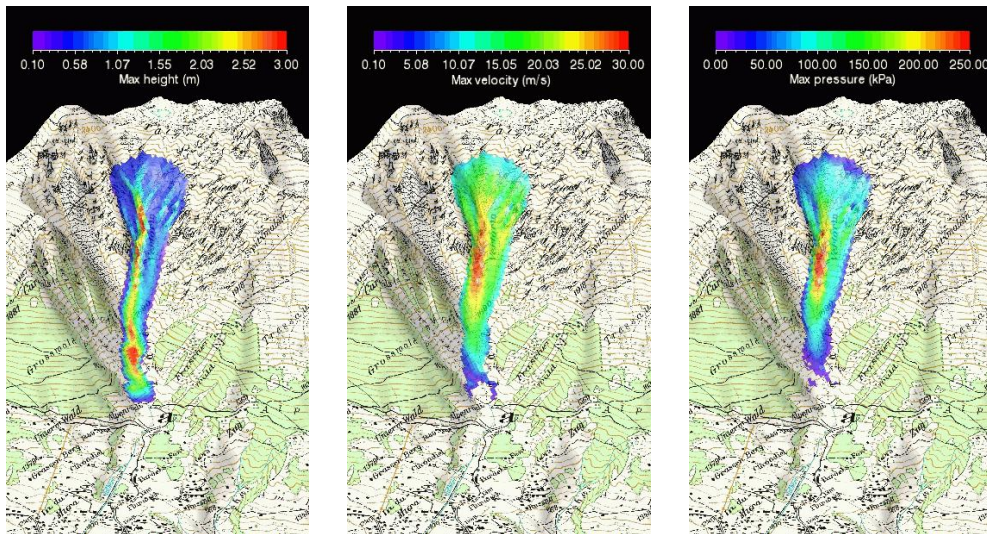


Figure 5.9: Results: Maximum values of flow height (left), velocity (middle) and pressure (right), topographic map ©2010 Swisstopo (BA091601).

End of exercise 5.2.a

Exercise 5.2.b: 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 double-click with the left mouse button.
- A window opens, displaying the line profile.

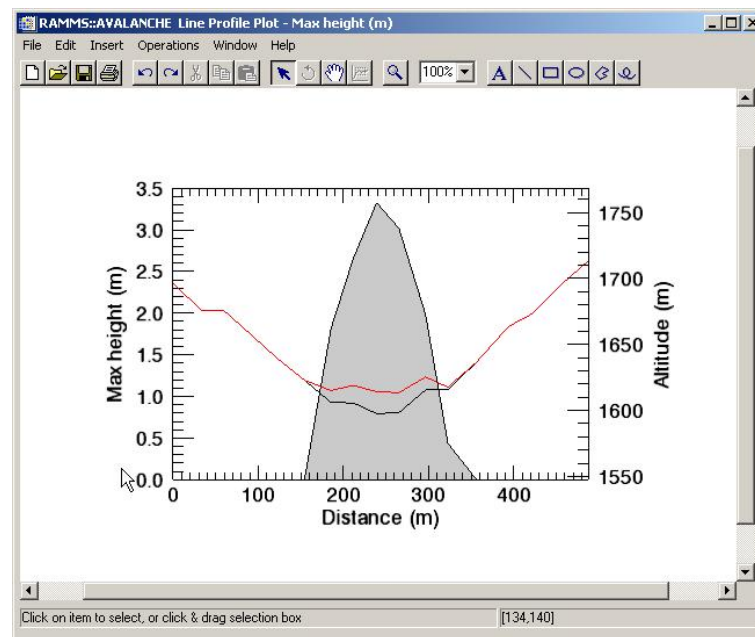


Figure 5.10: Line profile plot.

- filled grey area → active parameter (scale on left side).
 - red line → active parameter (multiplied by 50) added to the track profile (altitude, scale on the right side).
 - black line → track profile (altitude, scale on the right side).
 - bottom scale: projected profile distance (in m).
- If you change the active parameter, min or max values or the dump step in RAMMS, the plot is directly updated. You can also start the simulation and then watch the time variations in your line profile plot.
 - It makes sense to either draw a profile line perpendicular to the flow direction or to draw the line along the avalanche path. Basically every imaginable path is possible.

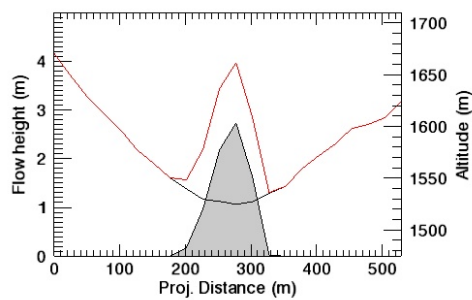
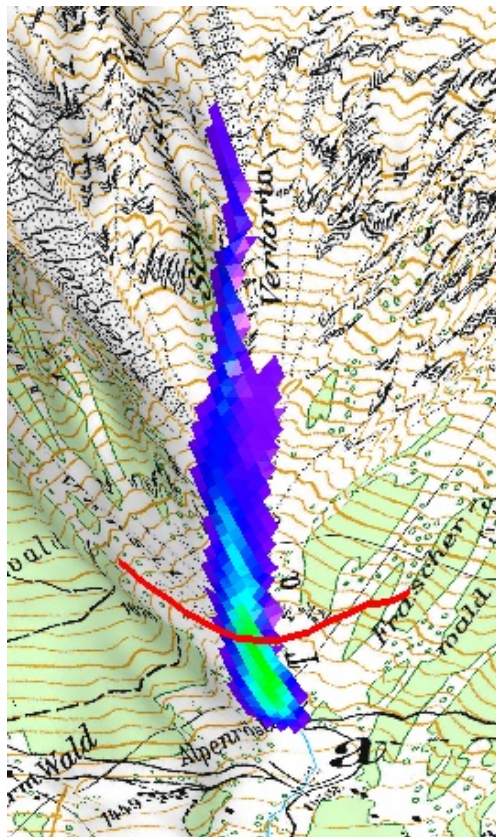


Figure 5.11: Line profile perpendicular to flow direction, topographic map ©2010 Swisstopo (BA091601).

