



**AvaFrame**

**AvaFrame developers**

**Feb 15, 2024**



# GENERAL

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>General</b>	<b>3</b>
2.1	Introduction	3
2.2	Installation	3
2.2.1	Operational Installation	4
2.2.1.1	Requirements	4
2.2.1.2	Setup AvaFrameConnector and run	4
2.2.1.3	Update Avaframe to a new release	4
2.2.2	Operationelle Installation (Deutsch)	5
2.2.2.1	Vorraussetzungen	5
2.2.2.2	Setup AvaFrameConnector	5
2.2.2.3	Update Avaframe auf eine neue Version	5
2.3	QGIS AvaFrameConnector	5
2.3.1	Operational	6
2.3.2	Experimental	6
2.3.3	Admin	6
2.4	Advanced Usage (Script)	7
2.4.1	Advanced (Script) Installation	7
2.4.1.1	Requirements	7
2.4.1.2	Setup AvaFrame	7
2.4.1.3	Update AvaFrame	8
2.4.2	First run	8
2.4.3	Workflow example	9
2.4.3.1	Initialize project	9
2.4.3.2	Input data	9
2.4.3.3	Building your run script	9
2.5	Configuration	10
2.5.1	Override configuration	11
2.5.2	Logging	11
2.5.3	Example runscripts	12
2.5.3.1	Derive input data	12
2.5.3.2	Create a new project	12
2.5.3.3	Generate idealized/generic topography data	12
2.5.3.4	Postprocessing	12
2.5.3.5	Visualisation	12
2.5.3.6	Testing	12
2.6	Testing	13
2.6.1	Tests for model verification	13
2.6.1.1	Rotational Symmetry	13

2.6.1.2	Dambreak problem	13
2.6.1.3	Similarity Solution	13
2.6.2	Tests for model validation	14
2.6.2.1	Idealized test cases	14
2.6.2.2	Real-world test cases	22
2.7	Data Visualisation	27
2.8	Frequently Asked Questions	28
2.8.1	Can the spatial resolution of simulations performed with com1DFA (dense flow) be changed?	28
2.9	Release Notes	29
2.9.1	1.3 (12. Okt 2022)	29
2.9.2	1.2 (07. July 2022)	29
2.9.3	1.1 (19. May 2022)	30
2.9.4	1.0.1 (20. April 2022)	31
2.9.5	1.0 (6. April 2022)	31
2.9.6	v0.6 (24. September 2021)	33
2.9.7	v0.5 (13. July 2021)	34
2.9.8	v0.4.1 (9. June 2021)	35
2.9.9	v0.4 (8. June 2021)	35
2.9.10	v0.3 (26. April 2021)	35
2.9.11	v0.2 (28. Dezember 2020)	35
2.9.12	v0.1 (06 November 2020)	36
2.10	API Reference	36
2.10.1	Computational Modules	36
2.10.1.1	com1DFA	36
2.10.1.2	com2AB	50
2.10.1.3	com3Hybrid	53
2.10.2	Input/Transformation Modules	53
2.10.2.1	in1Data	53
2.10.2.2	in2Trans	62
2.10.2.3	in3Utils	68
2.10.3	Analysis/Output/Helper Modules	109
2.10.3.1	ana1Tests	109
2.10.3.2	ana3AIMEC	130
2.10.3.3	ana4Stats	136
2.10.3.4	ana5Utils	144
2.10.3.5	log2Report	157
2.10.3.6	out1Peak	160
2.10.3.7	out3Plot	161
2.10.3.8	tmp1Ex.tmp1Ex	203
2.11	Development	203
2.11.1	Notes to developers	203
2.11.2	Our suggested git workflow	204
2.11.3	Build the documentation	205
2.11.4	How to test code	205
2.11.5	How to add a benchmark test	205
<b>3</b>	<b>Computational modules</b>	<b>207</b>
3.1	com1DFA: DFA-Kernel	207
3.1.1	Input	207
3.1.1.1	Release-, entrainment thickness settings	208
3.1.1.2	DEM input data	209
3.1.1.3	Dam input	209
3.1.2	Model configuration	210
3.1.3	Output	210



3.1.4	Parallel computation	211
3.1.5	To run	211
3.1.6	Theory	211
3.1.7	Numerics	212
3.2	com1DFAOrig: Original DFA-Kernel	212
3.2.1	C++ Executable	212
3.2.2	Input	213
3.2.3	Output	214
3.2.4	To run	214
3.2.5	Theory	214
3.2.6	Numerics	214
3.3	com2AB: Alpha Beta Model	215
3.3.1	Input	215
3.3.2	Outputs	215
3.3.3	To run	215
3.3.4	Theory	215
3.3.5	Procedure	217
3.3.6	Configuration parameters	217
3.4	com3Hybrid: Hybrid modeling	218
3.4.1	Input	218
3.4.2	Outputs	218
3.4.3	To run	219
3.4.4	Procedure	219
3.4.5	Configuration parameters	219
3.5	com4FlowPy: Flow-Py	220
3.5.1	Running the Code	220
3.5.2	Configuration	220
3.5.3	Input Files	221
3.5.4	Output	221
3.6	com5SnowSlide: Snow slide	221
3.6.1	Input	222
3.6.2	Initialization of bonds	222
3.6.3	To run	222
3.7	com6RockAvalanche: Rock Avalanche	223
3.7.1	Input	223
3.7.2	To run	223
<b>4</b>	<b>Input/transformation modules</b>	<b>225</b>
4.1	in1Data: Input data utilities	225
4.1.1	Get input data	225
4.1.1.1	getInputCom1DFA	225
4.1.2	computeFromDistribution	225
4.1.2.1	To run	225
4.2	in2Trans: Transformation Utilities	226
4.2.1	Working with ASCII files	226
4.2.2	Working with shapefiles	226
4.2.2.1	Reading shapefiles	226
4.3	in3Utils: Various Utilities Modules	226
4.3.1	geoTrans	226
4.3.2	Generate Topography	226
4.3.2.1	To run generateTopo	227
4.3.2.2	Theory	227
4.3.2.3	Configuration parameters	227
4.3.3	Get Release Area	229

4.3.3.1	To run getReleaseArea . . . . .	229
4.3.4	Initialize Project . . . . .	230
4.3.4.1	To run . . . . .	230
4.3.5	fileHandlerUtils . . . . .	230
<b>5</b>	<b>Analysis/helper modules</b>	<b>231</b>
5.1	ana1Tests: Testing . . . . .	231
5.1.1	Deviation/difference/error computation . . . . .	231
5.1.1.1	Uniform norm . . . . .	231
5.1.1.2	Euclidean norm . . . . .	231
5.1.2	Dambreak test . . . . .	232
5.1.2.1	To run . . . . .	233
5.1.3	Similarity solution . . . . .	233
5.1.3.1	To run similarity solution . . . . .	234
5.1.4	Energy line test . . . . .	235
5.1.4.1	Theory . . . . .	235
5.1.4.2	Procedure . . . . .	238
5.1.4.3	To run energy line test . . . . .	238
5.1.5	Rotation test . . . . .	238
5.1.5.1	Inputs . . . . .	239
5.1.5.2	To run rotation test . . . . .	241
5.2	ana3AIMEC: Aimec . . . . .	241
5.2.1	Inputs . . . . .	241
5.2.1.1	Optional inputs . . . . .	242
5.2.2	Outputs . . . . .	242
5.2.3	To run . . . . .	242
5.2.4	Theory . . . . .	243
5.2.4.1	Mean and max values along thalweg . . . . .	243
5.2.4.2	Runout point . . . . .	244
5.2.4.3	Runout length . . . . .	244
5.2.4.4	Mean and max indicators . . . . .	244
5.2.4.5	Area indicators . . . . .	244
5.2.4.6	Mass indicators . . . . .	245
5.2.5	Procedure . . . . .	245
5.2.5.1	Defining the reference simulation . . . . .	245
5.2.5.2	Perform thalweg-domain transformation . . . . .	245
5.2.5.3	Assign data . . . . .	246
5.2.5.4	Analyze results . . . . .	246
5.2.5.5	Plot and save results . . . . .	246
5.2.5.6	Domain overview plots . . . . .	246
5.2.5.7	Analysis summary plots . . . . .	248
5.2.5.8	Analysis on simulation level plots . . . . .	251
5.2.5.9	Comparison to reference simulation plots . . . . .	253
5.2.5.10	List of Aimec result variables . . . . .	253
5.2.6	Configuration parameters . . . . .	255
5.3	ana4Stats: Statistical analysis tools . . . . .	258
5.3.1	probAna - Probability maps . . . . .	258
5.3.1.1	To run - via QGIS Connector . . . . .	258
5.3.1.2	To run - example run scripts . . . . .	258
5.3.1.3	Theory . . . . .	260
5.3.2	getStats . . . . .	260
5.3.2.1	To run . . . . .	260
5.4	ana5Utils . . . . .	260
5.4.1	distanceTimeAnalysis: Visualizing the temporal evolution of flow variables . . . . .	260

5.4.1.1	To run	263
5.4.1.2	Theory	263
5.4.1.3	Thalweg-time diagram	263
5.4.1.4	Simulated Range-Time diagram	265
5.4.2	Automated path generation	265
5.4.2.1	Input	265
5.4.2.2	Outputs	265
5.4.2.3	To run automated path	265
5.4.2.4	Theory automated path	266
5.5	out1Peak: Peak plots	267
5.5.1	Plot all peak fields	267
5.5.2	Plot all fields	267
5.6	out3Plot: Plots	268
5.6.1	plotUtils	268
5.6.1.1	makeColorMap	268
5.6.2	outQuickPlot	269
5.6.2.1	generatePlot	269
5.6.2.2	quickPlotBench	271
5.6.2.3	quickPlotSimple	271
5.6.2.4	generateOnePlot	272
5.6.2.5	quickPlotOne	272
5.6.3	in1DataPlots	273
5.6.4	statsPlots	273
5.6.4.1	plotValuesScatter	273
5.6.4.2	plotValuesScatterHist	273
5.6.4.3	plotHistCDFDiff	273
5.6.5	particle analysis plots	273
5.6.5.1	To run	277
5.7	log2Reports: Create Reports	277
5.7.1	Generate Report	277
5.7.2	Generate Compare Report	279
<b>6</b>	<b>Theory</b>	<b>281</b>
6.1	com1DFA DFA-Kernel theory	281
6.1.1	Governing Equations for the Dense Flow Avalanche	281
6.1.1.1	Mass balance:	281
6.1.1.2	Momentum balance:	281
6.1.1.3	Boundary conditions:	282
6.1.1.4	Choice of the coordinate system:	282
6.1.1.5	Thickness averaged equations:	282
6.1.1.6	Non-dimensional Equations	285
6.1.1.7	Friction Model	287
6.1.1.8	Dam	290
6.2	com1DFA DFA-Kernel numerics	291
6.2.1	Discretization	291
6.2.1.1	Space discretization	291
6.2.1.2	Time step	291
6.2.2	Mesh and interpolation	291
6.2.2.1	Mesh	291
6.2.2.2	Interpolation	293
6.2.3	Neighbor search	294
6.2.4	SPH gradient	297
6.2.4.1	Method	297
6.2.5	Particle splitting and merging	299

6.2.5.1	Split ( <b>default</b> )	300
6.2.5.2	Split and merge	300
6.2.6	Artificial viscosity	300
6.2.6.1	SAMOS Artificial viscosity	300
6.2.6.2	Ata Artificial viscosity	301
6.2.7	Forces discretization	301
6.2.7.1	Lateral force	301
6.2.7.2	Bottom friction force	302
6.2.7.3	Added resistance force	302
6.2.7.4	Entrainment force	302
6.2.8	Adding forces	303
6.2.8.1	Adding artificial viscosity	303
6.2.8.2	Adding entrainment	303
6.2.8.3	Adding driving forces	303
6.2.8.4	Adding friction forces	303
6.3	com1DFA Algorithm and workflow	304
6.3.1	Algorithm graph	304
6.3.2	Initialization:	306
6.3.2.1	Initialize Mesh	306
6.3.2.2	Initialize release, entrainment and resistance areas	306
6.3.2.3	Initialize Dam	306
6.3.2.4	Initialize particles	306
6.3.2.5	Initialize fields	308
6.3.3	Time scheme and iterations:	308
6.3.4	Compute Forces:	309
6.3.4.1	Artificial viscosity	309
6.3.4.2	Compute friction forces	309
6.3.4.3	Compute driving force	310
6.3.4.4	Take entrainment into account	310
6.3.4.5	Compute lateral pressure forces	310
6.3.5	Update position	310
6.3.5.1	Take gravity and lateral pressure forces into account	310
6.3.5.2	Take friction into account	311
6.3.5.3	Update particle position	311
6.3.5.4	Take dam interaction into account	311
6.3.5.5	Correction step:	311
6.3.6	Add secondary release area	312
6.3.7	Update fields	312
6.3.7.1	Update particles flow thickness	312
6.3.8	Simulation outputs	312
6.4	com4FlowPy Theory	312
<b>7</b>	<b>References</b>	<b>317</b>
7.1	Data sources	317
7.2	Glossary	317
	<b>Bibliography</b>	<b>321</b>
	<b>Python Module Index</b>	<b>323</b>
	<b>Index</b>	<b>325</b>

## INTRODUCTION

AvaFrame is a cooperation between the Austrian Avalanche and Torrent Service (Wildbach- und Lawinenverbauung; WLW) and the Austrian Research Centre for Forests (Bundesforschungszentrum für Wald; BFW) within the Federal Ministry Republic of Austria: Agriculture, Regions and Tourism (BMLRT).

Our mission is to develop an open source framework (licensed with the European Union Public license (EUPL) ) for avalanche simulations which replicates and simplifies current simulation procedures and models at the WLW and make them accessible to the public domain.

Our aims are to make AvaFrame:

- applicable in an operational setting e.g. for hazard mapping or planning of mitigation measures within the WLW and civil engineering.
- extensible for scientific applications to be used to develop new methods, models and cover a wide range of scientific topics.
- easily usable and extendable by (inter-)national partners, academia and interested users.

AvaFrame is part of an BMLRT initiative to foster technological innovation within a strategic focus on the area of Austrian safety policies, digital innovations and climate mitigation strategies.

Information in German can be found on the [BMLRT Homepage](#).

A summary of the latest advances of the project can be found on the [Avaframe homepage](#) web-page.

Feel free to contact us at our [matrix room](#) if you have any questions or suggestions. AvaFrame is in an early development phase and all ideas and contributions are welcome!

If you want to cite our project, please use:



## 2.1 Introduction

AvaFrame is a cooperation between the Austrian Avalanche and Torrent Service (Wildbach- und Lawinenverbauung; WLV) and the Austrian Research Centre for Forests (Bundesforschungszentrum für Wald; BFW) within the Federal Ministry Republic of Austria: Agriculture, Regions and Tourism (BMLRT).

Our mission is to develop an open source framework (licensed with the European Union Public license (EUPL) ) for avalanche simulations which replicates and simplifies current simulation procedures and models at the WLV and make them accessible to the public domain.

Our aims are to make AvaFrame:

- applicable in an operational setting e.g. for hazard mapping or planning of mitigation measures within the WLV and civil engineering.
- extensible for scientific applications to be used to develop new methods, models and cover a wide range of scientific topics.
- easily usable and extendable by (inter-)national partners, academia and interested users.

AvaFrame is part of an BMLRT initiative to foster technological innovation within a strategic focus on the area of Austrian safety policies, digital innovations and climate mitigation strategies.

Information in German can be found on the [BMLRT Homepage](#).

A summary of the latest advances of the project can be found on the [Avaframe homepage](#) web-page.

Feel free to contact us at our [matrix room](#) if you have any questions or suggestions. AvaFrame is in an early development phase and all ideas and contributions are welcome!

If you want to cite our project, please use:

## 2.2 Installation

---

**Note:** There are two ways to install and use AvaFrame:

### Operational (GUI)

If you want the operational workflow, running *com1DFA* (dense flow avalanche) and *com2AB* (alpha beta) with standard settings from within *QGis*, head over to the [Operational Installation \(Deutsche Version\)](#) section. Use this if you want to:

- use the standard, well tested and calibrated setup for hazard mapping or similar

- use QGis as frontend
- have results in a short amount of time
- use the latest release

### Advanced (Script)

If you want to contribute and develop AvaFrame, head over to [Advanced \(Script\) Installation](#). Use this if you want to:

- work on the code itself
  - implement new features
  - change/improve existing code
  - have the latest development code. *Warning: might be unstable!*
- 

## 2.2.1 Operational Installation

This is the quick start for the operational AvaFrame setup with QGis as frontend.

### 2.2.1.1 Requirements

The prerequisites are:

- QGis: install from here: [QGis installation](#) (we recommend using the latest version)

### 2.2.1.2 Setup AvaFrameConnector and run

1. Open QGis from your start menu and go to Plugins -> Manage and Install Plugins
2. Search for *AvaFrameConnector* and install it
3. Wait for it to finish and restart QGis
4. Access the QGis - Avaframe connector via Toolbox -> AvaFrame -> Operational -> FullOperationalRun
5. Add the described data and run. Results will be loaded after a while (depending on the size of your DEM)

This tries to install the AvaFrame core package as well (since AvaFrameConnector version 1.2). If it fails, head over to the operational:Manual Installation instructions.

For more information about the functions in the AvaFrameConnector, see [connector:AvaFrameConnector \(GUI\)](#)

### 2.2.1.3 Update Avaframe to a new release

1. Restart/Open QGis from your start menu and go to Plugins -> Manage and Install Plugins
  2. Search for *AvaFrameConnector* and check whether it also needs updating
  3. Access the QGis - Avaframe connector via Toolbox -> AvaFrame
  4. Go to the *Admin* section, choose and run *Update*
-



## 2.2.2 Operationelle Installation (Deutsch)

Dies ist der Schnellstart für die Verwendung von AvaFrame mit QGIS als Frontend.

### 2.2.2.1 Voraussetzungen

Die Voraussetzungen sind:

- QGIS: [QGIS Installation \(download\)](#) (Wir empfehlen die aktuellste Version zu verwenden)

### 2.2.2.2 Setup AvaFrameConnector

1. Öffnen Sie QGIS über Ihr Startmenü und gehen Sie zu Erweiterungen -> Erweiterungen verwalten und installieren
2. Nach *AvaFrameConnector* suchen und installieren
3. Warten Sie, bis der Vorgang abgeschlossen ist und starten Sie QGIS neu
4. Rufen Sie den QGIS - AvaframeConnector über Verarbeitungswerkzeuge -> AvaFrame -> Operational -> FullOperationalRun auf
5. Fügen Sie die beschriebenen Daten hinzu und starten Sie den Connector. Die Ergebnisse werden nach einer Weile geladen (abhängig von der Größe Ihres DEM)

Hiermit wird versucht das AvaFrame-Kernpaket zu installieren (seit AvaFrameConnector Version 1.2). Wenn dies fehlschlägt, gehen Sie zu: `operationalGerman:Manuelle Installation`.

Für eine Kurzzusammenfassung der Funktionen des AvaFrameConnector, siehe `connector:AvaFrameConnector (GUI)`.

### 2.2.2.3 Update Avaframe auf eine neue Version

1. Starten Sie QGIS neu/öffnen Sie es über Ihr Startmenü und gehen Sie zu Plugins -> Plugins verwalten und installieren
2. Suchen Sie nach AvaFrameConnector und prüfen Sie, ob es aktualisiert werden muss
3. Rufen Sie den QGIS - AvaframeConnector über Verarbeitungswerkzeuge -> AvaFrame auf
4. Dann -> Admin -> Update aufrufen

## 2.3 QGIS AvaFrameConnector

The AvaFrameConnector allows QGIS users to access certain base workflows directly from QGIS. The connector only provides the interface to functions within the AvaFrame python package, and is developed separately, see the [QGISAF github repository](#). It makes use of the QGIS processing plugin, which is included in all current QGIS releases.

---

**Note:** ALL (!! ) data provided to the functions below HAVE to be in the same projection. It does not matter which projection it is as long as it is the same one.

---

### 2.3.1 Operational

**Alpha Beta (com2)**

Runs the alpha beta calculation via module com2AB. For more specific info about the inputs (shapefile attributes etc), see *Input*.

**Dense Flow (com1)**

Runs dense flow avalanche module com1DFA. For more specific info about the inputs (shapefile attributes etc), see *Input*. You can select multiple shapefiles for release areas, each will be calculated as one scenario. Each shapefile can contain multiple polygons. If you provide entrainment/resistance areas, each scenario will be calculated once without them (i.e null simulation) and once with them.

**Operational Run (com1 and com2)**

Runs com1DFA and additionally com2AB if profile and splitpoint are set. Additional info see description of *Dense Flow Standard (com1)* and *Alpha Beta (com2)*.

### 2.3.2 Experimental

These either require deeper knowledge about the AvaFrame python package, i.e. configuration files etc., or are not yet fully tested and might produce unwanted results.

**Layer rename**

Renames com1DFA result layers by adding the values of the given variable (from the configuration file)

**Probability analysis for directory (ana4)**

Runs the probability analysis on com1DFA results from a given avalanche directory without running any simulations. Please provide the base avalanche directory (i.e. the one containing INPUT, OUTPUT, \*.log etc)

**Probability run (ana4, com1)**

Runs probability simulations via module com1DFA. The release shape HAS TO HAVE a ci95 field containing the 95 percentile confidence interval (same unit as the release thickness). Multiple scenarios can be provided, final map includes variations from all scenarios combined. Release thickness and SamosAT friction mu are being varied, 40 variations per scenario.

**Release area stats(in1, com1)**

Returns info for release area statistics in a csv file. See output of QGis processing for the location of the file.

**Rock Avalanche (com6)**

Runs rock avalanche simulations via module com1DFA. For more info see com6RockAvalanche section in the documentation. Note that a release thickness raster the same size as the DEM is needed.

**Snow slide (com5)**

Runs snow slide simulations via module com1DFA. For more info see com5SnowSlide section in the documentation. The resistance layer is meant for building outlines.

### 2.3.3 Admin

**GetVersion**

Displays the current version of the AvaFrame python package (NOT the AvaFrameConnector)

**UpdateAvaFrame**

Updates the AvaFrame python package

## 2.4 Advanced Usage (Script)

After the installation, the next sections describe on how to get started. The sections configuration and logging describe the general methods we use, this is helpful to understand how you can change model parameters and similar.

### 2.4.1 Advanced (Script) Installation

This is a quick guide on how to install AvaFrame and the required dependencies on your machine. AvaFrame is developed on **Linux machines** (Ubuntu/Manjaro/Arch) with recent Python versions > 3.8. These instructions assume you are familiar with working in a terminal. This guide is currently described for *Linux* only, but expert users should be able to adapt it to *Windows*.

#### 2.4.1.1 Requirements

Install [git](#) and python, we suggest to work with miniconda/anaconda. For installation see [miniconda](#) or [anaconda](#). Some operating systems might require the python headers (e.g python-dev on ubuntu) or other supporting libraries/packages (e.g. Visual Studio on Windows).

#### 2.4.1.2 Setup AvaFrame

Create a new [conda environment](#) for AvaFrame, activate it and install pip, numpy and cython in this environment:

```
conda create --name avafame_env
conda activate avafame_env
conda install pip numpy cython
```

Clone the AvaFrame repository (in a directory of your choice: [YOURDIR]) and change into it:

```
cd [YOURDIR]
git clone https://github.com/avafame/AvaFrame.git
cd AvaFrame
```

Compile the cython com1DFA part. You might also have to install a c-compiler (gcc or similar) through your systems package manager:

```
python setup.py build_ext --inplace
```

**Warning:** You will have to do this compilation every time something changes in the cython code. We also suggest to do this everytime updates from the repositories are pulled.

Install avafame and its requirements:

```
pip install -e .
```

This installs avafame in editable mode, so every time you import avafame the current (local) version will be used.

Test it by starting python and do an `import avafame`. If no error comes up, you are good to go.

Head over to gettingstarted:First run for the next steps.

### 2.4.1.3 Update AvaFrame

To update go to your avaframe repository [YOURDIR]/Avaframe, pull the latest changes via:

```
git pull
```

and repeat the compilation step from above:

```
python setup.py build_ext --inplace
```

If there are updates on the requirements inside `setup.py`, it might be necessary to run:

```
pip install -e .
```

again to get the additional requirements installed.

---

## 2.4.2 First run

Follow these steps to run your first simulation:

- change into your AvaFrame directory (replace [YOURDIR] with your path from the installation steps):

```
cd [YOURDIR]/AvaFrame/avaframe
```

- run:

```
python runCom1DFA.py
```

- a similar output should show up:

```
logUtils - INFO - Started logging at: 03.11.2020 22:42:04
logUtils - INFO - Also logging to: data/avaParabola/runCom1DFA.log
runCom1DFA - INFO - MAIN SCRIPT
runCom1DFA - INFO - Current avalanche: data/avaParabola
...
```

This will perform a dense flow avalanche simulation using the `com1DFA` module. The results are saved to `data/avaParabola/Outputs/com1DFA`. For a first look at the results, go to the folder `reports`, there you can find a markdown report of the simulations performed including some plots of the results.

To display markdown files in a nice way use a markdown viewer of your choice. Some other options are:

- Use the Atom editor with a markdown plugin
- If you have *pandoc* installed use this to convert it to pdf/html
- Some browsers have markdown extensions you can install easily

### 2.4.3 Workflow example

The following example should make it easier for you to find your way in AvaFrame and setup your own AvaFrame workflow after you did the full setup. There is also a directory with examples for different workflows, see more here: [Example runscripts](#).

Make sure you change to your AvaFrame directory by:

```
cd [YOURDIR]/AvaFrame
```

Replace [YOURDIR] with the directory from your installation step.

#### 2.4.3.1 Initialize project

To create the folder where the input data lies and where the output results will be saved, specify the full path to the folder in the `local_avaframeCfg.ini` (which is a copy of `avaframeCfg.ini` that you need to create). So:

```
cd avaframe
cp avaframeCfg.ini local_avaframeCfg.ini
```

and edit `local_avaframeCfg.ini` with your favorite text editor and adjust the variable `avalancheDir`.

Then run

```
python runScripts/runInitializeProject.py
```

This will create a new directory with the input required by AvaFrame structured as described in [Initialize Project](#).

#### 2.4.3.2 Input data

Check the input data required by the different modules you want to use and fill the `Inputs/` inside the `[avalancheDir]` folder from the initialize step accordingly.

For example the `com1DFA` module needs input as described in [Input](#). You can also have a look at the default setting for the module you want to use (for example `com1DFACfg.ini` for module `com1DFA`). If you want to use different settings, create a `local_` copy of the `.ini` file and modify the desired parameters.

More information about the configuration can be found here: [configuration:Configuration](#)

#### 2.4.3.3 Building your run script

Create your own workflow by taking the `runOperational.py` script as template.

We suggest you copy it and adjust it to your liking. There are annotations in the code that should help you to understand the structure.

A lot more examples can be found in the `runScripts` directory (see also [Example runscripts](#)).

## 2.5 Configuration

In order to set the configurations required by all the modules within Avaframe, the python module `configparser` is used.

This is done in two steps. The first step fetches the main settings:

```
from avafame.in3Utils import cfgUtils
# Load avalanche directory from general configuration file
cfgMain = cfgUtils.getGeneralConfig()
avalancheDir = cfgMain['MAIN']['avalancheDir']
```

In the second step the specific settings to a given module are imported:

```
from avafame.tmp1Ex import tmp1Ex
# Load all input Parameters from config file
# get the configuration of an already imported module
# Write config to log file
cfg = cfgUtils.getModuleConfig(tmp1Ex)
```

The `in3Utils.cfgUtils.getModuleConfig()` function reads the settings from a configuration file (`tmpEx.ini` in our example) and writes these settings to the log file. The default settings can be found in the configuration file provided within each module.

It is possible to modify these settings, there are two options:

- provide the path to your own configuration file when calling `cfgUtils.getModuleConfig(moduleName, path to config file)`
- create a copy of the module configuration file called `local_` followed by the name of the original configuration file and set the desired values of the individual parameters.
- see configuration:Override configuration for additional options to modify configuration

So the order is as follows:

1. if there is a path provided, configuration is read from this file.
2. if there is no path provided, the `local_...` configuration file is read if it exists.
3. if there is no `local_...`, the `getModuleConfig` function reads the settings from the default configuration file with the default settings.

In the configuration file itself, there are multiple options to vary a parameter:

- replace the default parameter value with desired value
- provide a number of parameter values separated by `|` (e.g. `relTh=1.|2.|3.`)
- provide a number of parameter values using `start:stop:numberOfSteps` (e.g. `relTh=1.:3.:3`) - a single value can be added by appending `&4.0` for example

## 2.5.1 Override configuration

If tools of one module, let's call this module **A** for now, are called from another module **B**, there is the option to include an *collectionName\_A\_override* section in the configuration file of module **B**. In this case, the default configuration of module **A** is read and the parameters in the *A\_override* section in the module **B** configuration file are used to update the configuration settings of module **A**. This has the advantage of gathering all the configuration parameters used for one task in one configuration file. An example of this usage can be found in `ana1Tests/energyLineTestCfg.ini`.

## 2.5.2 Logging

In order to generate simulation logs and to control what is prompted to the terminal, we use the python module `logging`.

Let's have a look at the simple example in `runScripts.runTmp1Ex` and `tmp1Ex.tmp1Ex()` on how this is used within AvaFrame.

In your main script call:

```
from avaframe.in3Utils import logUtils
# log file name; leave empty to use default runLog.log
logName = 'runTmp1Ex'
# specify the working directory
avalancheDir = './'
# -----
# Start logging
log = logUtils.initiateLogger(avalancheDir, logName)
```

This will configure the logging (it sets the console output as well as the log file). In your modules/subscripts add:

```
import logging
log = logging.getLogger(__name__)
```

So you can use:

```
log.debug('Should be here')
log.info('DEM : %s',variable)
```

To get output that looks like this in your console:

```
tmp1Ex:DEBUG - Should be here
tmp1Ex:INFO - DEM : /path/to/DEM
```

and something similar in the `.log` file which is saved in `./runTmp1Ex.log` in this example. The logging configuration is set in `AvaFrame/avaframe/in3Utils/logging.conf`.

You can modify this `logging.conf` file to modify the levels or format of the messages to display ([python doc](#) will help you).

### 2.5.3 Example runscripts

In runScripts we provide ready-to-use scripts for different applications of the modules provided within AvaFrame.

#### 2.5.3.1 Derive input data

- `runScripts.runComputeDist`

#### 2.5.3.2 Create a new project

- `runScripts.runInitializeProject`

#### 2.5.3.3 Generate idealized/generic topography data

- `runScripts.runGenerateTopo`
- `runScripts.runGenProjTopoRelease`

#### 2.5.3.4 Postprocessing

- `runScripts.runAna3AIMEC`
- `runScripts.runAna3AIMECCompMods`
- `runScripts.runAna3AIMECIndi`
- `runScripts.runStatsExample`
- `runScripts.runProbAna`

#### 2.5.3.5 Visualisation

- `runScripts.runQuickPlotSimple`
- `runScripts.runQuickPlotOne`
- `runScripts.runPlotTopo`
- `runScripts.runExportToCsv`

#### 2.5.3.6 Testing

- `runScripts.runDamBreak`
- `runScripts.runSimilaritySol`
- `runScripts.runTestFP`
- `runScripts.runStandardTestsCom1DFAOrig`
- `runScripts.runComparisonModules`
- `runScripts.runFetchBench`
- `runScripts.runWriteDesDict`



## 2.6 Testing

In AvaFrame we provide a continuously growing test suite. This includes tests that target model verification and model validation. The former are used to test the numerical implementation of the mathematical model, whereas the latter are aiming at testing if the mathematical model employed is appropriate to describe the physical processes of the desired application, in our case the simulation of dense snow avalanches.

In the following, a brief description of the available tests is given.

---

**Note:** This section is currently under development and will be updated regularly.

---

---

**Note:** See this [example pdf](#) (25mb) how we apply these tests during development.

---

### 2.6.1 Tests for model verification

This section includes testing of bits and pieces of the model code up to tests that check whether the full workflow is numerically robust. However, at the ‘smallest scale’, where individual functions are tested, we also refer to the Section about *pytests*: [How to test code](#).

#### 2.6.1.1 Rotational Symmetry

In this test, a pyramid-shaped topography is used to test whether different grid alignments in respect to the topography and hence flow direction cause problems for the model implementation. For this test, we simply provide two pyramid shaped topographies, where one is rotated around the z-axis to change the orientation of the pyramid’s facets with respect to the mesh.

#### 2.6.1.2 Dambreak problem

This test is based on a Riemann problem, where the initial condition is described by two different states that are separated by a discontinuity. In [FM13] exact solutions are derived for this problem, based on the Savage-Hutter model to describe the granular flow. Here, we provide an implementation of one of the seven cases presented in [FM13]. In Test 2, the initial velocity is zero and a granular mass is suddenly released and flows downslope. For further details have a look at the module `ana1Tests.damBreak` and the Section [Dambreak test](#).

#### 2.6.1.3 Similarity Solution

This test makes use of semi-analytic solution of a sliding granular mass on an inclined plane. In [HSSN93] the semi-analytic solution is derived for this problem, based on the Savage-Hutter shallow flow model to describe the granular flow. Here, we provide an implementation of the similarity solution and compare it to the output of a numerical simulation. For further details have a look at the module `ana1Tests.simiSol` and the Section [Similarity solution](#).

## 2.6.2 Tests for model validation

In this section, tests that check whether the chosen mathematical model is suitable to represent the problem at hand can be found. Currently, we provide idealized and real-world topographies that can be used for avalanche simulations. Additionally, there are several functions that can be used to analyse the simulation results and compare them to other data such as results from other models or observations. For this, we refer to Sections [ana3AIMEC: Aimec](#) and [outQuickPlot](#).

### 2.6.2.1 Idealized test cases

We provide a range of idealized topographies to perform avalanche simulations including different release area scenarios. Some of the topographies also include entrainment and/or resistance areas. These can be found in `data/NameOfAvalanche`. The functions to produce these can be found in module `in3Utils/generateTopo.py`.

#### Bowl - BL

Bowl shaped topography. Used to test e.g. rotational symmetry.

#### Flat Plane - FP

#### Inclined Plane - IP

IP (Inclined plane)

#### Parabolic slope - PF

Parabolic slope with flat foreland

#### Hockeystick - HS

Hockeystick with linear slope and flat foreland and smooth transition

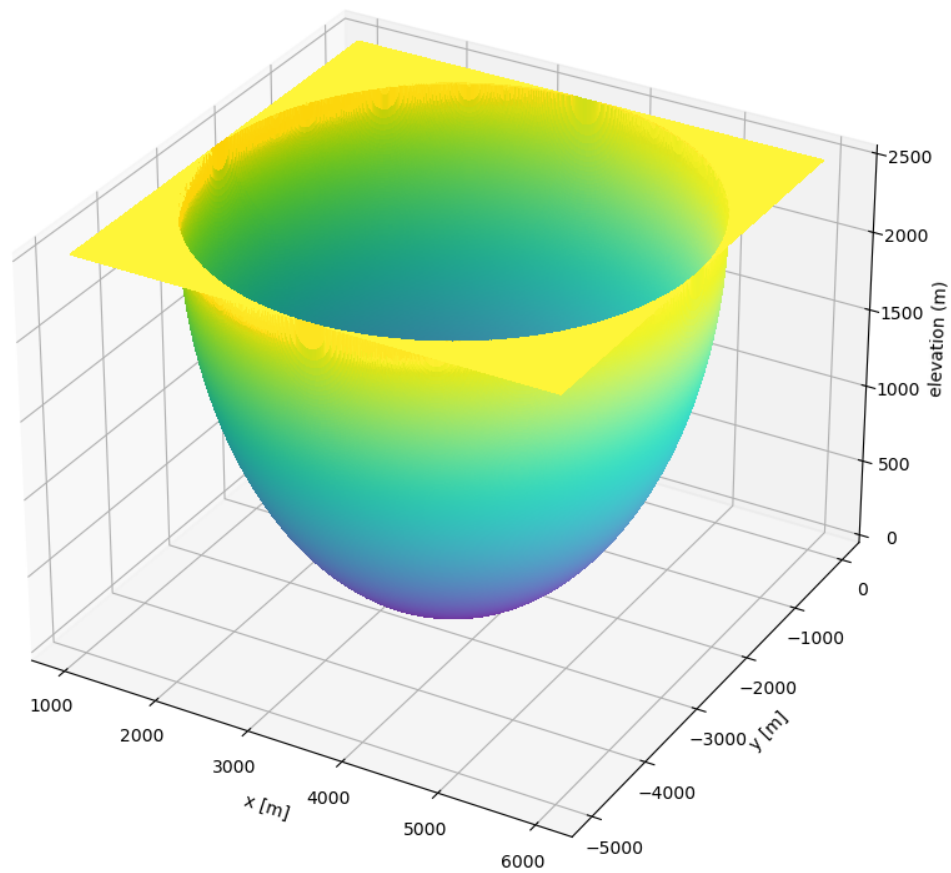
#### Helix - HX

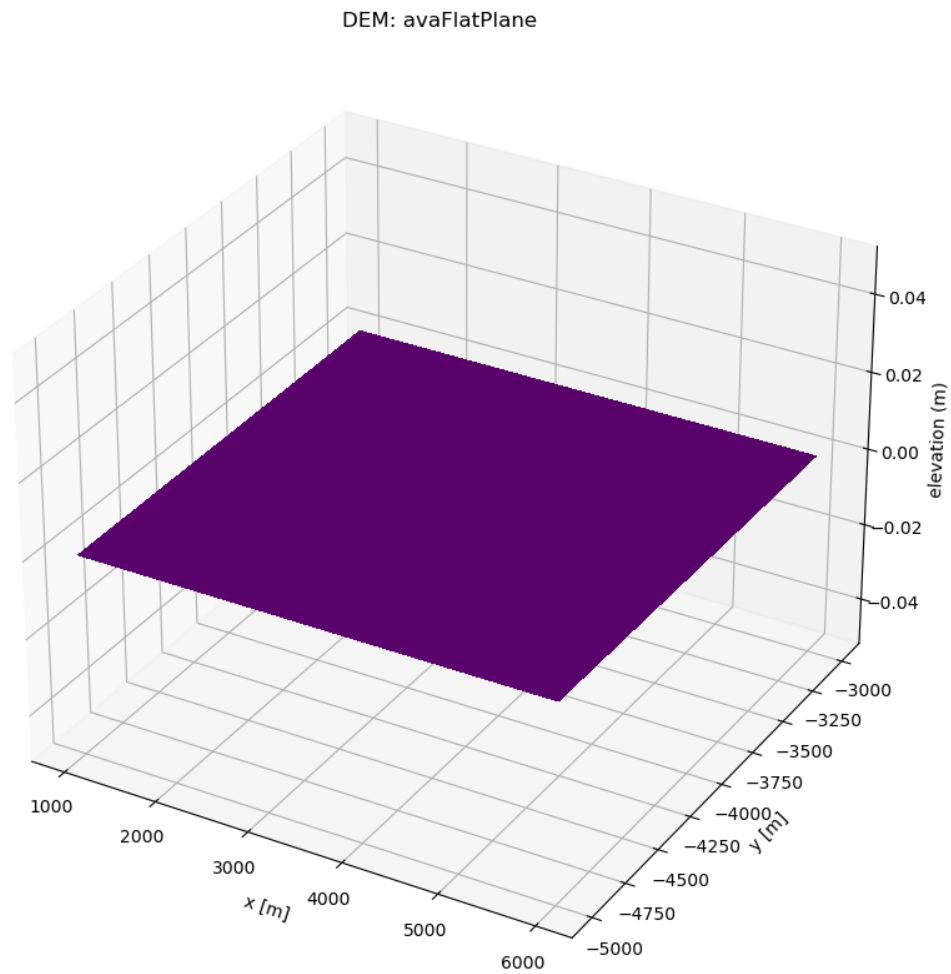
Helix-shaped topography

#### Pyramid - PY

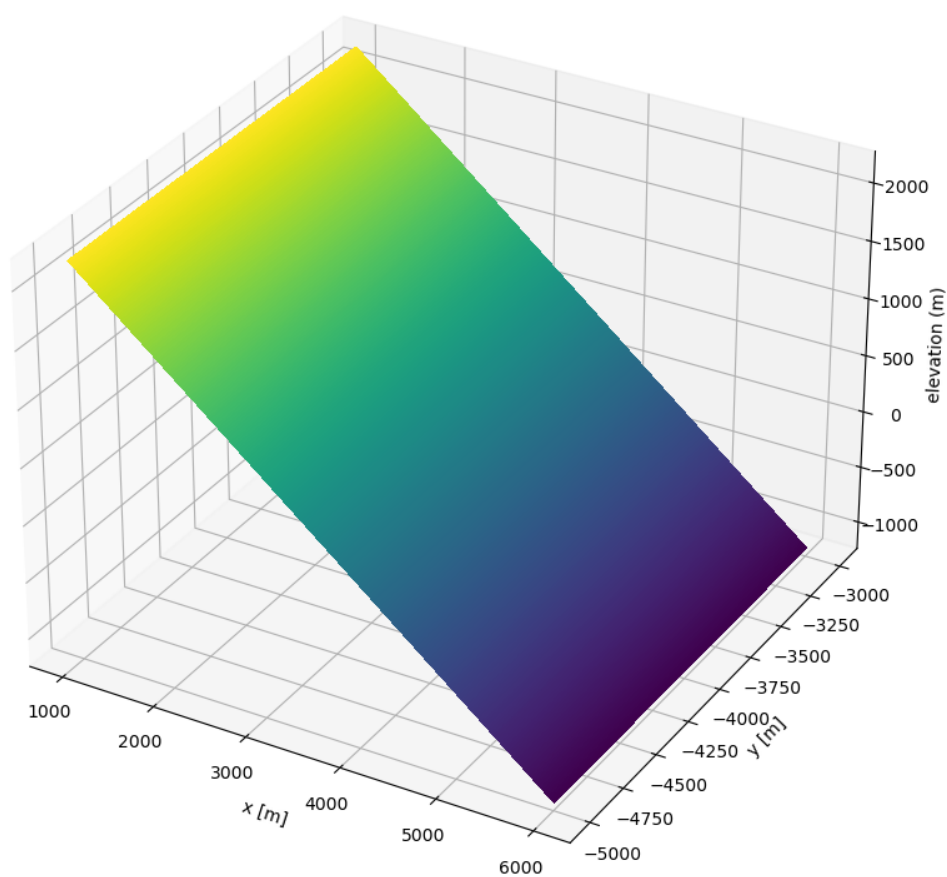
pyramid-shaped topography, optional with flat foreland

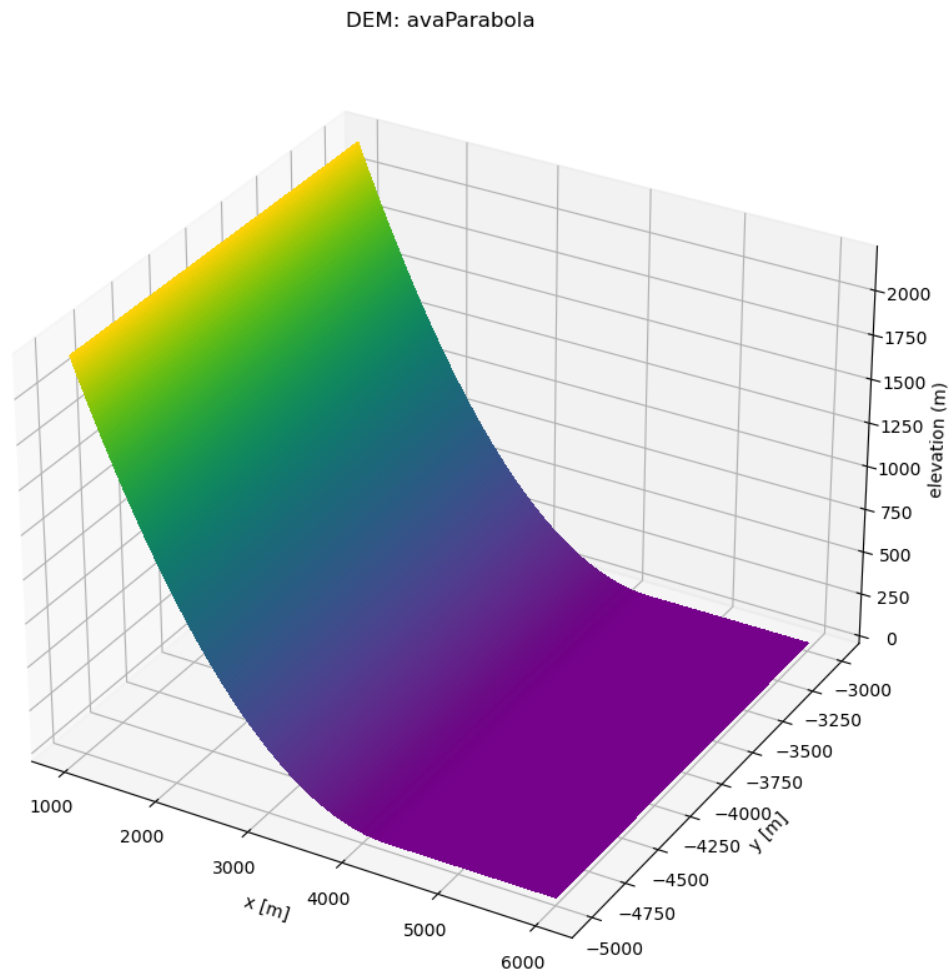
DEM: avaBowl



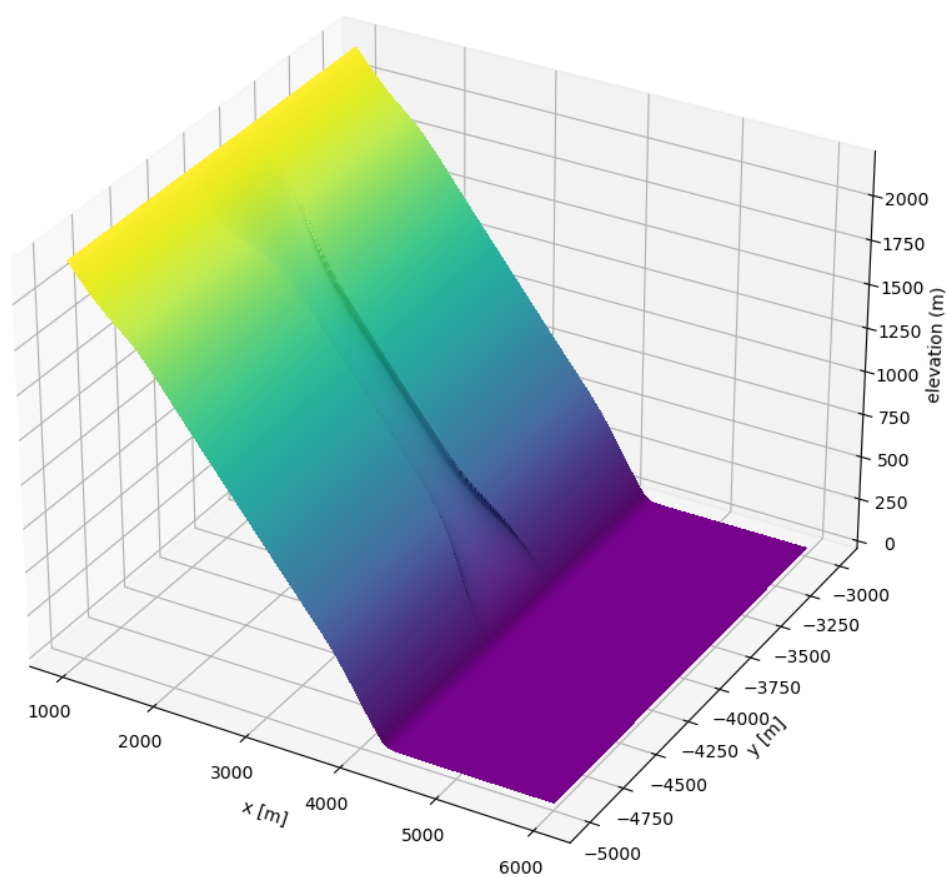


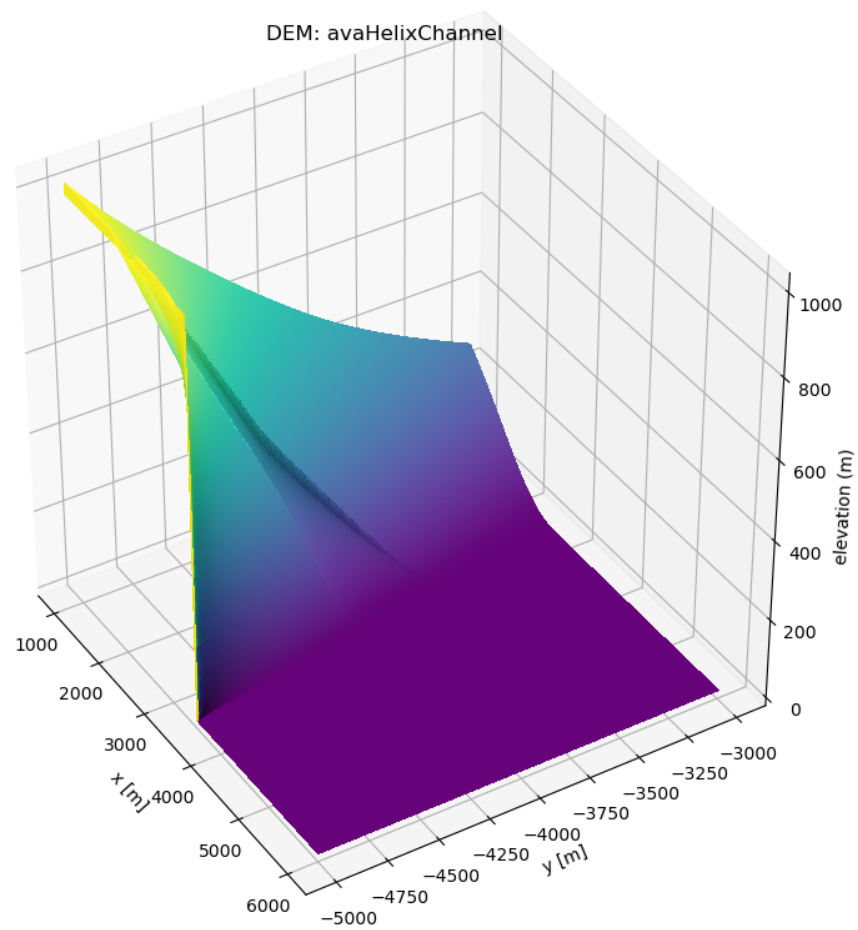
DEM: avalInclinedPlane





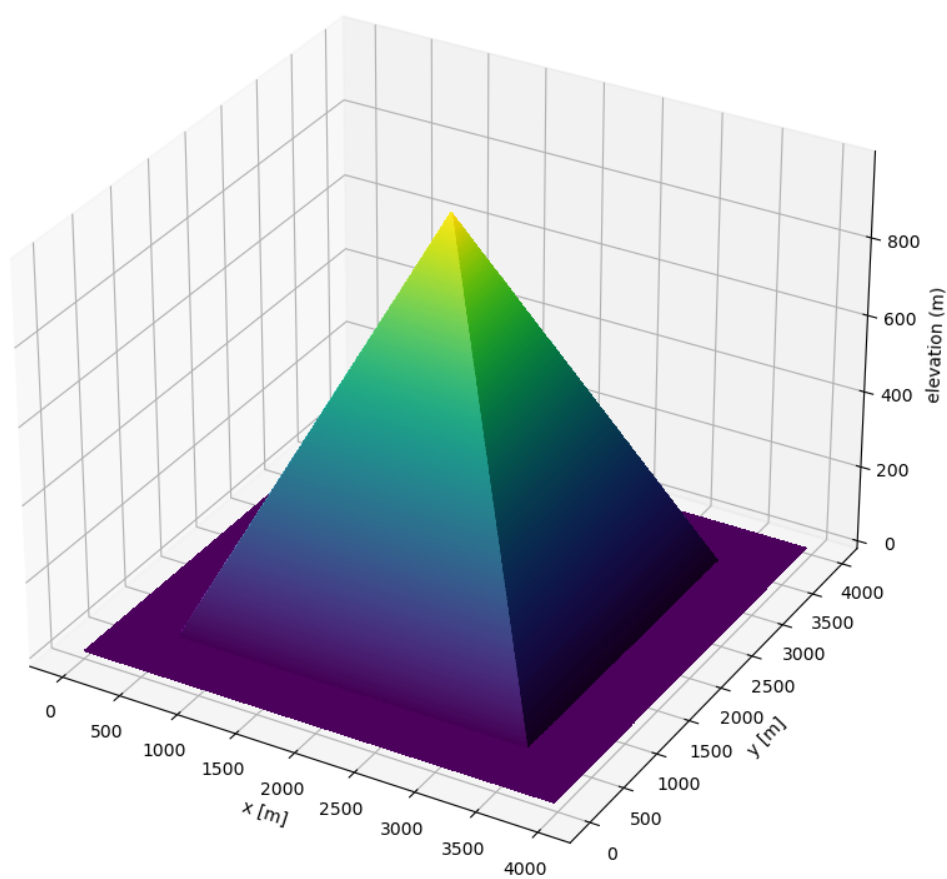
DEM: avaHockeyChannel







DEM: avaPyramid



### 2.6.2.2 Real-world test cases

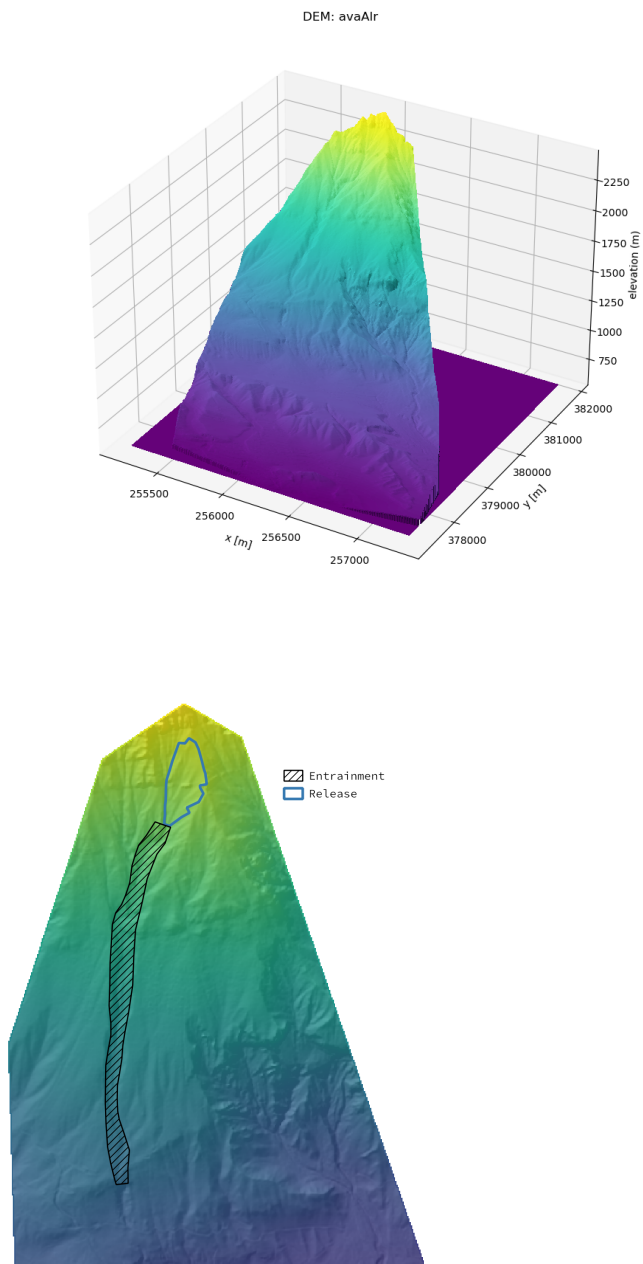
We provide a range of real-world topographies to perform avalanche simulations including different release area scenarios including entrainment areas. These can be found in `data/NameOfAvalanche`.