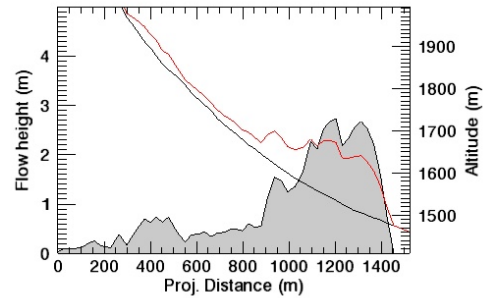
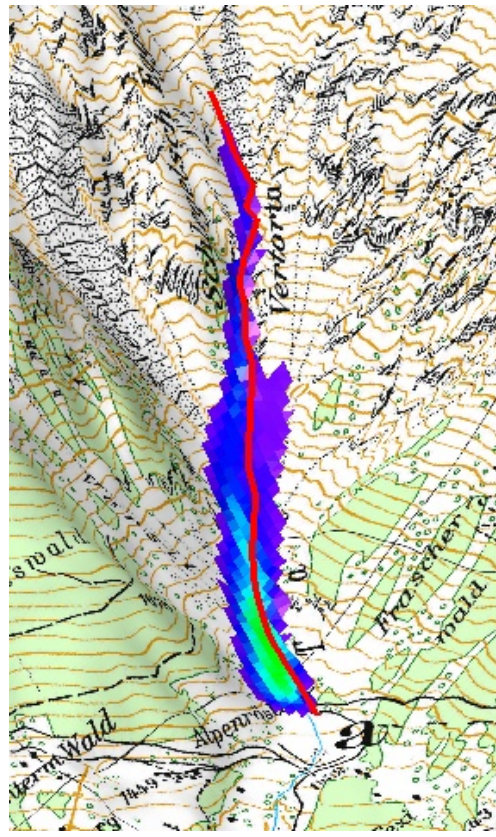


Figure 5.12: Line profile along the avalanche path, topographic map ©2010 Swisstopo (BA091601).

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

b) Load an existing line profile:

- Switch to 2D mode 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.

End of exercise 5.2.b

Exercise 5.2.c: How to create a time plot.

a) Select time plot point:

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

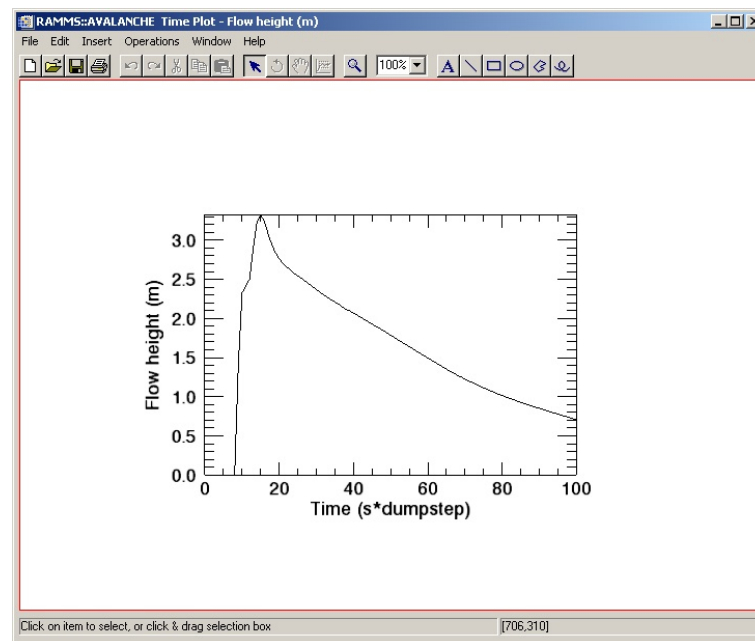


Figure 5.13: Time plot window.

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

b) Load a time plot:

- To reopen the time plot graph window of the last selected point, go on **Extras→Point...→Create Point Time Plot**.
- To open an arbitrary time plot that was saved anytime 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.

End of exercise 5.2.c

Exercise 5.2.d: Enter point coordinates and get a time plot.

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

End of exercise 5.2.d

5.3 Extras

5.3.1 Creating an image and a GIF animation

Image:

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

GIF animation:

Creating a GIF animation is of course only possible in output mode.

Click  and wait until the simulation stopped and a window opened. Enter a file name and location. The GIF animation folder as well as the corresponding gif animation file is saved in the simulation folder. In the avalanche tab in the preferences you can define the interval for the GIF animation (GIF animation interval (s)).

5.3.2 Creating a dam

RAMMS offers the possibility, to simulate a dam by increasing the altitude at the position where a dam should be considered.

Exercise 5.3.a: How to create a new DEM to simulate a dam.

- Create a polygon ("release area") where a dam is supposed to be build.
- Create a second, inner polygon, if you wish to have a two-stage dam.
- Go on **GIS/GRASS→Add DAM to DEM...**
- You will be asked to "Open dam file (*.rel)". Select the shapefile you want to use as the outer edge of the dam.
- The question pops up, if you want to "Open 2nd dam shapefile (inner polygon)?"
 - Click **No** to continue with the next step.
 - Click **Yes** to choose 2nd dam file (*.rel).
- Next step is to "Enter TOTAL elevation height of dam (m)". This is the elevation (masl.) of the dam crest.
- If you loaded an inner polygon file, you will be asked to "Enter intermediate height (m)" as well.
- At last you have to "Enter new DEM name". Your new DEM, containing the "dam" is created in the folder set as DEM directory (RAMMS preferences .

End of exercise 5.3.a

To run a simulation based on the new created DEM, you first have to create a new project. Do almost exactly the same as if creating a regular project without the dam information. The only important difference is, that you have to choose the correct DEM-file manually during step 2 of the project wizard.