---

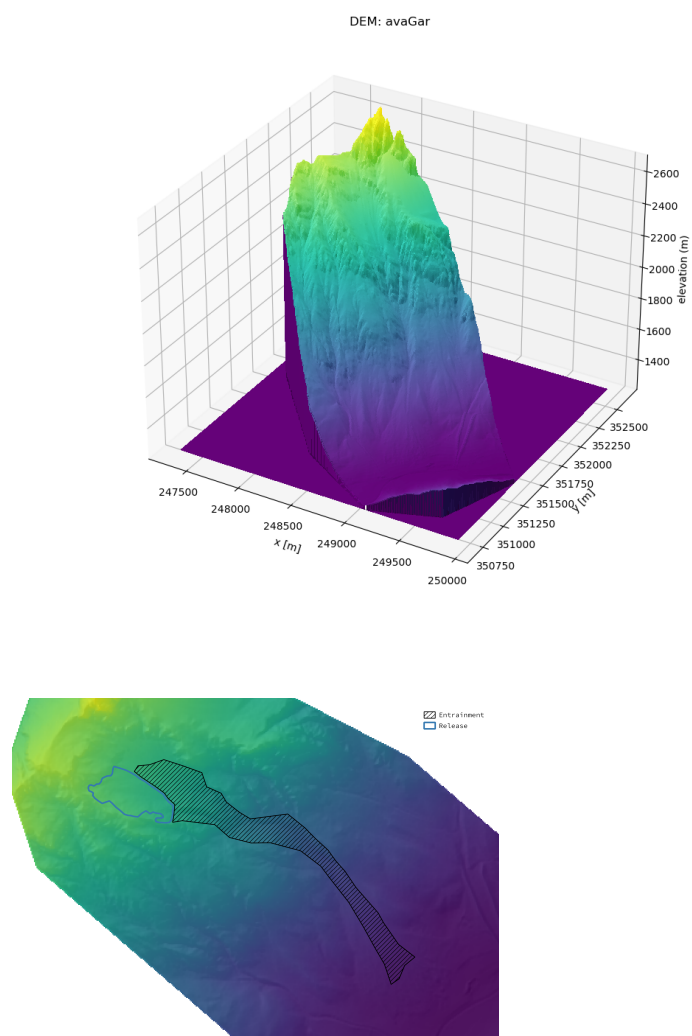
**Note:** All images give you a bigger version on click.

---

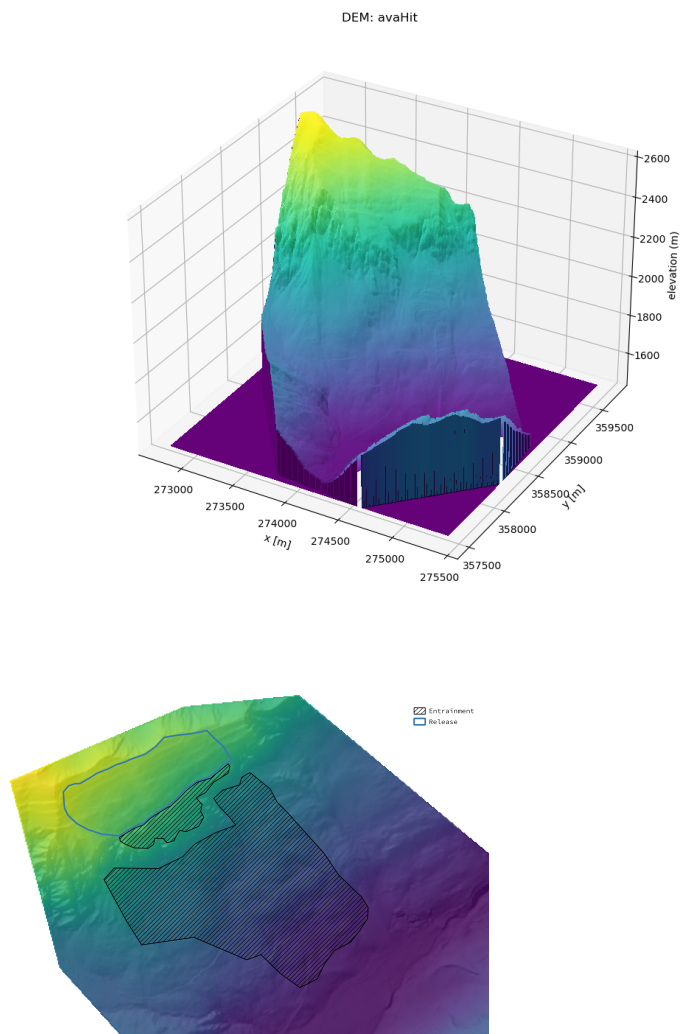
#### Air



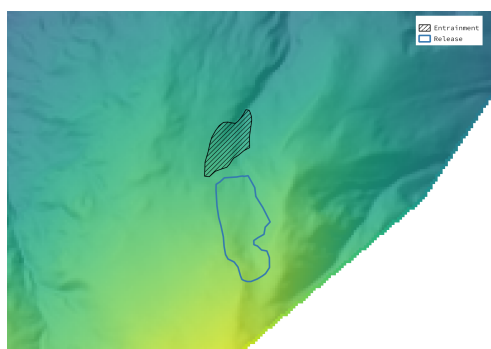
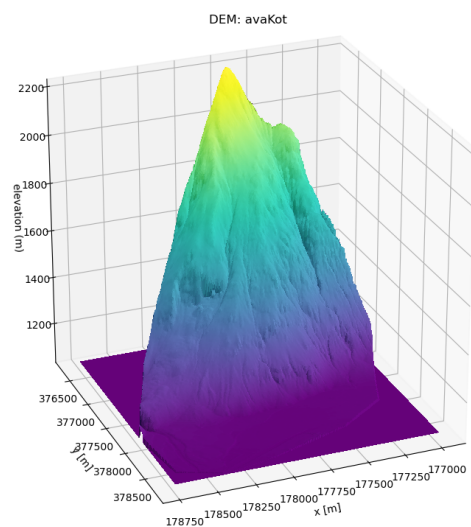
## Gar



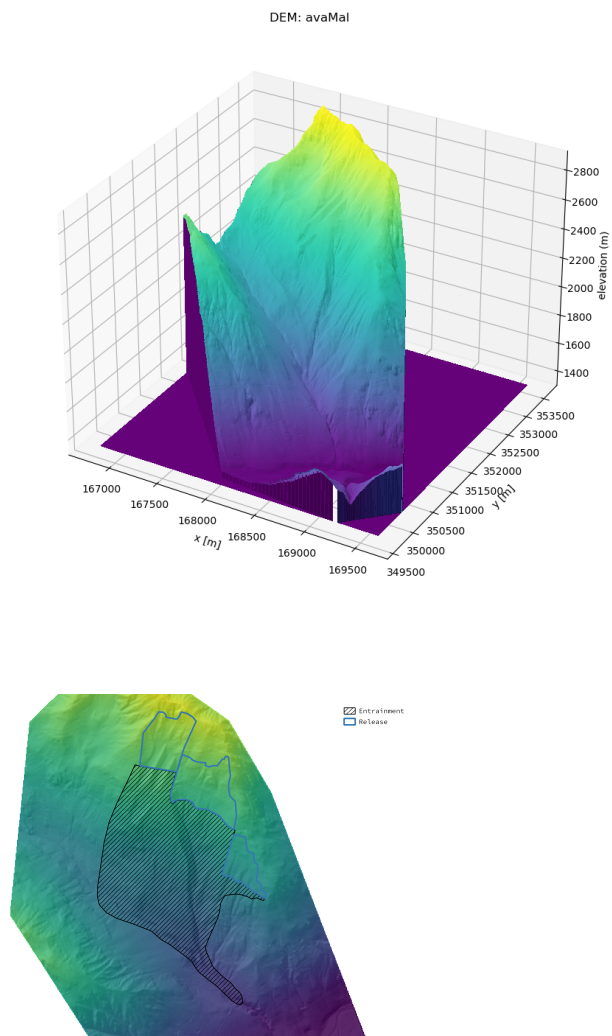
## Hit



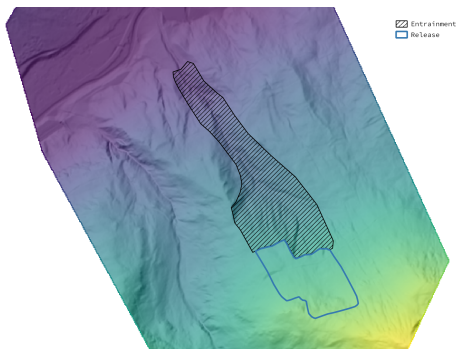
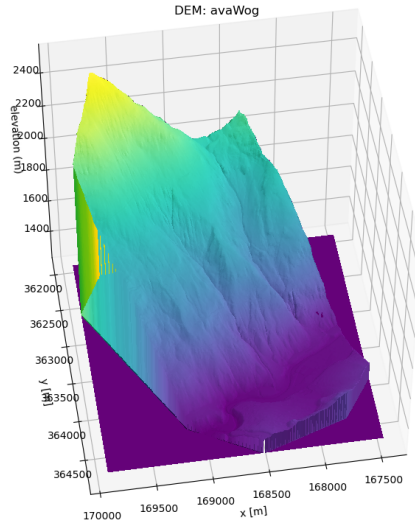
## Kot



## Mal



## Wog



## 2.7 Data Visualisation

Main functions for creating visualisations of AvaFrame simulation results can be found in [out3Plot](#) and [out1Peak](#). [com1DFA](#) also offers the possibility to export data on particle properties for visualisation using the open-source application [ParaView](#). In order to start analysing particle data of [com1DFA](#), follow these steps:

in your local copy of `com1DFA/com1DFACfg.ini`:

- *section [GENERAL]:* add *particles* to the *resType*
- *section [GENERAL]:* set which time steps you want to save at *tSteps*
- *section [VISUALISATION]:* set *writePartToCSV = True* and add the particles' properties you are interested in to the *particleProperties*

in ParaView

- open the particles file located in *data/avaDir/Outputs/com1DFA/particlesCSV* - there will be one available for each simulation performed containing the info on all saving time steps
- apply the filter **TableToPoints** and set X, Y, Z for the X, Y, Z columns
- switch to the **RenderView**: you can look at all the different particle properties you have saved for all exported time steps
- it is also possible to save an animation over all time steps

## 2.8 Frequently Asked Questions

---

**Note:** If you want to develop your own simulation workflow or develop new code we also suggest to have a look at the code examples provided in the *runScripts* directory.

---

### 2.8.1 Can the spatial resolution of simulations performed with com1DFA (dense flow) be changed?

The default setup of *com1DFA* is calibrated for medium to large dense flow snow avalanches in regards to hazard mapping. To provide calibrated results, new simulations with DEM cell sizes other than 5 meters are being remeshed by *com1DFA* to **5 meters** for computation.

It is possible to change the cell size, but keep in mind there are no tested/calibrated setups available. We want to list a few things that we strongly suggest to take into account when changing the mesh cell size:

- Start by changing the *meshCellSize* in the configuration file of *com1DFA* (*com1DFACfg.ini* or rather your local copy of it), and consider changing the *sphKernelRadius* to match the new mesh cell size.
- For the *SPH gradient*, required to compute the lateral forces, the number of particles per cell has to be adjusted. If the *sphKernelRadius* (see first point) is decreased, the number of particles should increase to ensure a reasonable estimate of the gradient. There are three options for setting the number of particles which is computing from the mass per particle (*massPerParticleDeterminationMethod*; see *Initialize particles*):
  - With the default setup *MPPDH* (mass per particle through release thickness), the number of particles per cell is independent of the mesh cell size. So no adjustment is necessary.
  - If *MPPKR* (mass per particles through number of particles per kernel radius) is chosen, the size of particles is adjusted to get a constant number of particles within an *sphKernelRadius*. This ensures a reasonable amount of particles for the gradient computation.
  - If *MPPDIR* (mass per particle set directly) is chosen, the number of particles per cell depends on the release snow mass within a cell. The number of particles is computed using the cell area and the release thickness. To ensure a reasonable amount of particles for decreased mesh size and *sphKernelRadius*, the *massPerParticle* value should be decreased accordingly. This ensures a reasonable amount of particles for the gradient computation.

Regarding DEM cell sizes: - if a cell size XX different than the default 5 meters is chosen - and the provided DEM in Inputs is of different mesh cell size than XX - the folder *remeshedRasters* will be checked for a matching DEM. - if no matching DEM is found -> remeshing takes place

Using the default settings, the subdirectory 'remeshedRasters' is cleaned when starting simulations, to ensure remeshing. This can be changed in the configuration file (further details are provided in *DEM input data*).



## 2.9 Release Notes

Please refer to [Github releases](#) for current release notes.

### 2.9.1 1.3 (12. Okt 2022)

Main change is the change of logic for secondary release areas. This was done to be able to expose this functions in the AvaFrameConnector. So it is now possible to include secondary release areas in dense flow simulations done via QGIS.

#### ENHANCEMENTS

- Change logic for secondary release areas: - Check for release and secondary release shapefiles - If only release available -> just run release - If both release and secondary release are available -> run with secondary release UNLESS secRelArea = False
  - ALL scenarios in release get the secondary release areas!
- Add rotational energy line test: helps to checks eg. for numerical grid independence
- Update ini file procedure for the energy line test and the rotation test
- Additional statistical plots
- New three panel plot of tt-diagram (plus animation)
- Add variation option for thickness settings and probrun based on normal distribution derived from ci and mean
- Add filtering option to aimec
- Add scenario name to configuration to be used for plotting example #757
- Add surface parallel coordinate computation to Aimec
- Improve operational installation instructions
- Add german version of operational installation

FIX - Contour legend bug with matplotlib 3.5.2 - Update installation instructions; fixes #764 - Bugs in deriving variation - Remeshing issue that lead to standard test differences (originated in commit 419c11f) - No calls to matplotlib for plotting purposes in com1DFA - Removes multiple printouts of config during run in, e.g. com1DFA - CompareConfig always honours the toPrint flag

Contributors:

Code: core team

### 2.9.2 1.2 (07. July 2022)

Main changes are the automatic split point generation and optional computation of fields inside the calculation loop. Furthermore renaming functions used for the QGIS AvaFrameconnector are included.

#### ENHANCEMENTS

- Add function for renaming simulations, i.e. adding info to the simName. Used for AvaFrameConnector
- Split cfgUtils: Utils contains all reading and writing, cfgHandling contains functions that do something with the cfgInfos
- Make computation of ppr, pta, P, TA and pke optional within the calculation loop . Only compute them if they are required as output

- Add automatic split point generation: - First run a DFA simulation, either using the runDFAModule flag in runComputeDFAPath.py or yourself (do not forget that you need to save particles or FD, FM, FV for multiple time steps)
  - What DFA parameters to use is not yet clear. I would use the BenMoussa time stepping and particles parameters, sphOption 2, explicit friction option 1, some artificial viscosity, and maybe activate curvature (with all this and a  $\mu=0.42$ , I get ok results)
  - Then runComputeDFAPath computes the mass average path, extends it and defines a splitPoint

#### FIX

- Particle exiting domain was not working properly with the new cython flag (wraparound boundcheck...)
- correct hybrid Path ini file
- Update fields is too slow #455
- Automatic split point or beta point finding #706
- Add pfe to output files #694
- Make cflTime and sphKernelRadius a common option #668
- Change benchmarks not read particles from file (particles files still exist)
- Add consistency check for com1DFA configuration
- Fix problems with filtering of simulations
- Aimec comparison checks
- Add missing mass files to benchmarks

Contributors:

Code: core team

### 2.9.3 1.1 (19. May 2022)

The benchmark and thickness release. There are two main changes:

- The benchmarks (i.e. reference results) are updated and originate from com1DFA. Previously these were produced by com1DFAOrig. (Attached zip file contains the standardTest report for the switch)
- All references to *depth* are now switched to *thickness*. This is done to be more consistent and precise. It also means result types switch from *pfd / fd* (peak flow depth / flow depth) to *pft / ft* (peak flow thickness / flow thickness). Note that this is a *naming* change; nothing changed regarding the computation!

#### ENHANCEMENTS

- benchmarks originate from com1DFA
- change all depth variables to thickness
- change the order of simHash within the result names; fixes Move simHash in filename #690
- path finding added; see issue #610. This will be fully introduced in version 1.2, including automatic split point generation - refactor path computation functions - allow computing path from particles or fields (if *from fields*: needs the FM=FlowMass) - runscript to compute a path from com1DFA results (requires that one saves some time steps)
- automate the benchmark updating process
- improve energy line plot

- set deleteOutput to False in runOperational; addresses User Feedback (CT) #715. This means it is possible to reuse the same directory in the QGis Connector, adding results to existing ones
- add real area to aimec analysis #695
- update hybrid and energy test
- add com3Hybrid documentation #618

#### FIX

- hybrid model #611
- refactor com2AB for clarity and readability #446
- address savefigFormat TODO in outAB #560
- only one makeDomainTransfo #700

Contributors:

Code: core team

### 2.9.4 1.0.1 (20. April 2022)

#### FIX

- #712 , missing init files

### 2.9.5 1.0 (6. April 2022)

#### ENHANCEMENTS

- adds avafame version to log
- appends date to logfile name
- update similarity solution plots
- re-add codecov
- add in addition to vary thickness values if read from shp - not just in percent but also in absolute value
- *ana1Test* energy line test
- *documentation* info on visualisation options (Paraview)
- update the pytest github action to version 3.9
- add ana5Hybrid, module that combines statistical module com2AB with the DFA module com1DFA
- new requirement shapely
- add release area info to benchmark ini files
- make AB optional in runOperational (related to QGis AvaFrameConnector)
- updates to ana1Tests
- hillshade and contours for peak plots
- documentation improvements
- reorder installation and get started documentation

- create distance-time diagrams of ava simulations from a reference point showing the avalanche front and the average values of a chosen result parameter (e.g. flow depth, flow velocity)
- *com1DFA* new flags/system for release thickness and entrainment thickness settings and options
- *com1DFA* add travel angle computation
- *com1DFA* release thickness percent variation option
- *com1DFA* unique simHash including info on release scenario with correct thickness
- *com1DFA* removed return parameters from *com1DFAMain*
- *com1DFA* update benchmark ini files
- *com1DFA* documentation for bottom friction and operator splitting
- *com1DFA* option to redistribute particles after initialisation in order to reduce SPH force
- *com1DFA* Implement Ata Viscosity and an SPH flow thickness computation
- *com1DFA* new splitting/merging of particles
- *com1DFA* enable to initialize particles with a non constant flow thickness
- *com1DFA* remove unmaintained leap frog time stepping scheme
- *com1DFA* new parameter: *cleanDEMremeshed*
- *com1DFA* add simulation DEM if remeshed to different *cellSize* #670
- *com1DFA* check for remeshed DEM, save remeshed DEM #675
- *com1DFA* enable to chose dem asc file for *com1DFA* #658
- *com1DFA* new parameter: *cleanDEMremeshed*
- *com1DFA* add simulation DEM if remeshed to different *cellSize* #670
- *com1DFA* check for remeshed DEM, save remeshed DEM #675
- *com1DFA* enable to chose dem asc file for *com1DFA* #658
- *ana4Prob* add example for performing a parameter variation run with prob analysis
- *ana4Prob* use default com module setup or specified in local - add variation for prob run
- *ana4Prob* perform analysis using *probabilityConfiguration* in *runScript*

#### FIX

- errors in *com2AB* documentation
- *tcpu* field in *com1DFA*
- ordering of dict for *analysisAdd*
- pytest errors related to matplotlib colors and legend
- particle splitting issue
- fix pypi related issues (pypi needs clean version tags)
- quickfix for shapely vs QGis problem with the *AvaFrameConnector*, see Linux QGis 3.24 crashes on *Connector* activation QGisAF#9
- move Release-version file for packaged releases
- change naming of log file: fix #689
- (hacky) solution to handle apostrophes in filenames #683

- allow choosing a tau0 in samosAT friction type (so far, tau0 was fixed and equal to 0)
- add tau0 to SamosAT friction #702
- address the wrong logName in runsript
- error running simulations one day after #701
- error on python 3.7 and QGis 3.12 #705
- python3-dev package required. #699

Contributors:

Code: core team, M. v. Busse (UIBK), M. Winkler (UIBK) Code review tt-diagram: A. Köhler (BFW)

## 2.9.6 v0.6 (24. September 2021)

### ENHANCEMENTS

- installation via pypi (pip install)
- connection to QGis (via plugin manager)
- function to interpolate data on mesh of different cellSize using splinesp
- testing via pytest extended
- more pathlib usage
- ASCII header is read as dict
- documentation contains FAQ page
- reworked installation instructions
- cleaner test reports/inis
- github action to deploy to pypi
- switch to codeclimate
- use consistent thickness attributes (shapefiles etc)
- *comIDFA* any resolution is possible now
- *comIDFA* split the getWeight function in two: first get cell and then get weights.
- *comIDFA* avoid possibility of segfault because particles exit too quickly the domain.
- *comIDFA* additional particles info: unique identifier for each particle and parent particles
- *comIDFA* central time step calling
- *comIDFA* additional options to set mass per particle directly or via release thickness
- *comIDFA* interpolation option for initialization of Hpart
- *comIDFA* read entrainment thickness
- *ana3AIMEC* override option for raster cellsize
- *ana3AIMEC* mass analysis plot even if more than 2 simulations

### FIX

- getTimeIndex problem if dtSave < actual dt
- better way to remove particles

- track particles exiting the computation domain
- fix issue save particles
- read aimec grid info from result files and not from dem
- add reasonString to removal of particles
- fix correct module name in AIMEC
- com2AB write out to shp

Contributors:

- **Code: core team**

### 2.9.7 v0.5 (13. July 2021)

#### ENHANCEMENTS

- filtering functions for com1DFA simulations
- flag to disable print at CFG reading
- new colormaps for ppr, pft, pfv
- *com1DFA* option to add friction explicitly using the method described in #273 .
- *com1DFA* Resistance force is added explicitly.
- *com1DFA* New method to get the release area
- *com2AB* function to write results to shapefile
- *ana3AIMEC* warning for empty runout zone
- *ana3AIMEC* enable simulation ordering/filtering

#### FIX

- beta angle issue i.e. distance below angle
- correct removal of particles
- AIMEC produces warning on empty runout area
- adapt quickplot to new naming scheme

Contributors:

- **Code: core team**
- **Colormaps: C.Tollinger**

DOI for this release:

### 2.9.8 v0.4.1 (9. June 2021)

Minor release to fix issue with zenodo

### 2.9.9 v0.4 (8. June 2021)

The switch release

This is a big release: we switched our dense flow module ‘com1DFA’ to the python version. This means that you now get to use the python version as default. However, the original version is still available in the module com1DFAOrig. The full documentation for the python com1DFA version as well as updated benchmarks will be released in the next version.

Module com2AB (AlphaBeta) received an update allowing for custom parameters.

Simulation naming and identification also received a major change, we introduced unique ID’s for each individual configuration.

Contributors:

- **Code: core team**

### 2.9.10 v0.3 (26. April 2021)

The AIMEC and Windows release

This release brings an AIMEC refactor, plenty of improvements related to the test cases and Windows capabilities. 3 new idealised/generic test case are included: flat plane, inclined slope and pyramid.

Com1DFAPy received a lot of advancement as well, e.g. parts of it are converted to cython to speed up computation times.

Documentation regarding our testing is included, see more at the [testing](#) page.

Contributors:

- **Code: core team**

DOI for this release:

### 2.9.11 v0.2 (28. Dezember 2020)

The testing release

Version 0.2 includes the first real world avalanches. It provides data for 6 avalanches, including topographies, release areas and benchmark results. To know more about our data sources, head over to [our data sources documentation](#). The existing test cases also received some updates by including multiple release areas and multiple scenarios per avalanche.

This release also is the first to include [API documentation](#) for our modules and functions. However not all functions are included yet.

Contributors:

- **Data: M.Granig, C. Tollinger**
- **Data: Land Tirol**
- **Code: core team**

## 2.9.12 v0.1 (06 November 2020)

Initial release.

This release is the result of several months of development.

Several people have contributed to this release, either directly or through code that was used as reference/basis:

- **Peter Sampl**, code base for com1DFA
- **Jan-Thomas Fischer**, code base AIMEC, code related to com1DFA
- **Michael Neuhauser**, code for helper and transformation utilities, com1DFA
- **Andreas Kofler**, code related to AIMEC and com1DFA

and the core team:

- **Anna Wirbel**
- **Matthias Tonnel**
- **Felix Oesterle**

## 2.10 API Reference

### 2.10.1 Computational Modules

<i>com1DFA</i>	Dense flow avalanche kernel (Python/cython)
<i>com2AB</i>	AlphaBeta kernel
<i>com3Hybrid</i>	Hybrid modelling: combining DFA simulation and statistical approaches

#### 2.10.1.1 com1DFA

Dense flow avalanche kernel (Python/cython)

#### Modules

<i>com1DFA.DFAtools</i>	Basic tools for getting grid normals, area and working with vectors.
<code>com1DFA.checkCfg</code>	
<code>com1DFA.com1DFA</code>	
<code>com1DFA.com1DFATools</code>	
<i>com1DFA.deriveParameterSet</i>	Create dictionary for parameter variations
<i>com1DFA.particleInitialisation</i>	functions for initialising particle distribution
<code>com1DFA.particleTools</code>	
<i>com1DFA.timeDiscretizations</i>	Functions regarding time discretization and time stepping for com1DFA



**com1DFA.DFAtools**

Basic tools for getting grid normals, area and working with vectors.

**Functions**

<i>crossProd</i>	Compute cross product of vector $u = (ux, uy, uz)$ and $v = (vx, vy, vz)$ .
<i>getAreaMesh</i>	Get area of grid cells.
<i>getNormalArray</i>	Interpolate vector field from grid to unstructures points
<i>getNormalMesh</i>	Compute normal to surface at grid points
<i>norm</i>	Compute the Euclidean norm of the vector $(x, y, z)$ .
<i>norm2</i>	Compute the square of the Euclidean norm of the vector $(x, y, z)$ .
<i>normalize</i>	Normalize vector $(x, y, z)$ for the Euclidean norm.
<i>scalProd</i>	Compute scalar product of vector $u = (ux, uy, uz)$ and $v = (vx, vy, vz)$ .

**com1DFA.DFAtools.crossProd**

**crossProd**(*ux, uy, uz, vx, vy, vz*)

Compute cross product of vector  $u = (ux, uy, uz)$  and  $v = (vx, vy, vz)$ .

**com1DFA.DFAtools.getAreaMesh**

**getAreaMesh**(*dem, num*)

Get area of grid cells.

**Parameters**

- **dem dict updated with** –

**Nx: 2D numpy array**

x component of the normal vector field on grid points

**Ny: 2D numpy array**

y component of the normal vector field on grid points

**Nz: 2D numpy array**

z component of the normal vector field on grid points

**header**

[dict] dem header (cellsize)

- **num (int)** – chose between 4, 6 or 8 (using then 4, 6 or 8 triangles) or 1 to use the simple cross product method (with the diagonals)

**Returns**

**areaRaster: 2D numpy array**

real area of grid cells

**Return type**

dem dict updated with

### com1DFA.DFAtools.getNormalArray

**getNormalArray**(*x*, *y*, *Nx*, *Ny*, *Nz*, *csz*)

Interpolate vector field from grid to unstructures points

Originally created to get the normal vector at location (x,y) given the normal vector field on the grid. Grid has its origin in (0,0). Can be used to interpolate any vector field. Interpolation using a bilinear interpolation

#### Parameters

- **x** (*numpy array*) – location in the x location of desired interpolation
- **y** (*numpy array*) – location in the y location of desired interpolation
- **Nx** (*2D numpy array*) – x component of the vector field at the grid nodes
- **Ny** (*2D numpy array*) – y component of the vector field at the grid nodes
- **Nz** (*2D numpy array*) – z component of the vector field at the grid nodes
- **csz** (*float*) – cellsize of the grid

#### Returns

- **nx** (*numpy array*) – x component of the interpolated vector field at position (x, y)
- **ny** (*numpy array*) – y component of the interpolated vector field at position (x, y)
- **nz** (*numpy array*) – z component of the interpolated vector field at position (x, y)

### com1DFA.DFAtools.getNormalMesh

**getNormalMesh**(*dem*, *num*)

Compute normal to surface at grid points

Get the normal vectors to the surface defined by a DEM. Either by adding the normal vectors of the adjacent triangles for each points (using 4, 6 or 8 adjacent triangles). Or use the next point in x direction and the next in y direction to define two vectors and then compute the cross product to get the normal vector

#### Parameters

- **dem** (*dict*) –  
**header :**  
dem header (cellsize, ncols, nrows)  
**rasterData**  
[2D numpy array] elevation at grid points
- **num** (*int*) – chose between 4, 6 or 8 (using then 4, 6 or 8 triangles) or 1 to use the simple cross product method (with the diagonals)

#### Returns

**dem** –

**dem dict updated with:**

**Nx: 2D numpy array**  
x component of the normal vector field on grid points

**Ny: 2D numpy array**  
y component of the normal vector field on grid points

**Nz: 2D numpy array**

z component of the normal vector field on grid points

**outOfDEM: 2D boolean numpy array**

True if the cell is out the dem, False otherwise

**Return type**

dict

## com1DFA.DFAtools.norm

**norm**(*x*, *y*, *z*)

Compute the Euclidean norm of the vector (*x*, *y*, *z*).

(*x*, *y*, *z*) can be numpy arrays.

**Parameters**

- **x** (*numpy array*) – x component of the vector
- **y** (*numpy array*) – y component of the vector
- **z** (*numpy array*) – z component of the vector

**Returns**

**norme** – norm of the vector

**Return type**

numpy array

## com1DFA.DFAtools.norm2

**norm2**(*x*, *y*, *z*)

Compute the square of the Euclidean norm of the vector (*x*, *y*, *z*).

(*x*, *y*, *z*) can be numpy arrays.

**Parameters**

- **x** (*numpy array*) – x component of the vector
- **y** (*numpy array*) – y component of the vector
- **z** (*numpy array*) – z component of the vector

**Returns**

**norme2** – square of the norm of the vector

**Return type**

numpy array

**com1DFA.DFAtools.normalize****normalize**(*x*, *y*, *z*)

Normalize vector (*x*, *y*, *z*) for the Euclidean norm.

(*x*, *y*, *z*) can be np arrays.

**Parameters**

- **x** (*numpy array*) – x component of the vector
- **y** (*numpy array*) – y component of the vector
- **z** (*numpy array*) – z component of the vector

**Returns**

- **x** (*numpy array*) – x component of the normalized vector
- **y** (*numpy array*) – y component of the normalized vector
- **z** (*numpy array*) – z component of the normalized vector

**com1DFA.DFAtools.scalProd****scalProd**(*ux*, *uy*, *uz*, *vx*, *vy*, *vz*)

Compute scalar product of vector *u* = (*ux*, *uy*, *uz*) and *v* = (*vx*, *vy*, *vz*).

**com1DFA.deriveParameterSet**

Create dictionary for parameter variations

## Functions

<i>appendShpThickness</i>	append thickness values to GENERAL section if read from shp and not varied
<i>checkRasterMeshSize</i>	check if cell size of raster in Inputs/ is same as desired meshCellSize if not - check for remeshed raster or remesh the raster If onlySearch is True: no remeshing is being done, only search for the right one
<i>checkResType</i>	Check if the resTypes asked for exist Warns the user if some do not, removes them from the resType list and updates the cfg
<i>checkThicknessSettings</i>	check if thickness setting format is correct
<i>createSimDict</i>	Create a simDict with all the simulations that shall be performed
<i>getParameterVariationInfo</i>	create a variation dictionary according to parameter variation given in cfg object
<i>getThicknessValue</i>	set thickness values according to settings chosen and add info to cfg if thFromShp = True - add in INPUT section thickness and id info and ci95 if thFromShp = False - check format of thickness value in GENERAL section
<i>getVariationDict</i>	Create a dictionary with all the parameters that shall be varied from the standard configuration; ONLY accounts for variations in section GENERAL and INPUT/releaseScenario
<i>setPercentVariation</i>	determine thickness value if set from percentVariation and set from shp file and update percentVariation value that is used for exactly this sim
<i>setRangeFromCiVariation</i>	determine thickness value if set from rangeFromCiVariation and set from shp file and update rangeFromCiVariation value that is used for exactly this sim
<i>setRangeVariation</i>	determine thickness value if set from rangeVariation and set from shp file and update rangeVariation value that is used for exactly this sim
<i>setThicknessValueFromVariation</i>	set thickness value for thickness parameter for all features if multiple according to desired variation
<i>setVariationForAllFeatures</i>	set thickness value for all features according to varType variation
<i>splitVariationToArraySteps</i>	split variation in percent to create a list of factors to set parameter value for variations or if a rangeVariation is given in absolute values or if distVariation create an info string on how the distribution can be build (e.g.
<i>validateVarDict</i>	Check if all parameters in variationDict exist in default configuration and are provided in the correct format
<i>writeToCfgLine</i>	write an array of values to a string of values separated by   for configuration

**com1DFA.deriveParameterSet.appendShpThickness****appendShpThickness**(*cfg*)

append thickness values to GENERAL section if read from shp and not varied

**Parameters**

**cfg** (*dict*) – configuration settings

**Returns**

**cfg** – updated configuration settings

**Return type**

dict

**com1DFA.deriveParameterSet.checkRasterMeshSize****checkRasterMeshSize**(*cfgSim, rasterFile, typeIndicator='DEM', onlySearch=False*)

check if cell size of raster in Inputs/ is same as desired meshCellSize if not - check for remeshed raster or remesh the raster If onlySearch is True: no remeshing is being done, only search for the right one

**Parameters**

- **cfgSim** (*dict*) – configuration settings of com module
- **rasterFile** (*str or pathlib path*) – to raster in Inputs/
- **typeIndicator** (*str*) – indicate which type the raster is. Possible values DEM or RELTH
- **onlySearch** (*bool*) – if True - only searching for remeshed DEM but not remeshing if not found

**Returns**

**pathToRaster** – path to raster with correct cellSize relative to Inputs/

**Return type**

str

**com1DFA.deriveParameterSet.checkResType****checkResType**(*fullCfg, section, key, value*)

Check if the resTypes asked for exist Warns the user if some do not, removes them from the resType list and updates the cfg

**Parameters**

- **fullCfg** (*configParser object*) – full configuration potentially including variations of parameter
- **section** (*str*) – section name
- **key** (*str*) – key name
- **value** (*str*) – corresponding value

**Returns**

**fullCfg** – full configuration updated with resType if this last one was modified

**Return type**

configParser object

## com1DFA.deriveParameterSet.checkThicknessSettings

**checkThicknessSettings**(*cfg*, *thName*)

check if thickness setting format is correct

### Parameters

- **cfg** (*configparser*) – configuration settings
- **thName** (*str*) – thickness parameter name (entTh, ...)

### Returns

**thicknessSettingsCorrect** – True if settings are provided in the correct format

### Return type

bool

## com1DFA.deriveParameterSet.createSimDict

**createSimDict**(*avalancheDir*, *com1DFA*, *cfgInitial*, *inputSimFiles*, *simNameExisting*)

Create a simDict with all the simulations that shall be performed

### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **com1DFA** (*module*) – computational module
- **cfgStart** (*configparser object*) – configuration settings for com1DFA
- **inputSimFiles** (*dict*) – dictionary with info in input files (release area, dem, ...)
- **simNameExisting** (*list*) – list with names of sims that already exist in outputs

### Returns

**simDict** – dictionary with info on simHash, releaseScenario, release area file path, simType and contains full configuration configparser object for simulation run

### Return type

dict

## com1DFA.deriveParameterSet.getParameterVariationInfo

**getParameterVariationInfo**(*avalancheDir*, *module*, *cfgStart*)

create a variation dictionary according to parameter variation given in cfg object

### Parameters

- **avalancheDir** (*str or pathlib Path*) – path to avalanche directory
- **module** (*module*)
- **cfgStart** (*configparser object*) – full configuration object

### Returns

- **modCfg** (*configparser object*) – configuration of simulations to be performed
- **variationDict** (*dict*) – dictionary with information on parameter variations

### com1DFA.deriveParameterSet.getThicknessValue

**getThicknessValue**(*cfg*, *inputSimFiles*, *fName*, *thType*)

set thickness values according to settings chosen and add info to *cfg* if *thFromShp* = True - add in INPUT section thickness and id info and *ci95* if *thFromShp* = False - check format of thickness value in GENERAL section

#### Parameters

- **cfg** (*configparser object*) – configuration settings
- **inputSimFiles** (*dict*) – dictionary with info on input files and attributes (id and thickness)
- **fName** (*str*) – name of scenario shp file (entrainment, release, ...)
- **thType** (*str*) – thickness parameter name (entTh, relTh, ...)

#### Returns

**cfg** – updated configuration settings with info on actual thickness values to be used for simulations

#### Return type

configparser object

### com1DFA.deriveParameterSet.getVariationDict

**getVariationDict**(*avaDir*, *fullCfg*, *modDict*)

Create a dictionary with all the parameters that shall be varied from the standard configuration; ONLY accounts for variations in section GENERAL and INPUT/releaseScenario

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **fullCfg** (*configParser object*) – full configuration potentially including variations of parameter
- **modDict** (*dict*) – info on modifications to standard configuration

#### Returns

**variationDict** – dictionary with the parameters that shall be varied and the chosen flags for the run

#### Return type

dict

### com1DFA.deriveParameterSet.setPercentVariation

**setPercentVariation**(*cfg*, *variationFactor*, *thNameId*)

determine thickness value if set from *percentVariation* and set from shp file and update *percentVariation* value that is used for exactly this sim

this is required for reproducing this sim when using its configuration file - so that in the *percentVariation* parameter it is only one value e.g. +50\$1 so +50% in 1 step

#### Parameters

- **cfg** (*configparser object*) – comModule configuration file with info on thickness settings
- **variationFactor** (*float*) – value of percent variation in terms of required multiplication of reference value (e.g. *variationFactor*= 0.5 - a variation of 50% performed by multiplication of reference value times *variationFactor*)



- **thNameId** (*str*) – name of thickness feature (e.g. relTh0 if release thickness and feature with id 0)

**Returns**

**variationIni** – percentVariation parameter value for this sim to be added in cfg file

**Return type**

str

**com1DFA.deriveParameterSet.setRangeFromCiVariation****setRangeFromCiVariation**(*cfg, variationFactor, thValue, ciValue*)

determine thickness value if set from rangeFromCiVariation and set from shp file and update rangeFromCiVariation value that is used for exactly this sim

this is required for reproducing this sim when using its configuration file - so that in the rangeFromCiVariation parameter it is only one value e.g. ci95\$4\$1 so -ci95m

**Parameters**

- **cfg** (*configparser object*) – comModule configuration file with info on thickness settings
- **variationFactor** (*str*) – value of range variation in terms of required addition to the reference value (e.g. variationFactor= chi95\$4\$0 - a variation of -ci95 value performed by addition of reference value + the variation value)
- **thValue** (*str*) – thickness value of thickness feature in meter
- **ciValue** (*str*) – ci value of thickness features in meter

**Returns**

- **variationValue** (*float*) – actual thickness value modified according to variation for feature
- **variationIni** (*str*) – rangeFromCiVariation parameter value for this sim to be added in cfg file

**com1DFA.deriveParameterSet.setRangeVariation****setRangeVariation**(*cfg, variationFactor, thNameId*)

determine thickness value if set from rangeVariation and set from shp file and update rangeVariation value that is used for exactly this sim

this is required for reproducing this sim when using its configuration file - so that in the rangeVariation parameter it is only one value e.g. +0.5\$1 so +0.5m in 1 step

**Parameters**

- **cfg** (*configparser object*) – comModule configuration file with info on thickness settings
- **variationFactor** (*float*) – value of range variation in terms of required addition to the reference value (e.g. variationFactor= +0.5 - a variation of +0.5m performed by addition of reference value + variationFactor)
- **thNameId** (*str*) – name of thickness feature (e.g. relTh0 if release thickness and feature with id 0)

**Returns**

**variationIni** – rangeVariation parameter value for this sim to be added in cfg file

**Return type**

str

**com1DFA.deriveParameterSet.setThicknessValueFromVariation****setThicknessValueFromVariation**(*key*, *cfg*, *simType*, *row*)

set thickness value for thickness parameter for all features if multiple according to desired variation

**Parameters**

- **key** (*str*) – thickness variation info
- **cfg** (*configparser object*) – configuration settings of comModule
- **simType** (*str*) – simulation type (null, ent, entres, ..)
- **row** (*pandas row*) – info on variation of parameters

**Returns****cfg** – updated dict with info on thickness**Return type**

dict

**com1DFA.deriveParameterSet.setVariationForAllFeatures****setVariationForAllFeatures**(*cfg*, *key*, *thType*, *varType*, *variationFactor*)

set thickness value for all features according to varType variation

**Parameters**

- **cfg** (*configparser*) – configuration settings of comModule for thickness
- **key** (*str*) – name of parameter
- **thType** (*str*) – thickness type (e.g. relTh, entTh, ...)
- **varType** (*str*) – type of variation (range or percent)
- **variationFactor** (*float or str (if Dist or rangeFromCi)*) – value used for variation

**Returns****cfg** – updated configuration settings regarding thickness settings**Return type**

configparser

**com1DFA.deriveParameterSet.splitVariationToArraySteps****splitVariationToArraySteps**(*value*, *key*, *fullCfg*)

split variation in percent to create a list of factors to set parameter value for variations or if a rangeVariation is given in absolute values or if distVariation create an info string on how the distribution can be build (e.g. of format typeOfDistribution\$numberOfSteps\$ci95value\$ci95\$support and append the step of the current variation in front)

**Parameters**

- **value** (*str*) – value read from configuration

- **key** (*str*) – name of parameter
- **fullCfg** (*configparser*) – full configuration settings

**Returns**

**itemsArray** – factor to change parameter values by multiplication (Percent) or addition (Range) or info string on how to build the distribution and which step to draw from it

**Return type**

numpy array

**com1DFA.deriveParameterSet.validateVarDict**

**validateVarDict** (*variationDict*, *standardCfg*)

Check if all parameters in variationDict exist in default configuration and are provided in the correct format

**Parameters**

- **variationDict** (*dict*) – dictionary with parameters that shall be varied and a list for the parameter values for each parameter
- **standardCfg** (*config Parser object*) – default model configuration

**Returns**

**variationDict** – cleaned variation dict that meets the required structure

**Return type**

dict

**com1DFA.deriveParameterSet.writeToCfgLine**

**writeToCfgLine** (*values*)

write an array of values to a string of values separated by | for configuration

**Parameters**

**values** (*numpy array*) – array of values

**Returns**

**valString** – string of array values separated by |

**Return type**

str

**com1DFA.particleInitialisation**

functions for initialising particle distribution

## Functions

<i>createReleaseBuffer</i>	add a buffer around release polygons to get boundary particles
<i>getIniPosition</i>	Redistribute particles so that SPH force reduces with fixed particles as boundaries
<i>resetMassPerParticle</i>	recompute mass of particles according to their location with respect to relRaster

### com1DFA.particleInitialisation.createReleaseBuffer

**createReleaseBuffer**(*cfg*, *inputSimLines*)

add a buffer around release polygons to get boundary particles

**Parameters**

- **cfg** (*configparser object*) – configuration settings
- **inputSimLines** (*dict*) – dictionary with input data info

**Returns**

**inputSimLines** – updated inputSimLines with releaseLineBuffer

**Return type**

dict

### com1DFA.particleInitialisation.getIniPosition

**getIniPosition**(*cfg*, *particles*, *dem*, *fields*, *inputSimLines*, *relThField*)

Redistribute particles so that SPH force reduces with fixed particles as boundaries

**Parameters**

- **cfg** (*configparser object*) – configuration settings
- **particles** (*dict*) – dictionary with particle properties
- **dem** (*dict*) – dictionary with dem header and data
- **fields** (*dict*) – dictionary with fields of result types
- **inputSimLines** (*dict*) – info on input files
- **relThField** (*numpy array*) – release thickness field

**Returns**

- **particles** (*dict*) – updated particles dict with new positions
- **fields** (*fields*) – updated fields dictionary

**com1DFA.particleInitialisation.resetMassPerParticle****resetMassPerParticle**(*cfg, particles, dem, relRaster, relThField*)

recompute mass of particles according to their location with respect to relRaster

**Parameters**

- **cfg** (*configparser object*) – configuration settings
- **particles** (*dict*) – dictionary with particles properties
- **dem** (*dict*) – dictionary with info on dem
- **relRaster** (*np.array*) – raster of release thickness values

**Returns****particles** – updated particles dictionary with new mass**Return type**

dict

**com1DFA.timeDiscretizations**

Functions regarding time discretization and time stepping for com1DFA

**Functions***getSphKernelRadiusTimeStep*

Compute the time step given the sph kernel radius and the cMax coefficient This is based on the article from Ben Moussa et Vila DOI:10.1137/S0036142996307119

**com1DFA.timeDiscretizations.getSphKernelRadiusTimeStep****getSphKernelRadiusTimeStep**(*dem, cfg*)

Compute the time step given the sph kernel radius and the cMax coefficient This is based on the article from Ben Moussa et Vila DOI:10.1137/S0036142996307119

**Parameters**

- **dem** (*dict*) – dem dictionary (with info about sph kernel radius and mesh size)
- **cfg** (*configparser*) – the cfg cith cMax

**Returns****dtStable** – corresponding time step**Return type**

float

### 2.10.1.2 com2AB

AlphaBeta kernel

#### Modules

---

<i>com2AB.com2AB</i>	Main module for Alpha beta
----------------------	----------------------------

---

#### com2AB.com2AB

Main module for Alpha beta

#### Functions

---

<i>calcABAngles</i>	Kernel function that computes the AlphaBeta model (angular results) for a given avaProfile and eqParameters
<i>calcABDistances</i>	Compute runout distances and points from angles computed in calcABAngles
<i>com2ABKern</i>	Compute AlpahBeta model for a given avapath
<i>com2ABMain</i>	Main AlphaBeta model function
<i>readABInputs</i>	Fetch inputs for AlpahBeta model
<i>setEqParameters</i>	Set alpha beta equation parameters

---

#### com2AB.com2AB.calcABAngles

##### calcABAngles(*avaProfile*, *eqParameters*, *dsMin*)

Kernel function that computes the AlphaBeta model (angular results) for a given avaProfile and eqParameters

##### Parameters

- **avaProfile** (*dict*) – dictionary with the name of the avapath, the x, y and z coordinates of the path
- **eqParameters** (*dict*) – AB parameter dictionary
- **dsMin** (*float*) – threshold distance [m] when looking for the Beta point

##### Returns

**avaProfile** – updated avaProfile with alpha, beta and other values resulting from the AlphaBeta model computation

##### Return type

dict

**com2AB.com2AB.calcABDistances****calcABDistances**(*avaProfile*, *name*)

Compute runout distances and points from angles computed in calcABAngles

**Parameters**

- **avaProfile** (*dict*) – dictionary with the name of the avapath, the x, y and z coordinates of the path
- **name** (*str*) – profile name

**Returns**

**avaProfile** – updated avaProfile with s index of alpha, and alphaSD points

**Return type**

dict

**com2AB.com2AB.com2ABKern****com2ABKern**(*avaPath*, *splitPoint*, *dem*, *eqParams*, *distance*, *dsMin*)

Compute AlphahBeta model for a given avapath

Call calcABAngles to compute the AlphaBeta model given an input raster (of the dem), an avalanche path and split points

**Parameters**

- **avaPath** (*dict*) – dictionary with the name of the avaPath, the x and y coordinates of the path
- **splitPoint** (*dict*) – dictionary split points
- **dem** (*dict*) – dem dictionary used to get the avaProfile from the avaPath
- **eqParams** (*dict*) – dict containing the AB model parameters (produced by setEqParameters and depends on the com2ABCfg.ini)
- **distance** (*float*) – line resampling distance
- **dsMin** (*float*) – threshold distance [m] when looking for the beta point

**Returns**

**avaProfile** – avaPath dictionary with AlphaBeta model results (path became a profile adding the z and s arrays. AB runout angles and distances)

**Return type**

dict

**com2AB.com2AB.com2ABMain****com2ABMain**(*cfg*, *avalancheDir*)

Main AlphaBeta model function

Loops on the given AvaPaths and runs com2AB to compute AlphahBeta model

**Parameters**

- **cfg** (*configparser*) – configparser with all requiered fields in com2ABCfg.ini
- **avalancheDir** (*str*) – path to directory of avalanche to analyze

**Returns**

- **pathDict** (*dict*) – dictionary with AlphaBeta inputs
- **dem** (*dict*) – dem dictionary used to get the avaProfile from the avaPath
- **splitPoint** (*dict*) – split point dict
- **eqParams** (*dict*) – dict containing the AB model parameters (produced by setEqParameters and depends on the com2ABCfg.ini)
- **resAB** (*dict*) – dictionary with AlphaBeta model results

**com2AB.com2AB.readABInputs****readABInputs**(*avalancheDir*, *path2Line*="", *path2SplitPoint*="")

Fetch inputs for AlpahBeta model

Get path to AlphaBeta model inputs (dem raster, avalanche path and split points)

**Parameters**

- **avalancheDir** (*str*) – path to directory of avalanche to analyze
- **path2Line** (*pathlib path*) – pathlib path to altrnative line (if empty, reading the line from the input directory Inputs/LINES/yourNameAB.shp)
- **path2SplitPoint** (*pathlib path*) – pathlib path to altrnative splitPoint (if empty, reading the point from the input directory Inputs/LINES/yourNameAB.shp)

**Returns****pathDict** – dictionary with path to AlphaBeta inputs (dem, avaPath, splitPoint)**Return type**

dict

**com2AB.com2AB.setEqParameters****setEqParameters**(*cfg*, *smallAva*)

Set alpha beta equation parameters

Set alpha beta equation parameters to - standard (default) - small avalanche (if smallAva==True) - custom (if cfgsetup('customParam') is True use the custom values provided in the ini file)

**Parameters**

- **cfg** (*configParser*) – if cfgsetup('customParam') is True, the custom parameters provided in the cfg are used in the AlphaBeta equation. (provide all 5 k1, k2, k3, k4 and SD values)
- **smallAva** (*boolean*) – True if the small avallanche AlphaBeta equation parameters should be used

**Returns****eqParameters** – k1, k2, k3, k4 and SD values to be used in the AlphaBeta equation**Return type**

dict



### 2.10.1.3 com3Hybrid

Hybrid modelling: combining DFA simulation and statistical approaches

#### Modules

---

`com3Hybrid.com3Hybrid`

---

## 2.10.2 Input/Transformation Modules

<i>in1Data</i>	Tools for fetching or generating input data
<i>in2Trans</i>	Tools for input transformations
<i>in3Utils</i>	Tools for geo transformations

### 2.10.2.1 in1Data

Tools for fetching or generating input data

#### Modules

<i>in1Data.computeFromDistribution</i>	Generate sample of values following a beta-pert or a uniform distribution
<i>in1Data.getInput</i>	Fetch input data for avalanche simulations

#### in1Data.computeFromDistribution

Generate sample of values following a beta-pert or a uniform distribution

#### Functions

<i>computeParameters</i>	Compute alpha, beta and mu
<i>computePert</i>	Compute the CDF and PDF of the Pert distribution using scipy betainc function
<i>extractFromCDF</i>	Extract a sample from the CDF with prescribed steps
<i>extractNormalDist</i>	create a normal distribution from given parameters and draw a sample
<i>extractUniform</i>	Extract sample of a uniform distriution
<i>getEmpiricalCDF</i>	Derive empirical CDF using numpy histogram and cum-sum
<i>getEmpiricalCDFNEW</i>	Derive empirical CDF using sorted sample

**in1Data.computeFromDistribution.computeParameters****computeParameters**(*a, b, c*)

Compute alpha, beta and mu

**in1Data.computeFromDistribution.computePert****computePert**(*a, b, c, x, alpha, beta*)

Compute the CDF and PDF of the Pert distribution using scipy betainc function

**in1Data.computeFromDistribution.extractFromCDF****extractFromCDF**(*CDF, CDFint, x, cfg*)

Extract a sample from the CDF with prescribed steps

**in1Data.computeFromDistribution.extractNormalDist****extractNormalDist**(*cfg*)

create a normal distribution from given parameters and draw a sample

**Parameters****cfg** (*configparser object or dict*) – configuration settings for computing distributions, std or ci95, minMaxInterval, flagMinMax**Returns**

- **CDFint** (*function object*) – interpolated CDF function
- **sampleVect** (*numpy array*) – sample values
- **pdf** (*numpy array*) – pdf values computed for normal distribution and support x
- **x** (*numpy array*) – support x

**in1Data.computeFromDistribution.extractUniform****extractUniform**(*a, c, x, cfg*)

Extract sample of a uniform distriution

**in1Data.computeFromDistribution.getEmpiricalCDF****getEmpiricalCDF**(*sample*)

Derive empirical CDF using numpy histogram and cumsum

**in1Data.computeFromDistribution.getEmpiricalCDFNEW****getEmpiricalCDFNEW**(*sample*)

Derive empirical CDF using sorted sample

**in1Data.getInput**

Fetch input data for avalanche simulations

**Functions**

<i>checkForMultiplePartsShpArea</i>	check if in polygon read from shape file holes are present, if so error and save a plot to Outputs/com1DFA procedure: check if polygon has several parts
<i>computeAreasFromLines</i>	compute the area of a polygon in xy using shapely
<i>computeAreasFromRasterAndLine</i>	compute the area covered by a polygon by creating a raster from polygon projected area and actual area using a dem info
<i>computeRelStats</i>	compute stats of a polygon and a dem actual area (taking slope into account), projected area, max, mean, min elevation, mean slope
<i>createReleaseStats</i>	create a csv file with info on release shp file on: max, mean and min elevation, slope and projected and real area
<i>fetchReleaseFile</i>	select release scenario, update configuration to only include thickness info of current scenario and return file path
<i>getAndCheckInputFiles</i>	Fetch fileExt files and check if they exist and if it is not more than one
<i>getDEMFromConfig</i>	get dem file path in avaDir/Inputs
<i>getDEMPath</i>	get the DEM file path from a provided avalanche directory
<i>getInputData</i>	Fetch input datasets required for simulation, duplicated function because
<i>getInputDataCom1DFA</i>	Fetch input datasets required for simulation, duplicated function because now fetch all available files simulation type set differently in com1DFA compared to com1DFAOrig: TODO: remove duplicate once it is not required anymore
<i>getInputPaths</i>	Fetch paths to dem and first release area shp file found
<i>getThicknessInputSimFiles</i>	add thickness of shapefiles to dictionary
<i>initializeDEM</i>	check for dem and load to dict
<i>initializeRelTh</i>	check for relThFile and load to dict (if relTh is read from file)
<i>readDEM</i>	read the ascii DEM file from a provided avalanche directory
<i>selectReleaseFile</i>	select release scenario
<i>updateThicknessCfg</i>	add available release scenarios to ini file and set thickness values in ini files

### in1Data.getInput.checkForMultiplePartsShpArea

**checkForMultiplePartsShpArea**(*avaDir*, *lineDict*, *modName*, *type=""*)

check if in polygon read from shape file holes are present, if so error and save a plot to Outputs/com1DFA  
procedure: check if polygon has several parts

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **lineDict** (*dict*) – dictionary with info read from shape file used: x, y, Start, Length, nParts, nFeatures
- **modName** (*str*) – name of computational module where to save error plots to Outputs sub-folder
- **type** (*str*) – type of shp file area (release, secondary release, entrainment, resistance)

#### Returns

- *error if number of parts is bigger 1*
- *save plot showing all parts of each polygon feature in Outputs/modName*

### in1Data.getInput.computeAreasFromLines

**computeAreasFromLines**(*line*)

compute the area of a polygon in xy using shapely

#### Parameters

**line** (*dict*) – dictionary with info on line x, y coordinates start, end of each line feature

#### Returns

**projectedAreas** – list of projected area for each polygon in line dict

#### Return type

list

### in1Data.getInput.computeAreasFromRasterAndLine

**computeAreasFromRasterAndLine**(*line*, *dem*)

compute the area covered by a polygon by creating a raster from polygon projected area and actual area using a dem info

#### Parameters

- **line** (*dict*) – dictionary with info on line x, y coordinates start, end of each line feature
- **dem** (*dict*) – dictionary with dem data, header and areaRaster

#### Returns

- **areaActual** (*float*) – actual area taking slope into account by using dem area
- **areaProjected** (*float*) – projected area in xy plane

**in1Data.getInput.computeRelStats****computeRelStats**(*line*, *dem*)

compute stats of a polygon and a dem actual area (taking slope into account), projected area, max, mean, min elevation, mean slope

**Parameters**

- **line** (*dict*) – dictionary with info on line (x, y, Start, Length, rasterData, ...)
- **dem** (*dict*) – dictionary with info on dem (header, rasterData, normals, areaRaster)

**Returns**

**lineDict** – dictionary with stats info: featureNames, zMax, zMin, Slope, Area, AreaP

**Return type**

dict

**in1Data.getInput.createReleaseStats****createReleaseStats**(*avaDir*, *cfg*)

create a csv file with info on release shp file on: max, mean and min elevation, slope and projected and real area

**Returns**

**fPath** – file path

**Return type**

pathlib Path

**in1Data.getInput.fetchReleaseFile****fetchReleaseFile**(*inputSimFiles*, *releaseScenario*, *cfgSim*, *releaseList*)

select release scenario, update configuration to only include thickness info of current scenario and return file path

**Parameters**

- **inputSimFiles** (*dict*) – dictionary with info on input data
- **releaseScenario** (*str*) – name of release scenario
- **cfgSim** (*conigparser object*) – configuration of simulation
- **releaseList** (*list*) – list of available release scenarios

**Returns**

- **releaseScenarioPath** (*pathlib path*) – file path to release scenario shp file
- **cfgSim** (*configparser object*) – updated cfg object, removed thickness info from not other release scenarios than used one and rename thickness values of chosen scenario to relThThickness, relThId, ...