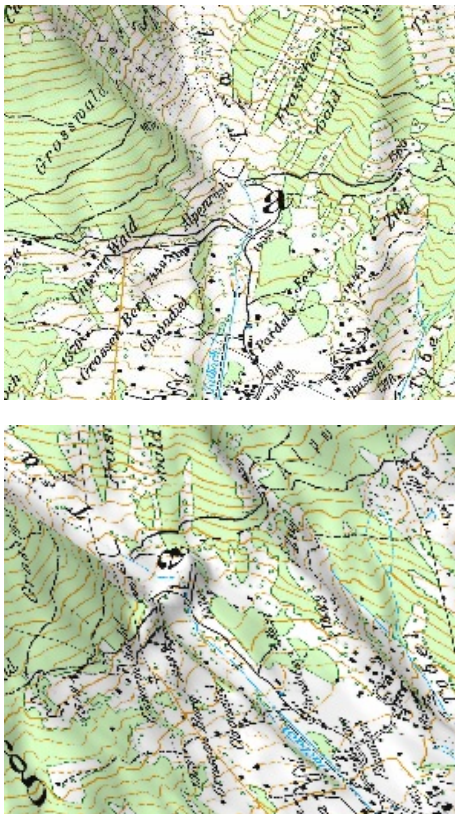


Figure 5.14: Project before (top) and after (bottom) creating a dam, topographic map ©2010 Swisstopo (BA091601).

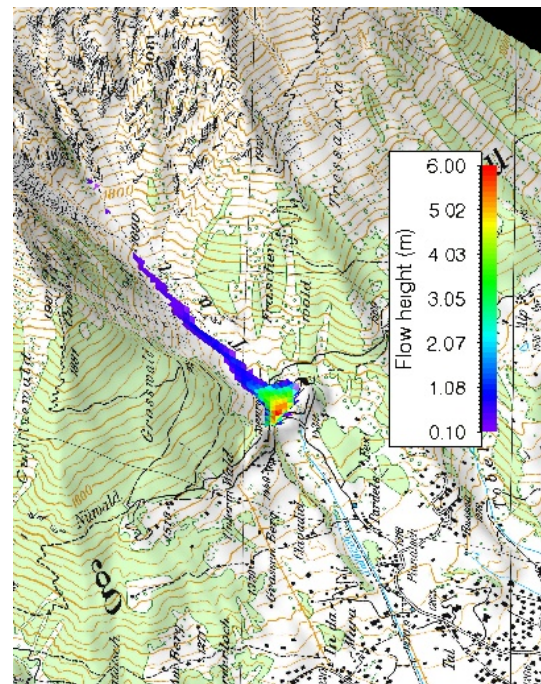


Figure 5.15: Avalanche being stopped by a dam, topographic map ©2010 Swisstopo (BA091601).

5.3.3 Creating a new DEM with avalanche deposition

In case you wish to simulate an avalanche flowing down over an old avalanche, you should take into account the deposition of the first avalanche, because the path of the second avalanche will be influenced by the changed terrain.

In output mode RAMMS provides the option of adding the snow height of an avalanche, at an arbitrary dump step, to the DEM. A new project can be created based on the updated DEM.

Exercise 5.3.b: How to add avalanche deposition to new DEM.

- The deposition height of the avalanche varies with time. So first run the simulation to the specific dump step.
- Go to **Results**→**DEM Adaptations**→**Add Deposition to DEM**.
- Enter a new name for the new DEM.
- The new DEM, containing the deposition information, is created. To run a simulation based on this DEM create a new project and manually choose the DEM file during step 2 of the wizard as explained above for the dam.

End of exercise 5.3.b

5.4 How to save input files and program settings

Once a project is created, it is saved under the name and location you entered during step 1 of the RAMMS::Avalanche Project Wizard (see figure 4.5 on page 29). The created inputfile has the ending *.av2.

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

Exercise 5.4.a: 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 and muxi-file was loaded but you wish to close the project before doing the calculation.
- If you wish to save a copy of your file under a new name, go on **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**.
- Continue working on the original file, not the just saved one!

b) Program settings:

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

End of Example 5.4.a

5.5 How to open input and output files

Exercise 5.5.a: How to open an input file.

- Close any active project file.
- Go to **Track**→**Open...**→**Inputfile** or click .
- A window opens to browse for an avalanche input file (*.av2).
- Click **Open** after file name was selected.
- The project will be opened.

_____ End of Example 5.5.a _____

Exercise 5.5.b: How to open an output file/avalanche simulation.

- Close any active project file.
- Go to **Track**→**Open...**→**Avalanche Simulation** or click .
- A window opens to browse for an avalanche simulation file (*.out.gz)
- Click **OK**.
- The simulation will be opened.

_____ End of Example 5.5.b _____

Exercise 5.5.c: How to load an optional shapefile.

- To load a shapefile, click .
- A window opens to browse for a shapefile (*.shp).
- Click **Open** after file was selected.

_____ End of Example 5.5.c _____

5.6 About RAMMS

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

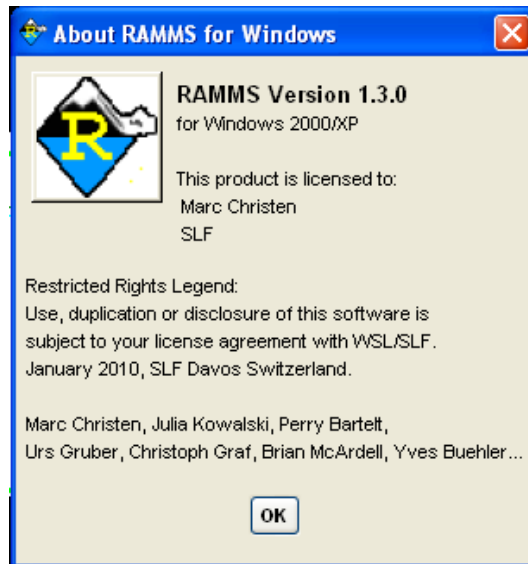


Figure 5.16: About RAMMS...