### **in1Data.getInput.getAndCheckInputFiles**

**getAndCheckInputFiles**(*inputDir*, *folder*, *inputType*, *fileExt*='shp')

Fetch fileExt files and check if they exist and if it is not more than one

Raises error if there is more than one fileExt file.

#### **Parameters**

- **inputDir** (*pathlib object or str*) – path to avalanche input directory
- **folder** (*str*) – subfolder name where the shape file should be located (SECREL, ENT or RES)
- **inputType** (*str*) – type of input (used for the logging messages). Secondary release or En-trainment or Resistance
- **fileExt** (*str*) – file extension e.g. shp, asc - optional default is shp

#### **Returns**

- **OutputFile** (*str*) – path to file checked
- **available** (*str*) – Yes or No depending of if there is a shape file available (if No, OutputFile is None)

### **in1Data.getInput.getDEMFromConfig**

**getDEMFromConfig**(*avaDir*, *fileName*='')

get dem file path in avaDir/Inputs

#### **Parameters**

- **avaDir** (*str or pathlib path*) – to avalancheDir
- **fileName** (*pathlib path*) – to dem file with filename in avaDir/Inputs

#### **Returns**

**demFile** – to dem file

#### **Return type**

pathlib path

### **in1Data.getInput.getDEMPath**

**getDEMPath**(*avaDir*)

get the DEM file path from a provided avalanche directory

#### **Parameters**

**avaDir** (*str*) – path to avalanche directory

#### **Returns**

**demFile** – full path to DEM .asc file

#### **Return type**

str (first element of list)

## in1Data.getInput.getInputData

### getInputData(*avaDir*, *cfg*)

**Fetch input datasets required for simulation, duplicated function because**

simulation type set differently in com1DFAOrig compared to com1DFA: TODO: remove duplicate once it is not required anymore

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **cfg** (*dict*) – configuration read from com1DFA simulation ini file

#### Returns

- **demFile[0]** (*str (first element of list)*) – list of full path to DEM .asc file
- **relFiles** (*list*) – list of full path to release area scenario .shp files
- **entFile** (*str*) – full path to entrainment area .shp file
- **resFile** (*str*) – full path to resistance area .shp file
- **wallFile** (*str*) – full path to wall line .shp file
- **entResInfo** (*flag dict*) – flag if Yes entrainment and/or resistance areas found and used for simulation

## in1Data.getInput.getInputDataCom1DFA

### getInputDataCom1DFA(*avaDir*)

Fetch input datasets required for simulation, duplicated function because now fetch all available files simulation type set differently in com1DFA compared to com1DFAOrig: TODO: remove duplicate once it is not required anymore

#### Parameters

**avaDir** (*str or pathlib object*) – path to avalanche directory

#### Returns

**inputSimFiles** – dictionary with all the input files

- **demFile** : str (first element of list), list of full path to DEM .asc file
- **relFiles** : list, list of full path to release area scenario .shp files
- **secondaryReleaseFile** : str, full path to secondary release area .shp file
- **entFile** : str, full path to entrainment area .shp file
- **resFile** : str, full path to resistance area .shp file
- **entResInfo** : flag dict

flag if Yes entrainment and/or resistance areas found and used for simulation flag True if a Secondary Release file found and activated

#### Return type

dict

## **in1Data.getInput.getInputPaths**

### **getInputPaths**(*avaDir*)

Fetch paths to dem and first release area shp file found

#### **Parameters**

**avaDir** (*str or pathlib object*) – path to avalanche directory

#### **Returns**

- **demFile** (*pathlib path*) – full path to DEM .asc file
- **relFiles** (*list*) – list of full paths to release area scenario .shp files found in *avaDir/Inputs/REL*
- **relFieldFiles** (*list*) – list of full paths to release area thickness .asc files found in *avaDir/Inputs/RELTH*

## **in1Data.getInput.getThicknessInputSimFiles**

### **getThicknessInputSimFiles**(*inputSimFiles*)

add thickness of shapefiles to dictionary

#### **Parameters**

**inputSimFiles** (*dict*) – dictionary with info on release and entrainment file paths

#### **Returns**

**inputSimFiles** – updated dictionary with thickness info read from shapefile attributes now includes one separate dictionary for each release, entrainment or secondary release scenario with a thickness and id value for each feature (given as list)

#### **Return type**

dict

## **in1Data.getInput.initializeDEM**

### **initializeDEM**(*avaDir, demPath=""*)

check for dem and load to dict

#### **Parameters**

- **avaDir** (*str or pathlib path*) – path to avalanche directory
- **demPath** (*str or pathlib Path*) – path to dem relative to Inputs - optional if not provided read DEM from Inputs

#### **Returns**

**demOri** – dem dictionary with header and data

#### **Return type**

dict



**in1Data.getInput.initializeRelTh****initializeRelTh**(*cfg, dOHeader*)

check for relThFile and load to dict (if relTh is read from file)

**Parameters**

- **cfg** (*configparser*) – used are avalancheDir and relThFile
- **demHeader** – header of dem to check for matching rows and numbers

**Returns**

- **relThFieldData** (*ndarray or str*) – with release thickness field data if not relThFromFile an empty string is returned
- **relThFile** (*path*) – updated path to relThFile (needed in case of remeshing)

**in1Data.getInput.readDEM****readDEM**(*avaDir*)

read the ascii DEM file from a provided avalanche directory

**Parameters****avaDir** (*str*) – path to avalanche directory**Returns****dem** – dict with header and raster data**Return type**

dict

**in1Data.getInput.selectReleaseFile****selectReleaseFile**(*inputSimFiles, releaseScenario*)

select release scenario

**Parameters**

- **inputSimFiles** (*dict*) – dictionary with info on input data
- **releaseScenario** (*str*) – name of release scenario

**Returns****inputSimFiles** – dictionary with info on input data updated with releaseScenario**Return type**

dict

## in1Data.getInput.updateThicknessCfg

### updateThicknessCfg(*inputSimFiles*, *cfgInitial*)

add available release scenarios to ini file and set thickness values in ini files

#### Parameters

- **inputSimFiles** (*dict*) – dictionary with info on release and entrainment file paths
- **cfgInitial** (*configParser object*) – with the current (and possibly overridden) configuration

#### Returns

- **inputSimFiles** (*dict*) – updated dictionary with thickness info read from shapefile attributes now includes one separate dictionary for each release, entrainment or secondary release scenario with a thickness and id value for each feature (given as list)
- **cfgInitial** (*configparser object*) – updated config object with release scenario, thickness info, etc.

## 2.10.2.2 in2Trans

Tools for input transformations

### Modules

<i>in2Trans.ascUtils</i>	ASCII file reader and handler
<i>in2Trans.shpConversion</i>	Conversion functions to read/ write Shape files

### in2Trans.ascUtils

ASCII file reader and handler

### Functions

<i>isEqualASHeader</i>	Test if two headers (A,B) are the same (except for noData Values)
<i>readASCdata2numpyArray</i>	Read ascii matrix as numpy array
<i>readASHeader</i>	return a class with information from an ascii file header
<i>readRaster</i>	Read raster file (.asc)
<i>writeResultToAsc</i>	Write 2D array to an ascii file with header and save to location of outFileName

**in2Trans.ascUtils.isEqualASCHheader****isEqualASCHheader**(*headerA*, *headerB*)

Test if two headers (A,B) are the same (except for noData Values)

**Parameters**

- **headerA** (*class*)
- **headerB** (*class*)

**Returns****boolean****Return type**

True if header A and B are equal (disregard the noData field)

**in2Trans.ascUtils.readASCdata2numpyArray****readASCdata2numpyArray**(*fName*)

Read ascii matrix as numpy array

**Parameters****fName** (*str or pathlib object*) – path to ascii file**Returns****-rasterdata** – 2D numpy array of ascii matrix**Return type**

2D numpy array

**in2Trans.ascUtils.readASCHheader****readASCHheader**(*fName*)

return a class with information from an ascii file header

**Parameters****fName** (*str or pathlib object*) – path to ascii file**Returns****headerInfo** – information that is stored in header (ncols, nrows, xllcenter, yllcenter, no-data\_value)**Return type**

class

**in2Trans.ascUtils.readRaster****readRaster**(*fName*, *noDataToNan=True*)

Read raster file (.asc)

**Parameters**

- **fName** (*str or pathlib object*) – path to ascii file
- **noDataToNan** (*bool*) – if True convert nodata\_values to nan and set nodata\_value to nan

**Returns**

**data** –

**-headerInfo: class**

information that is stored in header (ncols, nrows, xllcenter, yllcenter, nodata\_value)

**-rasterdata**

[2D numpy array] 2D numpy array of ascii matrix

**Return type**

dict

**in2Trans.ascUtils.writeResultToAsc**

**writeResultToAsc**(*header, resultArray, outFileName, flip=False*)

Write 2D array to an ascii file with header and save to location of outFileName

**Parameters**

- **header** (*class*) – class with methods that give cellsize, nrows, ncols, xllcenter yllcenter, no-data\_value
- **resultArray** (*2D numpy array*) – 2D numpy array of values that shall be written to file
- **outFileName** (*str*) – path incl. name of file to be written
- **flip** (*boolean*) – if True, flip the rows of the resultArray when writing

**in2Trans.shpConversion**

Conversion functions to read/ write Shape files

**Functions**

<i>SHP2Array</i>	Read shapefile and convert it to a python dictionary
<i>extractFeature</i>	Extract feature nFeature2Extract from featureIn
<i>getSHPProjection</i>	Fetch projection from shp file
<i>readLine</i>	Read line from .shp Use SHP2Array to read the shape file.
<i>readPoints</i>	Read points from .shp Use SHP2Array to read the shape file.
<i>readThickness</i>	Read shapefile and fetch info on features' ids and thickness values
<i>removeFeature</i>	Remove feature number nFeature2Remove from featureIn
<i>writeLine2SHPfile</i>	write a line to shapefile
<i>writePoint2SHPfile</i>	write a point to shapefile

## in2Trans.shpConversion.SHP2Array

**SHP2Array**(*infile*, *defname=None*)

Read shapefile and convert it to a python dictionary

The dictionary contains the name of the paths in the shape file, the np array with the coordinates of the feature points (all stacked in the same array) and information about the starting index and length of each feature

### Parameters

- **infile** (*str*) – path to shape file
- **defname** (*str*) – name to give to the feature in the shape file

### Returns

**SHPdata** –

**sks :**

projection information

**Name**

[str] list of feature names

**x**

[1D numpy array] np array of the x coord of the points in the features

**y**

[1D numpy array] np array of the y coord of the points in the features

**z**

[1D numpy array] np array of zeros and same size as the x and y coordinates array

**Start**

[1D numpy array] np array with the starting index of each feature in the coordinates arrays (as many indexes as features)

**Length**

[1D numpy array] np array with the length of each feature in the coordinates arrays (as many indexes as features)

**thickness (optional)**

[1D numpy array] np array with the (release or entrainment) thickness of each feature (as many values as features)

**id**

[list] list of oid as string for each feature

**ci95: list**

list of 95% confidence interval of thickness value

**nParts: list**

list of parts of polygon (added the total number of points as list item, so if multiple parts len>2)

**nFeatures: int**

number of features per line (parts)

### Return type

dict

### in2Trans.shpConversion.extractFeature

**extractFeature**(*featureIn*, *nFeature2Extract*)

Extract feature *nFeature2Extract* from *featureIn*

**Parameters**

- **featureIn** (*dict*) – shape file dictionary (structure produced by SHP2Array, readLine or readPoint)
- **nFeature2Extract** (*int*) – index of feature to extract from *featureIn*

**Returns**

**featureOut** – shape file dictionary with feature *nFeature2Extract*

**Return type**

dict

### in2Trans.shpConversion.getSHPProjection

**getSHPProjection**(*infile*)

Fetch projection from shp file

**Parameters**

**infile** (*str*) – path to shape file

**Returns**

**sks** – projection string (if available, None if not)

**Return type**

str

### in2Trans.shpConversion.readLine

**readLine**(*fname*, *defname*, *dem*)

Read line from .shp Use SHP2Array to read the shape file. Check if the lines are laying inside the dem extend

**Parameters**

- **fname** (*str*) – path to shape file
- **defname** (*str*) – name to give to the line in the shape file
- **dem** (*dict*) – dem dictionary

**Returns**

**Line** – Line['Name'] : list of lines names Line['Coord'] : np array of the coords of points in lines Line['Start'] : list of starting index of each line in Coord Line['Length'] : list of length of each line in Coord

**Return type**

dict

## in2Trans.shpConversion.readPoints

**readPoints**(*fname*, *dem*)

Read points from .shp Use SHP2Array to read the shape file. Check if the points are laying inside the dem extend

### Parameters

- **fname** (*str*) – path to shape file
- **defname** (*str*) – name to give to the points in the shape file
- **dem** (*dict*) – dem dictionary

### Returns

**Line** – Line['Name'] : list of points names Line['Coord'] : np array of the coords of points in points Line['Start'] : list of starting index of each point in Coord Line['Length'] : list of length of each point in Coord

### Return type

dict

## in2Trans.shpConversion.readThickness

**readThickness**(*infile*, *defname=None*)

Read shapefile and fetch info on features' ids and thickness values

### Parameters

- **infile** (*str*) – path to shape file
- **defname** (*str*) – name to give to the feature in the shape file

### Returns

- **thickness** (*list*) – list of strings with the (release or entrainment) thickness of each feature (as many values as features)
- **id** (*list*) – list of strings for oid of each feature in shp file
- **ci95** (*list*) – list of all ci95 values if provided

## in2Trans.shpConversion.removeFeature

**removeFeature**(*featureIn*, *nFeature2Remove*)

Remove feature number nFeature2Remove from featureIn

### Parameters

- **featureIn** (*dict*) – shape file dictionary (structure produced by SHP2Array, readLine or read-Point)
- **nFeature2Remove** (*int*) – index of feature to remove from featureIn

### Returns

**featureOut** – shape file dictionary without feature nFeature2Remove

### Return type

dict

### **in2Trans.shpConversion.writeLine2SHPfile**

**writeLine2SHPfile**(*lineDict*, *lineName*, *fileName*, *header=""*)

write a line to shapefile

#### **Parameters**

- **lineDict** (*dict*) – line dict
- **lineName** (*str*) – line name
- **fileName** (*str or pathlib path*) – path where the line will be saved line name
- **header** (*dict*) – optional argument ("" by default). If provided, header dictionary with 'xll-center' and 'yllcenter' to add to the line

#### **Returns**

**fileName** – path where the line has been saved

#### **Return type**

str

### **in2Trans.shpConversion.writePoint2SHPfile**

**writePoint2SHPfile**(*pointDict*, *pointName*, *fileName*)

write a point to shapefile

#### **Parameters**

- **pointDict** (*dict*) – point dict
- **pointName** (*str*) – point name
- **fileName** (*str or pathlib path*) – path where the point will be saved

#### **Returns**

**fileName** – path where the point has been saved

#### **Return type**

str

### **2.10.2.3 in3Utils**

Tools for geo transformations



## Modules

<i>in3Utils.cfgHandling</i>	Utilities for working with cfg info
<i>in3Utils.cfgUtils</i>	Utilities for handling configuration files
<i>in3Utils.fileHandlerUtils</i>	Directory and file handling helper functions
<i>in3Utils.generateTopo</i>	Create generic/idealised topographies
<i>in3Utils.geoTrans</i>	Operations and transformations of rasters and lines
<i>in3Utils.getReleaseArea</i>	Get release area corner coordinates for a rectangle of an area of approx.
<i>in3Utils.initialiseDirs</i>	Setup required directory structures by computational modules
<i>in3Utils.initializeProject</i>	Functions to initialize project, i.e create folder structure
<i>in3Utils.logUtils</i>	Defining a writable object to write the config to the log file

### in3Utils.cfgHandling

Utilities for working with cfg info

## Functions

<i>addInfoToSimName</i>	Add parameterName and value to simNames of simulation dataframe
<i>applyCfgOverride</i>	override configuration parameter values with the values provided in cfgWithOverrideParameters[modName_override] if addModValues True update the cfgWithOverrideParameters with the values for all parameters that are not provided in the override parameters
<i>errorDuplicateListEntry</i>	check if duplicate entries appear in a list and raise Assertion error using message
<i>fetchAndOrderSimFiles</i>	Filter simulations results using a list of parameters and a flag if in ascending or descending order
<i>fetchValidationString</i>	create a validation string to be checked if simDFrow matches filtering criteria (given by val)
<i>filterCom1DFAThicknessValues</i>	thickness settings different if read from shpfile - requires more complex filtering if read from shp - thickness values are provided per feature!! for example relTh = " but relTh0 = 1 is appended for feature with id 0, relTh1 for feature with id 1, etc.
<i>filterSims</i>	Filter simulations using a list of parameters and a pandas dataframe of simulation configurations if ~ is used as a prefix for a parameter - it is filtered according to values that do NOT match the value provided with the ~Parameter
<i>insertIntoSimName</i>	Add keys and values to name, in between parts of name split by index
<i>orderSimFiles</i>	Filter simulations results using a list of parameters and a flag if in ascending or descending order
<i>orderSimulations</i>	Order simulations dataframe using a list of parameters and a flag if in ascending or descending order
<i>removeSimsNotMatching</i>	remove simulations from simDF that do not match filtering criteria
<i>rewriteLocalCfgs</i>	fetch all override sections in cfgFull and write a local_NAMEOVERRIDE.ini configuration file for the available sections - naming is collection_module_override if no localCfgPath is provided, default saved to avalancheDir/Inputs/configurationOverrides where package refers to e.g.

### in3Utils.cfgHandling.addToSimName

**addToSimName**(*avalancheDir*, *csvString*="")

Add parameterName and value to simNames of simulation dataframe

E.g used as helper routine for renaming layernames in qgis

#### Parameters

- **avalancheDir** (*str*) – path to avalanche directory
- **csvString** – comma separated list with parameter names, as found in com1DFA ini file eg. 'mu,tau0,tEnd'

#### Returns

**simDF** – containing index, the parameters and the old and new name

#### Return type

dataframe

### in3Utils.cfgHandling.applyCfgOverride

**applyCfgOverride**(*cfgToOverride*, *cfgWithOverrideParameters*, *module*, *addModValues*=False)

override configuration parameter values with the values provided in *cfgWithOverrideParameters*[*modName\_override*] if *addModValues* True update the *cfgWithOverrideParameters* with the values for all parameters that are not provided in the override parameters

#### Parameters

- **cfgToOverride** (*configparser object*) – configuration of module of interest
- **cfgWithOverrideParameters** (*configparser object*) – full configuration settings containing a section *modName\_override* with parameter values that should be overridden in the *cfgToOverride*
- **module** – module of the *cfgToOverride* configuration OR pathlib path to module
- **addModValues** (*bool*) – if True add all parameters from *cfgToOverride* module to *cfgWithOverrideParameters* override section

#### Returns

- **cfgToOverride** (*configparser object*) – updated configuration of module
- **cfgWithOverrideParameters** (*configparser object*) – updated configuration of module

### in3Utils.cfgHandling.errorDuplicateListEntry

**errorDuplicateListEntry**(*listKeys*, *message*)

check if duplicate entries appear in a list and raise Assertion error using message

#### Parameters

- **listKeys** (*list*) – list with keys
- **message** (*str*) – message of error

### in3Utils.cfgHandling.fetchAndOrderSimFiles

**fetchAndOrderSimFiles**(*avalancheDir*, *inputDir*, *varParList*, *ascendingOrder*, *specDir*="", *resFiles*=False)

Filter simulations results using a list of parameters and a flag if in ascending or descending order

#### Parameters

- **avalancheDir** (*str*) – path to avalanche directory
- **inputDir** (*str*) – path to simulation results
- **varParList** (*str or list*) – simulation configuration parameters for ordering simulations
- **ascendingOrder** (*bool*) – True if simulations shall be ordered in ascending order regarding varPar
- **specDir** (*str*) – path to a directory where simulation configuration files can be found - optional

#### Returns

**dataDF** – dataFrame of simulation results (fileName, ... and values for parameters in varParList)

#### Return type

pandas dataFrame

### in3Utils.cfgHandling.fetchValidationString

**fetchValidationString**(*val*, *thIdList*, *thNames*, *simDFrow*)

create a validation string to be checked if simDFrow matches filtering criteria (given by val)

#### Parameters

- **val** (*str, float*) – value to be checked
- **thIdList** (*list*) – list with thickness feature ids
- **thNames** (*list*) – list with thickness feature names
- **simDFrow** (*pandas dataframe row*) – parameters of simulation

#### Returns

**validationString** – bool if simulation given by simDFrow matches filtering criteria

#### Return type

bool

### in3Utils.cfgHandling.filterCom1DFAThicknessValues

**filterCom1DFAThicknessValues**(*key*, *value*, *simDF*)

thickness settings different if read from shpfile - requires more complex filtering if read from shp - thickness values are provided per feature!! for example relTh = '' but relTh0 = 1 is appended for feature with id 0, relTh1 for feature with id 1, etc.

#### Parameters

- **key** (*str*) – name of parameter
- **value** (*list*) – list of values used for filtering
- **simDF** (*pandas dataframe*) – configuration info for each simulation

**Returns****simDF** – updated dataframe**Return type**

pandas data frame

**in3Utils.cfgHandling.filterSims****filterSims**(*avalancheDir*, *parametersDict*, *specDir*="", *simDF*="")

Filter simulations using a list of parameters and a pandas dataframe of simulation configurations if ~ is used as a prefix for a parameter - it is filtered according to values that do NOT match the value provided with the ~Parameter

**Parameters**

- **avalancheDir** (*str*) – path to avalanche directory
- **parametersDict** (*dict*) – dictionary with parameter and parameter values for filtering
- **specDir** (*str*) – path to a directory where simulation configuration files can be found - optional
- **simDF** (*pandas DataFrame*) – optional - if simDF already available

**Returns****simNameList** – list of simNames that match filtering criteria**Return type**

list

**in3Utils.cfgHandling.insertIntoSimName****insertIntoSimName**(*name*, *keys*, *values*, *index*)

Add keys and values to name, in between parts of name split by index

**Parameters**

- **name** (*str*) – name to extend
- **keys** (*list*) – list with keys
- **values** (*list*) – list with values
- **index** (*str*) – used to split name

**Returns****newName** – containing newName, with keys and values inserted after index**Return type**

string

### in3Utils.cfgHandling.orderSimFiles

**orderSimFiles**(*avalancheDir*, *inputDir*, *varParList*, *ascendingOrder*, *specDir*="", *resFiles*=False)

Filter simulations results using a list of parameters and a flag if in ascending or descending order

#### Parameters

- **avalancheDir** (*str*) – path to avalanche directory
- **inputDir** (*str*) – path to simulation results
- **varParList** (*str or list*) – simulation configuration parameters for ordering simulations
- **ascendingOrder** (*bool*) – True if simulations shall be ordered in ascending order regarding varPar
- **specDir** (*str*) – path to a directory where simulation configuration files can be found - optional

#### Returns

**dataDF** – dataFrame of simulation results (fileName, ... and values for parameters in varParList)

#### Return type

pandas dataFrame

### in3Utils.cfgHandling.orderSimulations

**orderSimulations**(*varParList*, *ascendingOrder*, *simDF*)

Order simulations dataframe using a list of parameters and a flag if in ascending or descending order

#### Parameters

- **varParList** (*str or list*) – simulation configuration parameters for ordering simulations
- **ascendingOrder** (*bool*) – True if simulations shall be ordered in ascending order regarding varPar
- **simDF** (*pandas dataframe*) – dataFrame of simulations (one line per simulation with fileName, ... and values for parameters in varParList)

#### Returns

**simDF** – sorted dataFrame of simulation results (fileName, ... and values for parameters in varParList)

#### Return type

pandas dataFrame

### in3Utils.cfgHandling.removeSimsNotMatching

**removeSimsNotMatching**(*simDF*, *key*, *value*)

remove simulations from simDF that do not match filtering criteria

#### Parameters

- **simDF** (*pandas dataframe*) – dataframe with one row per simulation and info on its characteristics, parameters used,..
- **key** (*str*) – name of parameter that shall be used for filtering
- **value** (*list*) – list of parameter values used for filtering

**Returns**

**simDF** – updated dataframe with only those simulations that match filtering criteria

**Return type**

pandas dataframe

**in3Utils.cfgHandling.rewriteLocalCfgs**

**rewriteLocalCfgs**(*cfgFull*, *avalancheDir*, *localCfgPath*="")

fetch all override sections in *cfgFull* and write a *local\_NAMEOVERRIDE.ini* configuration file for the available sections - naming is *collection\_module\_override* if no *localCfgPath* is provided, default saved to *avalancheDir/Inputs/configurationOverrides* where package refers to e.g. *ana1Tests*, *ana3AIMEC*, etc- and module to e.g. *energyLineTest.py*, *ana3AIMEC.py* so all python files inside the packages that have a *nameCfg.ini* file too

**cfgFull: configparser**

configuration with override sections for modules

**avalancheDir: pathlib path or str**

path to avalanche directory

**localCfgPath: pathlib Path**

optional - path to directory to store **local\_** cfg ini file to if not provided - **local\_** cfg ini file is saved to avalanche directory

**in3Utils.cfgUtils**

Utilities for handling configuration files

## Functions

<i>appendCgf2DF</i>	append simulation configuration to the simulation dataframe only account for sections GENERAL and INPUT
<i>appendTcpu2DF</i>	append Tcpu dictionary to the dataframe
<i>cfgHash</i>	UID hash of a config.
<i>compareTwoConfigs</i>	compare locCfg to defCfg and return a cfg object and modification dict Values are merged from locCfg to defCfg: - parameters already in defCfg get the value from locCfg - additional values in locCfg get added in the resulting Cfg
<i>convertConfigParserToDict</i>	create dictionary from configparser object
<i>convertDF2numerics</i>	convert a string DF to a numerical one
<i>convertDictToConfigParser</i>	create configparser object from dict
<i>convertToCfgList</i>	convert a list into a string where individual list items are separated by
<i>createConfigurationInfo</i>	Read configurations from all simulations configuration ini files from directory
<i>getDefaultModuleConfig</i>	Returns the default configuration for a given module returns a configparser object
<i>getGeneralConfig</i>	Returns the general configuration for avafame returns a configparser object
<i>getModPathName</i>	get the path and name of a module from imported module
<i>getModuleConfig</i>	Returns the configuration for a given module returns a configparser object
<i>getNumberOfProcesses</i>	Determine how many CPU cores to take for parallel tasks
<i>readAllConfigurationInfo</i>	Read allConfigurations.csv file as dataframe from directory
<i>readCfgFile</i>	Read configuration from ini file, if module is provided, module configuration is read from Outputs, if fileName is provided configuration is read from fileName
<i>readCompareConfig</i>	Read and optionally compare configuration files (if a local and default are both provided) and inform user of the eventual differences.
<i>setStrnanToNan</i>	set pandas element to np.nan if it is a string nan
<i>writeAllConfigurationInfo</i>	Write cfg configuration to allConfigurations.csv
<i>writeCfgFile</i>	Save configuration used to text file in Outputs/moduleName/configurationFiles/modName.ini or optional to filePath and with fileName
<i>writeDictToJson</i>	write a dictionary to a json file



**in3Utils.cfgUtils.appendCgf2DF****appendCgf2DF**(*simHash*, *simName*, *cfgObject*, *simDF*)

append simulation configuration to the simulation dataframe only account for sections GENERAL and INPUT

**Parameters**

- **simHash** (*str*) – hash of the simulation to append
- **simName** (*str*) – name of the simulation
- **cfgObject** (*configParser*) – configuration corresponding to the simulation
- **simDF** (*pandas dataframe*) – configuration dataframe

**Returns****simDF** – DFappended with the new simulation configuration**Return type**

pandas DataFrame

**in3Utils.cfgUtils.appendTcpu2DF****appendTcpu2DF**(*simHash*, *tCPU*, *tCPUDF*)

append Tcpu dictionary to the dataframe

**Parameters**

- **simHash** (*str*) – hash of the simulation corresponding to the tCPU dict to append
- **tCPU** (*dict*) – cpu time dict of the simulation
- **tCPUDF** (*pandas dataframe*) – tCPU dataframe

**Returns****simDF** – DFappended with the new simulation configuration**Return type**

pandas DataFrame

**in3Utils.cfgUtils.cfgHash****cfgHash**(*cfg*, *typeDict=False*)UID hash of a config. Given a configParser object *cfg*, or a dictionary - then *typeDict=True*, returns a uid hash**Parameters**

- **cfg** (*configParser object*)
- **typeDict** (*dict*) – dictionary
- **Returns**
- \_\_\_\_\_
- **uid** (*str*) – uid hash

### in3Utils.cfgUtils.compareTwoConfigs

**compareTwoConfigs**(defCfg, locCfg, toPrint=False)

compare locCfg to defCfg and return a cfg object and modification dict Values are merged from locCfg to defCfg:  
- parameters already in defCfg get the value from locCfg - additional values in locCfg get added in the resulting Cfg

**Parameters**

- **defCfg** (*configparser object*) – default configuration
- **locCfg** (*configuration object*) – configuration that is compared to defCfg
- **toPrint** (*bool*) – flag if config shall be printed to log

**Returns**

- **modInfo** (*dict*) – dictionary containing only differences from default
- **cfg** (*configParser object*) – contains combined config

### in3Utils.cfgUtils.convertConfigParserToDict

**convertConfigParserToDict**(cfg)

create dictionary from configparser object

### in3Utils.cfgUtils.convertDF2numerics

**convertDF2numerics**(simDF)

convert a string DF to a numerical one

**Parameters**

**simDF** (*pandas dataframe*) – dataframe

**Returns**

**simDF**

**Return type**

pandas DataFrame

### in3Utils.cfgUtils.convertDictToConfigParser

**convertDictToConfigParser**(cfgDict)

create configparser object from dict

### in3Utils.cfgUtils.convertToCfgList

**convertToCfgList**(*parameterList*)

convert a list into a string where individual list items are separated by |

**Parameters**

**parameterList** (*list*) – list of parameter values

**Returns**

**parameterString** – str with parameter values separated by |

**Return type**

str

### in3Utils.cfgUtils.createConfigurationInfo

**createConfigurationInfo**(*avaDir*, *comModule*='com1DFA', *standardCfg*='', *writeCSV*=False, *specDir*='')

Read configurations from all simulations configuration ini files from directory

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **standardCfg** (*dict*) – standard configuration for module - option
- **writeCSV** (*bool*) – True if configuration dataFrame shall be written to csv file
- **specDir** (*str*) – path to a directory where simulation configuration files can be found - optional

**Returns**

**simDF** – DF with all the simulation configurations

**Return type**

pandas DataFrame

### in3Utils.cfgUtils.getDefaultModuleConfig

**getDefaultModuleConfig**(*module*, *toPrint*=True)

Returns the default configuration for a given module returns a configParser object

**module object: module**

[the calling function provides the already imported] module eg.: from avafame.com2AB import com2AB leads to getModuleConfig(com2AB) whereas from avafame.com2AB import com2AB as c2 leads to getModuleConfig(c2)

### in3Utils.cfgUtils.getGeneralConfig

**getGeneralConfig**(*nameFile*='')

Returns the general configuration for avafame returns a configParser object

**Parameters**

**nameFile** (*pathlib path*) – optional full path to file, if empty use avafameCfg from folder one level up

### in3Utils.cfgUtils.getModPathName

#### getModPathName(*module*)

get the path and name of a module from imported module

##### Parameters

**module** (*imported module*)

##### Returns

- **modPath** (*pathlib path*) – path to directory where module is located
- **modName** (*str*) – name of module

### in3Utils.cfgUtils.getModuleConfig

#### getModuleConfig(*module, fileOverride="", modInfo=False, toPrint=True, onlyDefault=False*)

Returns the configuration for a given module returns a configparser object

##### module object: module

[the calling function provides the already imported] module eg.: from avafame.com2AB import com2AB leads to getModuleConfig(com2AB) whereas from avafame.com2AB import com2AB as c2 leads to getModuleConfig(c2) OR: pathlib Path to module (python file)

##### Str: fileOverride

[allows for a completely different file location. However note:] missing values from the default cfg will always be added!

##### modInfo: bool

true if dictionary with info on differences to standard config

##### onlyDefault: bool

if True, only use the default configuration

Order is as follows: fileOverride -> local\_MODULECfg.ini -> MODULECfg.ini

### in3Utils.cfgUtils.getNumberOfProcesses

#### getNumberOfProcesses(*cfgMain, nSims*)

Determine how many CPU cores to take for parallel tasks

##### Parameters

- **cfgMain** (*configuration object*) – the main avafame configuration
- **nSims** (*integer*) – number of simulations that need to be calculated

##### Returns

**nCPU** – number of cores to take

##### Return type

int

### in3Utils.cfgUtils.readAllConfigurationInfo

**readAllConfigurationInfo**(*avaDir*, *specDir*="", *configCsvName*='allConfigurations')

Read allConfigurations.csv file as dataframe from directory

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **specDir** (*str*) – path to a directory where simulation configuration files can be found - optional
- **configCsvName** (*str*) – name of configuration csv file

#### Returns

- **simDF** (*pandas DataFrame*) – DF with all the simulation configurations
- **simDFName** (*array*) – simName column of the dataframe

### in3Utils.cfgUtils.readCfgFile

**readCfgFile**(*avaDir*, *module*="", *fileName*="")

Read configuration from ini file, if module is provided, module configuration is read from Ouputs, if fileName is provided configuration is read from fileName

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **module** – module
- **fileName** (*str*) – path to file that should be read - optional

#### Returns

**cfg** – configuration that is from file

#### Return type

configParser object

### in3Utils.cfgUtils.readCompareConfig

**readCompareConfig**(*iniFile*, *modName*, *compare*, *toPrint*=True)

Read and optionally compare configuration files (if a local and default are both provided) and inform user of the eventual differences. Take the default as reference.

#### Parameters

- **iniFile** (*path to config file*) – Only one path if compare=False
- **compare** (*boolean*) – True if two paths are provided and a comparison is needed
- **toPrint** (*boolean*) – True (default) to print configuration to terminal. Differences to default will ALWAYS be printed

#### Returns

- **Output** (*ConfigParser object*) – contains combined config
- **modDict** (*dict*) – dictionary containing only differences from default

### in3Utils.cfgUtils.setStrnanToNan

**setStrnanToNan**(*simDF*, *simDFTest*, *name*)

set pandas element to np.nan if it is a string nan

**Parameters**

- **simDF** (*pandas dataframe*) – dataframe
- **simDFTest** (*pandas series*) – series of sim DF column named name replaced “.” with ” “
- **name** (*str*) – name of pandas dataframe column

**Returns**

**simDF** – updated pandas dataframe with np.nan values where string nan was

**Return type**

pandas dataframe

### in3Utils.cfgUtils.writeAllConfigurationInfo

**writeAllConfigurationInfo**(*avaDir*, *simDF*, *specDir*="", *csvName*='allConfigurations.csv')

Write cfg configuration to allConfigurations.csv

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **simDF** (*pandas dataframe*) – dataframe of the configuration
- **specDir** (*str*) – path to a directory where simulation configuration shall be saved - optional
- **csvName** (*str*) – name of csv file in which to save to - optional

**Returns**

**configFiles** – path where the configuration dataframe was saved

**Return type**

pathlib Path

### in3Utils.cfgUtils.writeCfgFile

**writeCfgFile**(*avaDir*, *module*, *cfg*, *fileName*="", *filePath*="")

Save configuration used to text file in Outputs/moduleName/configurationFiles/modName.ini or optional to filePath and with fileName

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **module** – module
- **cfg** (*configparser object*) – configuration settings
- **fileName** (*str*) – name of saved configuration file - optional
- **filePath** (*str or pathlib path*) – path where file should be saved to except file name - optional

**in3Utils.cfgUtils.writeDictToJson****writeDictToJson**(*inDict*, *outFilePath*)

write a dictionary to a json file

**in3Utils.fileHandlerUtils**

Directory and file handling helper functions

**Functions**

<i>checkCommonSims</i>	Check which files are common between local and full ExpLog
<i>checkIfFileExists</i>	test if file exists if not throw error
<i>checkPathlib</i>	check if pathlib.PurePath if not convert to
<i>exportcom1DFAOrigOutput</i>	Export the simulation results from com1DFA output to desired location
<i>extractLogInfo</i>	read log file and extract info on time, mass stop criterion
<i>extractParameterInfo</i>	Extract info about simulation parameters from the log file
<i>fetchFlowFields</i>	fetch paths to all desired flow fields within folder
<i>fileNotFoundMessage</i>	throw error if file not found with message and path
<i>getFilterDict</i>	Create parametersDict from ini file, for filtering simulations
<i>makeADir</i>	Check if a directory exists, if not create directory
<i>makeSimDF</i>	Create a dataframe that contains all info on simulations
<i>makeSimFromResDF</i>	Create a dataframe that contains all info on simulations in output/comModule/peakFiles
<i>readLogFile</i>	Read experiment log file and make dictionary that contains general info on all simulations
<i>splitIniValueToArraySteps</i>	read values in ini file and return numpy array or list if the items are strings; values can either be separated by   or provided in start:end:numberOfSteps format if separated by : or \$ also optional add one additional value using & if format of refVal\$percent\$steps is used - an array is created with +- percent of refVal in nsteps
<i>splitTimeValueToArrayInterval</i>	read save time step info from ini file and return numpy array of values

**in3Utils.fileHandlerUtils.checkCommonSims****checkCommonSims**(*logName*, *localLogName*)

Check which files are common between local and full ExpLog

**in3Utils.fileHandlerUtils.checkIfFileExists****checkIfFileExists**(*filePath*, *fileType=""*)

test if file exists if not throw error

**Parameters**

- **filePath** (*pathlib path*) – path to file
- **fileType** (*str*) – string for error message which kind of file is not found

**in3Utils.fileHandlerUtils.checkPathlib****checkPathlib**(*checkPath*)

check if pathlib.PurePath if not convert to

**Parameters**

- **checkPath** (*str or pathlib path*)
- **path to be checked**

**Returns****checkPath** – pathlib path version of checkPath**Return type**

pathlib path

**in3Utils.fileHandlerUtils.exportcom1DFAOrigOutput****exportcom1DFAOrigOutput**(*avaDir*, *cfg=""*, *addTSteps=False*)

Export the simulation results from com1DFA output to desired location

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **cfg** (*dict*) – configuration read from ini file that has been used for the com1DFAOrig simulation
- **addTSteps** (*bool*) – if True: first and last time step of flow thickness are exported

**in3Utils.fileHandlerUtils.extractLogInfo****extractLogInfo**(*fileName*)

read log file and extract info on time, mass stop criterion

**Parameters****fileName** (*str or pathlib path*) – path to log file**Returns****logDict** – dictionary with arrays for mass entrained mass time step and info on simulation run and stop criterion**Return type**

dict



**in3Utils.fileHandlerUtils.extractParameterInfo****extractParameterInfo**(*avaDir*, *simName*, *reportD*)

Extract info about simulation parameters from the log file

**Parameters**

- **avaDir** (*str*) – path to avalanche
- **simName** (*str*) – name of the simulation
- **reportD** (*dict*) – report dictionary

**Returns**

- **parameterDict** (*dict*) – dictionary listing name of parameter and value; release mass, final time step and current mast
- **reportD** (*dict*) – updated report dictionary with info on simulation

**in3Utils.fileHandlerUtils.fetchFlowFields****fetchFlowFields**(*flowFieldsDir*, *suffix=""*)

fetch paths to all desired flow fields within folder

**Parameters**

- **flowFieldsDir** (*str or pathlib path*) – path to flow field ascii files
- **suffix** (*str*) – suffix in flow field name to be searched for

**Returns****flowFields** – list of pathlib paths to flow fields**Return type**

list

**in3Utils.fileHandlerUtils.fileNotFoundMessage****fileNotFoundMessage**(*messageName*)

throw error if file not found with message and path

**Parameters****messageName** (*str*) – error message**in3Utils.fileHandlerUtils.getFilterDict****getFilterDict**(*cfg*, *section*)

Create parametersDict from ini file, for filtering simulations

**Parameters**

- **cfg** (*configParser object*) – configuration with information on filtering criteria
- **section** (*str*) – section of cfg where filtering criteria can be found

**Returns****parametersDict** – dictionary with parameter and parameter values for filtering simulation results

**Return type**  
dict

### **in3Utils.fileHandlerUtils.makeADir**

**makeADir**(*dirName*)

Check if a directory exists, if not create directory

**Parameters**

**dirName** (*str*) – path of directory that should be made

### **in3Utils.fileHandlerUtils.makeSimDF**

**makeSimDF**(*inputDir*, *avaDir*="", *simID*='simID')

Create a dataframe that contains all info on simulations

this can then be used to filter simulations for example

**Parameters**

- **inputDir** (*str*) – path to directory of simulation results
- **avaDir** (*str*) – optional - path to avalanche directory
- **simID** (*str*) – optional - simulation identification, depending on the computational module:  
com1DFA: simHash com1DFAOrig: Mu or parameter that has been used in parameter variation

**Returns**

**dataDF** – dataframe with full file path, file name, release area scenario, simulation type (null, entres, etc.), model type (dfa, ref, etc.), simID, result type (ppr, pft, etc.), simulation name, cell size and optional name of avalanche, optional time step

**Return type**

dataFrame

### **in3Utils.fileHandlerUtils.makeSimFromResDF**

**makeSimFromResDF**(*avaDir*, *comModule*, *inputDir*="", *simName*="")

Create a dataframe that contains all info on simulations in output/comModule/peakFiles

One line for each simulation - so all peakfiles that belong to one simulation are listed in one line that corresponds to that simulation

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **comModule** (*str*) – module used to create the results
- **inputDir** (*str*) – optional - path to directory of simulation results
- **simName** (*str*) – optional - key phrase to be found in the simulation result name

**Returns**

- **dataDF** (*dataFrame*) – dataframe with for each simulation, the full file path, file name, release area scenario, simulation type (null, entres, etc.), model type (dfa, ref, etc.), simID, path to result files (ppr, pft, etc.), simulation name, cell size and optional name of avalanche, optional time step
- **resTypeListAll** (*list*) – list of res types available for all simulations

### in3Utils.fileHandlerUtils.readLogFile

**readLogFile**(*logName*, *cfg=""*)

Read experiment log file and make dictionary that contains general info on all simulations

#### Parameters

- **logName** (*str*) – path to log file
- **cfg** (*dict*) – optional - configuration read from com1DFA simulation

#### Returns

**logDict** – dictionary with number of simulation (noSim), name of simulation (simName), parameter variation, full name

#### Return type

dict

### in3Utils.fileHandlerUtils.splitIniValueToArraySteps

**splitIniValueToArraySteps**(*cfgValues*, *returnList=False*)

read values in ini file and return numpy array or list if the items are strings; values can either be separated by | or provided in start:end:numberOfSteps format if separated by : or \$ also optional add one additional value using & if format of refVal\$percent\$steps is used - an array is created with +- percent of refVal in nsteps

#### Parameters

- **cfgValues** (*str*) – values of parameter to be read from ini file
- **returnList** (*bool*) – if True force to return values as list

#### Returns

**items** – values as 1D numpy array or list (in the case of strings)

#### Return type

1D numpy array or list

### in3Utils.fileHandlerUtils.splitTimeValueToArrayInterval

**splitTimeValueToArrayInterval**(*cfgValues*, *endTime*)

read save time step info from ini file and return numpy array of values

values can either be separated by | or provided in start:interval format

#### Parameters

- **cfgValues** (*str*) – time steps info
- **endTime** (*float*) – end time

**Returns**

**items** – time step values as 1D numpy array

**Return type**

1D numpy array

**in3Utils.generateTopo**

Create generic/idealised topographies

**Functions**

<i>addDrop</i>	Add a drop to a given topography
<i>bowl</i>	Compute coordinates of sphere with given radius (rBowl)
<i>computeCoordGrid</i>	
<i>createParabolaAxis</i>	create the s coordinate for a lined sloped from theta - phi from x axis
<i>flatplane</i>	Compute coordinates of flat plane topography
<i>generateTopo</i>	Compute coordinates of desired topography with given inputs
<i>getGridDefs</i>	
<i>getParabolaParams</i>	Compute parameters for parabola
<i>helix</i>	Compute coordinates of helix-shaped topography with given radius (rHelix)
<i>hockey</i>	Compute coordinates of an inclined plane with a flat foreland defined by total fall height z0, angle to flat foreland (meanAlpha) and a radius (rCirc) to smooth the transition from inclined plane to flat foreland
<i>inclinedplane</i>	Compute coordinates of inclined plane with given slope (meanAlpha)
<i>parabola</i>	Compute coordinates of a parabolically-shaped slope with a flat foreland defined by total fall height C, angle (meanAlpha) or distance (fLen) to flat foreland
<i>parabolaRotation</i>	Compute coordinates of a parabolically-shaped slope with a flat foreland defined by total fall height C, angle (meanAlpha) or distance (fLen) to flat foreland One parabolic slope in x direction, one sloped with 45° and one with 60°
<i>pyramid</i>	Generate a pyramid topography - in this case rectangular domain
<i>writeDEM</i>	Write topography information to file

**in3Utils.generateTopo.addDrop****addDrop**(*cfg*, *x*, *y*, *zv*)

Add a drop to a given topography

The drop is added in the x direction

**Parameters**

- **cfg** (*configparser*) – configuration for generateTopo
- **x** (*2D numpy array*) – x coordinate of the raster
- **y** (*2D numpy array*) – y coordinate of the raster
- **zv** (*2D numpy array*) – z coordinate of the raster

**Returns****zv** – z coordinate of the raster taking the drop into account**Return type**

2D numpy array

**in3Utils.generateTopo.bowl****bowl**(*cfg*)

Compute coordinates of sphere with given radius (rBwol)

**in3Utils.generateTopo.computeCoordGrid****computeCoordGrid**(*dx*, *xEnd*, *yEnd*)**in3Utils.generateTopo.createParabolaAxis****createParabolaAxis**(*phi*, *theta*, *r*, *zv*, *fLen*, *fFlat*)

create the s coordinate for a lined sloped from theta - phi from x axis

**in3Utils.generateTopo.flatplane****flatplane**(*cfg*)

Compute coordinates of flat plane topography

**in3Utils.generateTopo.generateTopo****generateTopo**(*cfg*, *avalancheDir*)

Compute coordinates of desired topography with given inputs

**in3Utils.generateTopo.getGridDefs****getGridDefs**(*cfg*)**in3Utils.generateTopo.getParabolaParams****getParabolaParams**(*cfg*)

Compute parameters for parabola

**in3Utils.generateTopo.helix****helix**(*cfg*)

Compute coordinates of helix-shaped topography with given radius (rHelix)

**in3Utils.generateTopo.hockey****hockey**(*cfg*)

Compute coordinates of an inclined plane with a flat foreland defined by total fall height z0, angle to flat foreland (meanAlpha) and a radius (rCirc) to smooth the transition from inclined plane to flat foreland

**in3Utils.generateTopo.inclinedplane****inclinedplane**(*cfg*)

Compute coordinates of inclined plane with given slope (meanAlpha)

**in3Utils.generateTopo.parabola****parabola**(*cfg*)

Compute coordinates of a parabolically-shaped slope with a flat foreland defined by total fall height C, angle (meanAlpha) or distance (fLen) to flat foreland

**in3Utils.generateTopo.parabolaRotation****parabolaRotation**(*cfg*)

Compute coordinates of a parabolically-shaped slope with a flat foreland defined by total fall height C, angle (meanAlpha) or distance (fLen) to flat foreland One parabolic slope in x direction, one sloped with 45° and one with 60°

**in3Utils.generateTopo.pyramid****pyramid(cfg)**

Generate a pyramid topography - in this case rectangular domain

**in3Utils.generateTopo.writeDEM****writeDEM(cfg, z, outDir)**

Write topography information to file

**in3Utils.geoTrans**

Operations and transformations of rasters and lines

**Functions**

<i>areaPoly</i>	Gauss's area formula to calculate polygon area
<i>cartToSpherical</i>	convert from cartesian to spherical coordinates
<i>checkOverlap</i>	Check if two rasters overlap
<i>checkParticlesInRelease</i>	remove particles laying outside the polygon
<i>checkProfile</i>	check that the avalanche profiles goes from top to bottom flip it if not and adjust the splitpoint in consequence
<i>computeAlongLineDistance</i>	compute distance along a dict of coordinates or a shapely lineString incrementally
<i>computeLengthOfLine2D</i>	compute distance along a line in 2D
<i>computeS</i>	compute s coordinate given a path (x, y)
<i>findAngleProfile</i>	Find the beta point: first point under the beta value given in prepareAngleProfile.
<i>findClosestPoint</i>	find the closest point of pointDict along line defined by xcoor and ycoor - only xy plane!
<i>findPointOnDEM</i>	find point on dem given a direction and a z value to reach
<i>findSplitPoint</i>	Finds the closest point in Points to the avaProfile and re- turns its projection on avaProfile.
<i>getCellsAlongLine</i>	Find all raster cells crossed by the line line has to be en- tirely contained on the raster extend.
<i>getNeighborCells</i>	Find the neighbour cells to a given cell
<i>isCounterClockWise</i>	Determines if a polygon path is mostly clockwise or counter clockwise
<i>makeCoordGridFromHeader</i>	Get x and y (2D) grid description vectors for a mesh with a given number of rows and columns, lower left center and cellSize.
<i>makeCoordinateGrid</i>	Create grid
<i>path2domain</i>	Creates a domain (irregular raster) along a path, given the path xyPath, a domain width and a raster cellsize
<i>pointInPolygon</i>	find particles within a polygon
<i>polygon2Raster</i>	convert line to raster

continues on next page

Table 2.1 – continued from previous page

<i>prepareAngleProfile</i>	Prepare inputs for findAngleProfile function Read profile (s, z), compute the slope Angle look for points for which the slope is under the given Beta value and that are located downstream of the splitPoint
<i>prepareArea</i>	convert shape file polygon to raster
<i>prepareLine</i>	Resample and project line on dem 1- Resample the avapath line with an interval of approximately distance in meters between points (projected distance on the horizontal plane).
<i>projectOnGrid</i>	Projects Z onto points (x,y) using a bilinear or nearest interpolation and returns the z coord
<i>projectOnRaster</i>	Projects Points on raster using a bilinear or nearest interpolation and returns the z coord (no for loop)
<i>remeshData</i>	compute raster data on a new mesh with cellSize using the specified remeshOption.
<i>remeshRaster</i>	change raster cell size by reprojecting on a new grid - first check if remeshed raster available
<i>resizeData</i>	Reproject raster on a grid of shape rasterRef
<i>rotate</i>	rotate a vector provided as start and end point with theta angle rotation counter-clockwise
<i>rotateRaster</i>	rotate clockwise a raster around (0, 0) with theta angle
<i>searchRemeshedRaster</i>	search if remeshed raster with correct name and cell size already available
<i>snapPtsToLine</i>	snap points to line in dataframe only considering x, y plane!

### in3Utils.geoTrans.areaPoly

#### areaPoly(X, Y)

Gauss's area formula to calculate polygon area

##### Parameters

- **X** (1D numpy array) – x coord of the vertices
- **Y** (1D numpy array) – y coord of the vertices
- **(Without repeating the first vertex!!!)**

##### Returns

**area** – Area of the polygon

##### Return type

float



**in3Utils.geoTrans.cartToSpherical****cartToSpherical**(*X*, *Y*, *Z*)

convert from cartesian to spherical coordinates

**Parameters**

- **X** (*float*) – x coordinate
- **Y** (*float*) – y coordinate
- **Z** (*float*) – z coordinate

**Returns**

- **r** (*float*) – radius
- **phi** (*float*) – azimuth angle [degrees]
- **theta** (*float*) – for elevation angle defined from Z-axis down [degrees]

**in3Utils.geoTrans.checkOverlap****checkOverlap**(*toCheckRaster*, *refRaster*, *nameToCheck*, *nameRef*, *crop=False*)

Check if two rasters overlap

**Parameters**

- **toCheckRaster** (*2D numpy array*) – Raster to check
- **refRaster** (*2D numpy array*) – reference Raster
- **nameToCheck** (*str*) – name of raster that might overlap
- **nameRef** (*str*) – name of reference raster
- **crop** (*boolean*) – if True, remove overlapping part and send a warning

**Returns**

**toCheckRaster** – if crop is True, return toCheckRaster without the overlapping part and send a warning if needed if crop is False, return error if Rasters overlap otherwise return toCheckRaster

**Return type**

2D numpy array

**in3Utils.geoTrans.checkParticlesInRelease****checkParticlesInRelease**(*particles*, *line*, *radius*)

remove particles laying outside the polygon

**Parameters**

- **particles** (*dict*) – particles dictionary
- **line** (*dict*) – line dictionary
- **radius** (*float*) – threshold val that decides if a point is in the polygon, on the line or very close but outside

**Returns**

**particles** – particles dictionary where particles outside of the polygon have been removed

**Return type**  
dict

### **in3Utils.geoTrans.checkProfile**

**checkProfile**(*avaProfile*, *projSplitPoint=None*)

check that the avalanche profiles goes from top to bottom flip it if not and adjust the splitpoint in consequence

**Parameters**

- **avaProfile** (*dict*) – line dictionary with x and y coordinates
- **projSplitPoint** (*dict*) – a point dictionary already projected on the *avaProfile*

**Returns**

- **avaProfile** (*dict*) – *avaProfile*, flipped if needed
- **projSplitPoint** (*dict*) – point dictionary

### **in3Utils.geoTrans.computeAlongLineDistance**

**computeAlongLineDistance**(*line*, *dim='2D'*)

compute distance along a dict of coordinates or a shapely *lineString* incrementally

**Parameters**

- **line** (*lineString shapely or dict*) – *lineString* object or dict with x, y, z keys and *np.array*s as items
- **dim** (*str*) – 2D only in xy, 3D in xyz

**Returns**

**distancePoints** – list of starting (at zero) distance along this line

**Return type**

list

### **in3Utils.geoTrans.computeLengthOfLine2D**

**computeLengthOfLine2D**(*x*, *y*)

compute distance along a line in 2D

**Parameters**

**x, y** (*np array*) – x, y coordinates of line

**Returns**

**s** – accumulated distance measured along line from point to point

**Return type**

*np array*

**in3Utils.geoTrans.computeS****computeS**(*avaPath*)

compute s coordinate given a path (x, y)

**Parameters**

**avaPath** (*dict*) – path dictionary with x and y coordinates as 1D numpy arrays

**Returns**

**avaPath** – path dictionary updated with s coordinate

**Return type**

dict

**in3Utils.geoTrans.findAngleProfile****findAngleProfile**(*tmp, ds, dsMin*)

Find the beta point: first point under the beta value given in prepareAngleProfile. Make sure that at least dsMin meters behind the point are also under the beta value otherwise keep searching

**Parameters**

- **tmp** (*1D numpy array*) – index array of point in profile with slope below the given beta angle and below the splitPoint
- **ds** (*1D numpy array*) – distance between points described in tmp
- **dsMin** (*float*) – threshold distance [m] for looking for the beta point (at least dsMin meters below beta degrees)

**Returns**

**idsAnglePoint** – index of beta point

**Return type**

int

**in3Utils.geoTrans.findClosestPoint****findClosestPoint**(*xcoor, ycoor, pointsDict*)

find the closest point of pointDict along line defined by xcoor and ycoor - only xy plane!

**Parameters**

- **xcoor, ycoor** (*np array*) – x and y coordinates of line
- **pointsDict** (*dict*) – a dictionary with coordinates of points with keys x and y

**Returns**

**indSplit** – index of closest point found on the line

**Return type**

int

### in3Utils.geoTrans.findPointOnDEM

**findPointOnDEM**(*dem*, *vDirX*, *vDirY*, *vDirZ*, *zHighest*, *xFirst*, *yFirst*, *zFirst*)

find point on dem given a direction and a z value to reach

#### Parameters

- **dem** (*dict*) – dem dict
- **vDirX**, **vDirY**, **vDirZ** (*floats*) – x, y and z components of the direction in which to extend
- **zHighest** (*float*) – z value to reach
- **xFirst**, **yFirst**, **zFirst** (*floats*) – x, y and z coordinates of the starting point

#### Returns

**xExtTop**, **yExtTop**, **zExtTop** – x, y and z coordinates of the point found

#### Return type

floats

### in3Utils.geoTrans.findSplitPoint

**findSplitPoint**(*avaProfile*, *Points*)

Finds the closest point in Points to the avaProfile and returns its projection on avaProfile.

#### Parameters

- **avaProfile** (*dict*) – line dictionary with x and y coordinates
- **Points** (*dict*) – a point dictionary

#### Returns

**projPoint** – point dictionary projected on the profile (if several points were give in input, only the closest point to the profile is projected)

#### Return type

dict

### in3Utils.geoTrans.getCellsAlongLine

**getCellsAlongLine**(*header*, *lineDict*, *addBuffer=True*)

Find all raster cells crossed by the line line has to be entierly contained on the raster extend. If addBuffer is True, add neighbour cells to the result based on <https://stackoverflow.com/a/35808540/15887086>

#### Parameters

- **header** (*dict*) – raster header
- **lineDict** (*dict*) – line dictionary
- **addBuffer** (*boolean*) – True to add a 1 cell buffer around the line

#### Returns

**lineDict** – line dictionary updated with the “cellsCrossed” 1D array (boolean array of 0 and 1 if the cell is crossed by the line or in its neighborhood)

#### Return type

dict

### in3Utils.geoTrans.getNeighborCells

**getNeighborCells**(*indX*, *indY*, *ncols*, *nrows*, *cellsArray*)

Find the neighbour cells to a given cell

#### Parameters

- **indX** (*int*) – x index of the cell for which you want to find the direct neighbors
- **indY** (*int*) – y index of the cell for which you want to find the direct neighbors
- **ncols** (*int*) – number of cols in the raster
- **nrows** (*int*) – number of rows in the raster
- **cellsArray** (*1D int array*) – boolean array of 0 and 1 if the cell is crossed by the line or in its neighborhood

#### Returns

**cellsArray** – updated boolean array of 0 and 1 if the cell is crossed by the line or in its neighborhood

#### Return type

1D int array

### in3Utils.geoTrans.isCounterClockWise

**isCounterClockWise**(*path*)

Determines if a polygon path is mostly clockwise or counter clockwise

<https://stackoverflow.com/a/45986805/15887086>

#### Parameters

**path** (*matplotlib.path*) – polygon path

#### Returns

**isCounterCloc1** – 1 if the path is counter clockwise, 0 otherwise

#### Return type

int

### in3Utils.geoTrans.makeCoordGridFromHeader

**makeCoordGridFromHeader**(*rasterHeader*, *cellSizeNew=None*, *larger=False*)

Get x and y (2D) grid description vectors for a mesh with a given number of rows and columns, lower left center and cellSize. If 'cellSizeNew' is not None use cellSizeNew instead of rasterHeader['cellsize'] Make sure the new grid is at least as big as the old one if larger=True (can happen if 'cellSizeNew' is not None)

#### Parameters

- **rasterHeader** (*dict*) – raster header with info on ncols, nrows, csz, xllcenter, yllcenter, no-data\_value
- **cellSizeNew** (*float*) – If not None, use cellSizeNew as cell size
- **larger** (*boolean*) – If True, make sure the extend of the (xGrid, yGrid) is larger or equal than the header one

#### Returns

- **xGrid, yGrid** (*2D numpy arrays*) – 2D vector of x and y values for mesh center coordinates (produced using meshgrid)
- **ncols, nrows** (*int*) – number of columns and rows

### **in3Utils.geoTrans.makeCoordinateGrid**

**makeCoordinateGrid**(*xllc, yllc, csz, ncols, nrows*)

Create grid

#### **Parameters**

- **xllc, yllc** (*float*) – x and y coordinate of the lower left center
- **csz** (*float*) – cell size
- **ncols, nrows** (*int*) – number of columns and rows

#### **Returns**

**xGrid, yGrid** – 2D vector of x and y values for mesh center coordinates (produced using meshgrid)

#### **Return type**

2D numpy arrays

### **in3Utils.geoTrans.path2domain**

**path2domain**(*xyPath, rasterTransfo*)

Creates a domain (irregular raster) along a path, given the path *xyPath*, a domain width and a raster cellsize

#### **Parameters:**

##### **xyPath: dict**

line dictionary with coordinates x and y

##### **rasterTransfo: dict**

**rasterTransfo['w']:** float

Domain width

**rasterTransfo['cellSizeSL']:** float

cellsize expected for the new raster

#### **Returns:**

##### **rasterTransfo: dict**

rasterTransfo updated with xp, yp Arrays determining a path of width w along a line

**rasterTransfo['DBXI']:**

x coord of the left boundary

**rasterTransfo['DBXr']:**

x coord of the right boundary

**rasterTransfo['DBYl']:**

y coord of the left boundary

**rasterTransfo['DBYr']:**

y coord of the right boundary

[Fischer2013] Fischer, Jan-Thomas. (2013). A novel approach to evaluate and compare computational snow avalanche simulation. Natural Hazards and Earth System Sciences. 13. 1655-. 10.5194/nhess-13-1655-2013. Uwe Schlifkowitz/ BFW, June 2011

### in3Utils.geoTrans.pointInPolygon

**pointInPolygon**(*demHeader, points, Line, radius*)

find particles within a polygon

#### Parameters

- **demHeader** (*dict*) – dem header dictionary
- **points** (*dict*) – points to check
- **Line** (*dict*) – line dictionary
- **radius** (*float*) – threshold val that decides if a point is in the polygon, on the line or very close but outside

#### Returns

**Mask** – Mask of particles to keep

#### Return type

1D numpy array

### in3Utils.geoTrans.polygon2Raster

**polygon2Raster**(*demHeader, Line, radius, th=""*)

convert line to raster

#### Parameters

- **demHeader** (*dict*) – dem header dictionary
- **Line** (*dict*) – line dictionary
- **radius** (*float*) – include all cells which center is in the polygon or close enough
- **th** (*float*) – thickness value of the line feature

#### Returns

**Mask** – updated raster

#### Return type

2D numpy array

### in3Utils.geoTrans.prepareAngleProfile

**prepareAngleProfile**(*beta*, *avaProfile*, *raiseWarning=True*)

Prepare inputs for findAngleProfile function Read profile (s, z), compute the slope Angle look for points for which the slope is under the given Beta value and that are located downstream of the splitPoint

#### Parameters

- **beta** (*float*) – beta angle in degrees
- **avaProfile** (*dict*) – profile dictionary, s, z and a split point(optional)
- **raiseWarning** (*bool*) – True to raise eventual warnings

#### Returns

- **angle** (*1D numpy array*) – profile angle
- **tmp** (*1D numpy array*) – index array of point in profile with slope bellow the given beta angle and bellow the splitPoint
- **ds** (*1D numpy array*) – distance between points discribed in tmp

### in3Utils.geoTrans.prepareArea

**prepareArea**(*line*, *dem*, *radius*, *thList=""*, *combine=True*, *checkOverlap=True*)

convert shape file polygon to raster

#### Parameters

- **line** (*dict*) – line dictionary
- **dem** (*dict*) – dictionary with dem information
- **radius** (*float*) – include all cells which center is in the polygon or close enough
- **thList** (*list*) – thickness values for all features in the line dictionary
- **combine** (*Boolean*) – if True sum up the rasters in the area list to return only 1 raster if False return the list of distinct area rasters this option works only if thList is not empty
- **checkOverlap** (*Boolean*) – if True check if features are overlapping and return an error if it is the case if False check if features are overlapping and average the value for overlapping areas (Attention: if combine is set to False, you do not see the result of the averaging since the list of raters was not affected by the averaging step)

#### Returns

contains either

- Raster: 2D numpy array, raster of the area (returned if relRHlist is empty OR if combine is set

to True) - RasterList: list, list of 2D numpy array rasters (returned if relRHlist is not empty AND if combine is set to False)

#### Return type

updates the line dictionary with the rasterData



## in3Utils.geoTrans.prepareLine

**prepareLine**(*dem*, *avapath*, *distance=10*, *Point=None*)

Resample and project line on dem 1- Resample the avapath line with an interval of approximately distance in meters between points (projected distance on the horizontal plane). 2- Make avalanche profile out of the path (affect a z value using the dem) 3- Get projection of points on the profil (closest point)

### Parameters

- **dem** (*dict*) – dem dictionary
- **avapath** (*dict*) – line dictionary
- **distance** (*float*) – resampling distance
- **Point** (*dict*) – a point dictionary (optional, can contain several point)

### Returns

- **avaProfile** (*dict*) – the resampled avapath with the z coordinate
- **projPoint** (*dict*) – point dictionary projected on the profile (if several points were give in input, only the closest point to the profile is projected)

## in3Utils.geoTrans.projectOnGrid

**projectOnGrid**(*x*, *y*, *Z*, *csz=1*, *xllc=0*, *yllc=0*, *interp='bilinear'*)

Projects Z onto points (x,y) using a bilinear or nearest interpolation and returns the z coord

### Parameters

- **x** (*array*) – x coord of the points to project
- **y** (*array*) – y coord of the points to project
- **Z** (*2D numpy array*) – raster data
- **csz** (*float*) – cellsize corresponding to the raster data
- **xllc** (*float*) – x coord of the lower left center of the raster
- **yllc** (*float*) – y coord of the lower left center of the raster
- **interp** (*str*) – interpolation option, between nearest or bilinear

### Returns

- **z** (*2D numpy array*) – projected data on the raster data
- **ioob** (*int*) – number of out of bounds indexes

## in3Utils.geoTrans.projectOnRaster

**projectOnRaster**(*dem*, *Points*, *interp='bilinear'*, *inData='rasterData'*, *outData='z'*)

Projects Points on raster using a bilinear or nearest interpolation and returns the z coord (no for loop)

### Parameters

- **dem** (*dict*) – dem dictionary
- **Points** (*dict*) – Points dictionary (x,y)

- **interp** (*str*) – interpolation option, between nearest or bilinear
- **inData** (*str*) – key in the dem dict of the 2D field to use for the interpolation.
- **outData** (*str*) – key in the Points dict to be updated with the interpolated data.

**Returns**

- **Points** (*dict*) – Points dictionary with z coordinate added or updated
- **ioob** (*int*) – number of out of bounds indexes

**in3Utils.geoTrans.remeshData**

**remeshData**(*rasterDict*, *cellSizeNew*, *remeshOption*='griddata', *interpMethod*='cubic', *larger*=True)

compute raster data on a new mesh with cellSize using the specified remeshOption.

remeshOption are to choose between 'griddata' or 'RectBivariateSpline' Only the 'griddata' works properly if the input data contains noData points, 'RectBivariateSpline' is faster but fails if input data contains noData points. The new mesh is as big or smaller as the original mesh if larger is False and bigger if larger is True

**Parameters**

- **rasterDict** (*dict*) – raster dictionary (with header and rasterData)
- **cellSizeNew** (*float*) – mesh size of new mesh
- **remeshOption** (*str*) – method used to remesh ('griddata' or 'RectBivariateSpline') Check the scipy documentation for more details default is 'griddata'
- **interpMethod** (*str*) – interpolation order to use for the interpolation ('linear', 'cubic' or 'quintic')
- **larger** (*Boolean*) – if true (default) output grid is at least as big as the input

**Returns**

**data** – remeshed data dict with data as numpy array and header info

**Return type**

dict

**in3Utils.geoTrans.remeshRaster**

**remeshRaster**(*rasterFile*, *cfgSim*, *typeIndicator*='DEM', *onlySearch*=False)

change raster cell size by reprojecting on a new grid - first check if remeshed raster available

the new raster is as big or smaller as the original raster and saved to Inputs/remeshedRasters as remeshed-TYPEINDICATORcellSize

Interpolation is based on griddata with a cubic method. Here would be the place to change the order of the interpolation or to switch to another interpolation method.

**Parameters**

- **rasterFile** (*str or pathlib path*) – file path to DEM in Inputs/
- **cfgSim** (*configParser*) – meshCellSizeThreshold : threshold under which no remeshing is done meshCellSize : desired cell size
- **typeIndicator** (*str*) – type of raster, possible options DEM or RELTH

- **onlySearch** (*bool*) – if True - only searching for remeshed DEM but not remeshing if not found

**Returns**

**pathRaster** – path of raster with desired cell size relative to Inputs/

**Return type**

str

**in3Utils.geoTrans.resizeData****resizeData**(*raster, rasterRef*)

Reproject raster on a grid of shape rasterRef

**Parameters**

- **raster** (*dict*) – raster dictionary
- **rasterRef** (*dict*) – reference raster dictionary

**Returns**

- **data** (*2D numpy array*) – reprojected data
- **dataRef** (*2D numpy array*) – reference data

**in3Utils.geoTrans.rotate****rotate**(*locationPoints, theta, deg=True*)

rotate a vector provided as start and end point with theta angle rotation counter-clockwise

**Parameters**

- **locationPoints** (*list*) – list of lists with x,y coordinate of start and end point of a line
- **theta** (*float*) – rotation angle of the vector from start point to end point - degree default
- **deg** (*bool*) – if true theta is converted to rad from degree

**Returns**

**rotatedLine** – list of lists of x,y coordinates of start and end point of rotated vector

**Return type**

list

**in3Utils.geoTrans.rotateRaster****rotateRaster**(*rasterDict, theta, deg=True*)

rotate clockwise a raster around (0, 0) with theta angle

**Parameters**

- **rasterDict** (*dict*) – raster dictionary
- **theta** (*float*) – rotation angle of the vector from start point to end point - degree default
- **deg** (*bool*) – if true theta is converted to rad from degree

**Returns**

**rotatedRaster** – rotated raster dictionary

**Return type**  
dict

### in3Utils.geoTrans.searchRemeshedRaster

**searchRemeshedRaster**(*rasterName*, *cfgSim*, *typeIndicator*='DEM')

search if remeshed raster with correct name and cell size already available

**Parameters**

- **rasterName** (*str*) – name of raster file in Inputs/
- **typeIndicator** (*str*) – which type of raster to handle, possible options Raster or RELTH
- **cfgSim** (*configparser object*) – configuration settings: *avaDir*, *meshCellSize*, *meshCellSizeThreshold*

**Returns**

- **pathRaster** (*pathlib path*) – to Raster
- **rasterFound** (*bool*) – flag if dem is found
- **allRasterNames** (*list*) – of all names of dems found in Inputs/remeshedRasters

### in3Utils.geoTrans.snapPtsToLine

**snapPtsToLine**(*dbData*, *projstr*, *lineName*, *pointsList*)

snap points to line in dataframe only considering x, y plane!

**Parameters**

- **dbData** (*pandas dataframe*) – dataframe with geometry info of events
- **lineName** (*str*) – name of line column except *projstr*
- **pointsList** (*list*) – list with point column names except *projstr*
- **projstr** (*str*) – projection string to append to all names

**Returns**

**dbData** – updated dataframe with ...\_snapped point column

**Return type**

pandas dataframe

### in3Utils.getReleaseArea

Get release area corner coordinates for a rectangle of an area of approx. 100 000 m2

This file is part of Avaframe.

## Functions

<i>correctOrigin</i>	Move the points so that the correct origin is set
<i>getCornersFP</i>	
<i>getCornersHS</i>	
<i>getCornersIP</i>	
<i>getReleaseArea</i>	Main function to compute release areas
<i>makexyPoints</i>	Make xy Points of release area from given extent and start and end points
<i>writeReleaseArea</i>	Write topography information to file

### in3Utils.getReleaseArea.correctOrigin

**correctOrigin**(*xyPoints*, *cfgT*)

Move the points so that the correct origin is set

### in3Utils.getReleaseArea.getCornersFP

**getCornersFP**(*cfgR*)

### in3Utils.getReleaseArea.getCornersHS

**getCornersHS**(*cfgR*, *cfgT*)

### in3Utils.getReleaseArea.getCornersIP

**getCornersIP**(*cfgR*, *cfgT*)

### in3Utils.getReleaseArea.getReleaseArea

**getReleaseArea**(*cfgT*, *cfgR*, *avalancheDir*)

Main function to compute release areas

### in3Utils.getReleaseArea.makexyPoints

**makexyPoints**(*x1*, *x2*, *y1*, *cfgR*)

Make xy Points of release area from given extent and start and end points

**in3Utils.getReleaseArea.writeReleaseArea****writeReleaseArea**(*xyPoints*, *demType*, *cfgR*, *outDir*)

Write topography information to file

**in3Utils.initialiseDirs**

Setup required directory structures by computational modules

**Functions**

---

<i>initialiseRunDirs</i>	Initialise Simulation run with input data
--------------------------	---

---

**in3Utils.initialiseDirs.initialiseRunDirs****initialiseRunDirs**(*avaDir*, *modName*, *cleanRemeshedRasters*)

Initialise Simulation run with input data

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **modName** (*str*) – name of module
- **cleanRemeshedRasters** (*bool*) – if True directory Inputs/DEMremeshed shall be cleaned

**Returns**

- **workDir** (*str*) – path to Work directory
- **outputDir** (*str*) – path to Outputs directory

**in3Utils.initializeProject**

Functions to initialize project, i.e create folder structure

This file is part of Avaframe.

**Functions**

---

<i>checkMakeDir</i>	combines base and dir, checks whether it exists, if not creates it
<i>cleanModuleFiles</i>	Cleans all generated files from the provided module in the outputs folder and work folder
<i>cleanRemeshedDir</i>	clean remeshedRasters folder in avaDir Inputs
<i>cleanSingleAvaDir</i>	Clean a single avalanche directory of the work and output directories
<i>createFolderStruct</i>	creates the folder structure with avalanche base path
<i>initializeFolderStruct</i>	Initialize the standard folder structure.

---

**in3Utils.initializeProject.checkMakeDir****checkMakeDir**(*base, dirName*)

combines base and dir, checks whether it exists, if not creates it

**in3Utils.initializeProject.cleanModuleFiles****cleanModuleFiles**(*avaDir, module, alternativeName="", deleteOutput=True*)

Cleans all generated files from the provided module in the outputs folder and work folder

**Parameters**

- **avaDir** (*path/string*) – Avalanche directory path
- **module** (*module object*) – The module do delete. e.g. from `avaframe.com2AB` import `com2AB` leads to `cleanModuleFiles(com2AB)` whereas from `avaframe.com2AB` import `com2AB` as `c2` leads to `cleanModuleFiles(c2)` Boolean to be able to avoid deletion of Outputs (true by default)

**in3Utils.initializeProject.cleanRemeshedDir****cleanRemeshedDir**(*avaDir*)clean remeshedRasters folder in `avaDir` Inputs**Parameters**

- **avaDir** (*str or pathlib patch*) – path to avalanche directory

**in3Utils.initializeProject.cleanSingleAvaDir****cleanSingleAvaDir**(*avaDir, deleteOutput=True*)

Clean a single avalanche directory of the work and output directories

**Parameters**

- **avaDir** (*path/string*) – Avalanche directory path
- **deleteOutput** (*boolean*) – If True (default), directory output and the log files are deleted

**in3Utils.initializeProject.createFolderStruct****createFolderStruct**(*pathAvaName*)

creates the folder structure with avalanche base path

## in3Utils.initializeProject.initializeFolderStruct

**initializeFolderStruct** (*pathAvaName*, *removeExisting=False*)

Initialize the standard folder structure. If *removeExisting* is true, deletes any existing folders! BEWARE!

### Parameters

- **pathAvaName** (*str*) – string with base avalanche path
- **removeExisting** (*boolean, optional, default: False*) – remove existing directory, use this to completely clean out the directory

## in3Utils.logUtils

Defining a writable object to write the config to the log file

This file is part of Avaframe.

## Functions

<i>initiateLogger</i>	Initiates logger object based on setup in logging.conf
<i>writeCfg2Log</i>	write a configparser object to log file

## in3Utils.logUtils.initiateLogger

**initiateLogger** (*targetDir*, *logName='runLog'*, *modelInfo=''*)

Initiates logger object based on setup in logging.conf

### Parameters

- **targetDir** (*str*) – folder to save log file to
- **logName** (*str*) – filename of log file; optional; defaults to runLog.log
- **modelInfo** (*str*) – if not empty add info on modelInfo to log

### Returns

**log**

### Return type

logging object

## in3Utils.logUtils.writeCfg2Log

**writeCfg2Log** (*cfg*, *cfgName='Unspecified'*)

write a configparser object to log file



### 2.10.3 Analysis/Output/Helper Modules

<i>ana1Tests</i>	Code for ana1Tests module
<i>ana3AIMEC</i>	Code for ana3AIMEC analysis module
<i>ana4Stats</i>	Tools for statistical analysis of simulation results
<i>ana5Utils</i>	Generate thalweg-time-diagrams and simulograms
<i>log2Report</i>	Tools for generating reports
<i>out1Peak</i>	Simple tools for visualising datasets.
<i>out3Plot</i>	Plotting
<i>tmplEx.tmplEx</i>	This is the template for new modules, with the bare minimal required files

#### 2.10.3.1 ana1Tests

Code for ana1Tests module

##### Modules

<i>ana1Tests.FPtest</i>	Flat plane test
<i>ana1Tests.analysisTools</i>	
<i>ana1Tests.damBreak</i>	Simple python script to reproduce analytic solution for a Riemann problem, following the derivations in Faccanoni and Mangeney (2012), Test 2, Case 1.2.
<i>ana1Tests.energyLineTest</i>	Energy line test This module runs a DFA simulation extracts the center of mass path and compares it to the analytic geometric/alpha line solution
<i>ana1Tests.rotationTest</i>	Rotation test
<i>ana1Tests.simiSolTest</i>	Similarity solution module
<i>ana1Tests.testUtilities</i>	Functions for handling benchmark tests data, filtering, sorting

#### ana1Tests.FPtest

Flat plane test

##### Functions

<i>getReleaseThickness</i>	define release thickness for Flat Plane solution test
<i>plotProfilesFPtest</i>	Plot flow thickness and gradient for FlatPlane simulation results
<i>postProcessFPcom1DFA</i>	get fields and particles dictionaries for given time step

**ana1Tests.FPtest.getReleaseThickness****getReleaseThickness**(*avaDir, cfg, demFile*)

define release thickness for Flat Plane solution test

**ana1Tests.FPtest.plotProfilesFPtest****plotProfilesFPtest**(*cfg, ind\_time, relDict, comSol*)

Plot flow thickness and gradient for FlatPlane simulation results

**Parameters**

- **cfg** (*configparser*)
- **ind\_time** (*int*) – time index for simiSol
- **relDict** (*dict*) – dictionary of release area info
- **comSol** (*dict*) – dictionary of simulation results and info (particles, fields, indices, time step)

**ana1Tests.FPtest.postProcessFPcom1DFA****postProcessFPcom1DFA**(*cfgGen, particles, fields, ind\_t, relDict*)

get fields and particles dictionaries for given time step

**ana1Tests.analysisTools****Functions**

<i>L2Norm</i>	Compute L2 norm of an array
<i>computeErrorAndNorm</i>	Compute error between two functions given their norm 2
<i>normL2Scal</i>	Compute L2 and Lmax norm of the error between the analytic and numerical solution
<i>normL2Vect</i>	Compute L2 and Lmax norm of the error between the analytic and numerical solution

**ana1Tests.analysisTools.L2Norm****L2Norm**(*norm2Array, cellSize, cosAngle*)

Compute L2 norm of an array

**Parameters**

- **norm2Array** (*numpy array*) – norm2 of the function
- **cellSize** (*float*) – grid cell size
- **cosAngle** (*float*) – cosine of the slope angle

**Returns****normL2** – normL2 of the function

**Return type**  
float

### ana1Tests.analysisTools.computeErrorAndNorm

**computeErrorAndNorm**(*localError*, *analyticalSol2*, *cellSize*, *cosAngle*)

Compute error between two functions given their norm 2

#### Parameters

- **localError** (*numpy array*) – norm2 of the error function
- **analyticalSol2** (*numpy array*) – norm2 of the reference function
- **cellSize** (*float*) – grid cell size
- **cosAngle** (*float*) – cosine of the slope angle

#### Returns

- **errorL2** (*float*) – L2 error
- **errorL2Rel** (*float*) – Relativ L2 error
- **errorMax** (*float*) – LMax error
- **errorMaxRel** (*float*) – Relativ LMax error

### ana1Tests.analysisTools.normL2Scal

**normL2Scal**(*analyticalSol*, *numericalSol*, *cellSize*, *cosAngle*)

Compute L2 and Lmax norm of the error between the analytic and numerical solution

#### Parameters

- **analyticalSol** (*numpy array*) – analytic solution array
- **numericalSol** (*numpy array*) – numericalSol solution array
- **cellSize** (*float*) – grid cell size
- **cosAngle** (*float*) – cosine of the slope angle

#### Returns

- **errorL2** (*float*) – L2 error
- **errorL2Rel** (*float*) – Relativ L2 error
- **errorMax** (*float*) – LMax error
- **errorMaxRel** (*float*) – Relativ LMax error

**ana1Tests.analysisTools.normL2Vect****normL2Vect** (*analyticalSol*, *numericalSol*, *cellSize*, *cosAngle*)

Compute L2 and Lmax norm of the error between the analytic and numerical solution

**Parameters**

- **analyticalSol** (*dictionary*) – analytic solution dictionary
  - fx: x component of the vector
  - fy: y component of the vector
  - fz: z component of the vector
- **numericalSol** (*dictionary*) – numericalSol solution dictionary
  - fx: x component of the vector
  - fy: y component of the vector
  - fz: z component of the vector
- **cellSize** (*float*) – grid cell size
- **cosAngle** (*float*) – cosine of the slope angle

**Returns**

- **errorL2** (*float*) – L2 error
- **errorL2Rel** (*float*) – Relativ L2 error
- **errorMax** (*float*) – LMax error
- **errorMaxRel** (*float*) – Relativ LMax error

**ana1Tests.damBreak**

Simple python script to reproduce analytic solution for a Riemann problem, following the derivations in Faccanoni and Mangeney (2012), Test 2, Case 1.2. but scaled up in size.

Here the instantaneous release of fluid from rest is described using incompressible, thickness-averaged mass and momentum conservation equations and a Coulomb-type friction law.

**Functions**

<i>analyzeResults</i>	Compare analytical and com1DFA results, compute error
<i>damBreakSol</i>	Compute analytical Flow thickness and velocity for dam break test case
<i>getDamExtend</i>	Get the extend where the analytic and simulation should be compared
<i>postProcessDamBreak</i>	loop on all DFA simulations and compare then to the analytic solution

## ana1Tests.damBreak.analyzeResults

**analyzeResults**(*avalancheDir*, *fieldsList*, *timeList*, *solDam*, *fieldHeader*, *cfg*, *outDirTest*, *simHash*, *simDFrow*)

Compare analytical and com1DFA results, compute error

### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **fieldsList** (*list*) – list of fields dictionaries
- **timeList** (*list*) – list of time steps
- **solDam** (*dict*) – analytic solution dictionary
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **cfg** (*configParser object*) – configuration setting for avalanche simulation including DAMBREAK section
- **outDirTest** (*pathlib path*) – path output directory (where to save the figures)
- **simHash** (*str*) – com1DFA simulation id
- **simDFrow** (*pandas object*) – com1DFA simulation rox corresponding to simHash

### Returns

- **hErrorL2Array** (*numpy array*) – L2 error on Flow thickness for saved time steps
- **hErrorLMaxArray** (*numpy array*) – LMax error on Flow thickness for saved time steps
- **vErrorL2Array** (*numpy array*) – L2 error on flow velocity for saved time steps
- **vErrorLMaxArray** (*numpy array*) – LMax error on flow velocity for saved time steps
- **tSave** (*float*) – time corresponding the errors

## ana1Tests.damBreak.damBreakSol

**damBreakSol**(*avaDir*, *cfg*, *cfgC*, *outDirTest*)

Compute analytical Flow thickness and velocity for dam break test case

for a granular flow over a dry rough sloping bed with the Savage Hutter model

### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **cfg** (*configParser object*) – main avafame configuration - here showPlot flag used
- **cfgC** (*configParser object*) – configuration setting for avalanche simulation including DAMBREAK section
- **outDirTest** (*pathlib path*) – path to output directory

### Returns

**solDam** –

analytic solution dictionary:

**tAna**: 1D numpy array  
time array

**h0: float**

release thickness

**hAna: 2D numpy array**

Flow thickness (rows for x and columns for time)

**uAna: 2D numpy array**

flow velocity (rows for x and columns for time)

**xAna: 2D numpy array**

extent of domain in the horizontal plane coordinate system (rows for x and columns for time)

**xMidAna: 1D numpy array**

middle of the material in x dir in the horizontal plane coordinate system (used to compute the error)

**Return type**

dict

### **ana1Tests.damBreak.getDamExtend**

**getDamExtend**(*fieldHeader, xM, cfgDam*)

Get the extend where the analytic and simulation should be compared

**Parameters**

- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **xM** (*float*) – x coordinate of the start of the comparizon domain
- **cfgDam** (*configParser object*) – configuration setting for the DAMBREAK section

**Returns**

- **xDamPlus** (*1D numpy array*) – x array corresponding to the comparizon domain
- **nColMid** (*int*) – index of the column corresponding to xM
- **nColMax** (*int*) – index of the column corresponding to xEnd
- **nRowMin** (*int*) – index of the row corresponding to yStart
- **nRowMax** (*int*) – index of the row corresponding to yEnd

### **ana1Tests.damBreak.postProcessDamBreak**

**postProcessDamBreak**(*avalancheDir, cfgMain, cfgDam, simDF, solDam, outDirTest*)

loop on all DFA simulations and compare then to the anlytic solution

**Parameters**

- **avalancheDir** (*str or pathlib path*) – avalanche directory
- **cfgMain** (*confiparser*) – avafameCfg configuration
- **cfgDam** (*configParser object*) – configuration setting for avalanche simulation including DAMBREAK section
- **simDF** (*pandas dataframe*) – configuration DF
- **solDam** (*dict*) – analytic solution dictionary

- **outDirTest** (*pathlib path*) – path to output directory

**Returns**

**simDF** – configuration DF appended with the analysis results

**Return type**

pandas dataframe

**ana1Tests.energyLineTest**

Energy line test This module runs a DFA simulation extracts the center of mass path and compares it to the analytic geometric/alpha line solution

**Functions**

<i>generateCom1DFAEnergyPlot</i>	Make energy test analysis and plot results
<i>getAlphaProfileIntersection</i>	Extend the profile path and compute the intersection between the theoretical energy line and the path profile The profile is extended by a line.
<i>getEnergyInfo</i>	Compute energy dots and errors
<i>getRunOutAngle</i>	Compute the center of mass runout angle
<i>mainEnergyLineTest</i>	This is the core function of the energyLineTest module This module extracts the center of mass path from a DFA simulation and compares it to the analytic geometric/alpha line solution

**ana1Tests.energyLineTest.generateCom1DFAEnergyPlot**

**generateCom1DFAEnergyPlot** (*avalancheDir*, *energyLineTestCfg*, *com1DFACfg*, *avaProfileMass*, *dem*, *fieldsList*, *simName*)

Make energy test analysis and plot results

**Parameters**

- **avalancheDir** (*pathlib*) – avalanche directory pathlib path
- **energyLineTestCfg** (*configParser*) – energy line test config
- **com1DFACfg** (*configParser*) – com1DFA config
- **avaProfileMass** (*dict*) – particle mass averaged properties
- **dem** (*dict*) – com1DFA simulation dictionary
- **fieldsList** (*list*) – field dictionary list
- **simName** (*str*) – simulation name

### ana1Tests.energyLineTest.getAlphaProfileIntersection

**getAlphaProfileIntersection**(*energyLineTestCfg*, *avaProfileMass*, *mu*, *csz*)

Extend the profile path and compute the intersection between the theoretical energy line and the path profile. The profile is extended by a line. The line slope is computed from the slope of the regression on the last points of the profile.

#### Parameters

- **energyLineTestCfg** (*configParser*) – energy test config
- **avaProfileMass** (*dict*) – particle mass average properties
- **mu** (*float*) – friction coefficient
- **csz** (*float*) – dem cell size

#### Returns

- **slopeExt** (*float*) – slope of the extrapolation line
- **sIntersection** (*float*) – s coord of the intersection between the line of slope mu and the mass average path profile
- **zIntersection** (*float*) – z coord of the intersection between the line of slope mu and the mass average path profile
- **coefExt** (*float*) – coefficient saying how long the path was extended to find the intersection

### ana1Tests.energyLineTest.getEnergyInfo

**getEnergyInfo**(*avaProfileMass*, *g*, *mu*, *sIntersection*, *zIntersection*, *runOutAngleDeg*, *alphaDeg*)

Compute energy dots and errors

#### Parameters

- **avaProfileMass** (*dict*) – particle mass average properties
- **g** (*float*) – gravity
- **mu** (*float*) – friction coefficient
- **sIntersection** (*float*) – s coord of the intersection between the line of slope mu and the mass average path profile
- **zIntersection** (*float*) – z coord of the intersection between the line of slope mu and the mass average path profile
- **runOutAngleRad** (*float*) – center of mass runout angle in radians
- **runOutAngleDeg** (*float*) – center of mass runout angle in degrees

#### Returns

- **zEne** (*numpy 1D array*) – energy height of the particle averaged time steps
- **u2Path** (*numpy 1D array*) – kinetic energy of the particle averaged time steps
- **sGeomL** (*2 element list*) – s coord (start and end) of the run out angle line
- **zGeomL** (*2 element list*) – z coord (start and end) of the run out angle line



- **resultEnergyTest** (*dict*) – zEnd, sEnd, runoutAngle as well as rmseVelocityElevation, runOutSError, runOutZError, runOutAngleError between theoretical solution and simulation result

### ana1Tests.energyLineTest.getRunOutAngle

**getRunOutAngle**(*avaProfileMass*, *indStart=0*, *indEnd=-1*)

Compute the center of mass runout angle

#### Parameters

- **avaProfileMass** (*dict*) – particle mass average properties
- **indStart** (*int*) – index of the start of the mass averaged path (to enable e.g. to discard the top extension). 0 by default - so full mass averaged path from top
- **indEnd** (*int*) – index of the start of the mass averaged pass (to discard the bottom extension). -1 by default

#### Returns

- **runOutAngleRad** (*float*) – Center of Mass runout angle in radians
- **runOutAngleDeg** (*float*) – Center of Mass runout angle in degrees

### ana1Tests.energyLineTest.mainEnergyLineTest

**mainEnergyLineTest**(*avalancheDir*, *energyLineTestCfg*, *comIDFACfg*, *simName*, *dem*)

This is the core function of the energyLineTest module This module extracts the center of mass path from a DFA simulation and compares it to the analytic geometric/alpha line solution

### ana1Tests.rotationTest

Rotation test

This module runs a DFA simulation for different grid alignments and compares the results

### Functions

<i>buildRotationTestReport</i>	Write the rotation test report
<i>initializeRotationTestReport</i>	Initialize dictionary that is used for markdown report generation for rotation test
<i>mainRotationTest</i>	This is the core function of the Rotation Test module
<i>rotatedDFAResults</i>	Rotate the DFA results

### ana1Tests.rotationTest.buildRotationTestReport

**buildRotationTestReport**(*avalancheDir*, *reportRotationTest*, *simDF*, *resAnalysisDF*, *aimecPlotDict*, *flagMass*)

Write the rotation test report

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **reportRotationTest** (*dict*) – report dictionary
- **simDF** (*dataFrame*) – DFA simulation dataFrame
- **resAnalysisDF** (*dataFrame*) – aimec results dataFrame
- **aimecPlotDict** (*dict*) – aimec plots dictionary
- **flagMass** (*boolean*) – Was a mass analysis conducted?

#### Return type

generates the markDown report

### ana1Tests.rotationTest.initializeRotationTestReport

**initializeRotationTestReport**(*avalancheDir*, *resTypeList*, *comModule*, *refSimName*, *flagMass*)

Initialize dictionary that is used for markdown report generation for rotation test

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **resTypeList** (*list*) – list of result types available
- **comModule** (*str*) – computation module name
- **refSimName** (*str*) – name of the reference simulation in the simDF
- **flagMass** (*boolean*) – Was a mass analysis conducted?

#### Returns

**reportRotationTest** – report dictionary

#### Return type

dict

### ana1Tests.rotationTest.mainRotationTest

**mainRotationTest**(*avalancheDir*, *energyLineTestCfg*, *com1DFACfg*, *dem*, *simDF*, *resTypeList*, *flagMass*, *refSimRowHash*, *comModule*)

This is the core function of the Rotation Test module

This module runs the energy line test and rotates the simulation results for the aimec analysis

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **energyLineTestCfg** (*configParser*) – energy line test configuration object
- **com1DFACfg** (*configParser*) – com1DFA configuration object
- **dem** (*dict*) – dem dictionary

- **simDF** (*dataFrame*) – DFA simulation dataFrame
- **resTypeList** (*list*) – list of result types available
- **flagMass** (*boolean*) – should the aimec mass analysis be done
- **refSimRowHash** (*str*) – row index of the reference simulation in the simDF
- **comModule** (*str*) – computation module used for the DFA simulation

#### Returns

- **simDF** (*dataFrame*) – DFA simulation dataFrame updated with the energy line test results
- **flagMass** (*boolean*) – should the aimec mass analysis be done (switched to true if it is an entrainment simulation)

### ana1Tests.rotationTest.rotatedDFAResults

**rotatedDFAResults**(*avalancheDir*, *simDF*, *rowSimHash*, *resTypeList*, *thetaRef*, *comModule*)

Rotate the DFA results

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **simDF** (*dataFrame*) – DFA simulation dataFrame
- **rowSimHash** (*str*) – row index (in the simDF dataframe) of the simulation to analyze
- **resTypeList** (*list*) – list of result types available
- **thetaRef** (*float*) – angle of the path of the reference simulation
- **comModule** (*str*) – computation module used for the DFA simulation

#### Return type

saves the rotated rasters to *avalancheDir* / 'Outputs' / *comModule* + 'Rotated' / 'peakFiles'

### ana1Tests.simiSolTest

Similarity solution module

This module contains functions that compute the similarity solution for a gliding avalanche on a inclined plane according to similarity solution from : Hutter, K., Siegel, M., Savage, S.B. et al. Two-dimensional spreading of a granular avalanche down an inclined plane Part I. theory. Acta Mechanica 100, 37–68 (1993). <https://doi.org/10.1007/BF01176861>

## Functions

<i>Ffunction</i>	Calculate right hand side of the differential equation : $dx/dt = F(x,t)$ F is discribed in Hutter 1993.
<i>analyzeResults</i>	Compare analytical and com1DFA results
<i>calcEarlySol</i>	Compute the early solution for $0 < t < t_1$ to avoid singularity in the Runge-Kutta integration process
<i>computeEarthPressCoeff</i>	Computes the earth pressure coefficients function of sng of f and g i.e depending on if we are in the active or passive case
<i>computeFCoeff</i>	Compute coefficients eq 3.2 for the function F
<i>computeH</i>	get flow thickness from f and g solutions
<i>computeU</i>	get flow velocity in x direction from f and g solutions
<i>computeV</i>	get flow velocity in y direction from f and g solutions
<i>computeXC</i>	get center of mass location
<i>defineEarthPressCoeff</i>	Define earth pressure coefficients
<i>getReleaseThickness</i>	Define release thickness for the similarity solution test
<i>getSimiSolParameters</i>	get flow thickness, flow velocity and center location of flow mass of similarity solution for required time step
<i>mainSimilaritySol</i>	Compute similarity solution :Parameters: <b>simiSolCfg</b> ( <i>pathlib path</i> ) -- path to simiSol configuration file
<i>odeSolver</i>	Solve the ODE using a Runge-Kutta method
<i>postProcessSimiSol</i>	loop on all DFA simulations and compare then to the analytic solution
<i>prepareParticlesFieldscom1DFA</i>	get fields and particles dictionaries for given time step, for com1DFA domain origin is set to 0,0 for particles - so info on domain is required

## ana1Tests.simiSolTest.Ffunction

**Ffunction**(*t, x, earthPressureCoefficients, zeta, delta, eps\_x, eps\_y*)

Calculate right hand side of the differential equation :  $dx/dt = F(x,t)$  F is discribed in Hutter 1993.