6 Program Overview

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

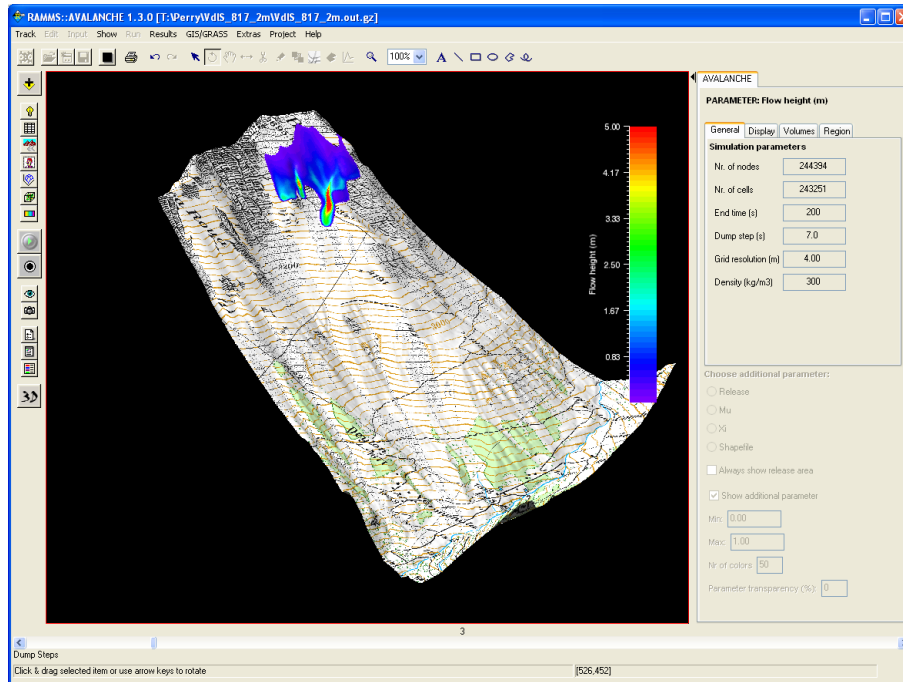


Figure 6.1: Graphical user interface (GUI).

6.1 The Graphical User Interface (GUI)

The graphical user interface (GUI), see figure 6.1 on page 78, consists of menu bar, horizontal and vertical toolbar, main window, time step slider, right and left status bar, colorbar and panel. They will be explained in the following sections.

6.1.1 The menu bar

Track

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

New... ► Project Wizard 

Start a new project, guided by the avalanche wizard (Ctrl+w).

New... ► Convert XYZ – > ASCII Grid

Convert laser scanning data into a ESRI ASCII grid.

Open... ► Input File 

Open an existing input file (*.av2) (Ctrl+O).

Open... ► Avalanche Simulation 

Open existing avalanche simulation (select the folder containing the simulation files) (Ctrl+A).

Close 

Close active file (input or output).

Save 

Save active file (Ctrl+S)

Save Copy As

Save a copy of the active file (e.g. *test.av2*) under a new name (e.g. *simulation1.av2*, works only in input mode). But, RAMMS stays with the active file (*test.av2*)!

Export... ► Export Display To Image File Create a Image of the active window in a chosen formate. You can chose the desired image format using the file extension (e.g. .png, .jpg, .gif, .tif etc.)

Export... ► Export GIF Animation Create a GIF Animation of the simulation (only in OUTPUT mode). Change GIF Animation Interval (s) in the Avalanche tab of the preferences.

Export... ► Export Display To EPS Create an EPS-file of the active window.

Backup... ► Backup RAMMS Version Make a backup of the current RAMMS version.

Backup... ► Backup Active Project Backup your active project. The user will be asked if he wants to include output files in the backup. This function is useful when having problems with a simulation. Make a backup and send the zip-file together with some explanations to ramms@slf.ch. Make sure that all your input data (release area shapefiles, MuXi files, domain files, etc...) is in the project folder.

Backup... ► Backup User-Defined Files/Folders Backup any folder or files you want.

Preferences  Change RAMMS preferences.

Log files... ► RAMMS Logfile (current) Show active RAMMS logfile.

Log files... ► RAMMS Logfile (last session) If RAMMS crashed, open this logfile

and copy/paste the content into an email to ramms@slf.ch.

Install CYGWIN/GRASS Install CYGWIN and GRASS.

Exit Exit RAMMS (Ctrl+Q).

Edit

This menu is used to edit colorbar and dataspace properties. It is active only in input mode.

Edit Colorbar Properties  Edit the colorbar properties.

Edit Dataspace Properties  Edit your dataspace properties.

Input

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

Global Parameters		Change the global parameters (avalanche return period, volume category)
Calculation Domain... ►	Draw New Calculation Domain	This activates the button to draw a new calculation domain. The mouse cursor changes to an arrow.
	Load Existing Domain	Load a existing calculation domain (*.dom) drawn and saved before
Release Area... ►	Draw New Release Areas	This activates the button to draw new release areas. The mouse cursor changes to an arrow.
	Load Existing Release Areas	Load an existing release area shapefile.
	View and Edit Release Areas	The mouse cursor changes to an arrow and you can select release areas to define the release height and to view release area information. This works only in 2D mode .
	Crop Release Area	If your release area shapefile consists of several polygons, you can crop some of them and create a new release shapefile.
Friction Values... ►	Load Existing MuXi File	Load afore created MuXi file (*.muxi.shp).
	Create New MuXi File (Automatic Procedure)	The DEM is analysed, classified and according to altitude, slope and curvature information, a new MuXi file is created.
	Show MuXi Classification	Shows the result of the MuXi classification.

Forest... ►	Draw New Forest Area	The mouse cursor changes to an arrow and you can draw new forest areas. Save the forest shapefile and then use the function below (Import Forest Area From SHAPEFILE) to import the forest cover.
	Show Active Forest Cover	If forest cover is taken into account, the corresponding shapefile is displayed. If your project uses no forest cover at the moment, RAMMS will tell you so.
	Import Forest Area From SHAPEFILE	If a forest shapefile has been drawn (using ) it can be imported using this function.
	Import Forest Area From ASCII Grid	If a forest ASCII grid is available, it can be imported using this function (0=no forest, 1 = forest).
	Remove Active Forest Cover	Remove the active forest raster data from the project.

Show

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

Show Lights	Show/hide light effects	
Show Grid	Show/hide computational grid	
Show Map	Show map	
Show Image	Show orthophoto/image	
Show Release Area/Simulation	Show/hide release area (input mode) or simulation results (output mode)	
Show Isotropic View	Switch between realistic (isotropic) and anisotropic view	
Show Colorbar	Show/hide colorbar	
Show Bottom Color	Show/hide 0-color	
Show Arrow	OUTPUT — Show/hide point arrow of time plot	
Show Line Profile	OUTPUT — Show/hide line of line profile	
Show Domain	Show/hide calculation domain area (only in input mode)	

Run

This menu is active only in **input mode**.