## Parameters

- **t** (*float*) – curent time
- **x** (*numpy array*) – initial condition, column vector of size 4
- **earthPressureCoefficients** (*numpy array*) – earth Pressure Coefficients
- **zeta** (*float*) – slope angle
- **delta** (*float*) – friction angle
- **eps\_x** (*float*) – scale in x dir
- **eps\_y** (*float*) – scale in y dir
- **Returns**
- **F** (*numpy array*) – column vector of size 4

## ana1Tests.simiSolTest.analyzeResults

**analyzeResults**(*avalancheDir, fieldsList, timeList, solSimi, fieldHeader, cfgMain, cfgSimi, outDirTest, simHash, simDFrow*)

Compare analytical and com1DFA results

### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **fieldsList** (*list*) – list of fields dictionaries
- **timeList** (*list*) – list of time steps
- **solSimi** (*dictionary*) – similarity solution
  - time: time array (without dimension)
  - time: time array (with dimension)
  - g\_sol: g array
  - g\_p\_sol: first derivativ of g array
  - f\_sol: f array
  - f\_p\_sol: first derivativ of f array
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **cfgMain** (*dict*) – main configuration
- **cfgSimi** (*dict*) – simisol configuration including SIMISOL section
- **outDirTest** (*pathlib path*) – path output directory (where to save the figures)
- **simHash** (*str*) – com1DFA simulation id

### Returns

- **hErrorL2Array** (*numpy array*) – L2 error on flow thickness for saved time steps
- **hErrorLMaxArray** (*numpy array*) – LMax error on flow thickness for saved time steps
- **vErrorL2Array** (*numpy array*) – L2 error on flow velocity for saved time steps
- **vErrorLMaxArray** (*numpy array*) – LMax error on flow velocity for saved time steps
- **tSave** (*float*) – time corresponding the errors

## ana1Tests.simiSolTest.calcEarlySol

**calcEarlySol**(*t, earthPressureCoefficients, x\_0, zeta, delta, eps\_x, eps\_y*)

Compute the early solution for  $0 < t < t_1$  to avoid singularity in the Runge-Kutta integration process

### Parameters

- **t** (*numpy array*) – time array
- **earthPressureCoefficients** (*numpy array*) – earth Pressure Coefficients
- **x\_0** (*numpy array*) – initial condition
- **zeta** (*float*) – slope angle
- **delta** (*float*) – friction angle

- **eps\_x** (*float*) – scale in x dir
- **eps\_y** (*float*) – scale in y dir

**Returns**

**solSimi** – similarity solution (for early times)

- **time**: time array (without dimension)
- **g\_sol**: g array
- **g\_p\_sol**: first derivativ of g array
- **f\_sol**: f array
- **f\_p\_sol**: first derivativ of f array

**Return type**

dictionary

**ana1Tests.simiSolTest.computeEarthPressCoeff**

**computeEarthPressCoeff**(*x*, *earthPressureCoefficients*)

Computes the earth pressure coefficients function of sng of f and g i.e depending on if we are in the active or passive case

**Parameters**

- **x** (*float*) – internal friction angle
- **earthPressureCoefficients** (*numpy array*) – [Kxact Kxpass Kyact(Kxact) Kypass(Kxact) Kyact(Kxpass) Kypass(Kxpass)]

**Returns**

- **K\_x** (*float*) – earth pressure coefficient in x direction
- **K\_y** (*float*) – earth pressure coefficient in y direction

**ana1Tests.simiSolTest.computeFCoeff**

**computeFCoeff**(*K\_x*, *K\_y*, *zeta*, *delta*, *eps\_x*, *eps\_y*)

Compute coefficients eq 3.2 for the function F

**Parameters**

- **K\_x** (*float*) – Kx earth pressure coef
- **K\_y** (*float*) – Ky earth pressure coef
- **zeta** (*float*) – slope angle
- **delta** (*float*) – friction angle
- **eps\_x** (*float*) – scale in x dir
- **eps\_y** (*float*) – scale in y dir

**Returns**

**A, B, C, D, E** – coefficients of eq 3.2

**Return type**

floats

**ana1Tests.simiSolTest.computeH****computeH**(*solSimi*, *x1*, *y1*, *i*, *L\_x*, *L\_y*, *H*, *AminusC*)

get flow thickness from f and g solutions

**Parameters**

- **solSimi** (*dictionary*) – similarity solution
- **x1** (*numpy array*) – x coordinate location desired for the solution
- **y1** (*numpy array*) – y coordinate location desired for the solution
- **i** (*int*) – time index
- **L\_x** (*float*) – scale in x dir
- **L\_y** (*float*) – scale in y dir
- **H** (*float*) – scale in z dir
- **AminusC** – A-C coefficient

**Returns****h** – h similarity solution at (x1, y1)**Return type**

numpy array

**ana1Tests.simiSolTest.computeU****computeU**(*solSimi*, *x1*, *y1*, *i*, *L\_x*, *U*, *AminusC*)

get flow velocity in x direction from f and g solutions

**Parameters**

- **solSimi** (*dictionary*) – similarity solution
- **x1** (*numpy array*) – x coordinate location desired for the solution
- **y1** (*numpy array*) – y coordinate location desired for the solution
- **i** (*int*) – time index
- **L\_x** (*float*) – scale in x dir
- **U** (*float*) – x velocity component scale
- **AminusC** – A-C coefficient

**Returns****u** – u similarity solution at (x1, y1)**Return type**

numpy array

**ana1Tests.simiSolTest.computeV****computeV**(*solSimi*, *x1*, *y1*, *i*, *L\_y*, *V*)

get flow velocity in y direction from f and g solutions

**Parameters**

- **solSimi** (*dictionary*) – similarity solution
- **x1** (*numpy array*) – x coordinate location desired for the solution
- **y1** (*numpy array*) – y coordinate location desired for the solution
- **i** (*int*) – time index
- **L\_y** (*float*) – scale in y dir
- **V** (*float*) – y velocity component scale

**Returns****v** – v similarity solution at (x1, y1)**Return type**

numpy array

**ana1Tests.simiSolTest.computeXC****computeXC**(*solSimi*, *x1*, *y1*, *i*, *L\_x*, *AminusC*)

get center of mass location

**Parameters**

- **solSimi** (*dictionary*) – similarity solution
- **x1** (*numpy array*) – x coordinate location desired for the solution
- **y1** (*numpy array*) – y coordinate location desired for the solution
- **i** (*int*) – time index
- **L\_x** (*float*) – scale in x dir
- **AminusC** – A-C coefficient

**Returns****xc** – x position of the center of the similarity solution pile**Return type**

numpy array

**ana1Tests.simiSolTest.defineEarthPressCoeff****defineEarthPressCoeff**(*phi*, *delta*)

Define earth pressure coefficients

**Parameters**

- **phi** (*float*) – internal friction angle
- **delta** (*float*) – bed friction angle



**Returns**

**earthPressureCoefficients** – [Kxact Kxpass Kyact(Kxact) Kypass(Kxact) Kyact(Kxpass) Kypass(Kxpass)]

**Return type**

numpy array

**ana1Tests.simiSolTest.getReleaseThickness****getReleaseThickness**(*avaDir*, *cfg*, *demFile*)

Define release thickness for the similarity solution test

Release area is defined as an ellipse with main radius Lx and Ly. Release thickness has a parabolic shape from relTh in the center to 0 on the edges

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **cfg** (*dict*) – configuration settings
- **demFile** (*str*) – path to DEM file

**Returns**

**relDict** – dictionary with info on release thickness distribution

**Return type**

dict

**ana1Tests.simiSolTest.getSimiSolParameters****getSimiSolParameters**(*solSimi*, *header*, *indTime*, *cfgSimi*, *Hini*, *gravAcc*)

get flow thickness, flow velocity and center location of flow mass of similarity solution for required time step

**Parameters**

- **solSimi** (*dict*) – similarity solution
- **header** (*dict*) – header dictionary with info about the extent and cell size
- **indTime** (*int*) – index for required time step in similarity solution
- **cfg** (*dict*) – configuration
- **Hini** (*float*) – initial release thickness
- **gravAcc** (*float*) – gravity acceleration

**Returns**

**simiDict** – dictionary of similarity solution with flow thickness, flow velocity, and center location in x for required time step

**Return type**

dict

**ana1Tests.simiSolTest.mainSimilaritySol****mainSimilaritySol**(*simiSolCfg*)

Compute similarity solution :Parameters: **simiSolCfg** (*pathlib path*) – path to simiSol configuration file

**Returns**

**solSimi** –

**similarity solution:**

timeAdim: time array (without dimension) time: time array (with dimension) g\_sol: g array  
g\_p\_sol: first derivativ of g array f\_sol: f array f\_p\_sol: first derivativ of f array

**Return type**

dictionary

**ana1Tests.simiSolTest.odeSolver****odeSolver**(*solver, dt, tEnd, solSimi*)

Solve the ODE using a Runge-Kutta method

**Parameters**

- **solver** (*ode instance*) – *ode* instance corresponding to out ODE
- **dt** (*float*) – time step
- **tEnd** (*float*) – end time
- **solSimi** (*dictionary*) – similarity solution (for early times)
  - time: time array (without dimension)
  - g\_sol: g array
  - g\_p\_sol: first derivativ of g array
  - f\_sol: f array
  - f\_p\_sol: first derivativ of f array

**Returns**

**solSimi** – similarity solution (completed with all time steps)

- time: time array (without dimension)
- g\_sol: g array
- g\_p\_sol: first derivativ of g array
- f\_sol: f array
- f\_p\_sol: first derivativ of f array

**Return type**

dictionary

## ana1Tests.simiSolTest.postProcessSimiSol

**postProcessSimiSol**(*avalancheDir*, *cfgMain*, *cfgSimi*, *simDF*, *solSimi*, *outDirTest*)

loop on all DFA simulations and compare then to the analytic solution

### Parameters

- **avalancheDir** (*str or pathlib path*) – avalanche directory
- **cfgMain** (*confiparser*) – avaframeCfg configuration
- **cfgSimi** (*dict*) – configuration for similarity sol
- **simDF** (*pandas dataframe*) – configuration DF
- **solSimi** (*dict*) – similarity solution
- **outDirTest** (*pathlib path*) – path to output directory

### Returns

**simDF** – configuration DF appended with the analysis results

### Return type

pandas dataframe

## ana1Tests.simiSolTest.prepareParticlesFieldscom1DFA

**prepareParticlesFieldscom1DFA**(*fields*, *particles*, *header*, *simiDict*, *axis*)

get fields and particles dictionaries for given time step, for com1DFA domain origin is set to 0,0 for particles - so info on domain is required

### Parameters

- **Fields** (*dictionary*) – fields dictionary
- **Particles** (*dictionary*) – particles dictionary
- **header** (*dict*) – header dictionary with info about the extend and cell size
- **simiDict** (*dict*) – dictionary with center location in x for similarity solution
- **axis** (*str*) – axis (x or y) for profile

### Returns

**com1DFASol** – dictionary with location of particles, flow thickness, flow velocity, fields, and index for x or y cut of domain at the required time step

### Return type

dict

## ana1Tests.testUtilities

Functions for handling benchmark tests data, filtering, sorting

## Functions

<i>createDesDictTemplate</i>	create an empty dictionary with all required test info keys
<i>fetchBenchmarkResults</i>	Fetch a list of all paths to result files in a benchmark test, default all result file paths, if resType list provided only for resTypes in list
<i>filterBenchmarks</i>	filter benchmarks according to characteristic
<i>getTestAvaDirs</i>	get a list of avalanche directories of tests
<i>readAllBenchmarkDesDicts</i>	get description dicts for all benchmark tests and add test name as key
<i>readDesDictFromJson</i>	read dict from json file
<i>writeDesDictToJson</i>	create a json file with all required test info from desdict

### ana1Tests.testUtilities.createDesDictTemplate

#### createDesDictTemplate()

create an empty dictionary with all required test info keys

### ana1Tests.testUtilities.fetchBenchmarkResults

#### fetchBenchmarkResults(*testName*, *resTypes*=[], *refDir*="")

Fetch a list of all paths to result files in a benchmark test, default all result file paths, if resType list provided only for resTypes in list

##### Parameters

- **refDir** (*str*) – path to benchmark test directory
- **resTypes** (*list*) – list of all resTypes that shall be returned

##### Returns

**refFiles** – list of paths to all result files found for benchmark test and resType

##### Return type

list

### ana1Tests.testUtilities.filterBenchmarks

#### filterBenchmarks(*testDictList*, *filterType*, *valuesList*, *condition*='or')

filter benchmarks according to characteristic

##### Parameters

- **testDictList** (*list*) – list of benchmark dictionaries
- **filterType** (*str*) – key which is used for filtering (options: TAGS, TYPE)
- **valuesList** (*list*) – list of values used for filtering
- **condition** (*str*) – if multiple values given in valuesList - if 'or' then all test dictionaries are returned that have either of the values, if 'and' then only those are returned that feature both values

##### Returns

**testList** – list of all the benchmark dictionaries that meet filter criterion

**Return type**

list

**ana1Tests.testUtilities.getTestAvaDirs****getTestAvaDirs**(*testList*)

get a list of avalanche directories of tests

**Parameters****testList** (*list*) – list of test dictionaries**Returns****avaDirs** – list of paths to avalanche directories**Return type**

list

**ana1Tests.testUtilities.readAllBenchmarkDesDicts****readAllBenchmarkDesDicts**(*info=False, inDir=""*)

get description dicts for all benchmark tests and add test name as key

**Parameters**

- **info** (*bool*) – True if info shall be printed to log
- **inDir** (*pathLib object*) – path to benchmarks directory

**Returns****testDictList** – list of test info dictionaries**Return type**

list

**ana1Tests.testUtilities.readDesDictFromJson****readDesDictFromJson**(*fileName*)

read dict from json file

**Parameters****fileName** (*str or pathlib path*) – path to json file**Returns****desDict** – dictionary read from json file**Return type**

dict

**ana1Tests.testUtilities.writeDesDictToJson****writeDesDictToJson**(*desDict*, *testName*, *outDir*)

create a json file with all required test info from desdict

**Parameters**

- **desDict** (*dict*) – description dictionary of test
- **testName** (*str*) – name of benchmark test
- **outDir** (*str or pathlib Path*) – path where json file is saved

**Returns****fileName** – path to json file**Return type**

pathlib path

**2.10.3.2 ana3AIMEC**

Code for ana3AIMEC analysis module

**Modules**

---

ana3AIMEC.aimecTools

---

[\*ana3AIMEC.ana3AIMEC\*](#)

AIMEC post processing workflow

---

ana3AIMEC.dfa2Aimec

---

**ana3AIMEC.ana3AIMEC**

AIMEC post processing workflow

## Functions

<i>addSLToParticles</i>	add aimec (thalweg) s,l coordinates to particle dicts use dem from sim corresponding to particle dicts by providing demFileName if demFileName="" standard dem in avaDir/Inputs is used
<i>aimecRes2ReportDict</i>	Gather aimec results and append them to report the dictionary
<i>aimecTransform</i>	Adding s projected and l projected in thalweg/aimec coordinate system to the particle dictionary
<i>computeSLParticles</i>	find the closest s, l coordinates in the aimec/thalweg coordinate system to a particles x,y location
<i>fullAimecAnalysis</i>	fetch all data required to run aimec analysis, setup pathDict and reference sim, read the inputs, perform checks if all required data is available, run main aimec
<i>mainAIMEC</i>	Main logic for AIMEC postprocessing
<i>postProcessAIMEC</i>	Apply domain transformation and analyse result data (for example pressure, thickness, velocity...)

### ana3AIMEC.ana3AIMEC.addSLToParticles

**addSLToParticles**(*avaDir*, *cfgAimec*, *demFileName*, *particlesList*, *saveToPickle=False*)

add aimec (thalweg) s,l coordinates to particle dicts use dem from sim corresponding to particle dicts by providing demFileName if demFileName="" standard dem in avaDir/Inputs is used

#### Parameters

- **avaDir** (*pathlib path*) – path to avalanche directory
- **cfgAimec** (*configparser object*) – configuration settings of aimec
- **demFileName** (*str*) – path including fileName and extension to demFile located in avaDir/Inputs
- **particlesList** (*list*) – list of particles dicts of sim
- **saveToPickle** (*bool*) – if updated particle dicts shall be saved to pickle

#### Returns

- **particlesList** (*list*) – list of particles dicts updated with s,l
- **rasterTransfo** (*dict*) – dictionary with info on rasterTransformation

### ana3AIMEC.ana3AIMEC.aimecRes2ReportDict

**aimecRes2ReportDict**(*resAnalysisDF*, *reportD*, *benchD*, *pathDict*)

Gather aimec results and append them to report the dictionary

#### Parameters

- **resAnalysisDF** (*dataFrame*) – dataframe with aimec analysis results
- **reportD** (*dict*) – report dictionary to be updated
- **benchD** (*dict*) – benchmark dictionary to be updated

- **pathDict** (*dict*) – dictionary with refSimRowHash

**Returns**

- **reportD** (*dict*) – report dictionary updated with the ‘Aimec analysis’ section
- **benchD** (*dict*) – benchmark dictionary updated with the ‘Aimec analysis’ section

**ana3AIMEC.ana3AIMEC.aimecTransform**

**aimecTransform**(*rasterTransfo, particle, demHeader, timeSeries=False*)

Adding s projected and l projected in thalweg/aimec coordinate system to the particle dictionary

**Parameters**

- **rasterTransfo** (*dict*) – domain transformation information
- **particle** (*dict*) – dictionary with particle properties, particles x,y coordinate origin is 0,0
- **demHeader** (*dict*) – dict with info on dem cellSize, xllcenter, ..
- **timeSeries** (*bool*) – if timeSeries consider that particles dict is provided as time series if False particles dict is provided for one time step only

**Returns**

**particle** – particle dictionary with s grid and l grid added

**Return type**

dict

**ana3AIMEC.ana3AIMEC.computeSLParticles**

**computeSLParticles**(*rasterTransfo, demHeader, particlesX, particlesY*)

find the closest s, l coordinates in the aimec/thalweg coordinate system to a particles x,y location

**Parameters**

- **rasterTransfo** (*dict*) – info on rasterTransformation, here gridx, gridy coordinates
- **demHeader** (*dict*) – dict with info on dem xllcenter, yllcenter
- **particlesX** (*np array*) – x coordinates of particles location for one time step but all particles (origin 0,0)
- **particlesY** (*np array*) – Y coordinates of particles location for one time step but all particles (origin 0,0)

**Returns**

**sList, lList** – list of s, l coordinates for respective particle

**Return type**

list



## ana3AIMEC.ana3AIMEC.fullAimecAnalysis

**fullAimecAnalysis**(*avalancheDir*, *cfg*, *inputDir*="", *demFileName*="")

fetch all data required to run aimec analysis, setup pathDict and reference sim, read the inputs, perform checks if all required data is available, run main aimec

### Parameters

- **avalancheDir** (*str or pathlib path*) – path to avalanche directory
- **cfg** (*configparser object*) – main aimec configuration settings
- **inputDir** (*str or pathlib path*) – optional- directory where peak files are located, if "", `avaDir/Outputs/comMod/peakFiles` is set
- **demFileName** (*pathlib path*) – path to dem file

### Returns

- **rasterTransfo** (*dict*) – domain transformation information
- **resAnalysisDF** (*dataFrame*) – results of ana3AIMEC analysis
- **plotDict** (*dict*) – dictionary for report
- **newRasters** (*dict*) – dictionary containing pressure, velocity and flow thickness rasters after transformation for the reference and the current simulation

## ana3AIMEC.ana3AIMEC.mainAIMEC

**mainAIMEC**(*pathDict*, *inputsDF*, *cfg*)

Main logic for AIMEC postprocessing

Reads the required files location for ana3AIMEC postprocessing given an avalanche directory Make the domain transformation corresponding to the input avalanche path Transform 2D fields (pressure, flow thickness ...) Analyze transformed rasters and mass Produce plots and report

### Parameters

- **pathDict** (*dict*) – dictionary with paths to dem and lines for Aimec analysis
- **inputsDF** (*dataFrame*) – dataframe with simulations to analyze and associated path to raster data
- **cfg** (*configparser*) – configparser with ana3AIMEC settings defined in `ana3AIMECCfg.ini`

### Returns

- **rasterTransfo** (*dict*) – domain transformation information
- **resAnalysisDF** (*dataFrame*) – results of ana3AIMEC analysis
- **plotDict** (*dict*) – dictionary for report

**ana3AIMEC.ana3AIMEC.postProcessAIMEC**

**postProcessAIMEC**(*cfg, rasterTransfo, pathDict, resAnalysisDF, newRasters, timeMass, simRowHash, contourDict*)

Apply domain transformation and analyse result data (for example pressure, thickness, velocity...)

Apply the domain transformation to peak results. Analyse them. Calculate runout, Max Peak Values, Average Peak Values... Get mass and entrainment

The output *resAnalysisDF* contains:

**-maxACrossMax: float**

max max A (A depends on what is in *resTypes*)

**-ACrossMax: 1D numpy array**

max A in each cross section (A depends on what is in *resTypes*)

**-ACrossMean: 1D numpy array**

mean A in each cross section (A depends on what is in *resTypes*)

**-xRunout: float**

x coord of the runout point calculated from the MAX peak result in each cross section (*runoutResType* provided in the ini file)

**-yRunout: float**

y coord of the runout point calculated from the MAX peak result in each cross section (*runoutResType* provided in the ini file)

**-sRunout: float**

projected runout distance calculated from the MAX peak result in each cross section (*runoutResType* provided in the ini file)

**-xMeanRunout: float**

x coord of the runout point calculated from the MEAN peak result in each cross section (*runoutResType* provided in the ini file)

**-yMeanRunout: float**

y coord of the runout point calculated from the MEAN peak result in each cross section (*runoutResType* provided in the ini file)

**-sMeanRunout: float**

projected runout distance calculated from the MEAN peak result in each cross section (*runoutResType* provided in the ini file)

**-elevRel: float**

elevation of the release area (based on first point with peak field > *thresholdValue*)

**-deltaH: float**

elevation fall difference between *elevRel* and altitude of run-out point

**-TP: float**

ref = True sim2 = True

**-FN: float**

ref = False sim2 = True

**-FP: float**

ref = True sim2 = False

**-TN: float**

ref = False sim2 = False

**-resTypeFieldMax: float**

max value of peak field resType raster (original raster read from result file (no coordinate transformation))

**-resTypeFieldMin: float**

min value of peak field resType raster (original raster read from result file (no coordinate transformation))  
0 values are masked

**-resTypeFieldMean: float**

mean value of peak field resType raster (original raster read from result file (no coordinate transformation))  
0 values are masked

**-resTypeFieldStd: float**

std value of peak field resType raster (original raster read from result file (no coordinate transformation))  
0 values are masked

if mass analysis is performed

**-relMass: float**

release mass

**-entMass: float**

entrained mass

**-finalMass: float**

final mass

**-relativMassDiff: float**

the final mass diff with ref (in %)

**-growthIndex: float**

growth index

**-growthGrad: float**

growth gradient

**Parameters**

- **cfg** (*configParser objec*) – parameters for AIMEC analysis
- **rasterTransfo** (*dict*) – transformation information
- **pathDict** (*dict*) – dictionary with paths to dem and lines for Aimec analysis
- **resAnalysisDF** (*dataFrame*) – dataframe with simulations to analyze and associated path to raster data
- **newRasters** (*dict*) – dictionary containing pressure, velocity and flow thickness rasters after transformation for the reference and the current simulation
- **timeMass** (*1D numpy array*) – time array for mass analysis (if flagMass=True, otherwise None)
- **simRowHash** (*str*) – dataframe hash of the current simulation to analyze
- **contourDict** (*dict*) – dictionary with one key per sim and its x, y coordinates for contour line of runoutresType for thresholdValue

**Returns**

- **resAnalysisDF** (*dataFrame*) – input DF with results from Aimec Analysis updated with results from current simulation
- **contourDict** (*dict*) – dictionary with one key per sim and its x, y coordinates for contour line of runoutresType for thresholdValue - updated with info for current simulation

### 2.10.3.3 ana4Stats

Tools for statistical analysis of simulation results

#### Modules

<code>ana4Stats.getStats</code>	Function to determine statistics of datasets
<code>ana4Stats.probAna</code>	This is a simple function for computing a probability map of all peak files of one parameter that exceed a particular threshold

#### ana4Stats.getStats

Function to determine statistics of datasets

#### Functions

<code>extractMaxValues</code>	Extract max values of result parameters and save to dictionary
-------------------------------	--

#### ana4Stats.getStats.extractMaxValues

**extractMaxValues**(*inputDir*, *avaDir*, *varPar*, *restrictType*="", *nameScenario*="", *parametersDict*="")

Extract max values of result parameters and save to dictionary

- optionally restrict data of peak fields by defining which result parameter with *restrictType*, provide *nameScenario* and a *parametersDict* to filter simulations

##### Parameters

- **inputDir** (*str*) – path to directoy where peak files can be found
- **avaDir** (*str*) – path to avalanche directoy
- **varPar** (*str*) – parameter that has been varied when performing simulations (for example *relTh*)
- **restrictType** (*str*) – optional -result type of result parameters that should be used to mask result fields (eg. *ppr*, *pft*, ..)
- **nameScenario** (*str*) – optional -parameter that shall be used for color coding of simulation results in plots (for example *releaseScenario*)
- **parametersDict** (*dict*) – optional -dictionary with parameter and parameter values to filter simulations

##### Returns

**peakValues** – dictionary that contains max values for all result parameters for each simulation

##### Return type

dict

## ana4Stats.probAna

This is a simple function for computing a probability map of all peak files of one parameter that exceed a particular threshold

### Functions

<i>cfgFilesGlobalApproach</i>	create configuration files with all parameter variations - drawn from full sample for performing sims with mod-Name comModule
<i>cfgFilesLocalApproach</i>	create configuration file for performing sims with mod-Name com module
<i>checkForNumberOfReferenceValues</i>	check if in reference configuration no variation option of varPar is set if set - throw error
<i>checkParameterSettings</i>	check if parameter settings in comMod configuration do not include variation for parameters to be varied
<i>createCfgFiles</i>	create all config files required to run com Module from parameter variations using paramValues
<i>createComModConfig</i>	create configuration file for performing sims with mod-Name com module
<i>createSample</i>	create a sample of parameters
<i>createSampleFromConfig</i>	Create a sample of parameters for a desired parameter variation, and draw nSample sets of parameter values if thickness values read from shp for comMod, convert sample values for these
<i>createSampleWithVariationForThParameters</i>	Create a sample of parameters for a desired parameter variation, and fetch thickness values from shp file and perform variation for each feature within shapefile but treating the features of one shapefile as not-independent
<i>createSampleWithVariationStandardParameters</i>	create a sample for a parameter variation using latin hypercube sampling
<i>fetchProbConfigs</i>	fetch configurations of prob run in order to filter simulations e.g.
<i>fetchStartCfg</i>	fetch start configuration of comMod if onlyDefault use default comModCfg.ini and if false check if there is a local_comModCfg.ini
<i>fetchThThicknessLists</i>	fetch the desired thickness shp file info on thickness, id and ci values of all available features in shp file
<i>fetchThicknessInfo</i>	Fetch input data for avaDir and thickness info
<i>makeDictFromVars</i>	create a dictionary with info on parameter variation for all parameter in varParList
<i>probAnalysis</i>	Compute probability map of a given set of simulation result exceeding a particular threshold and save to outDir
<i>updateCfgRange</i>	update cfg with a range for parameters in cfgProb

### ana4Stats.probAna.cfgFilesGlobalApproach

**cfgFilesGlobalApproach**(*avaDir*, *cfgProb*, *modName*, *outDir*)

create configuration files with all parameter variations - drawn from full sample for performing sims with mod-Name comModule

**Parameters**

- **cfgProb** (*configParser object*) – configuration settings
- **avaDir** (*pathlib path*) – path to avalanche directory
- **modName** (*module*) – computational module

**Returns**

**cfgFiles** – list of paths to newly generated configuration files for com module including parameter variations

**Return type**

list

### ana4Stats.probAna.cfgFilesLocalApproach

**cfgFilesLocalApproach**(*variationsDict*, *cfgProb*, *modName*, *outDir*)

create configuration file for performing sims with modName com module

**Parameters**

- **variationsDict** (*dict*) – dictionary with for each varName, varVariation, varSteps, and type of variation
- **cfgProb** (*configParser object*) – configuration settings
- **avaDir** (*pathlib path*) – path to avalanche directory
- **modName** (*module*) – computational module

**Returns**

**cfgFiles** – dictionary of paths to newly generated configuration files for com module for all parameters

**Return type**

dict

### ana4Stats.probAna.checkForNumberOfReferenceValues

**checkForNumberOfReferenceValues**(*cfgGen*, *varPar*)

check if in reference configuration no variation option of varPar is set if set - throw error

**Parameters**

- **cfgGen** (*configparser object*) – reference configuration settings
- **varPar** (*str*) – name of parameter to be checked

## ana4Stats.probAna.checkParameterSettings

**checkParameterSettings**(*cfg*, *varParList*)

check if parameter settings in comMod configuration do not include variation for parameters to be varied

### Parameters

- **cfg** (*configparser object*) – configuration settings
- **varParList** (*list*) – list of parameters (names) that shall be varied

## ana4Stats.probAna.createCfgFiles

**createCfgFiles**(*paramValuesDList*, *comMod*, *cfg*, *cfgPath=""*)

create all config files required to run com Module from parameter variations using paramValues

### Parameters

- **paramValuesDList** (*list*) –  
list of dictionaries with parameter names and values (array of all sets of parameter values,  
one row per value set)  
multiple dictionaries if multiple release area scenarios and thFromShp
- **comMod** (*com module*) – computational module
- **cfg** (*configparser object*) – configuration settings
- **cfgPath** (*str*) – path where cfg files should be saved to

### Returns

**cfgFiles** – list of cfg file paths for comMod including the updated values of the parameters to vary

### Return type

list

## ana4Stats.probAna.createComModConfig

**createComModConfig**(*cfgProb*, *avaDir*, *modName*)

create configuration file for performing sims with modName com module

### Parameters

- **cfgProb** (*configParser object*) – configuration settings
- **avaDir** (*pathlib path*) – path to avalanche directory
- **modName** (*module*) – computational module

### Returns

**cfgFiles** – list of paths to newly generated configuration files for com module including parameter variations

### Return type

list

**ana4Stats.probAna.createSample****createSample**(*cfgProb*, *varParList*)

create a sample of parameters

**Parameters**

- **cfgProb** (*configparser object*) – configuration settings
- **varParList** (*list*) – list of parameters used for creating a sample

**Returns****sample** – sample object of given dimension that can be adjusted to desired bounds**Return type**

scipy object

**ana4Stats.probAna.createSampleFromConfig****createSampleFromConfig**(*avaDir*, *cfgProb*, *comMod*)

Create a sample of parameters for a desired parameter variation, and draw nSample sets of parameter values if thickness values read from shp for comMod, convert sample values for these

**Parameters**

- **avaDir** (*pathlib path*) – path to avalanche directory
- **cfgProb** (*configparser object*) – configuration settings for parameter variation
- **comMod** (*computational module*) – module to perform then sims for parameter variation

**Returns****paramValuesDList** – list of paramValuesD (multiple if multiple release area scenarios)

- **names**: list, list of parameter names (that are varied)
- **values**: numpy nd array, as many rows as sets of parameter values and as many rows as parameters
- **typeList**: list, list of types of parameters (float, ...)
- **thFromIni**: str, str of parameter names where the base value is read from shape

**Return type**

list

**ana4Stats.probAna.createSampleWithVariationForThParameters****createSampleWithVariationForThParameters**(*avaDir*, *cfgProb*, *cfgStart*, *varParList*, *valVariationValue*, *varType*, *thReadFromShp*)

Create a sample of parameters for a desired parameter variation, and fetch thickness values from shp file and perform variation for each feature within shapefile but treating the features of one shapefile as not-independent

paramsValuesD dict in output list contains

**Parameters**

- **cfgProb** (*configparser object*) – configuration settings for parameter variation
- **cfgStart** (*configparser object*) – configuration settings for comMod without variation values



- **varParList** (*list*) – list of parameters that shall be varied
- **valVariationValue** (*list*) – list if value used for variation
- **varType** (*list*) – list of type of variation for each parameter (percent, range, rangefromci)

**Returns**

**paramValuesDList** – list of paramValuesD (multiple if multiple release area scenarios)

- names: list, list of parameter names (that are varied)
- values: numpy nd array, as many rows as sets of parameter values and as many rows as parameters
- typeList: list, list of types of parameters (float, ...)
- thFromIni: str, str of parameter names where the base value is read from shape

**Return type**

list

### ana4Stats.probAna.createSampleWithVariationStandardParameters

**createSampleWithVariationStandardParameters**(*cfgProb, cfgStart, varParList, valVariationValue, varType*)

create a sample for a parameter variation using latin hypercube sampling

**Parameters**

- **cfgProb** (*configparser object*) – configuration settings for parameter variation
- **cfgStart** (*configparser object*) – configuration settings for comMod without variation values
- **varParList** (*list*) – list of parameters that shall be varied
- **valVariationValue** (*list*) – list if value used for variation
- **varType** (*list*) – list of type of variation for each parameter (percent, range, rangefromci)

**Returns**

**paramValuesD** – dictionary used to pass parameter variation values

- names: list, list of parameter names (that are varied)
- values: numpy nd array, as many rows as sets of parameter values and as many rows as parameters
- typeList: list, list of types of parameters (float, ...)
- thFromIni: str, str of parameter names where the base value is read from shape

**Return type**

dict

## ana4Stats.probAna.fetchProbConfigs

### fetchProbConfigs(*cfg*)

fetch configurations of prob run in order to filter simulations e.g. to create probability maps for different scenarios

#### Parameters

**cfg** (*configparser object*) – configuration setting, here used: samplingStrategy, varParList

#### Returns

**probConfigs** – dictionary with one key per config and a dict per key with parameter and value

#### Return type

dict

## ana4Stats.probAna.fetchStartCfg

### fetchStartCfg(*comMod*, *cfgProb*)

fetch start configuration of comMod if onlyDefault use default comModCfg.ini and if false check if there is a local\_comModCfg.ini

#### Parameters

- **comMod** (*computational module*) – module where configuration is read from
- **cfgProb** (*configparser object*) – configuration settings of probAna with collection\_comMod\_override section

#### Returns

**cfgStart** – configuration object of comMod

#### Return type

configparser object

## ana4Stats.probAna.fetchThThicknessLists

### fetchThThicknessLists(*varPar*, *inputSimFiles*, *releaseFile*, *ciRequired=False*)

fetch the desired thickness shp file info on thickness, id and ci values of all available features in shp file

#### Parameters

- **varPar** (*str*) – name of thickness parameter
- **inputSimFiles** (*dict*) – dictionary with info in input data
- **ciRequired** (*bool*) – if True throw error if ci Values not provided

#### Returns

- **thicknessFeatureNames** (*list*) – list of names of thickness features
- **thValues** (*list*) – list of thickness values for all features
- **ciValues** (*list*) – list of ci values for all feature

## ana4Stats.probAna.fetchThicknessInfo

### fetchThicknessInfo(*avaDir*)

Fetch input data for *avaDir* and thickness info

#### Parameters

- **cfg** (*configparser object*) – configuration settings
- **avaDir** (*pathlib path or str*) – path to avalanche directory

#### Returns

**inputSimFilesAll** – dictionary with info on available input data (release areas, entrainment, and thickness info)

#### Return type

dict

## ana4Stats.probAna.makeDictFromVars

### makeDictFromVars(*cfg*)

create a dictionary with info on parameter variation for all parameter in *varParList*

#### Parameters

**cfg** (*configparser object*) – configuration settings, here *varParList*, *variationValue*, *numberOfSteps*

#### Returns

**variationsDict** – dictionary with for each *varName*, *varVariation*, *varSteps*, and type of variation

#### Return type

dict

## ana4Stats.probAna.probAnalysis

### probAnalysis(*avaDir*, *cfg*, *module*, *parametersDict*="", *inputDir*="", *probConf*="", *simDFActual*="")

Compute probability map of a given set of simulation result exceeding a particular threshold and save to *outDir*

#### Parameters

- **avaDir** (*str*) – path to avalanche directory
- **cfg** (*dict*) – configuration read from ini file of *probAna* function
- **module** – computational module that was used to run the simulations
- **parametersDict** (*dict*) – dictionary with simulation parameters to filter simulations
- **inputDir** (*str*) – optional - path to directory where data that should be analysed can be found in a subfolder called *peakFiles* and *configurationFiles*, required if not in module results
- **probConf** (*str*) – name of probability configuration
- **simDFActual** (*pandas dataframe*) – dataframe of simulation configurations that shall be used for prob analysis

**ana4Stats.probAna.updateCfgRange****updateCfgRange**(*cfg*, *cfgProb*, *varName*, *varDict*)update *cfg* with a range for parameters in *cfgProb***Parameters**

- **cfg** (*configparser object*) – configuration object to update
- **cfgProb** (*configParser object*) – configparser object with info on update
- **varName** (*str*) – name of parameter used for variation
- **varDict** (*dict*) – dictionary with *variationValue* and *numberOfSteps* for *varName*

**Returns****cfg** – updated configuration object**Return type**

configParser

**2.10.3.4 ana5Utils**

Generate thalweg-time-diagrams and simulograms

**Modules**

---

<i>ana5Utils.DFAPathGeneration</i>	Tools for generating an avalanche path from a DFA simulation
<i>ana5Utils.distanceTimeAnalysis</i>	functions to convert to a different coordinate system and produce a range-time diagram from simulation results options: 1) range-time diagram from radar's field of view capped to this 2) thalweg-time diagram from beta point of view along avalanche path

---

**ana5Utils.DFAPathGeneration**

Tools for generating an avalanche path from a DFA simulation

## Functions

<i>appendAverageStd</i>	append averaged to path
<i>extendDFAPath</i>	extend the DFA path at the top and bottom avaProfile with x, y, z, s information
<i>extendProfileBottom</i>	extend the DFA path at the bottom (runout area)
<i>extendProfileTop</i>	extend the DFA path at the top (release)
<i>generateAveragePath</i>	extract path from fields or particles
<i>getDFAPathFromField</i>	compute mass averaged path from fields
<i>getDFAPathFromPart</i>	compute mass averaged path from particles
<i>getParabolicFit</i>	fit a parabola on a set of (s, z) points
<i>getSplitPoint</i>	find the split point corresponding to an avalanche profile, with parabolic fit and the slopeSplitPoint
<i>resamplePath</i>	resample the path profile and keep track of indStartMassAverage and indEndMassAverage
<i>saveSplitAndPath</i>	Save avaPath and split point to directory
<i>weightedAvgAndStd</i>	Return the weighted average and standard deviation.

### ana5Utils.DFAPathGeneration.appendAverageStd

**appendAverageStd**(propList, avaProfile, particles, weights, naming="")

append averaged to path

#### Parameters

- **propList** (*list*) – list of properties to average and append
- **avaProfile** (*dict*) – path
- **particles** (*dict*) – particles dict
- **weights** (*numpy array*) – array of weights (same size as particles)
- **naming** (*list*) – optional - list of properties at source (if different than final naming in avaProfile)

#### Returns

**avaProfile** – averaged profile

#### Return type

dict

### ana5Utils.DFAPathGeneration.extendDFAPath

**extendDFAPath**(cfg, avaProfile, dem, particlesIni)

extend the DFA path at the top and bottom avaProfile with x, y, z, s information

#### Parameters

- **cfg** (*configParser*) – configuration object with:
  - extTopOption: int, how to extend towards the top? 0 for height point method, a for largest runout method
  - nCellsResample: int, resampling length is given by nCellsResample\*demCellSize
  - nCellsMinExtend: int, when extending towards the bottom, take points at more

than  $nCellsMinExtend * demCellSize$  from last point to get the direction -  $nCellsMaxExtend$ : int, when extending towards the bottom, take points at less than  $nCellsMaxExtend * demCellSize$  from last point to get the direction -  $factBottomExt$ : float, extend the profile from  $factBottomExt * sMax$

- **avaProfile** (*dict*) – profile to be extended
- **dem** (*dict*) – dem dict
- **particlesIni** (*dict*) – initial particles dict

**Returns**

**avaProfile** – extended profile at top and bottom (x, y, z).

**Return type**

dict

### ana5Utils.DFAPathGeneration.extendProfileBottom

**extendProfileBottom**(*cfg, dem, profile*)

extend the DFA path at the bottom (runout area)

Find the direction in which to extend considering the last point of the profile and a few previous ones but discarding the ones that are too close ( $nCellsMinExtend * csz < distFromLast \leq nCellsMaxExtend * csz$ ). Extend in this direction for a distance  $factBottomExt * length$  of the path.

**Parameters**

- **cfg** (*configParser*) –  $nCellsMinExtend$ : int, when extending towards the bottom, take points at more than  $nCellsMinExtend * demCellSize$  from last point to get the direction  
  
 $nCellsMaxExtend$ : int, when extending towards the bottom, take points at less than  $nCellsMaxExtend * demCellSize$  from last point to get the direction  $factBottomExt$ : float, extend the profile from  $factBottomExt * sMax$
- **dem** (*dict*) – dem dict
- **profile** (*dict*) – profile to extend

**Returns**

**profile** – extended profile

**Return type**

dict

### ana5Utils.DFAPathGeneration.extendProfileTop

**extendProfileTop**(*extTopOption, particlesIni, profile*)

extend the DFA path at the top (release)

Either towards the highest point in particlesIni ( $extTopOption = 0$ ) or the point leading to the longest runout ( $extTopOption = 1$ )

**Parameters**

- **extTopOption** (*int*) – decide how to extend towards the top if 0, extrapolate towards the highest point in the release if 1, extrapolate towards the point leading to the lonest runout
- **particlesIni** (*dict*) – initial particles dict

- **profile** (*dict*) – profile to extend

**Returns**

**profile** – extended profile

**Return type**

dict

**ana5Utils.DFAPathGeneration.generateAveragePath**

**generateAveragePath**(*avalancheDir*, *pathFromPart*, *simName*, *dem*, *addVelocityInfo*=False, *flagAvaDir*=True, *comModule*='com1DFA')

extract path from fields or particles

**Parameters**

- **avalancheDir** (*pathlib*) – avalanche directory pathlib path
- **pathFromPart** (*boolean*) – compute path from particles if True, from fields (FT, FM, FV) if False
- **simName** (*str*) – simulation name
- **dem** (*dict*) – com1DFA simulation dictionary
- **addVelocityInfo** (*boolean*) – True to add (u2, ekin, totEKin) to result Will only work if the particles (ux, uy, uz) exist or if 'FV' exists
- **flagAvaDir** (*bool*) – if True avalancheDir corresponds to an avalanche directory and data is read from avaDir/Outputs/comModule/particles or avaDir/Outputs/comModule/peakFiles depending on if pathFromPart is True or False
- **comModule** (*str*) – module that computed the particles or fields

**Returns**

- **avaProfileMass** (*dict*) – mass averaged profile (x, y, z, 's') if addVelocityInfo is True, kinetic energy and velocity information are added to the avaProfileMass dict (u2, ekin, totEKin)
- **particlesIni** (*dict*) – x, y coord of the initial particles or flow thickness field

**ana5Utils.DFAPathGeneration.getDFAPathFromField**

**getDFAPathFromField**(*fieldsList*, *fieldHeader*, *dem*)

compute mass averaged path from fields

Also returns the averaged velocity and kinetic energy associated The dem and fieldsList (FT, FV and FM) need to have identical dimensions and cell size. If FV is not provided, information about velocity and kinetic energy is not computed

**Parameters**

- **fieldsList** (*list*) – time sorted list of fields dict
- **fieldHeader** (*dict*) – field header dict
- **dem** (*dict*) – dem dict

**Returns**

**avaProfileMass** – mass averaged profile (x, y, z, 's') if 'FV' in fieldsList, kinetic energy and velocity information are added to the avaProfileMass dict (u2, ekin, totEKin)

**Return type**

dict

**ana5Utils.DFAPathGeneration.getDFAPathFromPart****getDFAPathFromPart**(*particlesList*, *addVelocityInfo=False*)

compute mass averaged path from particles

Also returns the averaged velocity and kinetic energy associated If *addVelocityInfo* is True, information about velocity and kinetic energy is computed

**Parameters**

- **particlesList** (*list*) – list of particles dict
- **addVelocityInfo** (*boolean*) – True to add (u2, ekin, totEKin) to result

**Returns**

**avaProfileMass** – mass averaged profile (x, y, z, 's', 'sCor') if *addVelocityInfo* is True, kinetic energy and velocity information are added to the *avaProfileMass* dict (u2, ekin, totEKin)

**Return type**

dict

**ana5Utils.DFAPathGeneration.getParabolicFit****getParabolicFit**(*cfg*, *avaProfile*, *dem*)

fit a parabola on a set of (s, z) points

first and last point match. Last constraint is given by *fitOption* (see below)

**Parameters**

- **cfg** (*configParser*) – configuration object with: *fitOption*: int how to optimize the fit 0 minimize distance between points 1 match the end slope
- **avaProfile** (*dict*) – profile to be extended
- **dem** (*dict*) – dem dict

**Returns**

**parabolicFit** – a, b, c coefficients of the parabolic fit ( $y = a \cdot x^2 + b \cdot x + c$ )

**Return type**

dict

**ana5Utils.DFAPathGeneration.getSplitPoint****getSplitPoint**(*cfg*, *avaProfile*, *parabolicFit*)

find the split point corresponding to an avalanche profile, with parabolic fit and the *slopeSplitPoint*

**Parameters**

- **cfg** (*configParser*) – configuration object with
  - *slopeSplitPoint*: float, desired slope at split point (in degrees)
  - *dsMin*: float, threshold distance [m] for looking for the



- split point (at least dsMin meters below split point angle)
- **avaProfile** (*dict*) – profile to be extended
- **parabolicFit** (*dict*) – a, b, c coefficients of the parabolic fit ( $y = a*a*x + b*x + c$ )

**Returns**

**splitPoint** – (x, y, z, zPra, s) at split point location.

**Return type**

dict

**ana5Utils.DFAPathGeneration.resamplePath**

**resamplePath**(*cfg, dem, avaProfile*)

resample the path profile and keep track of indStartMassAverage and indEndMassAverage

**Parameters**

- **cfg** (*configParser*) – configuration object with
  - nCellsResample: int, resampling length is given by nCellsResample\*demCellSize
- **dem** (*dict*) – dem dict
- **avaProfile** (*dict*) – profile to be resampled

**Returns**

**avaProfile** – resampled path profile

**Return type**

dict

**ana5Utils.DFAPathGeneration.saveSplitAndPath**

**saveSplitAndPath**(*avalancheDir, simDFrow, splitPoint, avaProfileMass, dem*)

Save avaPath and split point to directory

**Parameters**

- **avalancheDir** (*pathlib*) – avalanche directory pathlib path
- **simDFrow** (*pandas series*) – row of the siulation dataframe corresponding to the current simulation analyzed
- **splitPoint** (*dict*) – point dictionary
- **avaProfileMass** (*dict*) – line dictionary
- **dem** (*dict*) – com1DFA simulation dictionary

**Returns**

**avaProfile** – resampled path profile

**Return type**

dict

**ana5Utils.DFAPathGeneration.weightedAvgAndStd****weightedAvgAndStd**(*values*, *weights*)

Return the weighted average and standard deviation.

*values*, *weights* – Numpy ndarrays with the same shape.

**ana5Utils.distanceTimeAnalysis**

functions to convert to a different coordinate system and produce a range-time diagram from simulation results options:

1) range-time diagram from radar's field of view capped to this 2) thalweg-time diagram from beta point of view along avalanche path

## Functions

<i>approachVelocity</i>	compute maximal approach velocity based on front location (range) and time step
<i>createThalwegTimeInfoFromSimResults</i>	top level function to create mtiInfo dict from com Module simulation results required are flow variable fields for different time steps
<i>exportData</i>	save all info about range, time steps, average values to pickle for producing plots
<i>extractFrontAndMeanValuesRadar</i>	extract average values of flow parameter result at certain distance intervals (range gates) add info to be used for colorcoding range-time diagram
<i>extractFrontAndMeanValuesTT</i>	extract avalanche front and mean values of flow parameter result field used for colorcoding range-time diagram
<i>fetchRangeTimeInfo</i>	determine avalanche front and average values of flow parameter along path update mtiInfo dictionary with this information
<i>fetchTimeStepFromName</i>	split path name to fetch info on time step
<i>getRadarLocation</i>	fetch radar location and direction of of field of view coordinates
<i>importMTIData</i>	import mtiInfo data for creating range time plots, if no inputDir is provided, pickles are fetched from avaDir/Outputs/modName/distanceTimeAnalysis multiple pickles allowed- these carry also configuration info for distanceTimeAnalysis
<i>initializeRangeTime</i>	initialize generation of range-time diagram for visualizing simulation data
<i>maskRangeFull</i>	mask range (already masked with radar field of view) also with avalanche result field where NOT above threshold
<i>radarMask</i>	function to create masked array of radar range using dem and radar's field of view
<i>setDemOrigin</i>	reset original origin to simulation DEM - required if for ava sim is set to something different
<i>setupRangeTimeDiagram</i>	create setup for creating range-time diagrams from avalanche simulation results with respect to a radar's field of view
<i>setupThalwegTimeDiagram</i>	initialize parameters and prerequisites for generating thalweg-time diagrams

## ana5Utils.distanceTimeAnalysis.approachVelocity

### approachVelocity(mtiInfo)

compute maximal approach velocity based on front location (range) and time step

- cleans nan in range
- neglects anything behind maximal runoff
- neglects non-unique range values, e.g. the front did not move
- **cleans approach velocity: velocity in cell under test can not be higher** than the double of the mean from the surrounding cells

**Parameters**

**mtiInfo** (*dict*) – info on distance to front and time steps

**Returns**

- **maxVel** (*float*) – max value of approach velocity
- **rangeVel** (*float*) – range of max value of approach velocity
- **timeVel** (*float*) – time of max value of approach velocity

**ana5Utils.distanceTimeAnalysis.createThalwegTimeInfoFromSimResults**

**createThalwegTimeInfoFromSimResults**(*avalancheDir, cfgRangeTime, modName, index, simDF, dem*)

top level function to create mtiInfo dict from com Module simulation results required are flow variable fields for different time steps

**Parameters**

- **avalancheDir** (*str or pathlib path*) – path to avalanche directory
- **cfgRangeTime** (*configparser object*) – configuration settings for range time diagram
- **modName** (*str*) – name of computational module that has been used to create ava sim
- **index** (*str*) – index of current sim in simDF
- **simDF** (*dataFrame*) – dataframe with info on sim configuration
- **dem** (*dict*) – dictionary with simulation dem

**Returns**

- **cfgRangeTime** (*configparser object*) – configuration settings for range time
- **mtiInfo** (*dict*) – dictionary with info for creating range time plots

**ana5Utils.distanceTimeAnalysis.exportData**

**exportData**(*mtiInfo, cfgRangeTime, modName*)

save all info about range, time steps, average values to pickle for producing plots

**Parameters**

- **mtiInfo** (*dict*) – dictionary with rangeList, timeList, mti (average values along range gates or cross profiles)
- **cfgRangeTime** (*configparser*) – configuration settings of distance time
- **modName** (*str*) – name of computational module

## ana5Utils.distanceTimeAnalysis.extractFrontAndMeanValuesRadar

**extractFrontAndMeanValuesRadar**(*cfgRangeTime*, *flowF*, *mtiInfo*)

extract average values of flow parameter result at certain distance intervals (range gates) add info to be used for colorcoding range-time diagram

### Parameters

- **cfgRangeTime** (*configparser object*) – configuration settings for creating range-time diagram
- **flowF** (*numpy array*) – flow parameter result field
- **mtiInfo** (*dict*) – info here used: rArray, bmaskRadar rangeMasked, rangeGates, mti, timeList

### Returns

**mtiInfo** – updated mtiInfo dict with info on mti values

### Return type

dict

## ana5Utils.distanceTimeAnalysis.extractFrontAndMeanValuesTT

**extractFrontAndMeanValuesTT**(*cfgRangeTime*, *flowF*, *demHeader*, *mtiInfo*)

extract avalanche front and mean values of flow parameter result field used for colorcoding range-time diagram

### Parameters

- **cfgRangeTime** (*configparser object*) – configuration settings
- **flowF** (*numpy nd array*) – flow parameter result field
- **demHeader** (*dict*) – dictionary with info on DEM header used for locating fields
- **mtiInfo** (*dict*) – dictionary with arrays, lists for range-time diagram: mti for colorcoding (average values of range gates/ crossprofiles) timeList time step info rangeList range info rangeGates info on cross profile locations rangeRaster - info on distance from beta point in path following coordinate system rasterTransfo - information on domain transformation to path following coordinate system

### Returns

**mtiInfo** – updated dictionary with info on average value along crossprofiles (mti) and distance to front from reference point along path (rangeList)

### Return type

dict

## ana5Utils.distanceTimeAnalysis.fetchRangeTimeInfo

**fetchRangeTimeInfo**(*cfgRangeTime*, *cfg*, *dtRangeTime*, *t*, *demHeader*, *fields*, *mtiInfo*)

determine avalanche front and average values of flow parameter along path update mtiInfo dictionary with this information

### Parameters

- **cfgRangeTime** (*configparser*) – configuration settings for range-time diagram
- **cfg** (*configParser object*) – configuration settings of computational module

- **dtRangeTime** (*list*) – list of time steps where avalanche front shall be exported
- **t** (*float*) – actual simulation time step
- **demHeader** (*dict*) – dictionary with DEM header
- **fields** (*dict*) – dictionary with flow parameter result fields
- **mtiInfo** (*dict*) – info on time steps (*timeList*) and distance (*rangeList*) of avalanche front averaged values of result parameter (*mti*) for each range gate (*rangeGates*) for colorcoding optional info on masks for radar field of view (*rangeMasked*), etc.

**Returns**

- **mtiInfo** (*dict*) – updated dictionary
- **dtRangeTime** (*list*) – updated list of time steps

**ana5Utils.distanceTimeAnalysis.fetchTimeStepFromName****fetchTimeStepFromName**(*pathNames*)

split path name to fetch info on time step

**Parameters**

**pathName** (*pathlib path or list of paths*) – path to file including time step info as `_t%.2f_`

**Returns**

- **timeStep** (*list*) – actual time steps
- **indexTime** (*numpy array*) – index of timeStep list to order them in ascending order

**ana5Utils.distanceTimeAnalysis.getRadarLocation****getRadarLocation**(*cfg*)

fetch radar location and direction of field of view coordinates

**Parameters**

**cfg** (*configparser object*) – configuration settings

**Returns**

**radarFov** – array with x and y coordinates of radar location and point in direction of field of view

**Return type**

numpy array

**ana5Utils.distanceTimeAnalysis.importMTIData****importMTIData**(*avaDir, modName, inputDir="", simHash=""*)

import mtiInfo data for creating range time plots, if no *inputDir* is provided, pickles are fetched from *avaDir/Outputs/modName/distanceTimeAnalysis* multiple pickles allowed- these carry also configuration info for *distanceTimeAnalysis*

**Parameters**

- **avaDir** (*pathlib path or str*) – path to avalanche directory

- **modName** (*str*) – name of computational module that has been used to produce flow variable fields
- **inputDir** (*pathlib path or str*) – optional: path to pickle location
- **simHash** (*str*) – optional simulation ID

**Returns**

**mtiInfoDicts** – list of mtiInfo dictionaries where key name has been added with file Path

**Return type**

list

## ana5Utils.distanceTimeAnalysis.initializeRangeTime

**initializeRangeTime**(*modName, cfg, dem, simHash*)

initialize generation of range-time diagram for visualizing simulation data

**Parameters**

- **modName** – module name
- **cfg** (*configparser object*) – configuration settings from computational module
- **dem** (*dict*) – dictionary with DEM header and data
- **simHash** (*str*) – unique simulation ID

**Returns**

**mtiInfo** – info on time steps (timeList) and distance (rangeList) of avalanche front averaged values of result parameter (mti) for each range gate (rangeGates) for colorcoding x, y origin of DEM, cellSize and plotTitle optional info on masks for radar field of view (rangeMasked), etc.

**Return type**

dict

## ana5Utils.distanceTimeAnalysis.maskRangeFull

**maskRangeFull**(*flowF, threshold, rangeMasked*)

mask range (already masked with radar field of view) also with avalanche result field where NOT above threshold

**Parameters**

- **flowF** (*numpy array*) – flow variable result field
- **threshold** (*float*) – threshold of result field - masked where values NOT above this threshold
- **rangeMasked** (*numpy masked array*) – range to radar location masked with radar field of view

**Returns**

- **maskAva** (*numpy mask*) – mask where result field NOT above threshold
- **bmaskAvaRadar** (*numpy mask*) – mask where result field NOT above threshold and NOT in radar field of view
- **rMaskedAvaRadar** (*numpy masked array*) – masked range array with NOT in radar field of view and result field NOT above threshold

### ana5Utils.distanceTimeAnalysis.radarMask

**radarMask**(*demOriginal*, *radarFov*, *aperture*, *cfgRangeTime*)

function to create masked array of radar range using dem and radar's field of view

#### Parameters

- **dem** (*dict*) – DEM dictionary with header info and rasterData
- **radarFov** (*numpy array*) – radars field of view min and max points
- **aperture** (*float*) – aperature angle
- **cfgRangeTime** (*configparser object*) – configuration settings here used: avalancheDir info

#### Returns

- **radarRange** (*numpy masked array*) – masked array of radar range covering full DEM extent
- **rangeGates** (*numpy array*) – range gates

### ana5Utils.distanceTimeAnalysis.setDemOrigin

**setDemOrigin**(*demSims*)

reset original origin to simulation DEM - required if for ava sim is set to something different

#### Parameters

**demSims** (*dict*) – dictionary with header and data of DEM used to run ava simulations must contain a key called originalHeader - where the original origin is given

#### Returns

**demOriginal** – updated DEM with xllcenter and yllcenter set to original origin cellsize and nrows and columns kept from simulation DEM

#### Return type

dict

### ana5Utils.distanceTimeAnalysis.setupRangeTimeDiagram

**setupRangeTimeDiagram**(*dem*, *cfgRangeTime*)

create setup for creating range-time diagrams from avalanche simulation results with respect to a radar's field of view

#### Parameters

- **dem** (*dict*) – DEM dictionary with header and raster data
- **cfgRangeTime** (*configparser object*) – configuration settings for range-time diagrams

#### Returns

**rangeMasked** – radar range retrieved from DEM and masked with radar's field of view

#### Return type

numpy masked array



## ana5Utils.distanceTimeAnalysis.setupThalwegTimeDiagram

### setupThalwegTimeDiagram(*dem*, *cfgRangeTime*)

initialize parameters and prerequisites for generating thalweg-time diagrams

#### Parameters

- **dem** (*dict*) – dictionary with info on dem header and data
- **cfgRangeTime** (*confiparser object*) – configuration settings of range-time diagram

#### Returns

**mtiInfo** – dictionary with arrays, lists for range-time diagram: mti for colorcoding (average values of range gates/ crossprofiles) timeList time step info rangeList range info rangeGates info on cross profile locations rangeRaster - info on distance from beta point in path following coordinate system betaPoint - coordinates of start of runout zone point betaPointAngle - angle of beta point used

#### Return type

dict

## 2.10.3.5 log2Report

Tools for generating reports

### Modules

<code>log2Report.generateCompareReport</code>	Generate a markdown report for data provided in dictionary with format to compare two simulations
<code>log2Report.generateReport</code>	Generate a markdown report for data provided in dictionary

### log2Report.generateCompareReport

Generate a markdown report for data provided in dictionary with format to compare two simulations

### Functions

<code>copyAimecPlots</code>	copy the aimec plots to report directory
<code>copyQuickPlots</code>	copy the quick plots to report directory
<code>makeLists</code>	Create a list for table creation
<code>writeCompareReport</code>	Write a report in markdown format of the comparison of simulations to reference simulation results; report is saved to location of reportFile

**log2Report.generateCompareReport.copyAimecPlots****copyAimecPlots**(*plotFiles*, *testName*, *outDir*, *plotPaths*, *rel=""*)

copy the aimec plots to report directory

**log2Report.generateCompareReport.copyQuickPlots****copyQuickPlots**(*avaName*, *testName*, *outDir*, *plotListRep*, *rel=""*)

copy the quick plots to report directory

**log2Report.generateCompareReport.makeLists****makeLists**(*simDict*, *benchDict*)

Create a list for table creation

**log2Report.generateCompareReport.writeCompareReport****writeCompareReport**(*reportFile*, *reportD*, *benchD*, *avaName*, *cfgRep*)Write a report in markdown format of the comparison of simulations to reference simulation results; report is saved to location of *reportFile***Parameters**

- **reportFile** (*str*) – path to report file
- **reportD** (*dict*) – report dictionary with simulation info
- **benchD** (*str*) – dictionary with simulation info of reference simulation
- **avaName** (*str*) – name of avalanche
- **cfgRep** (*dict*) – dictionary with configuration info

**log2Report.generateReport**

Generate a markdown report for data provided in dictionary

**Functions**

<i>addLineBlock</i>	add lineblock to report
<i>copyPlots2ReportDir</i>	copy the plots to report directory The plots are in a dictionary: <code>plotDict = {'plot1': PurePath to plot1, 'plot2': PurePath to plot2...}</code>
<i>writeReport</i>	Write a report in markdown format for simulations, saved to <i>outDir</i>
<i>writeReportFile</i>	Create markdown report with blocks, tables, list according to type key

## log2Report.generateReport.addLineBlock

**addLineBlock**(*titleString*, *reportDKey*, *pfile*, *italicFont=False*, *onlyFirstLine=False*)

add lineblock to report

### Parameters

- **titleString** (*str*) – string that shall be added
- **reportDKey** (*dict*) – dictionary with info for string
- **pfile** (*file*)
- **italicFont** (*bool*) – if True write value in italic
- **onlyFirstLine** (*bool*) – if first item in reportDKey is added to pfile - break

## log2Report.generateReport.copyPlots2ReportDir

**copyPlots2ReportDir**(*reportDir*, *plotDict*)

copy the plots to report directory The plots are in a dictionary: plotDict = {'plot1': PurePath to plot1, 'plot2': PurePath to plot2... }

### Parameters

- **reportDir** (*pathlib path*) – path to directory where to copy the plots to
- **plotDict** (*dict*) – dict of the location of the plots to copy

## log2Report.generateReport.writeReport

**writeReport**(*outDir*, *reportDictList*, *reportOneFile*, *plotDict=""*, *standaloneReport=False*, *reportName='fullSimulationReport'*)

Write a report in markdown format for simulations, saved to outDir

### Parameters

- **outDir** (*str*) – path to output directory
- **reportDictList** (*list*) – list of report dictionaries from simulations
- **reportOneFile** (*boolean*) – True to write all info in the same file
- **plotDict** (*dict*) – optional dictionary with info on plots that shall be included in report
- **standaloneReport** (*bool*) – if True copy plots to reportDir
- **reportName** (*str*) – report file name, fullSimulationReport is the default value

## log2Report.generateReport.writeReportFile

### writeReportFile(*reportD*, *pfile*)

Create markdown report with blocks, tables, list according to type key

#### Parameters

- **reportD** (*dict*) – report dictionary
- **pfile** (*file object*) – file where markdown format report is written to

## 2.10.3.6 out1Peak

Simple tools for visualising datasets.

### Modules

---

<i>out1Peak.outPlotAllPeak</i>	Functions to plot 2D simulation results: plot of all peak files at once
--------------------------------	---

---

### out1Peak.outPlotAllPeak

Functions to plot 2D simulation results: plot of all peak files at once

### Functions

---

<i>addConstrainedDataField</i>	find fileName data, constrain data and demField to where there is data, create colormap, define extent, add hillshade contours, add to axes and add colorbar
<i>plotAllFields</i>	Plot all fields within given directory and save to outDir
<i>plotAllPeakFields</i>	Plot all peak fields and return dictionary with paths to plots with DEM in background

---

### out1Peak.outPlotAllPeak.addConstrainedDataField

#### addConstrainedDataField(*fileName*, *resType*, *demField*, *ax*, *cellSize*, *alpha=1.0*, *setLimits=False*, *oneColor=""*)

find fileName data, constrain data and demField to where there is data, create colormap, define extent, add hillshade contours, add to axes and add colorbar

#### Parameters

- **fileName** (*pathlib path*) – path to data
- **resType** (*str*) – name of result variable type
- **demField** (*numpy ndarray*) – array of dem data
- **ax** (*matplotlib axes object*) – axes where to add data plot to
- **cellSize** (*float*) – cellSize of data
- **alpha** (*float*) – from 0 transparent to 1 opaque for plot of constrained data

- **setLimits** (*bool*) – if True set limits of constrained data to plot
- **oneColor** (*str*) – optional to add a color for a single color for field

**Returns**

**ax** – axes updated

**Return type**

matplotlib axes object

**out1Peak.outPlotAllPeak.plotAllFields**

**plotAllFields**(*avaDir, inputDir, outDir, unit="", constrainData=True*)

Plot all fields within given directory and save to outDir

**Parameters**

- **avaDir** (*str*) – path to avalanche directoy
- **inputDir** (*str*) – path to input directoy
- **outDir** (*str*) – path to directoy where plots shall be saved to
- **unit** (*str*) – unit of result type

**out1Peak.outPlotAllPeak.plotAllPeakFields**

**plotAllPeakFields**(*avaDir, cfgFLAGS, modName, demData=""*)

Plot all peak fields and return dictionary with paths to plots with DEM in background

**Parameters**

- **avaDir** (*str*) – path to avalanche directoy
- **cfgFLAGS** (*str*) – general configuration, required to define if plots saved to reports directoy
- **modName** (*str*) – name of module that has been used to produce data to be plotted
- **demData** (*dictionary*) – optional - if not the dem in the avaDir/Inputs folder has been used but a different one

**Returns**

**plotDict** – dictionary with info on plots, like path to plot

**Return type**

dict

**2.10.3.7 out3Plot**

Plotting

## Modules

<code>out3Plot.amaPlots</code>	functions for plotting ama event data and geometry info
<code>out3Plot.in1DataPlots</code>	make plots for in1Data module
<code>out3Plot.outAIMEC</code>	Plotting and saving AIMEC results
<code>out3Plot.outAna1Plots</code>	
<code>out3Plot.outCom1DFA</code>	
<code>out3Plot.outCom3Plots</code>	
<code>out3Plot.outContours</code>	Plotting and saving contour line plots
<code>out3Plot.outDebugPlots</code>	
<code>out3Plot.outDistanceTimeAnalysis</code>	functions to plot range-time diagrams
<code>out3Plot.outParticlesAnalysis</code>	Tools to extract information on the avalanche simulations run in the Output files
<code>out3Plot.outQuickPlot</code>	Functions to plot 2D avalanche simulation results as well as comparison plots between two datasets of identical shape.
<code>out3Plot.outTopo</code>	Simple plotting for DEMs
<code>out3Plot.statsPlots</code>	plot statistics of simulations

### out3Plot.amaPlots

functions for plotting ama event data and geometry info

## Functions

<code>plotBoxPlot</code>	create a boxplot of all columns of dbData
<code>plotHist</code>	create a histogram of name column of dbData
<code>plotPathAngle</code>	make a two panel plot of avalanche thalweg, release point, runout point, origin, transit and deposition point and xy distances and computed angles

### out3Plot.amaPlots.plotBoxPlot

**plotBoxPlot**(*dbData*, *colList*, *outDir*, *namePlot*, *renameCols*=[])

create a boxplot of all columns of dbData

#### Parameters

- **dbData** (*pandas dataframe*) – dataframe with geometry info
- **colList** (*list*) – names of columns in dbData to plot
- **outDir** (*pathlib path or str*) – path to folder where plot shall be saved to
- **namePlot** (*str*) – name of plot to be added to boxplot

**out3Plot.amaPlots.plotHist****plotHist**(*dbData*, *name*, *cfgMain*, *colorcode*="")

create a histogram of name column of dbData

**Parameters**

- **dbData** (*pandas dataframe*) – dataframe with geometry info of events and name\_Line and name\_Angle
- **name** (*str*) – name of Line and Angle to plot
- **cfgMain** (*configparser*) – configuration settings
- **colorcode** (*str*) – name of column to colorcode data

**out3Plot.amaPlots.plotPathAngle****plotPathAngle**(*dbData*, *cfgMain*, *nameEvent*, *tickOrigin*='orig-transit')

make a two panel plot of avalanche thalweg, release point, runout point, origin, transit and deposition point and xy distances and computed angles

**Parameters**

- **dbData** (*pandas dataframe*) – dataframe with geometry info of events
- **cfgMain** (*configparser object*) – config settings here used: avalancheDir, projstr,
- **nameEvent** (*str*) – name of xy distance lines and angles
- **tickOrigin** (*str*) – origin for panel 2 Sxy distance origin

**out3Plot.in1DataPlots**

make plots for in1Data module

**Functions**

<i>plotAreaShpError</i>	plot polygon parts of polygon read from shp file to check if holes
<i>plotDist</i>	plot the CDF
<i>plotECDF</i>	make a comparison plot of desired CDF and empirical CDF of sample
<i>plotEmpCDF</i>	make a comparison plot of desired CDF and empirical CDF of sample
<i>plotEmpPDF</i>	make a comparison plot of desired CDF and empirical CDF of sample
<i>plotSample</i>	Generate bar plot of sample values
<i>plotSamplePDF</i>	make comparison plot of desired PDF and approximated sample PDF

**out3Plot.in1DataPlots.plotAreaShpError****plotAreaShpError**(*xFeat*, *yFeat*, *nParts*, *pathDict*)

plot polygon parts of polygon read from shp file to check if holes

**Parameters**

- **xFeat, yFeat** (*numpy array*) – x, y coordinates of polygon
- **nParts** (*list*) – indices of parts of polygon (2 make only an outer polygon as entire number of points is added)
- **pathDict** (*dict*) – dictionary with info on outDir, outFileName and title of plot

**out3Plot.in1DataPlots.plotDist****plotDist**(*workingDir*, *CDF*, *a*, *b*, *c*, *cfg*, *flagShow*)

plot the CDF

**out3Plot.in1DataPlots.plotECDF****plotECDF**(*workingDir*, *CDF*, *sample*, *cfg*, *methodAbbr*, *flagShow*)

make a comparison plot of desired CDF and empirical CDF of sample

**out3Plot.in1DataPlots.plotEmpCDF****plotEmpCDF**(*workingDir*, *CDF*, *CDFEmp*, *xSample*, *cfg*, *methodAbbr*, *flagShow*, *x=""*)

make a comparison plot of desired CDF and empirical CDF of sample

**out3Plot.in1DataPlots.plotEmpPDF****plotEmpPDF**(*workingDir*, *PDF*, *sampleVect*, *cfg*, *flagShow*, *x=""*)

make a comparison plot of desired CDF and empirical CDF of sample

**out3Plot.in1DataPlots.plotSample****plotSample**(*workingDir*, *sample*, *cfg*, *flagShow*)

Generate bar plot of sample values

**out3Plot.in1DataPlots.plotSamplePDF****plotSamplePDF**(*workingDir*, *sampleVect*, *kdeDict*, *PDF*, *cfg*, *flagShow*)

make comparison plot of desired PDF and approximated sample PDF



## out3Plot.outAIMEC

Plotting and saving AIMEC results

This file is part of Avaframe.

### Functions

<i>addLinePlot</i>	add a contour line with label only for line that has _0 in name
<i>addThalwegAltitude</i>	add thalweg-altitude plot to axes the thalweg z profile and the $mpfv^2/2g$ (velocity-altitude) colorcoded using the mpfv values the mpfv values come from the cross max values along the thalweg coordinate system (aimec)
<i>fetchContourLines</i>	fetch contour line of transformed field data
<i>getIndicesVel</i>	create indices of peak flow velocity cross max vector first above tresholdValue and first below threshold again
<i>plotContoursTransformed</i>	plot contour lines of all transformed fields colorcode contour lines with first parameter in varParList if not type string of value
<i>plotMaxValuesComp</i>	plot result type name1 vs name 2 and colorcode scenarios using hue add reference sim value with label
<i>plotThalwegAltitude</i>	create thalweg-altitude plot the thalweg z profile and the $mpfv^2/2g$ (velocity-altitude) colorcoded using the mpfv values the mpfv values come from the cross max values along the thalweg coordinate system (aimec) plot is saved to avaDir/Outputs/out3Plot/thalwegAltSimname
<i>plotVelThAlongThalweg</i>	plot the velocity and thickness cross max values along the thalweg, with pft x10 only plot every barInt value
<i>resultVisu</i>	plot the normalized area difference between reference and other simulations
<i>resultWrite</i>	This function writes the main Aimec results to a file (outputFile) in pathDict
<i>visuComparison</i>	Plot and save the comparison between current simulation and Reference in the runout area
<i>visuMass</i>	Plot and save the results from mass analysis
<i>visuRunoutComp</i>	Plot and save the Peak Fields distribution (max mean per cross section) after coordinate transformation
<i>visuRunoutStat</i>	Panel1 reference peak field with runout points of all sims and distribution of runout SXY Panel 2 crossMax values of peak field along thalweg for all sims Panel 3 mean, median and envelope of cross max values for all sims
<i>visuSimple</i>	Plot and save the Peak Pressure Peak Flow thickness and Peak speed fields after coord transfo
<i>visuTransfo</i>	Plot and save the domain transformation figure The left subplot shows the reference result raster with the outline of the new domain.

### out3Plot.outAIMEC.addLinePlot

**addLinePlot**(*contourDict*, *colorStr*, *labelStr*, *ax*, *key*, *zorder*="", *linestyle*='solid', *alpha*=1.0)

add a contour line with label only for line that has \_0 in name

#### Parameters

- **contourDict** (*dict*) – dict with x, y info of contourline coordinates
- **colorStr** (*str*) – color of line
- **labelStr** (*str*) – name of legend entry for line
- **ax** (*matplotlib axes object*) – axes where to add the lines too
- **key** (*str*) – name of the contour line
- **zorder** (*int*) – order of line on plot
- **linestyle** (*str*) – style of line plot: dashed, solid, dotted,..
- **alpha** (*float*) – between 0 and 1 to define how opaque the line

### out3Plot.outAIMEC.addThalwegAltitude

**addThalwegAltitude**(*ax1*, *rasterTransfo*, *pfvCrossMax*, *velocityThreshold*, *zMaxM*=*numpy.nan*)

add thalweg-altitude plot to axes the thalweg z profile and the  $\text{mpfv}^2/2g$  (velocity-altitude) colorcoded using the mpfv values the mpfv values come from the cross max values along the thalweg coordinate system (aimec)

#### Parameters

- **rasterTransfo** (*dict*) – dict with info on transformation from cartesian to thalweg coordinate system
- **pfvCrossMax** (*pandas dataframe series*) – cross profile max values of peak flow velocity transformed into thalweg coordinate system
- **zMaxM** (*float*) – optional - value to define max limit of y -axis

### out3Plot.outAIMEC.fetchContourLines

**fetchContourLines**(*rasterTransfo*, *inputs*, *level*, *contourDict*)

fetch contour line of transformed field data

#### Parameters

- **rasterTransfo** (*dict*) – raster transformation dictionary
- **inputs** (*dict*) – input data here needed the transformed field data and simName
- **level** (*float*) – the contour level
- **contourDict** (*dict*) – dictionary with x, y coordinates for each sim contour line

#### Returns

**contourDict** – updat. dictionary with name of contour line: x, y coordinates for each sim contour line -> added info of current simulation

#### Return type

dict

**out3Plot.outAIMEC.getIndicesVel****getIndicesVel**(*pfvCM*, *velocityThreshold*)

create indices of peak flow velocity cross max vector first above thresholdValue and first below threshold again

**Parameters**

- **pfvCM** (*numpy array*) – peak flow velocity cross max values along thalweg
- **velocityThreshold** (*float*) – threshold value

**Returns**

- **indVelStart** (*int*) – index where pfvCM first exceeds threshold
- **indVelZero** (*int*) – index where pfvCM first smaller than threshold but only further downstream than indVelStart

**out3Plot.outAIMEC.plotContoursTransformed****plotContoursTransformed**(*contourDict*, *pathDict*, *rasterTransfo*, *cfgSetup*, *inputsDF*)

plot contour lines of all transformed fields colorcode contour lines with first parameter in varParList if not type string of value

**Parameters**

- **contourDict** (*dict*) – dictionary with contourline coordinates
- **pathDict** (*dict*) – dictionary with info on project name
- **rasterTransfo** (*dict*) – raster transformation dictionary
- **cfgSetup** (*configparser*) – configuration settings for AIMECSETUP
- **inputsDF** (*pandas DataFrame*) – dataframe with one row per simulation and information on parameter (if no configuration file found, derived from simName) and available files

**out3Plot.outAIMEC.plotMaxValuesComp****plotMaxValuesComp**(*pathDict*, *resultsDF*, *name1*, *name2*, *hue=None*)

plot result type name1 vs name 2 and colorcode scenarios using hue add reference sim value with label

**resultsDF: pandas dataframe**

dataframe with one row per simulation with info on parameters and aimec analysis results

**name1: str**

name of result type one

**name2: str**

name of result type two

**hue: str**

name of parameter used for colorcoding

### out3Plot.outAIMEC.plotThalwegAltitude

**plotThalwegAltitude**(*pathDict*, *rasterTransfo*, *pfvCrossMax*, *simName*, *velocityThreshold*)

create thalweg-altitude plot the thalweg z profile and the  $\text{mpfv}^2/2g$  (velocity-altitude) colorcoded using the mpfv values the mpfv values come from the cross max values along the thalweg coordinate system (aimec) plot is saved to `avaDir/Outputs/out3Plot/thalwegAltSimname`

#### Parameters

- **avaDir** (*pathlib path*) – path to avalanche directory
- **rasterTransfo** (*dict*) – dict with info on transformation from cartesian to thalweg coordinate system
- **pfvCrossMax** (*pandas dataframe series*) – cross profile max values of peak flow velocity transformed into thalweg coordinate system
- **simName** (*str*) – simulation name

### out3Plot.outAIMEC.plotVelThAlongThalweg

**plotVelThAlongThalweg**(*pathDict*, *rasterTransfo*, *pftCrossMax*, *pfvCrossMax*, *cfgPlots*, *simName*)

plot the velocity and thickness cross max values along the thalweg, with pft x10 only plot every barInt value

#### Parameters

- **pathDict** (*dict*) – info on avalancheDir
- **rasterTransfo** (*dict*) – info on domain transformation
- **pftCrossMax** (*numpy nd array*) – peak flow thickness max values along cross profiles of thalweg coordinate system
- **pfvCrossMax** (*numpy nd array*) – peak flow velocity max values along cross profiles of thalweg coordinate system
- **cfgPlots** (*configparser object*) – used: `barInterval`: width to compute the values that should be used for plotting `velocityTreshold`: threshold for computation of alpha angle where to identify stop of avalanche
- **simName** (*str*) – simulation name

### out3Plot.outAIMEC.resultVisu

**resultVisu**(*cfgSetup*, *inputsDF*, *pathDict*, *cfgFlags*, *rasterTransfo*, *resAnalysisDF*)

plot the normalized area difference between reference and other simulations

**out3Plot.outAIMEC.resultWrite**

**resultWrite**(*pathDict*, *cfg*, *rasterTransfo*, *resAnalysisDF*)

This function writes the main Aimec results to a file (outputFile) in pathDict

**out3Plot.outAIMEC.visuComparison**

**visuComparison**(*rasterTransfo*, *inputs*, *pathDict*)

Plot and save the comparison between current simulation and Reference in the runout area

**Parameters**

- **rasterTransfo** (*dict*) – domain transformation information
- **inputs** (*dict*) –  
**input data for plot:**  
 ‘refData’ and ‘compData’ arrays ‘refRasterMask’ and ‘compRasterMask’ arrays ‘thresholdArray’ ‘i’ id of the simulation ‘runoutResType’ result type analyzed ‘diffLim’
- **pathDict** (*dict*) – dictionary with path to data to analyze

**out3Plot.outAIMEC.visuMass**

**visuMass**(*resAnalysisDF*, *pathDict*, *simRowHash*, *refSimRowHash*, *timeMass*)

Plot and save the results from mass analysis

**Parameters**

- **resAnalysis** (*dict*) –  
**mass results from Aimec analysis:**  
 entMassFlowArray: entrained mass array corresponding to the time array for each simulation to analyse totalMassArray: entrained mass array corresponding to the time array for each simulation to analyse entMass: final entrained for each simulation finalMass: final mass for each simulation time: time array
- **pathDict** (*dict*) – dictionary with path to data to analyze

**out3Plot.outAIMEC.visuRunoutComp**

**visuRunoutComp**(*rasterTransfo*, *resAnalysisDF*, *cfgSetup*, *pathDict*)

Plot and save the Peak Fields distribution (max mean per cross section) after coordinate transformation

**Parameters**

- **rasterTransfo** (*dict*) – domain transformation information
- **resAnalysis** (*dict*) –  
**results from Aimec analysis (for ppr, pft and pfv):**  
 PPRCrossMax: numpy array with max peak field along path for each file to analyse  
 PPRCrossMean: numpy array with mean peak field along path for each file to analyse
- **cfgSetup** (*configparser*) – configparser with ana3AIMEC settings defined in ana3AIMECCfg.ini ‘runoutResType’ and ‘thresholdValue’

- **pathDict** (*dict*) – dictionary with path to data to analyze

### out3Plot.outAIMEC.visuRunoutStat

**visuRunoutStat**(*rasterTransfo*, *inputsDF*, *resAnalysisDF*, *newRasters*, *cfgSetup*, *pathDict*)

Panel1 reference peak field with runout points of all sims and distribution of runout SXY Panel 2 crossMax values of peak field along thalweg for all sims Panel 3 mean, median and envelope of cross max values for all sims

#### Parameters

- **rasterTransfo** (*dict*) – domain transformation information
- **inputsDF** (*dataFrame*) – aimec inputs DF
- **resAnalysis** (*dataFrame*) –  
**results from Aimec analysis:**  
numpy array with max peak field (of the ‘runoutResType’) along path for each file to analyse runout for ‘runoutResType’
- **newRasters** (*dict*) – dictionary with new (s, l) raster for the ‘runoutResType’
- **cfgSetup** (*configparser*) – configparser with ana3AIMEC settings defined in ana3AIMECCfg.ini ‘percentile’, ‘runoutResType’ and ‘thresholdValue’
- **pathDict** (*dict*) – dictionary with path to data to analyze

### out3Plot.outAIMEC.visuSimple

**visuSimple**(*cfgSetup*, *rasterTransfo*, *resAnalysisDF*, *newRasters*, *pathDict*)

Plot and save the Peak Pressure Peak Flow thickness and Peak speed fields after coord transfo

#### Parameters

- **rasterTransfo** (*dict*) – domain transformation information
- **resAnalysis** (*dict*) –  
**results from Aimec analysis:**  
numpy array with the ‘runout’ for each simulation and the ‘thresholdValue’
- **newRasters** (*dict*) – dictionary with new (s, l) raster for ppr, pft and pft
- **pathDict** (*dict*) – dictionary with path to data to analyze

### out3Plot.outAIMEC.visuTransfo

**visuTransfo**(*rasterTransfo*, *inputData*, *cfgSetup*, *pathDict*)

Plot and save the domain transformation figure The left subplot shows the reference result raster with the outline of the new domain. The second one shows this same data in the (s,l) coordinate system define by the outline in the first plot.

#### Parameters

- **rasterTransfo** (*dict*) – domain transformation information
- **inputData** (*dict*) –

**inputData dictionary:**

slRaster: numpy array with (s,l) raster xyRaster: numpy array with (x,y) raster headerXY: header corresponding to xyRaster

- **cfgSetup** (*configparser*) – configparser with ana3AIMEC settings defined in ana3AIMECCfg.ini ‘runoutResType’
- **pathDict** (*dict*) – dictionary with path to data to analyze

**out3Plot.outAna1Plots****Functions**

<i>addContour2Plot</i>	Make a contour plot of flow thickness for analytical solution and simulation result
<i>addErrorTime</i>	plot error between a given com1DFA sol and the analytic sol function of time on ax1 and ax2
<i>first_nonzero</i>	Get index of first non zero value
<i>getLabel</i>	Get error plot title (relativ error or not?)
<i>getPlotLimits</i>	Get x and y axis limits for the profile and contour plots
<i>getTitleError</i>	Get error plot title (relativ error or not?)
<i>last_nonzero</i>	Get index of last non zero value
<i>makeContourSimiPlot</i>	
<i>plotComparisonDam</i>	Generate plots that compare the simulation results to the analytical solution
<i>plotDamAnaResults</i>	Create plots of the analytical solution for the given settings, including an animation
<i>plotDamBreakSummary</i>	Plot summary figure of the dambreak test
<i>plotErrorConvergence</i>	plot error between all com1DFA sol and analytic sol function of whatever you want
<i>plotErrorTime</i>	plot and save error between a given com1DFA sol and the analytic sol function of time
<i>plotPresentation</i>	plot error between all com1DFA sol in simDF and analytic sol function of whatever you want
<i>plotSimiSolSummary</i>	Plot summary figure of the similarity solution test
<i>plotTimeCPULog</i>	plot computation time function of nParts function of whatever (ini parameter given in the simDF) you want
<i>saveSimiSolProfile</i>	Generate plots of the comparison of DFA solution and simiSol

**out3Plot.outAna1Plots.addContour2Plot**

**addContour2Plot** (*ax1, fieldFT, simiDict, fieldHeader, limits, nLevels=9*)

Make a contour plot of flow thickness for analytical solution and simulation result

**out3Plot.outAna1Plots.addErrorTime**

**addErrorTime**(*ax1, time, hErrorL2Array, hErrorLMaxArray, vhErrorL2Array, vhErrorLMaxArray, relativ, t*)  
plot error between a given com1DFA sol and the analytic sol function of time on ax1 and ax2

**Parameters**

- **ax1** (*matplotlib axis*) – axis where the erro should be plotted
- **time** (*1D numpy array*) – time array
- **hErrorL2Array** (*1D numpy array*) – flow thickness L2 error array
- **hErrorLMaxArray** (*1D numpy array*) – flow thickness LMax error array
- **vhErrorL2Array** (*1D numpy array*) – flow momentum L2 error array
- **vhErrorLMaxArray** (*1D numpy array*) – flow momentum LMax error array
- **relativ** (*str*)
- **t** (*float*) – time for vertical line

**out3Plot.outAna1Plots.first\_nonzero**

**first\_nonzero**(*arr, axis, invalid\_val=-1*)  
Get index of first non zero value

**Parameters**

- **arr** (*numpy array*) – data array
- **axis** (*int*) – axis along which you want to get the index

**Return type**

index of first non zero in axis direction

**out3Plot.outAna1Plots.getLabel**

**getLabel**(*start, end, dir="", vert=True*)  
Get error plot title (relativ error or not?)

**out3Plot.outAna1Plots.getPlotLimits**

**getPlotLimits**(*cfgSimi, fieldsList, fieldHeader*)  
Get x and y axis limits for the profile and contour plots

**Parameters**

- **cfgSimi** (*configparser*) – simiSol cfg
- **fieldsList** (*list*) – list of fields Dictionaries
- **fieldHeader** (*dict*) – field header dictionary



**out3Plot.outAna1Plots.getTitleError****getTitleError**(*relativ*, *ending=""*)

Get error plot title (relativ error or not?)

**out3Plot.outAna1Plots.last\_nonzero****last\_nonzero**(*arr*, *axis*, *invalid\_val=-1*)

Get index of last non zero value

**Parameters**

- **arr** (*numpy array*) – data array
- **axis** (*int*) – axis along which you want to get the index

**Return type**

index of last non zero in axis direction

**out3Plot.outAna1Plots.makeContourSimiPlot****makeContourSimiPlot**(*avalancheDir*, *simHash*, *fieldFT*, *limits*, *simiDict*, *fieldHeader*, *tSave*, *outDirTest*)**out3Plot.outAna1Plots.plotComparisonDam****plotComparisonDam**(*cfg*, *simHash*, *fields0*, *fieldsT*, *header*, *solDam*, *tSave*, *limits*, *outDirTest*)

Generate plots that compare the simulation results to the analytical solution

**Parameters**

- **cfgC** (*configParser object*) – configuration setting for avalanche simulation including DAMBREAK section
- **simHash** (*str*) – simulation hash
- **fields0** (*dict*) – initial time step field dictionary
- **fieldsT** (*dict*) – tSave field dictionary
- **header** (*dict*) – fields header dictionary
- **solDam** (*dict*) –

**analytic solution dictionary:****tAna: 1D numpy array**

time array

**h0: float**

release thickness

**hAna: 2D numpy array**

Flow thickness (rows for x and columns for time)

**uAna: 2D numpy array**

flow velocity (rows for x and columns for time)

**xAna: 2D numpy array**

extent of domain in the horizontal plane coordinate system (rows for x and columns for time)

**xMidAna: 1D numpy array**

middle of the material in x dir in the horizontal plane coordinate system (used to compute the error)

- **tSave** (*float*) – time step of analysis
- **limits** (*dict*) – y extend for profile plots
- **outDirTest** (*pathlib path*) – path to output directory

**out3Plot.outAna1Plots.plotDamAnaResults**

**plotDamAnaResults**(*t, x, xMiddle, h, u, tSave, cfg, outDirTest*)

Create plots of the analytical solution for the given settings, including an animation

**out3Plot.outAna1Plots.plotDamBreakSummary**

**plotDamBreakSummary**(*avalancheDir, timeList, fieldsList, fieldHeader, solDam, hErrorL2Array, hErrorLMaxArray, vErrorL2Array, vErrorLMaxArray, outDirTest, simDFrow, simHash, cfg*)

Plot summary figure of the dambreak test

**Parameters**

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **timeList** (*list*) – list of time steps
- **fieldsList** (*list*) – list of fields dictionaries
- **timeList** (*list*) – list of time steps
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **solDam** (*dict*) – analytic solution dictionary
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **hErrorL2Array** (*numpy array*) – L2 error on flow thickness for saved time steps
- **hErrorLMaxArray** (*numpy array*) – LMax error on flow thickness for saved time steps
- **vErrorL2Array** (*numpy array*) – L2 error on flow velocity for saved time steps
- **vErrorLMaxArray** (*numpy array*) – LMax error on flow velocity for saved time steps
- **outDirTest** (*pathlib path*) – path output directory (where to save the figures)
- **simDFrow** (*dataFrame*) – data frame row corresponding to simHash
- **simHash** (*str*) – com1DFA simulation id
- **cfg** (*configParser object*) – configuration setting for avalanche simulation including DAMBREAK section

### out3Plot.outAna1Plots.plotErrorConvergence

**plotErrorConvergence**(*simDF*, *outDirTest*, *cfgSimi*, *xField*, *yField*, *coloredBy*, *sizedBy*, *logScale=False*, *fit=False*)

plot error between all com1DFA sol and analytic sol function of whatever you want

The coloredBy, sizedBy can not be corresponding to non numeric parameters.

#### Parameters

- **simDF** (*dataFrame*) – the simulation data with the postprocessing results
- **outDirTest** (*str or pathlib*) – output directory
- **cfgSimi** (*configparser*) – the cfg
- **xField** (*str*) – column of the simDF to use for the x axis
- **yField** (*str*) – column of the simDF to use for the y axis
- **coloredBy** (*str*) – column of the simDF to use for the colors
- **sizedBy** (*str*) – column of the simDF to use for the marker size
- **logScale** (*boolean*) – If you want a loglog scale
- **fit** (*boolean*) – if True add power law regression

### out3Plot.outAna1Plots.plotErrorTime

**plotErrorTime**(*time*, *hErrorL2Array*, *hErrorLMaxArray*, *vhErrorL2Array*, *vhErrorLMaxArray*, *relativ*, *t*, *outputName*, *outDirTest*)

plot and save error between a given com1DFA sol and the analytic sol function of time

#### Parameters

- **time** (*1D numpy array*) – time array
- **hErrorL2Array** (*1D numpy array*) – flow thickness L2 error array
- **hErrorLMaxArray** (*1D numpy array*) – flow thickness LMax error array
- **vhErrorL2Array** (*1D numpy array*) – flow momentum L2 error array
- **vhErrorLMaxArray** (*1D numpy array*) – flow momentum LMax error array
- **relativ** (*str*)
- **t** (*float*) – time for vertical line
- **outputName** (*str*) – figure name
- **outDirTest** (*str or pathlib*) – output directory

### out3Plot.outAna1Plots.plotPresentation

**plotPresentation**(*simDF*, *outDirTest*, *cfgSimi*, *xField*, *yField*, *coloredBy*, *sizedBy*, *logScale=False*, *fit=False*)

plot error between all com1DFA sol in *simDF* and analytic sol function of whatever you want

The *coloredBy*, *sizedBy* can not be corresponding to non numeric parameters.

#### Parameters

- **simDF** (*dataFrame*) – the simulation data with the postprocessing results
- **outDirTest** (*str* or *pathlib*) – output directory
- **cfgSimi** (*configparser*) – the cfg
- **xField** (*str*) – column of the *simDF* to use for the x axis
- **yField** (*str*) – column of the *simDF* to use for the y axis
- **coloredBy** (*str*) – column of the *simDF* to use for the colors
- **sizedBy** (*str*) – column of the *simDF* to use for the marker size
- **logScale** (*boolean*) – If you want a loglog scale
- **fit** (*boolean*) – if True add power law regression

### out3Plot.outAna1Plots.plotSimiSolSummary

**plotSimiSolSummary**(*avalancheDir*, *timeList*, *fieldsList*, *fieldHeader*, *simiDict*, *hErrorL2Array*, *hErrorLMaxArray*, *vhErrorL2Array*, *vhErrorLMaxArray*, *outDirTest*, *simDFrow*, *simHash*, *cfgSimi*)

Plot summary figure of the similarity solution test

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **timeList** (*list*) – list of time steps
- **fieldsList** (*list*) – list of fields dictionaries
- **timeList** (*list*) – list of time steps
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **simiDict** (*dict*) – analytic solution dictionary
- **fieldHeader** (*dict*) – header dictionary with info about the extend and cell size
- **hErrorL2Array** (*numpy array*) – L2 error on flow thickness for saved time steps
- **hErrorLMaxArray** (*numpy array*) – LMax error on flow thickness for saved time steps
- **vErrorL2Array** (*numpy array*) – L2 error on flow velocity for saved time steps
- **vErrorLMaxArray** (*numpy array*) – LMax error on flow velocity for saved time steps
- **outDirTest** (*pathlib path*) – path output directory (where to save the figures)
- **simDFrow** (*dataFrame*) – data frame row corresponding to *simHash*
- **simHash** (*str*) – com1DFA simulation id
- **cfgSimi** (*configParser object*) – SIMISOL configuration

### out3Plot.outAna1Plots.plotTimeCPULog

**plotTimeCPULog**(*simDF*, *outDirTest*, *cfgSimi*, *xField*, *coloredBy*, *sizedBy*, *logScale=False*)

plot computation time function of nParts function of whatever (ini parameter given in the simDF) you want

#### Parameters

- **simDF** (*dataFrame*) – the simulation data with the postprocessing results
- **outDirTest** (*str or pathlib*) – output directory
- **cfgSimi** (*configparser*) – the cfg
- **xField** (*str*) – column of the simDF to use for the x axis
- **coloredBy** (*str*) – column of the simDF to use for the colors
- **sizedBy** (*str*) – column of the simDF to use for the marker size
- **logScale** (*boolean*) – If you want a loglog scale

### out3Plot.outAna1Plots.saveSimiSolProfile

**saveSimiSolProfile**(*cfgMain*, *cfgSimi*, *fields*, *limits*, *simiDict*, *tSave*, *header*, *outDirTest*, *simHash*)

Generate plots of the comparison of DFA solution and simiSol

#### Parameters

- **cfgMain** (*configparser*) – main cfg
- **cfgSimi** (*configparser*) – simiSol cfg
- **fieldsList** (*list*) – list of fields Dictionaries
- **solSimi** (*dict*) – dictionary with simiarty solution
- **tSave** (*float*) – time corresponding to fields
- **header** (*dict*) – field header dictionary
- **outDirTest** (*str or pathlib*) – output directory
- **simHash** (*str*) – hash of the current simulation

### out3Plot.outCom1DFA

## Functions

<code>addDem2Plot</code>	Add dem to the background of a plot
<code>addParticles2Plot</code>	Update axes with particles
<code>addResult2Plot</code>	Add raster data to a plot
<code>createContourPlot</code>	create a contour line plot of all simulations of current run
<code>fetchContCoors</code>	fetch coordinates of contour line
<code>plotAllPartAcc</code>	Plot time series of tracked particles :Parameters: * <b>outDirData</b> ( <i>str</i> ) -- path to output directory * <b>particlesList</b> ( <i>list</i> ) -- list with dictionary of all particles time steps particles * <b>cfg</b> ( <i>dict</i> ) -- configuration read from ini file * <b>Tsave</b> ( <i>list</i> ) -- list of saving time step info * <b>cuSimName</b> ( <i>str</i> ) -- name of current sim
<code>plotParticles</code>	Plot particles on dem
<code>plotReleaseScenarioView</code>	plot release polygon, area with thickness on dem hill-shade saved to <code>avaDir/Outputs/com1DFA/reports</code>
<code>plotTrackParticle</code>	Plot time series of tracked particles
<code>plotTrackParticleAcceleration</code>	Plot time series of tracked particles :Parameters: * <b>outDirData</b> ( <i>str</i> ) -- path to output directory * <b>trackedPartProp</b> ( <i>dict</i> ) -- dictionary with time series of the wanted properties for tracked particles * <b>cfg</b> ( <i>dict</i> ) -- configuration read from ini file * <b>cuSimName</b> ( <i>str</i> ) -- name of simulation
<code>updateTrackPart</code>	Update axes with particles (tracked particles are highlighted in red)

**out3Plot.outCom1DFA.addDem2Plot**

**addDem2Plot**(*ax*, *dem*, *what*='slope', *extent*="", *origHeader*=False)

Add dem to the background of a plot

**Parameters**

- **ax** (*matplotlib ax object*)
- **dem** (*dict*) – dem dictionary with normal information
- **what** (*str*) – what information about the dem will be plotted? slope: use the dem slope (computed from the normals) to color the plot z : use the elevation to color the plot
- **origHeader** (*bool*) – if True use originalHeader and not header

**out3Plot.outCom1DFA.addParticles2Plot**

**addParticles2Plot**(*particles*, *ax*, *dem*, *whatS*='m', *whatC*='h', *colBarResType*="")

Update axes with particles

**Parameters**

- **particles** (*dict*) – particles dictionary
- **ax** (*matplotlib ax object*)
- **dem** (*dict*) – dem dictionary with normal information

- **whatS** (*str*) – which particle property should be used for the markersize
- **whatC** (*str*) – which particle property should be used for the marker color

### out3Plot.outCom1DFA.addResult2Plot

**addResult2Plot**(*ax, header, rasterData, resType, colorbar=True, contour=False*)

Add raster data to a plot

#### Parameters

- **ax** (*matplotlib ax object*)
- **header** (*dict*) – raster header dictionary
- **rasterData** (*2D numpy array*) – data to plot
- **resType** (*str*) – what kind of result is it? ppr, pft...
- **colorbar** (*bool*) – If true add the colorbar

### out3Plot.outCom1DFA.createContourPlot

**createContourPlot**(*reportDictList, avalancheDir, simDF*)

create a contour line plot of all simulations of current run

#### Parameters

- **reportDictList** (*list*) – list of com1DFA dictionary with info on contour dicts for each sim
- **avalancheDir** (*str or pathlib path*) – path to avalanche directory
- **simDF** (*pandas DataFrame*) – dataframe with one row per simulation performed and its parameter settings

#### Returns

**reportDictList** – updated reportDictList - deleted contours dict

#### Return type

list

### out3Plot.outCom1DFA.fetchContCoors

**fetchContCoors**(*demHeader, flowF, cfgVisu, simName*)

fetch coordinates of contour line

#### Parameters

- **demHeader** (*dict*) – dictionary of dem nrows, ncols, cellsize
- **flowF** (*np.ndarray*) – field data used to compute contour line
- **cfgVisu** (*configparser object*) – configuration settings for visualisation, here used: contour-ResType, thresholdValue
- **simName** (*str*) – simName
- **Returns**
- \_\_\_\_\_

- **contDictXY** (*dict*) – dictionary with simName and subDict with x, y coordinates of contour line contourResType and thresholdValue

### out3Plot.outCom1DFA.plotAllPartAcc

**plotAllPartAcc**(*outDirData, particlesList, cfg, Tsave, cuSimName*)

Plot time series of tracked particles :Parameters: \* **outDirData** (*str*) – path to output directory

- **particlesList** (*list*) – list with dictionary of all particles time steps particles
- **cfg** (*dict*) – configuration read from ini file
- **Tsave** (*list*) – list of saving time step info
- **cuSimName** (*str*) – name of current sim

### out3Plot.outCom1DFA.plotParticles

**plotParticles**(*particlesList, cfg, dem*)

Plot particles on dem

#### Parameters

- **particlesList** (*list*) – list or particles dictionaries
- **cfg** (*dict*) – configuration read from ini file
- **dem** (*dict*) – dem dictionary with normal information

### out3Plot.outCom1DFA.plotReleaseScenarioView

**plotReleaseScenarioView**(*avaDir, releaseLine, dem, titleFig, cuSimName*)

plot release polygon, area with thickness on dem hillshade saved to *avaDir/Outputs/com1DFA/reports*

#### Parameters

- **avaDir** (*str*) – path to ava directory
- **dem** (*dict*) – dict with dem header and data
- **releaseLine** (*dict*) – dict with raster of release line and x,y coors
- **titleFig** (*str*) – title of figure
- **cuSimName** (*str*) – name of simulation

### out3Plot.outCom1DFA.plotTrackParticle

**plotTrackParticle**(*outDirData, particlesList, trackedPartProp, cfg, dem, cuSimName*)

Plot time series of tracked particles

#### Parameters

- **outDirData** (*str*) – path to output directory
- **particlesList** (*list*) – list or particles dictionaries



- **trackedPartProp** (*dict*) – dictionary with time series of the wanted properties for tracked particles
- **cfg** (*dict*) – configuration read from ini file
- **dem** (*dict*) – dem dictionary with normal information
- **cuSimName** (*str*) – name of current simulation

### out3Plot.outCom1DFA.plotTrackParticleAcceleration

**plotTrackParticleAcceleration**(*outDirData*, *trackedPartProp*, *cfg*, *cuSimName*)

Plot time series of tracked particles :Parameters: \* **outDirData** (*str*) – path to output directory

- **trackedPartProp** (*dict*) – dictionary with time series of the wanted properties for tracked particles
- **cfg** (*dict*) – configuration read from ini file
- **cuSimName** (*str*) – name of simulation

### out3Plot.outCom1DFA.updateTrackPart

**updateTrackPart**(*particles*, *ax*, *dem*)

Update axes with particles (tracked particles are highlighted in red)

### out3Plot.outCom3Plots

#### Functions

<i>generateCom1DFAPathPlot</i>	Make energy test analysis and plot results
<i>hybridPathPlot</i>	Update path plot with result of curent iteration
<i>hybridProfilePlot</i>	Update profile plot with result of curent iteration

### out3Plot.outCom3Plots.generateCom1DFAPathPlot

**generateCom1DFAPathPlot**(*avalancheDir*, *cfgPath*, *avaProfileMass*, *dem*, *parabolicFit*, *splitPoint*, *simName*)

Make energy test analysis and plot results

#### Parameters

- **avalancheDir** (*pathlib*) – avalanche directory pathlib path
- **cfgPath** (*configParser*) – path finding config
- **avaProfileMass** (*dict*) – particle mass averaged properties
- **dem** (*dict*) – com1DFA simulation dictionary
- **fieldsList** (*list*) – field dictionary list
- **simName** (*str*) – simulation name

**out3Plot.outCom3Plots.hybridPathPlot****hybridPathPlot** (*avalancheDir, dem, resultsHybrid, fields, particles, muArray*)

Update path plot with result of curent iteration

**out3Plot.outCom3Plots.hybridProfilePlot****hybridProfilePlot** (*avalancheDir, resultsHybrid*)

Update profile plot with result of curent iteration

**out3Plot.outContours**

Plotting and saving contour line plots

**Functions**

<i>createRasterContourDict</i>	create a dict with contour line coordinates for an ascii file
<i>plotContoursFromDict</i>	plot contour lines of two contourLine dicts only plot lines that are available within ref and save to file
<i>readShpLines</i>	derive a dict with x, y coordinates for each contourline according to given contour lines dependent on contLine name format here used resType_unit_level (for example: pft_m_1.0)

**out3Plot.outContours.createRasterContourDict****createRasterContourDict** (*inFile, levels*)

create a dict with contour line coordinates for an ascii file

**Parameters**

- **inFile** (*pathlib path*) – path to ascii file
- **levels** (*list*) – list of levels for contour lines

**Returns****contourDict** –**dictionary with keys: name of contourlines (format: resType\_unit\_level)**

x: x coordinates y: y coordinates contourlevel: float of contour line value

**Return type**

dict

**out3Plot.outContours.plotContoursFromDict**

**plotContoursFromDict**(*contourDictRef*, *contourDictSim*, *pathDict*, *levels*, *multiplePlots=True*)

plot contour lines of two contourLine dicts only plot lines that are available within ref and save to file

**Parameters**

- **contourDict** (*dict*) – dictionary with contourline coordinates for reference
- **contourDict** (*dict*) – dictionary with contourline coordinates for simulation
- **pathDict** (*dict*) – dictionary with info on avaDir, outDir, title, parameter, unit
- **levels** (*list*) – list of contour levels
- **multiplePlots** (*bool*) – if True one plot for each contour level

**out3Plot.outContours.readShpLines**

**readShpLines**(*shpData*, *layerName='name'*)

derive a dict with x, y coordinates for each contourline according to given contour lines dependent on contLine name format here used resType\_unit\_level (for example: pft\_m\_1.0)

**Parameters**

- **shpData** (*dict*) – shp file data extracted with SHP2Array
- **layerName** (*str*) – name that is used to name contourline and to identify lines in shpData

**Returns**

**contourDict** – contour line dictionary with name and in dict x, y coordinates

**Return type**

dict

**out3Plot.outDebugPlots****Functions**

<i>analysisPlots</i>	create analysis plots during simulation run
<i>plotAreaDebug</i>	
<i>plotBondsSnowSlideFinal</i>	With snowSlide option on, plot the bonds between particles as well as the particles properties
<i>plotBufferRelease</i>	plot release lines with added bufferLine
<i>plotContours</i>	
<i>plotFindAngle</i>	helper plot for the getSplitPoint, findAngleProfile and prepareAngleProfile functions Plots the slope angle and elevation function of s
<i>plotPartAfterRemove</i>	
<i>plotPartIni</i>	
<i>plotPathExtBot</i>	Plot the extended path towards the bottom of the avalanche
<i>plotPathExtTop</i>	Plot the extended path towards the top of the release
<i>plotPosition</i>	
<i>plotProfile</i>	plot profile, add beta info
<i>plotRemovePart</i>	
<i>plotSlopeAngle</i>	plot slope angle along a profile, add beta info
<i>plotVolumeRelease</i>	create a plot of the release line raster, the relThField for release thickness, releaseLineField - combination of relThField and release line raster mask

**out3Plot.outDebugPlots.analysisPlots**

**analysisPlots**(*particlesList*, *fieldsList*, *cfg*, *demOri*, *dem*, *outDir*)

create analysis plots during simulation run

**out3Plot.outDebugPlots.plotAreaDebug**

**plotAreaDebug**(*header, avapath, Raster*)

**out3Plot.outDebugPlots.plotBondsSnowSlideFinal**

**plotBondsSnowSlideFinal**(*cfg, particles, dem, inputSimLines=""*)

With snowSlide option on, plot the bonds between particles as well as the particles properties

**out3Plot.outDebugPlots.plotBufferRelease**

**plotBufferRelease**(*inputSimLines, xBuffered, yBuffered*)

plot release lines with added bufferLine

**out3Plot.outDebugPlots.plotContours**

**plotContours**(*fig, ax, t, header, data, Cmap, unit, last=False*)

**out3Plot.outDebugPlots.plotFindAngle**

**plotFindAngle**(*avaProfile, angleProf, parabolicProfile, anglePara, s0, sEnd, splitPoint, indSplitPoint*)

helper plot for the getSplitPoint, findAngleProfile and prepareAngleProfile functions Plots the slope angle and elevation function of s

**out3Plot.outDebugPlots.plotPartAfterRemove**

**plotPartAfterRemove**(*points, xCoord0, yCoord0, mask*)

**out3Plot.outDebugPlots.plotPartIni**

**plotPartIni**(*particles, dem*)

**out3Plot.outDebugPlots.plotPathExtBot**

**plotPathExtBot**(*profile, xInterest, yInterest, zInterest, xLast, yLast*)

Plot the extended path towards the bottom of the avalanche

**out3Plot.outDebugPlots.plotPathExtTop****plotPathExtTop**(*profile, particlesIni, xFirst, yFirst, zFirst, dz1*)

Plot the extended path towards the top of the release

**out3Plot.outDebugPlots.plotPosition****plotPosition**(*fig, ax, particles, dem, data, Cmap, unit, plotPart=False, last=False*)**out3Plot.outDebugPlots.plotProfile****plotProfile**(*s, z, idsBetaPoint*)

plot profile, add beta info

**out3Plot.outDebugPlots.plotRemovePart****plotRemovePart**(*xCoord0, yCoord0, header, X, Y, Mask, mask*)**out3Plot.outDebugPlots.plotSlopeAngle****plotSlopeAngle**(*s, angle, idsBetaPoint*)

plot slope angle along a profile, add beta info

**out3Plot.outDebugPlots.plotVolumeRelease****plotVolumeRelease**(*releaseLine, relThField, releaseLineField*)

create a plot of the release line raster, the relThField for release thickness, releaseLineField - combination of relThField and release line raster mask

**out3Plot.outDistanceTimeAnalysis**

functions to plot range-time diagrams

## Functions

<i>addRangeTimePlotToAxes</i>	add plot range-time diagram with avalanche front and colorcoded average values of result parameter
<i>addTitleBox</i>	add velocity legend title and boundary box and diagonal lines if maxVel=True add max velocity line in red
<i>addVelocityValues</i>	add velocity values as labels
<i>animationPlot</i>	3 panel plot: result in x,y, result in s, l, tt diagram
<i>getMaxVelocityPoint</i>	get point of max approach velocity in axes coordinates
<i>getVelocityPoints</i>	get points for legend creation (box and sloping lines)
<i>plotMaskForMTI</i>	plot masks used to compute averages for range-time diagram with radar field of view
<i>plotRangeRaster</i>	create plot of distance from start of runout area point extracted avalanche front and result field
<i>plotRangeTime</i>	plot range-time diagram with avalanche front and color-coded average values of result parameter
<i>radarFieldOfViewPlot</i>	Create radar field of view plot
<i>rangeTimeVelocityLegend</i>	set legend in range time diagram for velocity in terms of steepness of front position

### out3Plot.outDistanceTimeAnalysis.addRangeTimePlotToAxes

**addRangeTimePlotToAxes** (*mtiInfo*, *cfgRangeTime*, *ax*)

add plot range-time diagram with avalanche front and colorcoded average values of result parameter

#### Parameters

- **mtiInfo** (*dict*) – dictionary with average values for range to reference point (*mti*), *timeStep* list, list with distance to reference point of avalanche front (*rangeList*)
- **cfgRangeTime** (*configparser object*) – configuration settings for range time diagram - here used *avalancheDir*, *rangeTimeResType*, *simHash*, and from plots *width*, *height*, *lw*, ..
- **ax** (*matplotlib axes object*) – axes where plot shall be added to

### out3Plot.outDistanceTimeAnalysis.addTitleBox

**addTitleBox** (*ax*, *width*, *height*, *lw*, *point*, *textsize*, *maxVel=True*)

add velocity legend title and boundary box and diagonal lines if maxVel=True add max velocity line in red

#### Parameters

- **ax** (*matplotlib axes object*) – axes object for legend
- **width** (*float*) – fractional percentage of legend width
- **height** (*float*) – fractional percentage of legend height
- **lw** (*float*) – linewidth
- **point** (*list*) – list of points coordinates for velocity legend
- **textsize** (*float*) – size of legend entries
- **maxVel** (*bool*) – if True add max velocity line in red

### out3Plot.outDistanceTimeAnalysis.addVelocityValues

**addVelocityValues**(*ax*, *dataPoint*, *point*, *first=True*, *inititalNrOfTextElements=numpy.nan*)

add velocity values as labels

#### Parameters

- **ax** (*matplotlib axis object*)
- **dataPoint** (*list*) – list of points coordinates for velocity label locations
- **point** (*list*) – list of points coordinates for velocity lines
- **first** (*bool*) – if True text elements are created - initially required, if False then only changed
- **inititalNrOfTextElements**

### out3Plot.outDistanceTimeAnalysis.animationPlot

**animationPlot**(*demData*, *data*, *cellSize*, *resType*, *cfgRangeTime*, *mtiInfo*, *timeStep*)

3 panel plot: result in x,y, result in s, l, tt diagram

#### Parameters

- **demData** (*dict*) – info on dem with header and rasterData
- **data** (*numpy array*) – result type data
- **cellSize** (*float*) – cell size of result type and dem
- **resType** (*str*) – name of result type, e.g. FV, FT
- **cfgRangeTime** (*configparser object*) – configuration of range time diagram settings
- **mtiInfo** (*dict*) – info dictionary for color plot of tt diagram, also info on domain transformation into s,l
- **timeStep** (*float*) – actual time step of sim result

### out3Plot.outDistanceTimeAnalysis.getMaxVelocityPoint

**getMaxVelocityPoint**(*width*, *height*, *maxVel*, *diagVel*)

get point of max approach velocity in axes coordinates

#### Parameters

- **width** (*float*) – fractional percentage of legend width
- **height** (*float*) – fractional percentage of legend height
- **maxVel** (*float*) – maximum approach velocity value
- **diagVel** (*float*) – x/y ratio of axes

#### Returns

**point** – x, y axes coordinates of maximum velocity

#### Return type

tuple



### out3Plot.outDistanceTimeAnalysis.getVelocityPoints

**getVelocityPoints**(xMax, yMax, width, height, xinterval, yinterval)

get points for legend creation (box and sloping lines)

if xMax, yMax, xinterval, yinterval are all 1 -> points refer to axes coordinates, that range from 0 to 1

if given true values, e.g. in seconds and meters, points are in data coordinates and can be used for velocity calculus etc.