Run Avalanche Simulation Opens the module parameter window to change parameters and to start the calculation



Results

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

Flow Height		Shows flow height of the avalanche for every time step.
Flow Velocity		Shows flow velocity of the avalanche for every time step.
Flow Pressure		Shows flow pressure of the avalanche for every time step.
Flow Momentum		Shows flow momentum of the avalanche for every time step.
Max Values... ►	Max Flow Height	Displays the maximum flow height for each cell.
	Max Velocity	Displays the maximum velocity for each cell.
	Max Pressure	Displays the maximum pressure for each cell.
	Max Momentum	Displays the maximum momentum for each cell.
DEM Adaptations ►	Add Deposition to DEM	Adds the deposition of an avalanche simulation to a new DEM.
Flow Analysis... ►	Summary of Moving Mass	Summarizes the Moving Mass.
Friction Parameters ►	Mu	Display mu for this simulation.
	Xi	Display xi for this simulation.

GIS/GRASS

This menu contains GIS and GRASS functions.

Import Polygon Shapefile

Import a ESRI GIS polygon shapefile.



Convert Shapefile...

Converts a normal polygon shapefile into a RAMMS release file or a RAMMS forest file. This function makes only sense in the **input** mode.

Export Results As Shapefile

Export the active results to an ESRI GIS shapefile for later use in a GIS program.

Export Results As ASCII Grid

Export the active results to an ESRI ASCII Grid for later use in a GIS program.

Add Dam to DEM...

Adds a Dam to the DEM. You have to specify relative Dam height or absolute Dam elevation.

Show Slope Angle (°)

Display the slope angles.

Show Curvature (1/m)

Display the curvatures.

Show Contour Plot

Display a contour plot.

Resample Slope/Curvature

Normally, slope and curvatures are calculated for a grid resolution of 10m. You can change this resolution by using this function.

GRASS Advanced...

These are advanced GRASS functions. You should only use them if you know what you are doing.

GRASS Logfiles...

Logfiles of all the GRASS functions are provided here. If your MuXi file looks strange, have a look at the MuXi Logfile.

Extras

Add/Change or Remove map	Add or change the topographic map of your project. The maps have to be located in your distribution's 'Map' folder, see section 3.5 for details. If not, you can browse for the maps.
Add/Change or Remove Imagery	Add, change or remove the imagery used for visualization of your project. The images have to be located in your distribution's 'IMAGE' folder, see section 3.5 for details. If not, you can browse for the images.
Point...	Used to select points in OUTPUT mode, save point locations and export Time Plot Data.
Profile...	Used to draw a profile in OUTPUT mode, save profile points and export Profile Plot Data.
Save Active Position	Save your current state of view, as well as the enabled and disabled visualizations.
Reload Position	Reload your saved position.
Google Earth...	This function exports release areas, muxi files and your results to Google Earth. See Map Options for map settings for areas outside of Switzerland.
Map Options	Dialogue window to enter map settings (projection, datum, offsets...) for areas outside of Switzerland.

Project

This menu contains the project input and output logfiles.

Project Input Log

Displays the input logfile.

Project Output Log

Displays the output logfile. The input logfile is appended to the output logfile.

Help

Manuals... ►	User Manual (PDF)	RAMMS User Manual
	Friction Parameter Table (PDF)	RAMMS friction parameter table
RAMMS web...		RAMMS Homepage at http://ramms.slf.ch
Updates...		Download RAMMS updates manually or directly from the web
About RAMMS...		About dialog window
License Agreement...		RAMMS License Agreement
Release Notes		Information about the RAMMS release

6.1.2 Horizontal toolbar



Project wizard: open avalanche wizard for creating a new avalanche project.



Open input file.



Open simulation.



Save copy as: save the active file under a new name (input).



Close: close the active file (input and output).



Print: displays the Windows print manager.



Undo, Redo.



Arrow (move and resize), Rotate, Move.



Simulation Results: Choose this function and move the arrow over the topography → x-, y- and z-Coordinates of the mouse position are shown in the lower right status bar (see below).

OUTPUT | If you move the arrow over the simulation data, the active parameter is shown as well (see right value in the figure below). If you click once with the left mouse button at a point of interest, a new window pops up called 'RAMMS::Avalanche Time Plot <Active Parameter>'.



INPUT, 2D | Crop Release Areas: Click this button to make a selection of the release area polygons you want to crop.



INPUT, 2D | View and Edit Release Areas.



INPUT, 2D | Create new release area: specify new polygon-points by clicking with left mouse button, for the last polygon-point use a double-click to finish. The user is asked if he wants to draw more release areas. At last, he has to specify a new filename for the release area.



INPUT, 2D | Create new forest area: specify new polygon-points by clicking with left mouse button, for the last polygon-point use a double-click to finish. The user is asked if he wants to draw more forest areas. At last, he has to specify a new filename for the forest area.



INPUT, 2D | Create new domain area: specify a new domain polygon by clicking with left mouse button, for the last polygon-point use a double-click to finish. A dialogue box will then ask the user for a new domain name (e.g. test).



OUTPUT, 2D | Line Profile: Select the topography, until the Line-Profile-Button is active. Click the button and then move the cursor to the start point of your profile. Click the left mouse button and move the cursor to the next position of your profile. At the end position of your profile double click with the left mouse button. A new window pops up called 'RAMMS::Avalanche Line Profile Plot Active Parameter'. This line profile plot is linked to your simulation. If you change the parameter or if you change the max-value in the avalanche-panel, the changes are adapted in the line profile plot!



Zoom tools.



Annotation tools: text, line, rectangle, oval, polygon, freehand.

6.1.3 Vertical toolbar



Add shapefile (*.shp).



Show/hide lights.



Show/hide mesh.



Show map.



Show image.



INPUT | Show/hide release area (or other active parameter).

OUTPUT | Show/hide simulation.



Switch between isotropic and anisotropic view.



Show/hide colorbar.



INPUT | Run Simulation.

OUTPUT | Animate Simulation / Continue Simulation.

Stop/Pause Simulation ().



OUTPUT | End Simulation: skip to last dump step of simulation.



Create a screenshot of the main window.



OUTPUT | Create GIF animation.



Edit colorbar properties.



Edit dataspace properties.



Change RAMMS preferences (e.g. working directory).