**Note:** used to be an internal function to rangeTimeVelocityLegend,  
so use with care.

#### Parameters

- **xMax** (*float*) – max x extent of data (timeSteps)
- **yMax** (*float*) – max y extent of data (rangeGates)
- **width** (*float*) – fractional percentage of legend width
- **height** (*float*) – fractional percentage of legend height
- **xinterval** (*float*) – interval on x
- **yinterval** (*float*) – interval on y

#### Returns

**points** – list of point coordinates

#### Return type

list

### out3Plot.outDistanceTimeAnalysis.plotMaskForMTI

**plotMaskForMTI**(cfgRangeTime, bmaskRange, bmaskAvaRadar, bmaskAvaRadarRangeslice, mtiInfo)

plot masks used to compute averages for range-time diagram with radar field of view

#### Parameters

- **cfgRangeTime** (*configparser settings*) – configuration settings for range time diagram, avalancheDir
- **bmaskRange** (*numpy mask*) – mask for radar range slice for range gate and rgWidth
- **bmaskAvaRadar** (*numpy mask*) – mask for result parameter below threshold and out of radar field of view
- **bmaskAvaRadarRangeslice** (*numpy mask*) – full mask combination

### out3Plot.outDistanceTimeAnalysis.plotRangeRaster

**plotRangeRaster**(*slRaster*, *rasterTransfo*, *cfgRangeTime*, *rangeRaster*, *cLower*)

create plot of distance from start of runout area point extracted avalanche front and result field

#### Parameters

- **slRaster** (*numpy array*) – transformed result field
- **rasterTransfo** (*dict*) – info on coordinate transformation
- **cfgRangeTime** (*configparser object*) – configuration settings of range time diagram
- **rangeRaster** (*np array*) – distance to runout area point in s coordinate
- **cLower** (*int*) – index of avalanche front

### out3Plot.outDistanceTimeAnalysis.plotRangeTime

**plotRangeTime**(*mtiInfo*, *cfgRangeTime*)

plot range-time diagram with avalanche front and colorcoded average values of result parameter

#### Parameters

- **mtiInfo** (*dict*) – dictionary with average values for range to reference point (mti), timeStep list, list with distance to reference point of avalanche front (rangeList)
- **cfgRangeTime** (*configparser object*) – configuration settings for range time diagram - here used avalancheDir, rangeTimeResType, simHash, and from plots width, height, lw, ..

### out3Plot.outDistanceTimeAnalysis.radarFieldOfViewPlot

**radarFieldOfViewPlot**(*radarFov*, *radarRange*, *cfgRangeTime*, *rangeGates*, *dem*)

Create radar field of view plot

#### Parameters

- **radarFov** (*numpy array*) – list with radar location and end point of field of view, x and y coors
- **radarRange** (*masked array*) – masked array of DEM with radar field of view - showing distance to radar
- **cfgRangeTime** (*configparser object*) –  
configuration settings section - here used avalancheDir, simHash, aperture angle [degree]
- **rangeGates** (*numpy array*) – range gates of radar field of view
- **dem** (*dict*) – dictionary with dem header and data

## out3Plot.outDistanceTimeAnalysis.rangeTimeVelocityLegend

**rangeTimeVelocityLegend**(*ax*, *maxVel*, *width*, *height*, *lw*, *textsize*)

set legend in range time diagram for velocity in terms of steepness of front position  
connects to figure callback to react to zoom/pan events

### Parameters

- **ax** (*matplotlib axes object*) – figure axes object
- **maxVel** (*float*) – maximum approach velocity value
- **width** (*float*, \*0.25) – fractional percentage of legend width
- **height** (*float*) – fractional percentage of legend height
- **lw** (*float*) – linewidth for legend line
- **textsize** (*float*) – textsize for legend entries

## out3Plot.outParticlesAnalysis

Tools to extract information on the avalanche simulations run in the Output files

### Functions

<i>addPeakFieldConstrained</i>	use out1Peak functions to add plot of a peak field
<i>addTrOrMe</i>	add a line plot of x: prop1, y: prop2 and label if label=True
<i>plotParticleMotionTracking</i>	Create plot showing particle properties over time and along avalanche thalweg in light blue envelope for all particles (filled between min and max values) in dark blue the values for tracked particles
<i>plotParticleThalwegAltitudeVelocity</i>	plot peak flow fields and velocity thalweg envelope
<i>plotThalwegTimeAltitudes</i>	Create plot showing the resType peak field with thalweg, thalweg vs altitude with max peak field values along thalweg derived from peak fields and the tt-diagram
<i>readMeasuredParticleData</i>	fetch data on measured particles from pickle in Inputs/measuredParticles only one pickle file allowed if no pData (name of particle file) is provided
<i>velocityEnvelope</i>	compute the velocity envelope from particle values
<i>velocityEnvelopeThalweg</i>	function to generate the velocity envelope along the thalweg from simulated flows

### out3Plot.outParticlesAnalysis.addPeakFieldConstrained

**addPeakFieldConstrained**(*avaDir, modName, simName, resType, demData, ax, alpha, oneColor=""*)

use out1Peak functions to add plot of a peak field

#### Parameters

- **avaDir** (*pathlib path*) – path to avalanche dir
- **modName** (*str*) – name of computational module used to produce result peak fields
- **simName** (*str*) – name of simulation
- **resType** (*str*) – result variable name to look for peak field
- **demData** (*dict*) – dict with info on dem used to run sim of peak field data
- **ax** (*matplotlib axes object*) – axes where plot shall be added to
- **alpha** (*float*) – value for transparency from 0-1
- **oneColor** (*empty str*) – optional to add a color for a single color for field

### out3Plot.outParticlesAnalysis.addTrOrMe

**addTrOrMe**(*ax, pDict, prop1, prop2, cmap, label=False, zorder=1, lineStyle='-'*)

add a line plot of x: prop1, y: prop2 and label if label=True

#### Parameters

- **ax** (*matplotlib axes*) – axes where plot should be added to
- **prop1** (*str*) – name of property in pDict used for x axis
- **prop2** (*str*) – name of property in pDict used for y axis
- **cmap** (*matplotlib colormap*) – cmap to be used for multiple particles in pDict
- **label** (*bool*) – if True add label to lines for legend
- **zorder** (*int*) – order of the plot object on the axes

### out3Plot.outParticlesAnalysis.plotParticleMotionTracking

**plotParticleMotionTracking**(*avalancheDir, simName, dictVelAltThalweg, trackedPartProp, dictVelEnvelope, demSim, modName, rasterTransfo, measuredData=""*)

Create plot showing particle properties over time and along avalanche thalweg in light blue envelope for all particles (filled between min and max values) in dark blue the values for tracked particles

panel 1: map view of flow variable peak field panel 2: particle trajectoryLengthXYZ vs time panel 3: particle velocityMagnitude vs time panel 4: particle acceleration vs time panel 5: particle trajectoryLengthXYZ vs thalweg Sxy panel 6: particle velocity vs thalweg Sxy panel 7: particle acceleration vs thalweg Sxy

#### Parameters

- **avalancheDir** (*pathlib path or str*) – path to avalanche directory
- **simName** (*str*) – name of simulation
- **dictVelAltThalweg** (*dict*) – dict with velocity and altitude envelope info for all particles
- **trackedPartProp** (*dict*) – dict with time series of tracked particle properties

- **dictVelEnvelope** (*dict*) – dict with velocity envelope info
- **demSim** (*dict*) – dict with sim dem info
- **modName** (*str*) – name of computational module that has been used to produce the sims
- **measuredData** (*dict*) –

**dict data: with time series of measured data (same format as trackedPartProp)**  
 labelName: str name of label for plots t: 1D array of time steps

### out3Plot.outParticlesAnalysis.plotParticleThalwegAltitudeVelocity

**plotParticleThalwegAltitudeVelocity**(*avalancheDir, simIndex, simDF, rasterTransfo, dictVelAltThalweg, resTypePlots, modName, demData*)

plot peak flow fields and velocity thalweg envelope

#### Parameters

- **avalancheDir** (*pathlib path or str*) – path to avalanche directory
- **simIndex** (*str*) – index of current simulation
- **rasterTransfo** (*dict*) – info on domain transformation from xy to sl thalweg
- **dictVelAltThalweg** (*dict*) – info on velocity and altitude of particles along thalweg s
- **resTypePlots** (*list*) – list of result types that shall be plotted, one plot for each resType
- **modName** (*str*) – name of com module used to perform sims
- **demData** (*dict*) – dict of dem from sim

### out3Plot.outParticlesAnalysis.plotThalwegTimeAltitudes

**plotThalwegTimeAltitudes**(*avalancheDir, simIndex, simDF, rasterTransfo, pfvCrossMax, modName, demSim, mtiInfo, cfgRangeTime, velocityThreshold, measuredData=""*)

Create plot showing the resType peak field with thalweg, thalweg vs altitude with max peak field values along thalweg derived from peak fields and the tt-diagram

#### Parameters

- **avalancheDir** (*str or pathlib path*) – path to avalanche directory
- **simIndex** (*str*) – index of current sim in simDF
- **simDF** (*dataFrame*) – dataframe with one row per simulation, and all model config parameters
- **rasterTransfo** (*dict*) – dict with info on transformation from cartesian to thalweg coordinate system
- **dictRaster** (*dict*) – dict with info on peak fields
- **modName** (*str*) – name of com module used to perform sims
- **demSim** (*dict*) – dict with info on dem used for sims
- **measuredData** (*dict*) –

**dict data: with time series of measured data (same format as trackedPartProp)**  
 labelName: str name of label for plots t: 1D array of time steps

### out3Plot.outParticlesAnalysis.readMeasuredParticleData

**readMeasuredParticleData**(*avalancheDir*, *demHeader*, *pData*="")

fetch data on measured particles from pickle in Inputs/measuredParticles only one pickle file allowed if no *pData* (name of particle file) is provided

#### Parameters

- **avalancheDir** (*pathlib path*) – path to avalanche directory
- **demHeader** (*dict*) – currently *xllcenter*, *yllcenter* required to set x, y coordinates to origin 0,0
- **pData** (*str*) – name of mesured particle file

#### Returns

**mParticles** – dict with info on measured particles properties (*velocityMag*, *x*, *y*, *z*, *uAcc*, *t*) all properties except *t* are of shape: *m*x*n* matrix, where *m* refers to the time steps and *n* to the individual particles *t* is a vector of the time step values corresponding to *m* *label* is a list of names of the labels for the measured particles corresponding to *n*

#### Return type

dict

### out3Plot.outParticlesAnalysis.velocityEnvelope

**velocityEnvelope**(*particlesTimeArrays*)

compute the velocity envelope from particle values

#### Parameters

**particlesTimeArrays** (*dict*) – dict with time series for particle properties

#### Returns

**dictVelEnvelope** – max, mean, min values of velocity over all particles for each time step velocity, acceleration, min and max of trajectoryLengthXYZ of particles for each time step

#### Return type

dict

### out3Plot.outParticlesAnalysis.velocityEnvelopeThalweg

**velocityEnvelopeThalweg**(*particlesTimeArrays*)

function to generate the velocity envelope along the thalweg from simulated flows

#### Parameters

**particlesTimeArrays** (*dict*) – dict with time series of particle properties

#### Returns

**dictVelAltThalweg** – dict with min, max, mean, ... values of particles elevation, velocity and acceleration for each thalweg coordinate

#### Return type

dict

## out3Plot.outQuickPlot

Functions to plot 2D avalanche simulation results as well as comparison plots between two datasets of identical shape.

### Functions

<i>generateOnePlot</i>	Create plot of ascii dataset
<i>generatePlot</i>	Create comparison plots of two ascii datasets
<i>plotAllContours</i>	Create a plot of resType level contour lines for all resType files found in avaDir/Outputs/modName/peakFiles or specDir optional file format needs to be of type _resType.asc
<i>plotContours</i>	plot contour lines of all transformed fields
<i>quickPlotBench</i>	Plot simulation result and compare to reference solution (two raster datasets of identical dimension) and save to Outputs/out3Plot within avalanche directoy.
<i>quickPlotOne</i>	Plots one raster dataset and a cross profile
<i>quickPlotSimple</i>	Plot two raster datasets of identical dimension and difference between two datasets

## out3Plot.outQuickPlot.generateOnePlot

**generateOnePlot**(*dataDict*, *outDir*, *cfg*, *plotDict*)

Create plot of ascii dataset

#### Parameters

- **dataDict** (*dict*) – dictionary with info of the dataset to be plotted
- **outDir** (*pathlib path*) – path to dictionary where plots shall be saved to
- **cfg** (*dict*) – main configuration settings
- **plotDict** (*dict*) – dictionary with information about plots, for example release area...

#### Returns

**plotDict** – updated plot dictionary with path to plot

#### Return type

dict

## out3Plot.outQuickPlot.generatePlot

**generatePlot**(*dataDict*, *avaName*, *outDir*, *cfg*, *plotDict*, *crossProfile=True*)

Create comparison plots of two ascii datasets

This function creates two plots, one plot with four panels with, first dataset, second dataset, the absolute difference of the two datasets and the absolute difference capped to a smaller range of differences (ppr: +- 100kPa, pft: +- 1m, pfv: +- 10ms-1). The difference plots also include an insert showing the histogram and the cumulative density function of the differences. The second plot shows a cross- and along profile cut of the two datasets. The folder and simulation name of the datasets has to be passed to the function.

#### Parameters

- **dataDict** (*dict*) –

dictionary with info on both datasets to be plotted:

**name1:** **str**

string with the name of the first data set

**name2:** **str**

string with the name of the second data set

**data1:** **2D numpy arrays**

raster of the first data sets

**data2:** **2D numpy arrays**

raster of the first second sets

**cellSize:** **float**

cell size

**suffix:** **str**

optional information about the data type compared ('ppr', 'pft', 'pfv', 'P', 'FV', 'FT', 'Vx'...)

- **avaName** (*str*) – name of avalanche
- **outDir** (*pathlib path*) – path to dictionary where plots shall be saved to
- **cfg** (*configParser*) – `cfg['FLAGS'].getboolean('showPlot')`
- **plotDict** (*dict*) – dictionary with information about plots, for example release area

#### Returns

**plotDict** – updated plot dictionary with info about e.g. min, mean, max of difference between datasets

#### Return type

dict

### out3Plot.outQuickPlot.plotAllContours

**plotAllContours**(*avaDir, modName, resType, level, specDir=""*)

Create a plot of `resType` level contour lines for all `resType` files found in `avaDir/Outputs/modName/peakFiles` or `specDir` optional file format needs to be of type `_resType.asc`

#### Parameters

- **avaDir** (*str or pathlib path*) – path to avalanche directory
- **modName** (*str*) – name of computational module
- **resType** (*str*) – result variable type
- **level** (*float*) – contour line level
- **specDir** (*str or pathlib path*) – path to peak files folder - optional



**out3Plot.outQuickPlot.plotContours****plotContours**(*contourDict*, *resType*, *thresholdValue*, *pathDict*, *addLegend=True*)

plot contour lines of all transformed fields

**Parameters**

- **contourDict** (*dict*) – dictionary with contourline coordinates
- **resType** (*str*) – result type
- **thresholdValue** (*float*) – value for contour level
- **pathDict** (*dict*) – dictionary with info on project name, ...
- **addLegend** (*bool*) – if True add legend to plot

**out3Plot.outQuickPlot.quickPlotBench****quickPlotBench**(*avaDir*, *simNameRef*, *simNameComp*, *refDir*, *compDir*, *cfg*, *suffix*)

Plot simulation result and compare to reference solution (two raster datasets of identical dimension) and save to Outputs/out3Plot within avalanche directoy.

figure 1, plot raster data for dataset1, dataset2 and their difference, their difference limited to specified range, including a histogram and the cumulative density function of the differences

figure 2, plot cross and longprofiles for both datasets (ny\_loc and nx\_loc define location of profiles)

plots are saved to Outputs/out3Plot

**Parameters**

- **avaDir** (*str or pathlib path*) – path to avalanche directory
- **simNameRef** (*str*) – name of reference simulation
- **simNameComp** (*str*) – name of comparison simulation
- **refDir** (*str or pathlib path*) – path to reference file
- **compDir** (*str or pathlib path*) – path to comparison file
- **cfg** (*dict*) – global configuration settings
- **suffix** (*str*) – result type

**Returns**

**plotList** – plot dictionaries (path to plots, min, mean and max difference between plotted datasets, max and mean value of reference dataset )

**Return type**

dict

### out3Plot.outQuickPlot.quickPlotOne

**quickPlotOne**(*avaDir*, *datafile*, *cfg*, *locVal*, *axis*, *resType=""*)

Plots one raster dataset and a cross profile

figure 1: plot raster data for dataset and profile -plot is saved to Outputs/out3Plot

#### Parameters

- **avaDir** (*str or pathlib Path*) – path to avalanche directory
- **datafile** (*str or pathlib path*) – path to data file
- **cfg** (*dict*) – configuration including flags for plotting
- **locVal** (*float*) – location of cross profile
- **resType** (*str*) – result parameter type e.g. ‘pft’ - optional

### out3Plot.outQuickPlot.quickPlotSimple

**quickPlotSimple**(*avaDir*, *inputDir*, *cfg*)

Plot two raster datasets of identical dimension and difference between two datasets

figure 1: plot raster data for dataset1, dataset2 and their difference figure 2: plot cross and longprofiles for both datasets (ny\_loc and nx\_loc define location of profiles) -plots are saved to Outputs/out3Plot

Be aware: files are being sorted after getting them from the directory! (Important for the differences)

#### Parameters

- **avaDir** (*str or pathlib path*) – path to avalanche directory
- **inputDir** (*str or pathlib path*) – path to directory of input data (only 2 raster files allowed)
- **cfg** (*configParser object*) – global configuration settings

### out3Plot.outTopo

Simple plotting for DEMs

### Functions

<i>plotDEM3D</i>	Plots the DEM from the avalancheDir in cfg alongside it
<i>plotGeneratedDEM</i>	Plot DEM with given information on the origin of the DEM
<i>plotReleasePoints</i>	

---

**out3Plot.outTopo.plotDEM3D****plotDEM3D**(*cfg*, *showPlot=False*)Plots the DEM from the avalancheDir in *cfg* alongside it**Parameters**

- **cfg** (*configparser object*) – the main configuration
- **showPlot** (*boolean*) – If true shows the matplotlib plot

**out3Plot.outTopo.plotGeneratedDEM****plotGeneratedDEM**(*z*, *nameExt*, *cfg*, *outDir*, *cfgMain*)

Plot DEM with given information on the origin of the DEM

**out3Plot.outTopo.plotReleasePoints****plotReleasePoints**(*xv*, *yv*, *xyPoints*, *demType*)**out3Plot.statsPlots**

plot statistics of simulations

**Functions**

<i>plotDistFromDF</i>	create a dist plot from dataframe for <i>name1</i> on x axis and <i>name2</i> on y axis, optionally colorcoded with scenario name and filtered with <i>parametersDict</i>
<i>plotHistCDFDiff</i>	Produce histogram and CDF plot of the raster difference of two simulations
<i>plotProbMap</i>	plot probability maps including contour lines
<i>plotSample</i>	plot the parameter sample only if two parameters are varied
<i>plotThSample</i>	plot the parameter sample if generated for <i>th</i> thickness values read from shp file one plot for each release Scenario using <i>simDF</i>
<i>plotThSampleFromVals</i>	plot the parameter sample if generated for thickness values read from shp file one plot for each release Scenario - hence <i>paramValuesD</i> should be for one release scenario only
<i>plotValuesScatter</i>	Produce scatter plot of statistical measures (eg.
<i>plotValuesScatterHist</i>	Produce scatter and marginal kde plot of max values, for one set of simulations or multiple
<i>resultHistPlot</i>	create a histogram of values and optional colorcode using scenario name and option to filter simulations using <i>parametersDict</i>

**out3Plot.statsPlots.plotDistFromDF****plotDistFromDF**(*cfg*, *dataDF*, *name1*, *name2*, *scenario*="", *parametersDict*="", *type*="")

create a dist plot from dataframe for name1 on x axis and name2 on y axis, optionally colorcoded with scenario name and filtered with parametersDict

**Parameters**

- **cfg** (*configparser object*) – configuration settings here outDir
- **dataDF** (*dataframe*) – dataframe with one line per simulation and info on model parameters and results
- **name1** (*str*) – column name of dataDF to use for plot x axis
- **name2** (*str*) – column name of dataDF to use for plot y axis
- **scenario** (*str*) – optional name of column used to colorcode points in plots for type=scatter or kde for type=dist
- **parametersDict** (*dict*) – optional - dictionary filter criteria
- **type** (*str*) – optional - type of plot dist or scatter

**Returns**

**plotPath** – path to figure

**Return type**

pathlib path

**out3Plot.statsPlots.plotHistCDFDiff****plotHistCDFDiff**(*dataDiffPlot*, *ax1*, *ax2*, *insert*='True', *title*=["", ""])

Produce histogram and CDF plot of the raster difference of two simulations

**Parameters**

- **dataDiffPlot** (*2D numpy array*) – raster of the difference of the two simulations
- **ax1** (*axes*) – axes for the histogram plot
- **ax2** (*axes*) – axes for the CDF plot
- **insert** (*boolean*) – true if the plots are in inserted axes (size of the labels is then smaller)
- **title** (*list*) – if not inserts, title for the plots

**out3Plot.statsPlots.plotProbMap****plotProbMap**(*avaDir*, *inDir*, *cfgFull*, *demPlot*=False)

plot probability maps including contour lines

**Parameters**

- **avaDir** (*str*) – path to avalanche directory
- **inDir** (*str*) – path to datasets that shall be plotted
- **cfgFull** (*configParser object*) – configuration settings for probAna keys used: name, cmap-Type, levels, unit

- **demPlot** (*bool*) – if True plot dem in background with contourlines for elevation that is found in `avaDir/Inputs`

### out3Plot.statsPlots.plotSample

**plotSample**(*paramValuesD*, *outDir*, *releaseScenario=""*)

plot the parameter sample only if two parameters are varied

#### Parameters

- **paramValuesD** (*dict*) – dictionary with parameter names and sets of values
- **outDir** (*pathlib path*) – path where to save the plot

### out3Plot.statsPlots.plotThSample

**plotThSample**(*simDF*, *name1*, *thName*, *outDir*)

plot the parameter sample if generated for th thickness values read from shp file one plot for each release Scenario using `simDF`

#### Parameters

- **simDF** (*pandas DataFrame*) – dataframe with one row per sim and info on sim configuration
- **name1** (*str*) – name of parameter that is varied for x-axis
- **thName** (*str*) – name of the thickness parameter (`relTh`, `entTh`, `secondaryRelTh`)
- **outDir** (*pathlib path*) – path to folder where to save plot

### out3Plot.statsPlots.plotThSampleFromVals

**plotThSampleFromVals**(*paramValuesD*, *outDir*)

plot the parameter sample if generated for thickness values read from shp file one plot for each release Scenario - hence `paramValuesD` should be for one release scenario only

#### Parameters

- **paramValuesD** (*dict*) – dict with info on parameter variation and initial values of parameters
- **outDir** (*pathlib path*) – path to folder for saving plot

### out3Plot.statsPlots.plotValuesScatter

**plotValuesScatter**(*peakValues*, *resType1*, *resType2*, *cfg*, *avalancheDir*, *statsMeasure='max'*, *flagShow=False*)

Produce scatter plot of statistical measures (eg. max values) (`resType1` und `resType2`), for one set of simulations or multiple

#### Parameters

- **peakDictList** (*dict*) – `peakValues` dictionary that contain max values of peak parameters and parameter variation info
- **resType1** (*str*) – result parameter 1, 'ppr', 'pft', 'pfv'
- **resType2** (*str*) – result parameter 1, 'ppr', 'pft', 'pfv'

- **cfg** (*dict*) – configuration, for now contains output location and varPar: parameter that is varied to perform a set of simulations
- **statsMeasure** (*str*) – statistical measure for plotting, options: max, mean, min, std
- **flagShow** (*bool*) – if True show plot

### out3Plot.statsPlots.plotValuesScatterHist

**plotValuesScatterHist**(*peakValues*, *resType1*, *resType2*, *cfg*, *avalancheDir*, *statsMeasure*='max', *flagShow*=False, *flagHue*=False)

Produce scatter and marginal kde plot of max values, for one set of simulations or multiple

#### Parameters

- **peakValues** (*dict*) – peakValues dictionary that contain max values of peak parameters and parameter variation info
- **resType1** (*str*) – result parameter 1, 'ppr', 'pft', 'pfv'
- **resType2** (*str*) – result parameter 1, 'ppr', 'pft', 'pfv'
- **cfg** (*dict*) – configuration, for now contains output location and varPar: parameter that is varied to perform a set of simulations
- **statsMeasure** (*str*) – statistical measure for plotting, options: max, mean, min, std
- **flagShow** (*bool*) – if True show plot

### out3Plot.statsPlots.resultHistPlot

**resultHistPlot**(*cfg*, *dataDF*, *xName*="", *scenario*="", *stat*='count', *parametersDict*="")

create a histogram of values and optional colorcode using scenario name and option to filter simulations using parametersDict

#### Parameters

- **cfg** (*configparser object*) – configuration info here used outDir
- **dataDF** (*dataFrame*) – dataFrame with info on simulation results and configuration one line per simulation (e.g. aimec resAnalysisDF)
- **xName** (*str*) – column name for x axis
- **scenario** (*str*) – column name used to colorcode values
- **stat** (*str*) – statistical measure to show (percent, probability, density, count, frequency), default count
- **parametersDict** (*dict*) – optional - dictionary filter criteria, parameter name and list of values

#### Returns

**plotPath** – path to figure

#### Return type

pathlib path

### 2.10.3.8 tmp1Ex.tmp1Ex

This is the template for new modules, with the bare minimal required files

#### Functions

<i>tmp1ExMain</i>	Main function for module tmp1Example
-------------------	--------------------------------------

### tmp1Ex.tmp1Ex.tmp1ExMain

#### tmp1ExMain(*cfg*)

Main function for module tmp1Example

##### Parameters

- **foo** (*int, float, str, or tf.Tensor*) – The foo to bar, which has a really really, reeeeeeeeeeeeeeeeeeally unnecessarily long multiline description.
- **bar** (*str*) – Bar to use on foo
- **baz** (*float*) – Baz to frobnicate

##### Returns

The frobnicated baz

##### Return type

float

## 2.11 Development

### 2.11.1 Notes to developers

Here you can find notes on design principles and how to contribute to AvaFrame.

First a few general remarks. We are aware there's no right or wrong here, we decided to stick to a few principles. If you contribute code, please try to observe these:

#### AF-DESIGN-1: Use SI units

Units in APIs should be SI units: seconds, meters, etc.

#### AF-STYLE-1: Line length

If you want a guide for line length — we think line breaking at 120 characters makes more sense than line breaking at 80.

#### AF-STYLE-2: Naming

Use short, but descriptive enough names for variables. Use longer descriptive names for things with bigger scopes (functions, modules). Main theme: as short as possible, but as long as necessary.

Please use lowerCamelCase (e.g.: myFancyVariable) for naming, avoid underscores.

#### AF-STYLE-3: Commenting

Exposed API functions need documentation comments, most importantly a docstring, other code only needs to be commented as necessary. Write your code clearly and use sensible names to reduce the need for comments.

### AF-STYLE-4: No dead code

Unused code, commented out code, functions that are never called, etc, should be removed from the project to reduce the cognitive load of reading the source code. Old code is available in the source history if it is needed.

### AF-STYLE-5: Use spaces for indentation

Use 4 spaces as indentation throughout the python code.

---

## 2.11.2 Our suggested git workflow

Clone repository:

```
git clone https://github.com/avaframe/AvaFrame.git
```

Clones the repository to your local machine into the directory AvaFrame. Sets the repository to track to *origin*

Branch:

```
git checkout -b myAwesomeFeature
```

This changes your working directory to the myAwesomeFeature branch. Try to keep any changes in this branch specific to one bug or feature. You can have many branches and switch in between them using the git checkout command.

Work on it and from time to time commit your changes using following commands as necessary:

```
git add
git commit
```

To update this branch, you need to retrieve the changes from the master branch:

```
git rebase origin master
```

or:

```
git checkout master
git pull
git checkout myAwesomeFeature
git rebase master
```

This replays all your changes on the current status of the master (i.e main) branch. If conflicts arise, now is the time to solve them.

Push your changes to the main repository:

```
git push origin
```

Once you feel you are done, start a pull request on [github.com](https://github.com).

Pull requests are reviewed and handled. Once the pull request is included into the master, the local myAwesomeFeature branch can be deleted (the one in the main repository/origin will be handled by the pull request):

```
git checkout master
git branch -d myAwesomeFeature
```

---



### 2.11.3 Build the documentation

If you want to work on the documentation you need to install *Sphinx*. If you have followed the conda installation using `avaframe_env_spec.txt`, you can omit the following steps. If not, you can install Sphinx, the *ReadTheDocs* theme, and the *sphinxcontrib-bibtex*, which we use to include references, by running:

```
pip install sphinx
pip install sphinx-rtd-theme
pip install sphinxcontrib-bibtex
```

In order to build the documentation you need to install make

```
sudo apt install make
```

Then go to the docs\ directory and run:

```
make html
```

Html files of the documentation can be found in the `_build` directory.

### 2.11.4 How to test code

AvaFrame uses pytest to test code. If you add new code, consider including a pytest for it in `Avaframe/avaframe/tests/`. In order to perform the pytests, just run:

```
pytest
```

and you should see something like:

```
=====test session starts =====
platform linux -- Python 3.8.3, pytest-5.4.3, py-1.9.0, pluggy-0.13.1
collected 1 item
tests/test_tmp1Ex.py . [100%]

=====1 passed in 0.02s =====
```

### 2.11.5 How to add a benchmark test

AvaFrame offers an expanding benchmark test suite. At the moment this test suite includes avalanche simulations for various idealised topographies. The `runStandardTestsCom1DFA.py` facilitates running all the available benchmark tests for com1DFA at once. With this script, the avalanche simulations are performed, plotted and a report of the comparison between simulation results and the benchmark data is generated. If you plan to add a new benchmark test case, follow these steps

- first chose a name, we suggest to start it with `ava` (for now let's refer to it as `NameOfAvalanche`)
- add all the required input data in `data/NameOfAvalanche`. Follow the required directory structure which can be generated using: *Initialize Project*

as a next step, you need to add the benchmark results:

- go to `AvaFrame/benchmarks` and add the subdirectory named after your test name

- add benchmark data i.e. peak values of result parameters as ascii files. This data will be used as reference for the new test!
- add the configuration file as `NameOfAvalanche_com1DFACfg.ini`
- add a json file with required info on benchmark test - you can use the example provided in `runScripts/runWriteDesDict.py`
- go to `AvaFrame/benchmarks/simParametersDict.py` and add a simulation dictionary that contains all the info on the new benchmark

Now, you are ready to go! Move to `AvaFrame/avaframe` and run:

```
python runStandardTestsCom1DFA.py
```

You can check out the markdown-style report of the comparison at: `tests/reports/standardTestsReportPy.md`.

## COMPUTATIONAL MODULES

### 3.1 com1DFA: DFA-Kernel

*com1DFA* is a module for dense flow (snow) avalanche computations (DFA) . It is a python and cython implementation of the DFA C++ implementation samosAT (Snow Avalanche Modeling and Simulation- Advanced Technologies) developed by the Austrian government in cooperation with the company AVL List GmbH in Graz (see [com1DFAOrig: Original DFA-Kernel](#)). Calculations are based on the thickness integrated governing equations and solved numerically using the smoothed particle hydrodynamics (sph) method. Please note the use of *thickness averaged/integrated* instead of *depth averaged/integrated* for clarity and consistency.

Dense flow avalanche simulations can be performed for different release area scenarios, with or without entrainment and/or resistance areas, and is controlled via a configuration file. The configuration can be modified in order to change any of the default settings and also allows to perform simulations for varying parameters all at once.

---

**Note:** The configuration provided with com1DFA is well-tested and applied for hazard mapping (in Austria). If you change configuration parameters, be aware that unwanted/unexpected/spurious side-effects might appear. This is especially true if you switch to something far outside the intended range (i.e. changing density from snow to something like rock). Furthermore, be aware that the parameters are calibrated in connection, so changing one might necessitate also changing other connected parameters!

---

#### 3.1.1 Input

DFA simulations are performed within an avalanche directory, organized with the folder structure described below.

---

**Note:** An avalanche directory can be created by running: `runInitializeProject.py`, which creates the required folder structure:

```
NameOfAvalanche/  
  Inputs/  
    REL/      - release area scenario  
    RES/      - resistance areas  
    ENT/      - entrainment areas  
    POINTS/   - split points  
    LINES/    - avalanche paths  
    SECREL/   - secondary release areas  
  Outputs/  
  Work/
```

In the directory `Inputs`, the following files are required. Be aware that ALL inputs have to be provided in the same projection:

- digital elevation model as .asc file with [ESRI grid format](#)
- release area scenario as (multi-) polygon shapefile (in `Inputs/REL`; multiple features are possible)
  - the release area name should not contain an underscore, if so ‘\_AF’ is added.
  - recommended attributes are *name*, *thickness* (see [Release-, entrainment thickness settings](#)) and *ci95* (see [probAna - Probability maps](#))
  - ALL features within one shapefile are released at the same time (and interact), this is what we refer to as *scenario*
  - if you want to simulate different scenarios with the same features, you have to copy them to separate shapefiles

and the following files are optional:

- one entrainment area (multi-) polygon shapefile (in `Inputs/ENT`)
  - marks the (multiple) areas where entrainment can occur.
  - attribute *thickness* (see [Release-, entrainment thickness settings](#))
- one resistance area (multi-) polygon shapefile (in `Inputs/RES`)
  - marks the (multiple) areas where resistance is considered
- one secondary release area (multi-) polygon shapefile (in `Inputs/SECREL`)
  - can have multiple release areas, each as one feature
  - same setup as the release area scenario (see above)
  - features will release as soon as at least one particle enters its area

### 3.1.1.1 Release-, entrainment thickness settings

Release, entrainment and secondary release thickness can be specified in two different ways:

#### 1. Via **shape file**:

- add an attribute called *thickness* for each feature
- important: ALL features have to have a single thickness value, which can differ between features
- for entrainment area only: if the thickness value is missing, the thickness value is taken from *entThIfMissingInShp* (default 0.3 m) in the configuration file
- for backwards compatibility, the attribute ‘d0’ also works, but we suggest to use *thickness* in new projects
- set the flag *THICKNESSFromShp* (i.e. *relThFromShp*, *entThFromShp*, *secondaryRelthFromShp*) to True in the configuration file (default is True)
- a parameter variation can be added with the *THICKNESSPercentVariation* parameter in the configuration file in the form of *+-percentage\$numberOfSteps*. Provided a + a positive variation will be performed, if - is given, only a negative variation is performed. If no sign is given: both directions will be used. Additionally, a variation can be added with the *THICKNESSRangeVariation* parameter in the configuration file in the form of *+-range\$numberOfSteps*. Provided a + a positive variation will be performed, if - is given, only a negative variation is performed. If no sign is given: both directions will be used. Furthermore, there is the option to vary the thickness in a range of +- the 95% confidence interval value, which is also read from the shape file

(requires an attribute called ci95). In order to use this variation, set the 'THICKNESSRangeFromCiVariation' to ci95\$numberOfSteps.

## 2. Via **configuration file (ini)**:

- set the flag 'THICKNESSFromShp' to False
- provide your desired thickness value in the respective THICKNESS parameter (i.e. relTh, entTh or secondaryRelth)
- in addition to the *THICKNESSPercentVariation* and *THICKNESSRangeVariation* options (see option 1) and the standard variation options in configuration:Configuration, you can also directly set e.g. *relTh = 1.\$50\$2, referenceValue\$+-percentage\$numberOfSteps*, resulting in a variation of relTh from 0.5 to 1.5m in two steps.

Only available for release thickness:

## 3. Via **release thickness file**:

- set the flag 'relThFromShp' to False
- set the flag 'relThFromFile' to True
- save a raster file with info on release thickness as .asc file in Inputs/RELTH the number of rows and columns must match the DEM raster with desired meshCellSize

### 3.1.1.2 DEM input data

Regarding the DEM data: if the DEM in Inputs is not of cell size 5 meters, it is remeshed to a cell size of 5 meters. However, it is also possible to specify a desired cell size in the configuration file (parameter *meshCellSize*). In this case, also consider reading *Can the spatial resolution of simulations performed with com1DFA (dense flow) be changed?*. If the cell size of the DEM in Inputs is equal to the desired mesh cell size, the DEM is used without modification. If the cell sizes do not match, several options are available:

- cleanremeshedRasters = True, directory Inputs/remeshedRasters is cleaned, and the DEM in Inputs/ is remeshed to the desired cell size - this is the default setting
- cleanremeshedRasters = False and a DEM including the name of the DEM in Inputs/ and the desired cell size is found in Inputs/remeshedRasters - this DEM is used without modification
- cleanremeshedRasters = False and no matching DEM is found in Inputs/remeshedRasters - the DEM in Inputs/ is remeshed to the desired cell size

If the DEM in Inputs/ is remeshed, it is then saved to Inputs/remeshedRasters and available for subsequent simulations.

### 3.1.1.3 Dam input

The com1DFA module provides the option to take the effect of dams into account. This is done using an ad-hoc method based on particles being reflected/deflected by a dam wall.

The dam is described by the crown line, the slope and the restitution coefficient:

- crown line as shape file (use the line type and enable the “additional dimensions” option in order to specify the z coordinate). The z coordinate corresponds to the absolute height (terrain elevation plus dam height). The dam is then located on the left side of the dam (when one travels from the first point to the last point of the shapefile line). The dam shape files live in the avaDir/Inputs/DAM/ directory (only one file is allowed).
- the slope of the dam (in degrees °) between the horizontal plane and the wall to be provided in the shape file as an attribute (default value is 60° in the provided examples: avaSlide, avaKot and avaBowl)

- the restitution coefficient ( $\alpha_{\text{rest}}$ ), a float between 0 (no reflection in the normal direction) and 1 (full reflection) to be specified in the ini file (default value is 0)

### 3.1.2 Model configuration

The model configuration is read from a configuration file: `com1DFA/com1DFACfg.ini`. In this file, all model parameters are listed and can be modified. We recommend to create a local copy and keep the default configuration in `com1DFA/com1DFACfg.ini` untouched. For this purpose, in `AvaFrame/avaframe/` run:

```
cp com1DFA/com1DFACfg.ini com1DFA/local_com1DFACfg.ini
```

and modify the parameter values in there. For more information see `configuration:Configuration`.

It is also possible to perform multiple simulations at once, with varying input parameters.

### 3.1.3 Output

Using the default configuration, the simulation results are saved to: *Outputs/com1DFA* and include:

- raster files of the peak values for pressure, flow thickness and flow velocity (*Outputs/com1DFA/peakFiles*)
- raster files of the peak values for pressure, flow thickness and flow velocity for the initial time step (*Outputs/com1DFA/peakFiles/timeSteps*)
- markdown report including figures for all simulations (*Outputs/com1DFA/reports*)
- mass log files of all simulations (*Outputs/com1DFA*)
- configuration files for all simulations (*Outputs/com1DFA/configurationFiles*)

optional outputs

- pickles of particles properties (*Particle properties*.) for saving time steps if particles are added to the list of `resTypes` in your local copy of `com1DFACfg.ini`
- a csv file of specified particle properties for the saving time steps if particles are added to the list of `resTypes` in your local copy of `com1DFACfg.ini` and if in the VISUALISATION section `writePartToCsv` is set to `True`

However, in the configuration file, it is possible to change the result parameters and time Steps that shall be exported. The result types that can be chosen to be exported are (all correspond to fields except the particles):

- ppr - peak pressure
- pfv - peak flow velocity
- pft - peak flow thickness
- pta - peak travel angle
- FV - flow velocity
- FT - flow thickness
- P - pressure
- FM - flow mass
- Vx, Vy, Vz - velocity x-, y- and z-component
- TA - travel angle
- particles (*Particle properties*)

Have a look at the designated subsection Output in `com1DFA/com1DFACfg.ini`.

### 3.1.4 Parallel computation

If multiple runs of com1DFA are to be executed, these will be calculated in parallel via multiprocessing. So each task itself is calculated on only one core, but different tasks are run at the same time.

This happens if you have one of the following (or a combination of them):

- multiple scenarios (multiple input release shapefiles)
- multiple runtypes, i.e null variant and entrainment/resistance variant (e.g.: simTypeList = null|ent)
- some kind of parameter variation (e.g.: relTh = 1.0|1.5|1.7)

The number of CPU cores is controlled in the main `avaframeCfg.ini` file. By default a maximum of 50 percent of your available cores is being utilized. However you can set a different number if needed. For sequential execution set nCPU to 1.

### 3.1.5 To run

- first go to AvaFrame/avaframe
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- create an avalanche directory with required input files - for this task you can use [Initialize Project](#)
- copy `com1DFA/com1DFACfg.ini` to `com1DFA/local_com1DFACfg.ini` and if desired change configuration settings
- if you are on a develop installation, make sure you have an updated compilation, see `developinstall:Setup AvaFrame`
- run:

```
python3 runCom1DFA.py
```

### 3.1.6 Theory

The governing equations of the dense flow avalanche are derived from the incompressible mass and momentum balance on a Lagrange control volume ([Zw2000] [ZwK1Sa2003]). Assuming the avalanche is much longer and larger than thick, it is possible to integrate the governing equations over the thickness of the avalanche and operate some simplifications due to the shape of the avalanche. This leads, after some calculation steps described in details in Theory [Governing Equations for the Dense Flow Avalanche](#) to:

$$\begin{aligned} \frac{dV(t)}{dt} &= \frac{d(A_b \bar{h})}{dt} = \frac{\rho_{\text{ent}}}{\rho_0} w_f h_{\text{ent}} \|\bar{\mathbf{u}}\| \\ \frac{d\bar{u}_i}{dt} &= g_i + \frac{K_{(i)}}{\bar{\rho} A \bar{h}} \oint_{\partial A} \left( \frac{\bar{h} \sigma^{(b)}}{2} \right) n_i dl - \delta_{i1} \frac{\tau^{(b)}}{\bar{\rho} \bar{h}} - C_{\text{res}} \bar{\mathbf{u}}^2 \frac{\bar{u}_i}{\|\bar{\mathbf{u}}\|} - \frac{\bar{u}_i}{A \bar{h}} \frac{d(A \bar{h})}{dt} + \frac{F_i^{\text{ent}}}{\bar{\rho} A \bar{h}} \\ \bar{\sigma}_{33}^{(b)} &= \rho \left( g_3 - \bar{u}_1^2 \frac{\partial^2 b}{\partial x_1^2} \right) \bar{h} \end{aligned}$$

### 3.1.7 Numerics

Those equations are solved numerically using a **SPH** method ([LL10, Sam07]). **SPH** is a mesh free method where the basic idea is to divide the avalanche into small mass particles. The particles interact with each other according to the equation of motion described in *Theory* and the chosen kernel function. This kernel function describes the domain of influence of a particle (through the smoothing length parameter). See theory *com1DFA DFA-Kernel theory* for further details.

## 3.2 com1DFAOrig: Original DFA-Kernel

com1DFAOrig is a simulation tool for dense flow (snow) avalanches (DFA). It is a python wrapper function calling the samos-AT C++ (Snow Avalanche Modeling and Simulation- Advanced Technologies) DFA simulation code developed by the Austrian government in cooperation with the company AVL List GmbH in Graz. This code served as base for the implementation of the python DFA simulation module *com1DFA* presented in *com1DFA: DFA-Kernel*. *com1DFA* module is now the default DFA simulation module in AvaFrame.

The calculation of the DFA is based on the depth integrated governing equations and solved numerically using the smoothed particle hydrodynamic (sph) method.

Dense flow avalanche simulations can be performed for different release area scenarios, with or without entrainment and/or resistance areas. There is the option to vary the internal friction parameter or the release snow thickness.

### 3.2.1 C++ Executable

The computation of the com1DFAOrig dense flow avalanche module relies on a C++ executable. The executable (for now 64bit linux and windows) and needed files are available in this [git repository](#). To install, change into your directory [YOURDIR] from the AvaFrame installation above and clone the repository:

```
cd [YOURDIR]
git clone https://github.com/avaframe/com1DFA_Exe
```

Rename the executables according to your operating system, i.e. for Linux do:

```
mv com1DFA_Exe/com1DFA_x86_64.exe com1DFA_Exe/com1DFA.exe
mv com1DFA_Exe/SHPCnv_linux.exe com1DFA_Exe/SHPCnv.exe
```

for Windows do:

```
mv com1DFA_Exe/com1DFA_win64.exe com1DFA_Exe/com1DFA.exe
mv com1DFA_Exe/SHPCnv_win.exe com1DFA_Exe/SHPCnv.exe
```

Go to the com1DFAOrig directory of the AvaFrame repository from above and copy the configuration file:

```
cd AvaFrame/avaframe/com1DFAOrig
cp com1DFACfg.ini local_com1DFACfg.ini
```

Open the local\_com1DFACfg.ini file in your preferred text editor and change the com1Exe variable to reflect your paths, i.e.:

```
com1Exe = [YOURDIR]/com1DFA_Exe/com1DFA.exe -files [YOURDIR]/com1DFA_Exe/files/AK_
↳Attributes
```



**Attention:** We suggest to use the full path.

To test go to [YOURDIR], change into the com1DFA\_Exe repository and run the executable:

```
cd [YOURDIR]
cd com1DFA_Exe
./com1DFA.exe -files files/AK_Attributes/
```

The output should start like this:

```
Setting config files directory: files/AK_Attributes/ (src/SW_Workspace.cpp:3435)
./com1DFA.exe -files files/AK_Attributes/ (src/SW_Workspace.cpp:3453)
=====
./com1DFA.exe
Compiled Oct 19 2020 21:34:15
...
```

Exit by pressing q

### 3.2.2 Input

The module requires an avalanche directory, that follows a specified folder structure. This avalanche directory can be created by running `runInitializeProject.py`. In the directory *Inputs*, the following files are required:

- digital elevation model as .asc file -> use [ESRI grid format](#)
- release area scenario as shapefile (in *Inputs/REL*); multiple are possible -> the shapefile name should not contain an underscore, if so ‘\_AF’ is added

and the following files are optional:

- entrainment area as shapefile (in *Inputs/ENT*)
- resistance area as shapefile (in *Inputs/RES*)
- secondary release area as shapefile (in *Inputs/SECREL*)

The simulation settings area defined in the configuration file `com1DFAOrig/com1DFAOrigCfg.ini`:

- `com1Exe` - path to com1DFA executable
- `flagOut` - print full model output
- `simTypeList` - simulation types that shall be performed (options: null, ent, res, entres, available; if multiple, separate by ‘|’)
- `releaseScenario` - name of release area scenario shapefile (with or without extension -shp, if multiple, separate by ‘|’)
- `flagVarPar` - perform parameter variation
- `varPar` - parameter to be varied
- `varParValues` - values for parameter variation

### 3.2.3 Output

The simulation results are saved to: *Outputs/com1DFAOrig* and include:

- raster files of the peak values for pressure, flow thickness and flow velocity (*Outputs/com1DFAOrig/peakFiles*)
- reports of all simulations (*Outputs/com1DFAOrig/reports*)
- log files of all simulations
- experiment log that lists all simulations

### 3.2.4 To run

**Attention:** Please refer to the instructions in C++ Executable on how to get the necessary C++ executable and setup the correct paths.