Change view to 2D / Change view to 3D ().

6.1.4 Main window

All input and output related visualizations are displayed in the main window.

6.1.5 Time step slider (only OUTPUT)

The time step slider can be moved manually to change the active time step.

6.1.6 Left status bar

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

6.1.7 Right status bar

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

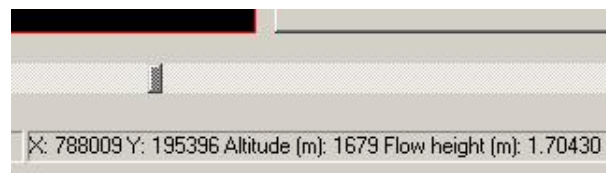


Figure 6.2: Position information and triangle simulation results in the right status bar.

6.1.8 Colorbar

In general, the colorbar appears at the right edge of the main window (see figure 6.1) and can be moved and resized (see exercise *"Editing the colorbar"* on page 49).

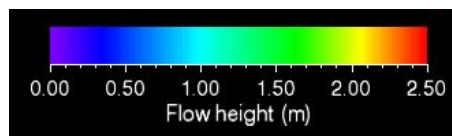


Figure 6.3: Colorbar

6.1.9 Panel

An avalanche panel is displayed on the right side of the RAMMS GUI (see fig. 6.4), and consists of four tabs (General, Display, Volumes and Region). This panel changes interactively, depending on what parameter is displayed in the main window. The current parameter (A-1) and all visualizations (A-2) are additional information/functions on the avalanche panel.

General Tab

The General tab (A) shows some important simulation parameters, such as: nr. of nodes, nr. of cells, end time (s), dump step (s), grid resolution (m) and avalanche density (kg/m³).

AVALANCHE

PARAMETER: Release height (m) **A**

General Display Volumes Region

Simulation parameters

Nr. of nodes: 9409

Nr. of cells: 9216

End time (s): 300

Dump step (s): 5.0

Grid resolution (m): 25.00

Density (kg/m3): 300

Choose additional parameter:

☒ Release: T1_release_area_RAMMS_rep.shp

☐ Mu: muxi_L100.shp

☐ Xi: muxi_L100.shp

☐ Shapefile

☒ Always show release area

☒ Show additional parameter

Min: 0.00

Max: 1.00

Nr of colors: 50

Parameter transparency (%): 0

Colorbar MIN and MAX values **B**

General Display Volumes Region

Min: 0.00

Max: 4.77

Nr of colors: 50

Transparency (%): 0

Hit ENTER to confirm!

Values < 0.095 are not displayed!!

Volumes and areas **C**

General Display Volumes Region

Proj. rel area (m2): 56117.3

Incl. rel area (m2): 71458.0

Release volume (m3): 71458.0

Release mass (t): 21437.4

Density (kg/m3): 300.0

Edit release areas

Project region information **D**

General Display Volumes Region

X extent (m): 781549.5 - 783949.5

Y extent (m): 187024.5 - 189424.5

Z extent (m): 1556.9 - 2530.6

Proj. area (km2): 5.760

Figure 6.4: Avalanche panel.

Display Tab

The Display tab (B) shows parameters that are important for the display (colors, transparency) of results and polygon shapefiles. The Min and Max values as well as the number of colors influence directly the colorbar and the visualization. The transparency of the simulation results can be changed on the avalanche-display-panel. 0% means no transparency, 100% means total transparency, see figure below (fig. 6.5). The colorbar is divided into n (nr. of colors) different colors, where the lowest color is normally not displayed. The bottom line informs the user of the range of the values that are not displayed in the current visualization (only in output mode).

Volumes Tab

The Volumes tab (C) gives the user information about the release area, such as projected release area, 3D release area, estimated release volume (input mode), release mass (input)

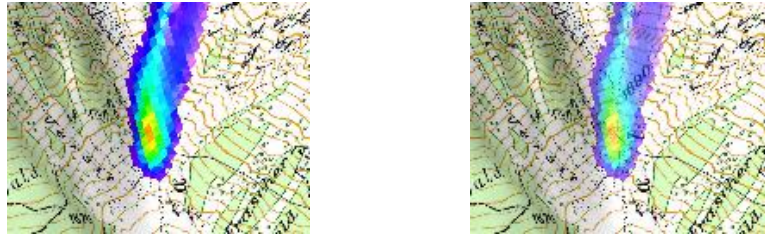


Figure 6.5: No transparency (left) and 40% transparency (right) of simulation result.

and calculated release volume (output). The input release volume is estimated. Normally, the calculated release area is 1% – 7% larger than the estimated values.

Region Tab

The Region Tab (D) gives information about min and max X-, Y-coordinates and the altitude limits as well as an information about the region area in km².

6.2 File management

6.2.1 Software RAMMS

After the installation of RAMMS, the installation directory contains the following folders:

<installation_directory>

- <bin>: This folder contains all RAMMS executable-files.
- <bmp>: Bitmap-files, used for toolbar buttons.
- <defaults>: This folder contains some default files.
- <grass>: This folder contains all GRASS shell scripts used for terrain analysis.
- <IDL71>: This folder contains the IDL Virtual Machine distribution.
- <Manual>: This folder contains the manuals and publications.
- <Temp>: This folder contains some temporary files and the CYGWIN/GRASS installation

Additionally, the installation directory contains also the following files:

<installation_directory>

- <ramms.ico>: The RAMMS icon.
- <ramms.sav>: The main compiled program file.
- <ramms_init.sav>: Additional compiled program file.
- <ramms.exe>: Start RAMMS with this exe-file.
- <ramms.ini>: The ini-file belongs to the ramms.exe-file.
- <uninstall.exe>: Uninstall RAMMS with this exe-file.
- <uninstall.ini>: The ini-file belongs to the uninstall.exe-file.

6.2.2 Organizing your data

Recommendation to organize your RAMMS-data (this is only a recommendation, but it proved to be a good way):

Create a folder 'RAMMS' (not in your installation directory, use another drive for your projects and data) e.g. *D:\RAMMS* and in there the following folders:

- Projects
- DEM
- FOREST
- Maps

- OrthoPhoto

It is important, that the DEM and forest files, as well as your georeferenced maps and imagery, are located in the appropriate folders. These folders are the ones selected in the RAMMS preferences (how to set the correct folders see section 3.5).

IMPORTANT: There should be no blanks or special characters in all the above directories. For example, do not specify a working directory like *C:\Documents and Settings\Eigene Dateien\Data* etc...this will not work for RAMMS.

List of Figures

3.1	Installation - Welcome dialog window.	6
3.2	Installation - Readme dialog window.	6
3.3	Installation - License agreement dialog window.	7
3.4	Installation - Destination directory dialog window.	7
3.5	Installation - Installing files dialog window.	8
3.6	Installation - Finished installing files dialog window.	8
3.7	Installation - Finished installation dialog window.	9
3.8	RAMMS icon.	9
3.9	RAMMS program group.	9
3.10	<i>RAMMS</i> start window.	11
3.11	RAMMS licensing window.	11
3.12	Enter user name and company name.	12
3.13	Personal license request file <i>RAMMS_request_Muster Test.txt</i>	12
3.14	Personal license key file <i>RAMMS_license_Muster Test.txt</i>	13
3.15	Continue Cygwin/GRASS installation?	13
3.16	Choose Cygwin/GRASS installation zip-file.	14
3.17	Choose Cygwin/GRASS installation directory.	15
3.18	Cygwin/GRASS - Extract files.	16
3.19	Cygwin/GRASS - Add to registry.	16
3.20	Cygwin/GRASS - Registered successfully.	16
3.21	Cygwin/GRASS - Automatic scripts.	17
3.22	Error message	17
3.23	IDL Visual Studio Merge Modules - Welcome dialog window.	18
3.24	IDL Visual Studio Merge Modules - Ready to install the Program.	18
3.25	IDL Visual Studio Merge Modules - Installing...	19
3.26	Installation - Destination directory dialog window.	19
3.27	General tab of RAMMS preferences.	21
3.28	Avalanche tab of RAMMS preferences.	21
3.29	RAMMS preferences.	22
3.30	Browse for the correct folder.	22

4.1	Project extent (area of interest) - topographic map ©2010 Swisstopo (BA091601).	24
4.2	Example ESRI ASCII Grid.	25
4.3	Example ASCII X,Y,Z single space data.	26
4.4	RAMMS Avalanche Project Wizard: Step 1 of 4.	28
4.5	Step 1 of the RAMMS project wizard: Project information.	29
4.6	Window to browse for a new project location.	29
4.7	Step 2 of the RAMMS project wizard: GIS information.	29
4.8	Project coordinates: lower left and upper right corner of project area - ©2010 swisstopo (BA091601).	30
4.9	Step 3 of the RAMMS project wizard: Project boundary coordinates. . . .	30
4.10	Step 4 of the RAMMS project wizard: Project summary.	31
4.11	Created project files.	34
4.12	"Active" project with lines and corners for resizing, topographic map ©2010 Swisstopo (BA091601)	36
4.13	"Active" project with rotation axes, topographic map ©2010 swisstopo (BA091601).	37
4.14	3D view of example model, topographic map ©2010 swisstopo (BA091601).	38
4.15	2D view of example model, topographic map ©2010 swisstopo (BA091601).	38
4.16	RAMMS global parameters.	39
4.17	Project with emerging release area.	41
4.18	Release area and volume information.	43
4.19	Release area information window.	44
4.20	Calculation domain in green confines the area of interest and saves calcula- tion time.	45
4.21	RAMMS Voellmy-fluid friction model.	46
4.22	RAMMS — Automatic MuXi Procedure.	47
4.23	The avalanche 'Display' panel.	49
4.24	The colorbar properties window.	50
4.25	RAMMS Project Input Log Information.	53
4.26	Region extent (X-, Y- and Z-Coordinates, total area).	54
4.27	Window to choose map image.	55
5.1	General information.	58
5.2	Calculation parameters	59
5.3	Friction values MuXi.	59
5.4	Release information.	59
5.5	Stop criteria.	60
5.6	Status window of calculation.	61

5.7	Main window in output mode, topographic map ©2010 Swisstopo (BA091601).	61
5.8	MuXi settings for a variable calculation.	62
5.9	Results: Maximum values of flow height (left), velocity (middle) and pressure (right), topographic map ©2010 Swisstopo (BA091601).	64
5.10	Line profile plot.	65
5.11	Line profile perpendicular to flow direction, topographic map ©2010 Swisstopo (BA091601).	66
5.12	Line profile along the avalanche path, topographic map ©2010 Swisstopo (BA091601).	66
5.13	Time plot window.	67
5.14	Project before (top) and after (bottom) creating a dam, topographic map ©2010 Swisstopo (BA091601).	71
5.15	Avalanche beeing stopped by a dam, topographic map ©2010 Swisstopo (BA091601).	71
5.16	About RAMMS...	75
6.1	Graphical user interface (GUI).	78
6.2	Position information and triangle simulation results in the right status bar.	92
6.3	Colorbar	92
6.4	Avalanche panel.	93
6.5	No transparency (left) and 40% transparency (right) of simulation result.	94

List of Exercises

3.5.a: Working directory	22
4.3.a: How to create a new project.	28
4.4.a: Moving and resizing the model	36
4.4.b: Rotating the model	37
4.4.c: How to switch between 2D and 3D mode	38
4.6.a: How to create a new release area.	41
4.6.b: How to load an existing release area.	42
4.6.c: Specify release height and view release information	44
4.8.a: How to create a new MuXi file.	48
4.8.b: How to load an existing MuXi file.	48
4.9.a: Editing the colorbar	49
4.10.a: How to create a new FOREST file.	52
4.12.a: How to add or change maps.	55
4.12.b: How to add or change remote sensing imagery.	56
5.1.a: How to run a constant avalanche calculation.	58
5.1.b: How to run a variable MuXi avalanche calculation.	62
5.2.a: Displaying max values.	64
5.2.b: How to draw a line profile.	65
5.2.c: How to create a time plot.	67
5.2.d: Enter point coordinates and get a time plot.	68
5.3.a: How to create a new DEM to simulate a dam.	70
5.3.b: How to add avalanche deposition to a new DEM.	72

5.4.a: How to save input files and program settings manually.	73
5.5.a: How to open an input file.	74
5.5.b: How to open an output file/avalanche simulation.	74
5.5.c: How to load an optional shapefile.	74

Index

A

Add Dam to DEM, [86](#)
Add/Change Image, [87](#)
Add/Change Map, [87](#)
Avalanche wizard, [28](#)

B

Backup Active Project, [79](#)
Backup RAMMS Version, [79](#)
Backup User-Defined Files/Folders, [79](#)

C

Calculation Domain, [45](#)
Colorbar, [92](#)
 Edit, [49](#)
Convert Shapefile..., [86](#)
Create New MuXi File (Automatic Procedure), [81](#)
Crop Release Areas, [81](#)

D

Dam, [70](#), [72](#)
DEM
 Create new, [70](#), [72](#)
Draw New Calculation Domain, [81](#)
Draw New Forest Area, [82](#)
Draw New Release Area, [81](#)
dump step slider, [91](#)

E

Edit Colorbar Properties, [80](#)
Edit Dataspace Properties, [80](#)

Exit, [80](#)

Export Display To EPS, [79](#)
Export Display To Image File, [79](#)
Export GIF Animation, [79](#)
Export Result To Google Earth, [87](#)
Export Results As ASCII Grid, [86](#)
Export Results As Shapefile, [86](#)

F

File management, [95](#)
Files
 *.tfw, [55](#)
 world, [55](#)
Flow Height, [85](#)
Flow height, [63](#)
 Max flow height, [63](#)
Flow Momentum, [85](#)
Flow momentum, [63](#)
 Max momentum, [63](#)
Flow Pressure, [85](#)
Flow pressure, [63](#)
 Max pressure, [63](#)
Flow Velocity, [85](#)
Flow velocity, [63](#)
 Max velocity, [63](#)
Forest, [51](#)
 Create FOREST-file, [51](#)
 Influence on friction parameters, [51](#)
Forest information, [25](#)
Friction parameter, [46](#)
 Create new MuXi file, [47](#)

- Load MuXi file, [47](#)
- Mu, [46](#)
- Xi, [46](#)
- Friction Parameters..., [85](#)

G

- Global Parameters, [81](#)
- Global parameters, [26](#), [39](#)
- Graphical User Interface, [78](#)
 - Colorbar, [78](#)
 - Main window, [78](#)
 - Menu bar, [78](#)
 - Edit, [80](#)
 - Extras, [87](#)
 - GIS/GRASS, [86](#)
 - Help, [88](#)
 - Input, [80](#)
 - Project, [88](#)
 - Results, [84](#)
 - Run, [84](#)
 - Show, [83](#)
 - Track, [78](#)
 - Panel, [78](#)
 - Status bar, [78](#)
 - Time step slider, [78](#)
 - Toolbar buttons, [78](#)
- GRASS Advanced..., [86](#)
- GRASS Logfiles..., [86](#)

I

- Import Forest Area From ASCII Grid, [82](#)
- Import Forest Area From SHAPEFILE, [82](#)
- Import GIS Polygon Shapefile, [86](#)
- Install CYGWIN/GRASS, [80](#)
- Installation, [5](#)

L

- Licensing Methods, [10](#)

- Load
 - MuXi file, [47](#)
 - Release file, [40](#)
- Load Existing Domain, [81](#)
- Load Existing MuXi File, [81](#)
- Load Existing Release Area, [81](#)

M

- Main window, [91](#)
- Map
 - Add, [55](#)
 - Change, [55](#)
 - Load new, [55](#)
- Map Options, [87](#)
- Max Flow Height, [85](#)
- Max Momentum, [85](#)
- Max Pressure, [85](#)
- Max Velocity, [85](#)
- Move, [36](#)

O

- Open
 - Input file, [74](#)
 - Output file, [74](#)
 - Shapefile, [74](#)
- Orthophoto
 - Add, [55](#)
 - Change, [55](#)
 - Load new, [56](#)

P

- Panel
 - Avalanche Panel, [92](#)
 - Display, [92](#)
 - General, [92](#)
 - Region, [92](#)
 - Transparency, [93](#)
 - Volumes, [92](#)
- Point..., [87](#)

-
- Preferences, [20](#), [22](#), [79](#)
 - Set DEM directory, [20](#)
 - Set FOREST directory, [20](#)
 - Set Map directory, [20](#)
 - Set Orthophoto directory, [20](#)
 - Set Working directory, [20](#), [22](#)
 - Profile..., [87](#)
 - Project
 - Create new, [28](#)
 - General, [23](#)
 - Information, [53](#)
 - Input data, [24](#)
 - Model input data, [24](#)
 - Scenarios, [23](#)
 - Wizard, [27](#)
 - R**
 - RAMMS Logfile (current), [79](#)
 - RAMMS Logfile (last session), [79](#)
 - Release area, [40](#)
 - Create manually, [40](#)
 - Load, [40](#)
 - Release information, [44](#)
 - shapefile, [40](#)
 - Release information, [26](#)
 - Reload Position, [87](#)
 - Remove Active Forest Cover, [82](#)
 - Resample Slope/Curvature, [86](#)
 - Resize, [36](#)
 - Results, [63](#)
 - Flow height, [63](#)
 - Flow momentum, [63](#)
 - Flow pressure, [63](#)
 - Flow velocity, [63](#)
 - Line profile, [63](#), [64](#)
 - Max values, [63](#), [64](#)
 - Time plot, [64](#)
 - Run calculation
 - Calculation Domain, [45](#)
 - Constant, [57](#)
 - Variable, [63](#)
 - S**
 - Save
 - automatically, [73](#)
 - manually, [73](#)
 - Save Active Position, [87](#)
 - Shapefile
 - Load, [74](#)
 - Show Active Forest Cover, [82](#)
 - Show Contour Plot, [86](#)
 - Show Curvature, [86](#)
 - Show MuXi Classification, [81](#)
 - Show Slope Angle, [86](#)
 - Status bar
 - Left, [92](#)
 - Right, [92](#)
 - System Requirements, [5](#)
 - T**
 - TFW-files, [55](#)
 - Toolbar
 - Horizontal, [89](#)
 - Vertical, [91](#)
 - Transparency, [93](#)
 - V**
 - View and Edit Release Area, [81](#)
 - W**
 - world-files, [55](#)