- first go to `AvaFrame/avaframe`
- create an avalanche directory with required input files - for this task you can use *Initialize Project*
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- run:

```
python3 com1DFAOrig/runCom1DFA.py
```

### 3.2.5 Theory

The governing equations of the dense flow avalanche are derived from the incompressible mass and momentum balance on a Lagrange control volume ([Zwi00, ZKS03]). Assuming the avalanche is much longer and larger than thick, it is possible to integrate the governing equations over the thickness of the avalanche and operate some simplifications due to the shape of the avalanche. This leads, after some calculation steps described in details in Theory *Governing Equations for the Dense Flow Avalanche* to:

$$\begin{aligned}\frac{dV(t)}{dt} &= \frac{d(A_b \bar{h})}{dt} = \frac{\rho_{\text{ent}}}{\rho_0} w_f h_{\text{ent}} \|\bar{\mathbf{u}}\| \\ \frac{d\bar{u}_i}{dt} &= g_i + \frac{K_{(i)}}{\bar{\rho} A \bar{h}} \oint_{\partial A} \left( \frac{\bar{h} \sigma^{(b)}}{2} \right) n_i dl - \delta_{i1} \frac{\tau^{(b)}}{\bar{\rho} \bar{h}} - C_{\text{res}} \bar{\mathbf{u}}^2 \frac{\bar{u}_i}{\|\bar{\mathbf{u}}\|} - \frac{\bar{u}_i}{A \bar{h}} \frac{d(A \bar{h})}{dt} + \frac{F_i^{\text{ent}}}{\bar{\rho} A \bar{h}} \\ \bar{\sigma}_{33}^{(b)} &= \rho \left( g_3 - \bar{u}_1^2 \frac{\partial^2 b}{\partial x_1^2} \right) \bar{h}\end{aligned}$$

### 3.2.6 Numerics

Those equations are solved numerically using a **SPH** method ([LL10, Sam07]). **SPH** is a mesh free method where the basic idea is to divide the avalanche into small mass particles. The particles interact with each other according to the equation of motion described in *Theory* and the chosen kernel function. This kernel function describes the domain of influence of a particle (through the smoothing length parameter). See theory `theoryCom1DFA:Numerics` for further details.

### 3.3 com2AB: Alpha Beta Model

*com2AB* calculates the runout of an avalanche according to the statistical  $\alpha - \beta$  model. An avalanche is defined by its DEM (digital elevation model), a path and a split point. The runout is calculated according to the  $\alpha - \beta$  model calibrated for avalanches in the Austrian Alps. It is also possible to adapt the model parameters for other regions.

#### 3.3.1 Input

Please be aware that all input data have to be provided in the same projection.

- digital elevation model as .asc file with [ESRI grid format](#)
- set of avalanche paths as (multi-) line shapefile. I.e there can be multiple paths in the shapefile
  - recommended attribute *name*
- Split points as (multi-) point shapefile. I.e. there can be multiple split points in the shapefile. For each profile/path the closest split point is considered.

#### 3.3.2 Outputs

- profile plot with alpha, beta and runout points
- txt file with angle and coordinates of the different points

#### 3.3.3 To run

- go to AvaFrame/avaframe
- copy `com2AB/com2ABCfg.ini` to `com2AB/local_com2ABCfg.ini` and edit (if not, default values are used)
- make sure all the required inputs are available in the avalanche directory
- enter the path to the desired dataset in `local_avaframeCfg.ini`
- run:

```
python3 runCom2AB.py
```

#### 3.3.4 Theory

The snow avalanche runout distance is calculated using a statistical model based on data collected for real avalanches ([BDL83, LB80, Wag16]). An equation of the following type is fitted to the data:

$$\alpha_j = k_1\beta + k_2z'' + k_3H_0 + k_4 + jSD$$

where  $H_0$  is the elevation loss of the quadratic fit of the avalanche profile.  $z''$  is the curvature of this same quadratic fit.  $\beta$  is the angle of the line between the  $10^\circ$  point (first point where the avalanche profiles slope is under  $10^\circ$ ) and the starting point. The coefficients  $(k_1, k_2, k_3, k_4)$  and the standard deviation  $SD$  are calculated during the fitting process. Index  $j = \{-1, -2, 0, 1\}$  and  $\alpha_j = \alpha + jSD$ . These coefficients obviously depend on the initial set of chosen data.  $\alpha_0 = \alpha$  is the angle between the stopping and the starting point of the avalanche.  $\alpha_j = \alpha + jSD$  takes into account the variability of the process. The values of the SD used are based on normal distribution. It is important to note that a bigger runout angle leads to a shorter runout distance. This means that  $\alpha_{-1} = \alpha - SD$  leads to a longer runout. In other words, the probability of the runout being shorter than  $s_{\alpha_{-1}}$  corresponding to  $\alpha_{-1}$  is approximately 83%.

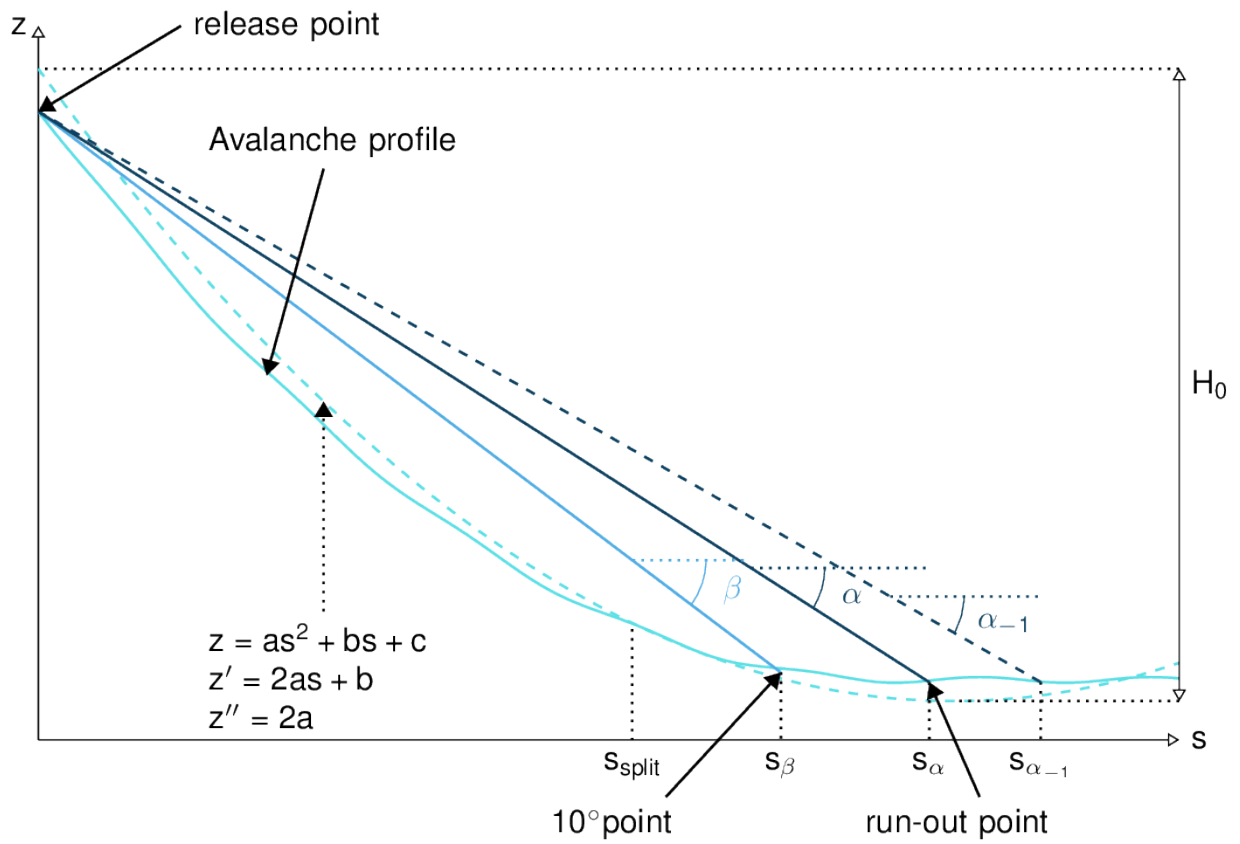


Fig. 3.1: Topographical variables for the calculation of  $\alpha$

In this module, the coefficients  $(k_1, k_2, k_3, k_4)$  and the standard deviation  $SD$  are already known, they are simply used in the  $\alpha$  equation to calculate the runout on a new profile.

### 3.3.5 Procedure

Preprocessing :

- The avalanche path (x,y) is first resampled. Default value for resampling is distance=10m (maximal horizontal distance between two points). Note that it does not make much sense to decrease this value to be smaller than the raster grid resolution. We then introduce the curvilinear coordinate  $s$  which represents the projected horizontal distance along the path.
- The avalanche path is projected on the DEM to generate the profile using a bi-linear interpolation on the DEM to the point of interest.
- The split point (which is not necessarily given on the avalanche path) is projected on the avalanche path.

From this we obtain the (x,y,z) and (s,z) coordinates of the avalanche profile.

AlphaBeta Model:

- Find the  $10^\circ$  point from (s,z).
- Calculate  $\beta$ .
- Calculate the  $\alpha_j$  angles using the adequate standard, small avalanche or custom parameter set.

Postprocessing:

- Plot and save results.

### 3.3.6 Configuration parameters

#### **distance**

resampling distance. The given avalanche path is resampled with a step  $\leq 10$ m (default).

#### **dsMin**

float. Threshold distance [m]. When looking for the beta point make sure at least dsMin meters after the beta point also have an angle below  $10^\circ$  (dsMin=30m as default).

#### **smallAva**

boolean (False as default) if True apply  $(k_1, k_2, k_3, k_4, SD)$  set of small avalanches or False, standard avalanches

#### **customParam**

boolean (False as default). Enables to choose custom  $(k_1, k_2, k_3, k_4, SD)$ . If True, the values from the configuration file are used

#### **k1**

float. Use this value if customParam=True

#### **k2**

float. Use this value if customParam=True

#### **k3**

float. Use this value if customParam=True

#### **k4**

float. Use this value if customParam=True

**SD**

float. Use this value if `customParam=True`

**PlotPath**

Plot Avalanche path on raster; default False

**PlotProfile**

Plot profile; default False

**SaveProfile**

Save profile to file; default True

**WriteRes**

Write result to file: default True

## 3.4 com3Hybrid: Hybrid modeling

*com3Hybrid* is a computational module that combines the dense flow avalanche (DFA) simulation model (*com1DFA*) and the Alpha-Beta statistical one (*com2AB*), taking advantage of their strength and trying to reduce their weaknesses. The weakness of the DFA simulation lies in its required inputs, among which is the friction parameter  $\mu$ . The disadvantage of the statistical model lies in the necessary path, which is used to extract an avalanche profile and compute a runout angle. The idea here is to determine the avalanche path in an automated way by:

- first, running a DFA simulation
- computing a mass averaged path from the results
- use the path to compute the runout angle corresponding to this specific avalanche
- using a coulomb friction method, the friction parameter  $\mu$  is extracted from the runout
- the resulting  $\mu$  is used in the input for a new DFA simulation.

This iteration process can be repeated multiple times until “convergence”, which means until the  $\mu$  or  $\alpha$  value stops varying.

### 3.4.1 Input

- raster of the DEM (.asc file)
- a release feature (shapefile) in `Inputs/REL`
- Split point (shapefile). This requirement is planned to be automated as well.

### 3.4.2 Outputs

- The avalanche path specific for the input topography
- the  $\mu$  value ( $\alpha$  travel angle) specific for the input topography
- results from the DFA simulation for the input topography
- a combined plot showing the results of the DFA simulation and Alpha-Beta model

### 3.4.3 To run

- go to AvaFrame/avaframe
- copy com3Hybrid/com3HybridCfg.ini to com3Hybrid/local\_com3HybridCfg.ini and edit (if not, default values are used)
- put your com1DFA and com2AB settings in com3Hybrid/hybridModel\_com1DFACfg.ini and com3Hybrid/hybridModel\_com2ABCfg.ini (these files replace the local .ini files in com1DFA and com2AB)
- make sure all the required inputs are available in the avalanche directory
- enter the path to the desired dataset in local\_avaframeCfg.ini
- run:

```
python3 runScripts/runCom3Hybrid.py
```

### 3.4.4 Procedure

1. A first com1DFA simulation is run using the com3Hybrid/hybridModel\_com1DFACfg.ini configuration and the  $\mu$  provided in com3Hybrid/local\_com3HybridCfg.ini.
2. The mass averaged path is computed from the DFA simulation using the [ana5Utils.DFAPathGeneration](#). From this we obtain the (x,y,z) and (s,z) coordinates of the avalanche profile (the path is extended towards the release and in the runout). This path is saved to avalancheDir/Inputs/LINES and replaces the old path. A copy is also saved to avalancheDir/Outputs/com3Hybrid.
3. com2AB module is run using the com3Hybrid/hybridModel\_com2ABCfg.ini to compute the  $\mu = \tan \alpha$  friction parameter corresponding to this specific avalanche and area.
4. The com3Hybrid/hybridModel\_com1DFACfg.ini is updated with the new  $\mu = \tan \alpha$  value and com1DFA simulation is run again.
5. Steps 2. and 3. are repeated to get the new avalanche path and  $\mu = \tan \alpha$  value.

This process is repeated as long as the  $\alpha = \arctan \mu$  value varies from more than `alphaThreshold` which you can specify in com3Hybrid/local\_com3HybridCfg.ini (this is a value in degrees) and this for a maximum of `nIterMax` iterations.

### 3.4.5 Configuration parameters

The parameters for path generation (section PATH of com3Hybrid/local\_com3HybridCfg.ini) are described in [Automated path generation](#).

The parameters for the DFA simulation are to be specified in com3Hybrid/hybridModel\_com1DFACfg.ini. This file replaces the local\_com1DFACfg.ini that is usually used to run a com1DFA simulation.

The parameters for the  $\alpha - \beta$  model are to be specified in com3Hybrid/hybridModel\_com2ABCfg.ini. This file replaces the local\_com2ABCfg.ini that is usually used to run a com2AB simulation.

## 3.5 com4FlowPy: Flow-Py

---

**Note:** THIS MODULE IS CURRENTLY UNDER HEAVY DEVELOPMENT!

The code is not automatically tested in any way and not included in the code coverage!

It also does not adhere to the AvaFrame coding and naming conventions (yet)...

Use at your own risk and if you want to contribute to the modules improvement, you are very welcome to do so!

---

Flow-Py is an open source tool to compute gravitational mass flows (GMF) run out and intensity. The main objective is to compute the spatial extent of GMF, which consists of the starting, transit and runout zones of GMF on the surface of a three dimensional terrain. The resulting run out is mainly dependent on the terrain and the location of the starting/release point. No time-dependent equations are solved in the model. com4FlowPy uses existing statistical-data-based approaches for solving the routing and stopping of GMF.

The tool has been designed to be computationally light, allowing the application on a regional scale including a large number of GMF paths. The implementation allows users to address specific GMF research questions by keeping the parameterization flexible and the ability to include custom model extensions and add-ons.

### 3.5.1 Running the Code

**You are required to install rasterio and gdal separately, since they are not included in the general AvaFrame requirements.**

If you have trouble installing GDAL or rasterio on Windows use these links to get the required version directly from their website, first install *GDAL* and then *rasterio*.

*GDAL*: <https://www.lfd.uci.edu/~gohlke/pythonlibs/#gdal>

*rasterio*: <https://www.lfd.uci.edu/~gohlke/pythonlibs/#rasterio>

Once the required libraries are installed the model runs via the `runCom4FlowPy.py` script.

### 3.5.2 Configuration

The configuration can be found in `com4FlowPyCfg.ini`

- `alpha_angle` (controls the run out angle induced stopping and routing)
- `exponent` (controls concentration of routing flux and therefore the lateral spread)
- working directory path
- path to DEM raster (.tiff or .asc)
- path to release raster (.tiff or .asc)
- (Optional) flux threshold (positive number) `flux_threshold=xx` (limits spreading with the exponent)
- (Optional) Max  $Z_{\Delta}$  (positive number) `max_z_delta=xx` (max kinetic energy height, turbulent friction)



### 3.5.3 Input Files

All raster files (DEM, release, ...) must be in the .asc or .tif format.

All rasters need the same resolution (normal sizes are e.g. 5x5 or 10x10 meters).

All Layers need the same spatial extend, with no data values < 0 (standard no data values = -9999).

The locations identified as release areas need values > 0. (see release.tif in examples)

### 3.5.4 Output

All outputs are in the .tif raster format in the same resolution and extent as the input raster layers.

- **z\_delta**: the maximum z\_delta of all paths for every raster cell (geometric measure of process magnitude, can be associated to kinetic energy/velocity)
- **Flux**: The maximum routing flux of all paths for every raster cell
- **sum\_z\_delta**: z\_delta summed up over all paths on every raster cell
- **Cell\_Counts**: number of paths that route flux through a raster cell
- **Flow Path Travel Angle, FP\_TA**: the gamma angle along the flow path
- **Straight Line Travel Angle, SL\_TA**: Saves the gamma angle, while the distances are calculated via a straight line from the release cell to the current cell

## 3.6 com5SnowSlide: Snow slide

The com5SnowSlide computational module provides the option to add an elastic cohesion force between particles based on com1DFA. This can be used for example to simulate small snow slides.

The cohesion between particles is modeled using elastic bonds with maximum strain rate. This means that the cohesion force  $\mathbf{F}_{lk}$  exerted by  $l$  on  $k$  is expressed by:

$$\mathbf{F}_{lk} = -\mathbf{F}_{kl} = \frac{L - L_0}{L_0} A_{kl} E \mathbf{e}_{kl}$$

Where  $L$  and  $L_0$  are the distances between the particles  $k$  and  $l$  at a time  $t$  and at rest (initial time step),  $E$  is the elastic modulus of the material,  $A_{kl}$  is the contact area between particles  $k$  and  $l$  and  $\mathbf{e}_{kl}$  is the unit vector

$$\epsilon = \frac{L - L_0}{L_0}$$

represents the strain ( $\epsilon$  is positive in the case of elongation and negative in compression). If the strain exceeds a critical value  $\epsilon_c$  the bond between the particles  $k$  and  $l$  breaks.

The contact area  $A_{kl}$  between particles  $k$  and  $l$  reads:

$$A_{kl} = h_{kl} d_{kl} = \frac{h_k + h_l}{2} \frac{L}{\sqrt{3}}$$

$h_{kl}$  represents the height of the contact surface and  $d_{kl}$  represents the length of the contact surface in the horizontal plane assuming that each particle has 6 neighbours.

### 3.6.1 Input

The standard inputs required to perform a simulation run using *com1DFA* can be found here: *Input*. There is a run script to perform a snow slide com1DFA run: `runCom5SnowSlide.py`, and the configuration settings can be found in `com5SnowSlide/com5SnowSlideCfg.ini`. The snow slide-specific parameters are:

- snowSlide is activated by setting `snowSlide` to 1
- the maximum strain before breaking of the bond `cohesionMaxStrain`
- the Young modulus in N/m<sup>2</sup> `cohesiveSurfaceTension`

However, also several other parameters, for example the particle initialization method, friction model and parameters, are changed compared to the default configuration of *com1DFA* and listed in the `com5SnowSlide/com5SnowSlideCfg.ini` in the `com1DFA_override` section.

Please note that the provided default setup for snow slide calculation is defined with the following premises:

- slope gradient in the release area from approx. 28° to approx. 50°.
- a return level for hazard mapping of 150-yr of snow depth as the initial value.
- movement as an intact unit without turbulence
- height difference from approx. 30 m to max. 60-80 m
- valid for altitude levels > 500 m and < 1,500 m sea level

### 3.6.2 Initialization of bonds

If the elastic cohesion (`snowSlide` set to 1) is activated, the bonds between particles are initialized. The construction of the bonds is done by building a triangular mesh on the particles (used as a point cloud) and the Delaunay *triangulation function* from matplotlib. The length  $L_0$  of the bonds at rest (initial distance between two bonded particles) is also initialized here.

---

**Note:** It is recommended to use the triangular (`triangular`) initialization option (*Initialize particles*) rather than the random or the uniform one in the case where cohesion is activated.

---

### 3.6.3 To run

- first go to `AvaFrame/avaframe`
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- create an avalanche directory with required input files - for this task you can use *Initialize Project*
- copy `com5SnowSlide/com5SnowSlideCfg.ini` to `com5SnowSlide/local_com5SnowSlideCfg.ini` and if desired change configuration settings
- if you are on a develop installation, make sure you have an updated compilation, see `installation:Setup AvaFrame`
- run:

```
python3 runCom5SnowSlide.py
```

## 3.7 com6RockAvalanche: Rock Avalanche

**Warning:** This is highly experimental and not tested!

The com6RockAvalanche computational module provides an override setting for com1DFA targeting the simulation of rock avalanches.

### 3.7.1 Input

The standard inputs required to perform a simulation run using `com1DFA` can be found here: [Input](#). However there is one main difference: com6RockAvalanche NEEDS a release thickness raster file. This file has to have the exact same dimensions as the topography file. There is a run script to perform a rock avalanche com1DFA run: `runCom6RockAvalanche.py`, and the configuration settings can be found in `com6RockAvalanche/com6RockAvalancheCfg.ini`.

### 3.7.2 To run

- first go to `AvaFrame/avaframe`
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- create an avalanche directory with required input files - for this task you can use [Initialize Project](#)
- copy `com6RockAvalanche/com6RockAvalancheCfg.ini` to `com6RockAvalanche/local_com6RockAvalancheCfg.ini` and if desired change configuration settings
- if you are on a develop installation, make sure you have an updated compilation, see `installation:Setup AvaFrame`
- run:

```
python3 runCom6RockAvalanche.py
```



## INPUT/TRANSFORMATION MODULES

### 4.1 in1Data: Input data utilities

#### 4.1.1 Get input data

*in1Data.getInput* module provides functions to fetch or generate input data for avalanche simulations within AvaFrame. The main functions are described in more detail in the following sections. A detailed description of all the small helper functions is provided in *in1Data.getInput*

##### 4.1.1.1 getInputCom1DFA

`in1Data.getInput.getInputCom1DFA()` fetches the input data from an avalanche director required to start avalanche simulations with *com1DFA*. This data consists of a digital elevation model (DEM), a release scenario shapefiles, optional entrainment and resistance area shapefiles and is returned as a dictionary.

#### 4.1.2 computeFromDistribution

*in1Data.computeFromDistribution* is collection of functions that facilitates retrieving a sample of values that are distributed following a uniform or Pert distribution. This is useful if you want to perform avalanche simulations for a range of release thickness values (or any other parameter), that should be distributed following a specific distribution. Besides returning a sample of values following the desired distribution, various plots can be generated detailing the characteristics of the sample. The required input parameters can be found in the respective configuration file `in1Data.computeFromDistributionCfg.ini`. Detailed information on the functions can be found in *in1Data.computeFromDistribution*.

##### 4.1.2.1 To run

An example of how a parameter set distributed following a uniform or Pert distribution can be generated, is provided by `runScripts/runComputeDist.py`.

- first go to `AvaFrame/avaframe`
- copy `in1Data/computeFromDistributionCfg.ini` to `in1Data/local_computeFromDistributionCfg.ini` (if not, the standard settings are used)
- adjust path to the desired `NameOfAvalanche/` folder in your local copy of `avaframeCfg.ini`
- run:

```
python3 runScripts/runComputeDist.py
```

## 4.2 in2Trans: Transformation Utilities

### 4.2.1 Working with ASCII files

*in2Trans.ascUtils* is a module created to handle raster ASCII files. It contains different functions to read ASCII files and write the data to a numpy array, to compare raster file headers or to write a raster to an ASCII file. A description of the functions is available in *in2Trans.ascUtils*.

### 4.2.2 Working with shapefiles

*in2Trans.shpConversion* is a module for handling shapefiles. It contains different functions to read shapefiles and convert them to a python dictionary. It also provides functions to extract or remove a feature from the shapefile dictionary.

#### 4.2.2.1 Reading shapefiles

Shapefiles are converted to a python dictionary. The dictionary has information about the number of features as well as the coordinates of the points. The output dictionary SHPdata looks like this:

```
SHPdata['Name'] = ['nameFeature1', 'nameFeature2', 'nameFeature3']
SHPdata['x'] = [xCoordsssFeature1, xCoordsssFeature2, xCoordsssFeature3]
SHPdata['y'] = [yCoordsssFeature1, yCoordsssFeature2, yCoordsssFeature3]
SHPdata['z'] = [zerosss, zerosss, zerosss]
SHPdata['Start'] = [indexStartFeature1, indexStartFeature2, indexStartFeature3]
SHPdata['Length'] = [lenghtFeature1, lenghtFeature2, lenghtFeature3]
SHPdata['sks'] = 'ProjectionInformation'
```

A description of the functions is available in *in2Trans.shpConversion*

## 4.3 in3Utils: Various Utilities Modules

### 4.3.1 geoTrans

The *in3Utils.geoTrans* module provides useful functions to operate transformations, comparison or interpolation on rasters, lines, points... Further information about the available functions can be found in *in3Utils.geoTrans*

### 4.3.2 Generate Topography

The *in3Utils.generateTopo* module provides functions to generate DEM files for idealized/generic topographies that can be used for snow avalanche simulations.

This module can generate the following topographies:

- flat plane (FP)
- inclined plane of constant slope (IP)
- parabola - parabolic slope transitioning into a flat foreland (PF)
- hockey stick - inclined plane of constant slope with a smooth transition to a flat foreland (HS)
- bowl-shaped topography (BL)

- helix-shaped topography (HX)
- pyramid-shaped topography (PY)

Extra features can be added to the above topographies:

- **a channel can be introduced (then set `channel=True`).**  
This channel can also be set to widen towards the top and bottom of the channel (then set `narrowing=True`). The channel can be added by either adding a 'channel layer' (max thickness=channel depth) on top of the topography (`topoAdd=True`) or by cutting them into the original topography (`topoAdd=False`).
- **in case of the parabola topography, a dam can be added by setting `dam=True`.**

This module returns a 3D plot of the generated topography as well as an .asc file of the DEM data. The input parameters are defined in the respective configuration file `in3Utils.generateTopoCfg.ini`. Detailed information on the individual functions used to create the topographies can be found in [in3Utils.generateTopo](#)

#### 4.3.2.1 To run generateTopo

- first go to `AvaFrame/avaframe`
- copy `in3Utils/generateTopoCfg.ini` to `in3Utils/local_generateTopoCfg.ini` and set the desired parameter values (if not, the default values are used)
- run:

```
python3 runScripts/runGenerateTopo.py
```

#### 4.3.2.2 Theory

Topographies are generated using inclined and flat planes, parabolas, spheres and circles. Channels are introduced as half-sphere shaped features with smooth transition from the no-channel area to the channel using cumulative distribution functions.

#### 4.3.2.3 Configuration parameters

In the case of the pyramid-shaped topography, the domain extent is defined by the max elevation ( $z_0$  - elevation of the apex point) and the slope of the pyramid facets (meanAlpha)

##### Domain parameters:

- dx**  
DEM spatial resolution [m]
- xEnd**  
total horizontal extent of the domain [m]
- yEnd**  
total vertical extent of the domain [m]

##### Topography parameters:

- flens**  
distance to the point where the slope transitions into a flat plane [m]
- meanAlpha**  
slope angle from the max. elevation to the start of the flat plane [°] - or slope of the inclined plane [°]

**C**  
total fall height [m]

**rBowl**  
bowl radius [m]

**rHelix**  
radius for helix [m]

**z0**  
max elevation [m]

**zElev**  
elevation of the flat plane [m]

**rCirc**  
radius of the smoothing circle [m]

**demType**  
topography types (FP, IP, PF, HS, BL, HX, PY - explanation given in the introductory description)

**flatx**  
extent of the flat foreland for the pyramid in x

**flaty**  
extent of the flat foreland for the pyramid in y

**phi**  
rotation angle for the pyramid

**Flags for channels and plotting:**

**channel**  
True - introduce channel; False - no channel

**narrowing**  
True - channel is wide at start and end and narrow in the middle part; False - channel is uniform

**topoAdd**  
True - add channel layer; False: cut channel into original topography;

**flagRot**  
True - rotate pyramid along z-axis

**Channel parameters:**

**cRadius**  
standard channel radius

**cInit**  
start and end half width of channel that is narrower in the middle part

**cff**  
standard deviation sigma

**cMustart**  
mean mu - represents upper part of the channel

**cMuend**  
mean mu - represents lower part of the channel



### 4.3.3 Get Release Area

`in3Utils.getReleaseArea` generates a release area for a topography created with `in3Utils.generateTopo`, this function is available for the following topographies:

- flat plane (FP)
- inclined plane (IP)
- parabola (PF)
- hockey stick (HS)

The release areas are defined as rectangular features build by four corner points, which are based on the following conditions:

- prescribed vertical stretch in meters (difference in altitude)
- prescribed volume of the release area
- lower margin is located where the slope angle falls below 30°
- if slope does not fall below 30 °, the upper margin is located xStart away from the upper margin of the DEM

The release areas can be saved as shapefile, .nxyz and .txt file. The required input parameters can be set in the respective configuration files `in3Utils/getReleaseAreaCfg.ini` and `in3Utils.generateTopoCfg.ini`. Detailed information on the individual functions used to create the release areas can be found in `in3Utils.getReleaseArea`

#### 4.3.3.1 To run getReleaseArea

Following these steps, you can generate an avalanche test case including a DEM and a simple release area.

- first go to `AvaFrame/avaframe`
- copy `in3Utils/generateTopoCfg.ini` and `in3Utils/getReleaseAreaCfg.ini` to `in3Utils/local_generateTopoCfg.ini` and `in3Utils/local_getReleaseAreaCfg.ini` and set desired parameter values (if not, the default values are used)
- run:

```
python3 runGenProjTopoRelease.py
```

#### Parameters:

- hr**  
release area vertical stretch [m]
- vol**  
volume of snow in release area [m3]
- dh**  
release snow thickness [m]
- xStart**  
upper margin of the release area distance in x from origin [m]
- lenP**  
number of release area polygon points
- outputtxt**  
True - copy the output to txt file

**xExtent**

horizontal extent of release area for flat plane

**alphaStop**

slope angle that defines lower margin of release area

**relNo**

number of release area for name

**relName**

name of release area feature in shapefile

### 4.3.4 Initialize Project

`in3Utils.initializeProject` provides functions to initialize a project, create the required directory structure and delete specified files or directories.

The main function `in3Utils.initializeProject.initializeFolderStruct()`, creates the folder structure required to perform avalanche simulations:

```
NameOfAvalanche/  
  Inputs/  
    ENT/          - entrainment areas  
    LINES/        - avalanche paths  
    POINTS/       - split points  
    REL/          - release area scenario  
    RES/          - resistance areas  
    SECREL/ - secondary release areas  
    .asc          - DEM  
  Outputs/  
  Work/
```

The path to this folder is specified in the configuration file `avaframeCfg.ini`, with the parameter `avalancheDir`.

#### 4.3.4.1 To run

- first go to `AvaFrame/avaframe`
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- run:

```
python3 runInitializeProject.py
```

### 4.3.5 fileHandlerUtils

`in3Utils.fileHandlerUtils` gathers useful functions to create directories, read log files, extract information from logs, fetch and export data and fetch simulation info into a `dataFrame`. Details on these functions can be found in `in3Utils.fileHandlerUtils`.

## ANALYSIS/HELPER MODULES

### 5.1 ana1Tests: Testing

In this module we gather tests with analytical or semi-analytical solution. These tests are used to evaluate the precision and accuracy of the com1DFA numerical solution.

#### 5.1.1 Deviation/difference/error computation

In order to either assess the accuracy of a numerical method or to compare results it will be necessary to compute errors. The methods used to compute deviations are described in this section.

Two deviation/difference/error measures are used. The first one is based on the  $\mathcal{L}_{max}$  norm (uniform norm), the second on the Euclidean norm ( $\mathcal{L}_2$  norm). In both cases, the aim is to measure the deviation between a numerical solution and reference solution on an interval (one or two dimensional). Let  $f_{num}$  be the numerical solution and  $f_{ref}$  the reference solution defined on an interval  $\Omega$ . The deviation is defined by  $\epsilon(x) = f_{num}(x) - f_{ref}(x)$

##### 5.1.1.1 Uniform norm

The  $\mathcal{L}_{max}$  norm measures the largest absolute value:

$$\mathcal{L}_{max}(\epsilon(x)) = \sup_{x \in \Omega} (|\epsilon(x)|)$$

This will give an idea of the largest difference between the two solutions. It can be applied to one or two dimensional results.

It can also be normalized by dividing the uniform norm of the deviation by the uniform norm of the reference:

$$\frac{\mathcal{L}_{max}(\epsilon(x))}{\mathcal{L}_{max}(f_{ref}(x))}$$

##### 5.1.1.2 Euclidean norm

The  $\mathcal{L}_2$  norm is defined by:

$$\mathcal{L}_2(\epsilon) = \int_{x \in \Omega} ||\epsilon(x)||^2 dx$$

This norm will give an overall measure of the deviations. It is useful to normalize the norm of the deviation either by dividing with the norm of the reference solution:

$$\frac{\mathcal{L}_2(\epsilon(x))}{\mathcal{L}_2(f_{ref}(x))}$$

or by the measure of the interval ( $\mathcal{L}_2(1) = \int_{x \in \Omega} dx$ ):

$$\sqrt{\frac{\mathcal{L}_2(\epsilon(x))}{\mathcal{L}_2(1)}}$$

The first normalization approach will give a relative deviation in % whereas the second will give an average deviation of  $f$  on  $\Omega$ .

### 5.1.2 Dambreak test

The Dambreak test compares the results of DFA simulations to the analytical solution of the “dam break” problem. In this test, a granular mass (Coulomb material) is suddenly released from rest on an inclined plane. In the case of a thickness integrated model as derived by Savage and Hutter (e.g. in [HSSN93]), an analytical solution exists. This solution is described in [FM13] and corresponds to a Riemann problem.

The problem considered has the following initial conditions:

$$(h, \mathbf{u})(x, t = 0) = \begin{cases} (h_0, \mathbf{0}), & \text{if } x \leq 0 \\ (0, \mathbf{0}), & \text{if } x > 0 \end{cases}$$

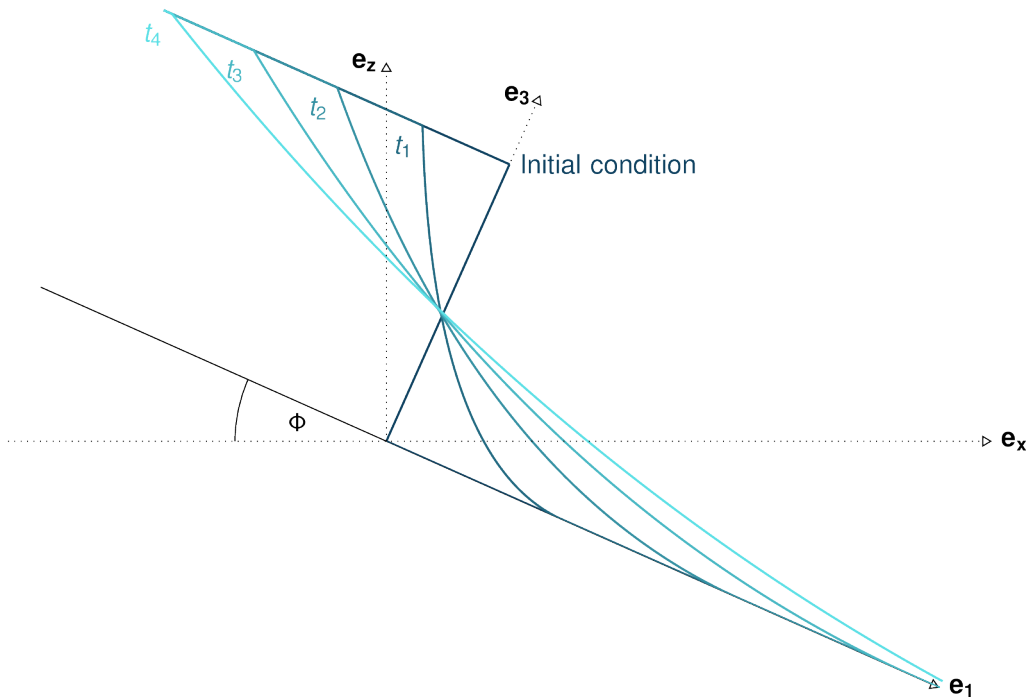


Fig. 5.1: Dam break theoretical evolution

The functions computing the analytical solution and comparing it to the simulation results can be found in `ana1Tests.damBreak` and `ana1Tests.analysisTools`. Plotting routines are located in `out3Plot.outAna1Plots`. The input data for this test case can be found in `data/avaDamBreak`.

This test produces a summary figure combining a comparison between the analytical solution and simulation result (cross cut along the flow direction) as well as a map view and an error measure plot as shown in Fig. 5.2.

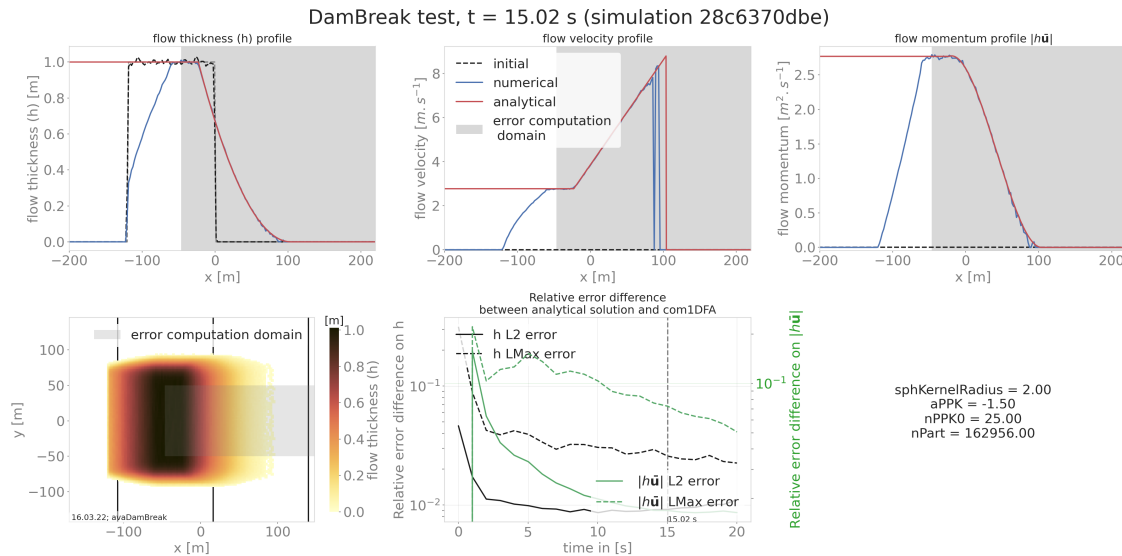


Fig. 5.2: Summary figure produced by the damBreak test (here for a cell size of 2m)

Another optional result is the comparison cross cut figure for all saved time steps as shown in the following animated figure.

Fig. 5.3: Time evolution of the flow thickness, velocity and momentum

### 5.1.2.1 To run

An example on how to apply this test is provided in `runScripts/runDamBreak` and `runScripts/runAnalyzeDamBreak`. The required input files are located in `data/avaDamBreak` (including the configuration file `data/avaDamBreak/Inputs/damBreak_com1DFACfg.ini`). In this configuration file, there is a specific section 'DAMBREAK' providing the required input parameters to compute the analytical solution. In order to run the test example:

- in `AvaFrame/avaframe` run:

```
python3 runScripts/runDamBreak.py
python3 runScripts/runAnalyzeDamBreak.py
```

## 5.1.3 Similarity solution

The similarity solution is one of the few cases where a semi-analytic solution can be derived for solving the thickness integrated equations. It is a useful test case for validating simulation results coming from the dense flow avalanche computational module. This semi-analytic solution can be derived under very strict conditions and making one major assumption on the shape of the solution (symmetry/anti-symmetry of the solution around the x and y axis). The full development of the conditions and assumptions as well as the derivation of the solution is presented in details in [HSSN93]. The term semi-analytic is here used because the method enables to transform the PDE (partial differential equation) of the problem into an ODE using a similarity analysis method. Solving the ODE still requires a numerical integration but this last one is more accurate (when conducted properly) and requires less computation power than solving the PDE.

In this problem, we consider an avalanche governed by a dry friction law (Coulomb friction) flowing down an inclined plane. The released mass is initially distributed in an ellipse with a parabolic depth shape. This mass is suddenly released at  $t = 0$  and flows down the inclined plane.

The `ana1Tests.simiSol` module provides functions to compute the semi-analytic solution and to compare it to the output from the DFA computational module as well as some plotting routines to visualize this solution in `out3Plot.outAna1Plots`.

Comparing the results from the DFA module to the similarity solution leads to the following plots:

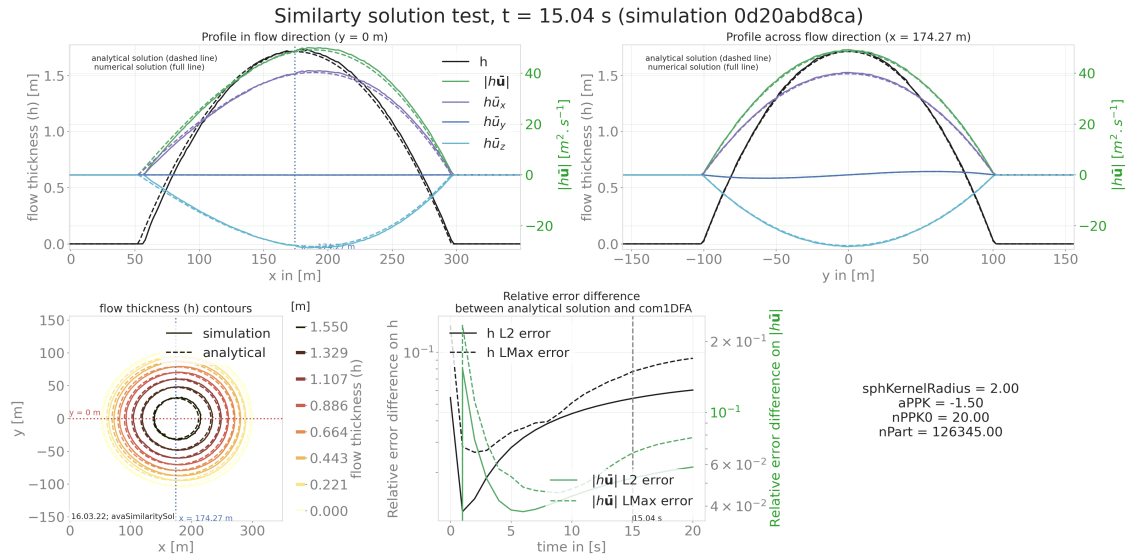


Fig. 5.4: Summary figure produced by the simiSol test (here for a cell size of 2m)

Fig. 5.5: Time evolution of the flow thickness contours in the x, y domain

Fig. 5.6: Time evolution of the profile in and across flow direction

### 5.1.3.1 To run similarity solution

A workflow example is given in `runScripts/runSimilaritySol`, where the semi-analytical solution is computed and avalanche simulations are performed and both results are then compared. The input data for this example can be found in `data/avaSimilaritySol` with the configuration settings of `com1DFA` including a section 'SIMISOL' (see `data/avaSimilaritySol/Inputs/simiSol_com1DFACfg.ini`).

The `out3Plot.outAna1Plots` function generate profile plots for the flow thickness and momentum in both flow and cross flow directions. The simulation results are plotted alongside the analytical solution for the given time step.

### 5.1.4 Energy line test

The Energy line test compares the results of the DFA simulation to a geometrical solution that is related to the total energy of the system. Solely considering Coulomb friction this solution is motivated by the first principle of energy conservation along a simplified topography. Here friction force only depends on the slope angle. The analytical runout is the intersection of the path profile with the geometrical ( $\alpha$ ) line defined by the friction angle ( $\delta$ ). From the geometrical line it is also possible to extract information about the flow mass averaged velocity at any time or position along the path profile.

#### 5.1.4.1 Theory

Applying the energy conservation law to a material block flowing down a slope with Coulomb friction and this between two infinitesimally close time steps reads:

$$\begin{aligned} E_{t+dt}^{tot} - E_t^{tot} &= E_{t+dt}^{kin} + E_{t+dt}^{pot} - (E_t^{kin} + E_t^{pot}) = \delta E_{fric} \\ &= \frac{1}{2}mv_{t+dt}^2 + mgz_{t+dt} - \frac{1}{2}mv_t^2 - mgz_t \\ &= \mathbf{F}_{fric} \cdot \mathbf{dl} = -\mu \|\mathbf{N}\| \frac{d\mathbf{l}}{dl} \cdot \mathbf{dl} = -\mu mg(\mathbf{e}_z \cdot \mathbf{n}) dl \end{aligned}$$

where  $\delta E_{fric}$  is the energy dissipation due to friction,  $\mathbf{N}$  represents the normal (to the slope surface) component of the gravity force,  $\mathbf{n}$  the normal vector to the slope surface and  $\mathbf{dl}$  is the vector representing the distanced traveled by the material between  $t$  and  $t + dt$ . The normal vector reads  $\mathbf{e}_z \cdot \mathbf{n} = \cos(\theta)$ , where  $\theta$  is the slope angle.  $m$  represents the mass of the material,  $g$  the gravity,  $\mu = \tan \delta$  the friction coefficient and friction angle,  $z$ , respectively  $v$  the elevation respectively velocity of the material block. Finally, in the 2D case,  $dl = \frac{ds}{\cos(\theta)}$ , which means that the material is flowing in the steepest slope direction ( $ds$  is the horizontal component of  $\mathbf{dl}$ ).

Integrating the energy conservation between the start and a time  $t$  reads:

$$\begin{aligned} E_t^{tot} - E_{t=0}^{tot} &= \frac{1}{2}mv_t^2 + mgz_t - \frac{1}{2}m0v_{t=0}^2 - mgz_0 \\ &= \int_{t'=0}^{t'=t} \delta E_{fric} dt' = - \int_{s'=s_0}^{s'=s_t} \mu mg ds' = -\mu mg(s_t - 0s_0) \end{aligned}$$

Speaking in terms of altitude, the energy conservation equation can be rewritten:

$$z_0 = z_t + \frac{v_t^2}{2g} + s_t \tan \alpha \quad (5.1)$$

This result is illustrated in the following figure.

Considering a system of material blocks flowing down a slope with to Coulomb friction: we can sum the previous equation Eq.5.1 of each block after weighting it by the block mass. This leads to the mass average energy conservation equation:

$$\bar{z}_0 = \bar{z}_t + \frac{\overline{v_t^2}}{2g} + \bar{s}_t \tan \alpha \quad (5.2)$$

where the mass average  $\bar{a}$  value of a quantity  $a$  is:

$$\bar{a} = \frac{\sum_k m_k a_k}{\sum_k m_k}$$

This means that the mass averaged quantities also follow the same energy conservation law when expressed in terms of altitude. The same figure as in Fig. 5.7 can be drawn for the center of mass profile path.

The aim is to take advantage of this energy conservation line to evaluate the DFA simulation. Computing the mass averaged path profile for the particles in the simulation and comparing it to the  $\alpha$  line allows to compute the error

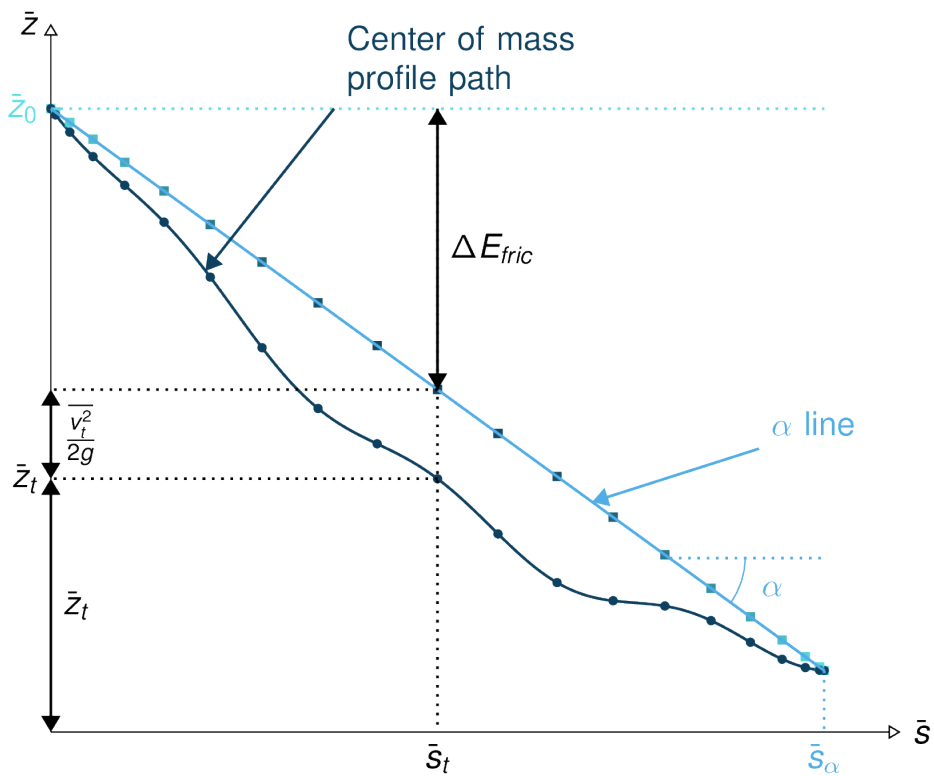
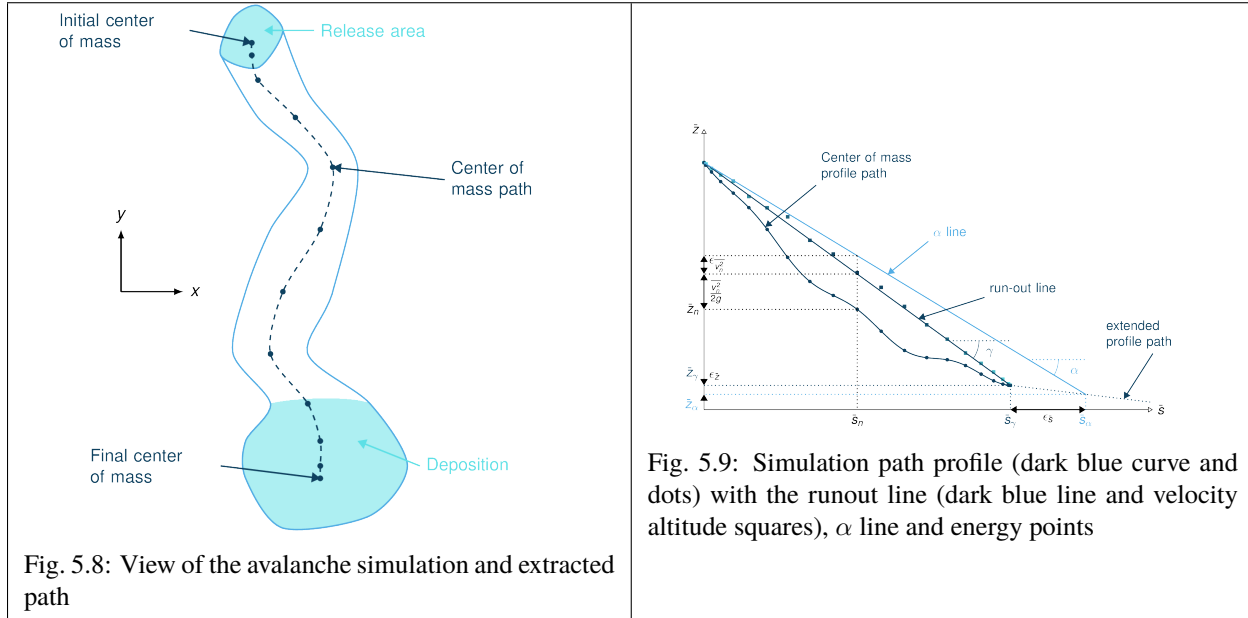


Fig. 5.7: Center of mass profile (dark blue line with the dots) with on top, the energy line (light blue) and the velocity altitude points (colored squared)



compared to the energy line runout. This also applies to the error on the velocity altitude. The following figures illustrate the concept.



From the different mass averaged simulation quantities and the theoretical  $\alpha$  line it is possible to extract four error indicators. The first three related to the runout point defined by the intersection between the  $\alpha$  line and the mass averaged path profile (or its extrapolation if the profile is too short) and the last one is related to the velocity :

- The horizontal distance between the runout point and the end of the path profile defines the  $\epsilon_s = \bar{s}_\gamma - \bar{s}_\alpha$  error in meters.
- The vertical distance between the runout point and the end of the path profile defines the  $\epsilon_z = \bar{z}_\gamma - \bar{z}_\alpha$  error in meters.
- The angle difference between the  $\alpha$  line angle and the DFA simulation runout line defines the  $\epsilon_\alpha = \gamma - \alpha$  angle error.
- The Root Mean Square Error (RMSE) between the  $\alpha$  line and the DFA simulation energy points defines an error on the velocity altitude  $\frac{\overline{v^2}}{2g}$ .

## Limitations and remarks

It is essential to stay where the assumptions of this test hold. Indeed, one of the important hypotheses when developing the energy solution, is that the material is flowing in the steepest slope direction (i.e. where  $dl = \frac{ds}{\cos(\theta)}$  theta holds). If this hypothesis fails (as illustrated in Fig. 5.10), then it is not possible to develop the analytic energy solution anymore. In the 3D case, the distance vector  $dl$  traveled by the particles reads  $dl = \frac{ds}{\cos(\gamma)}$ , where  $\gamma$  is the angle between the  $dl$  vector and the horizontal plane which can differ from the slope angle  $\theta$ . In this case, the energy solution is not the solution of the problem anymore and can not be used as reference.

It is also possible with this test to observe the effect of terms such as curvature acceleration, artificial viscosity or pressure gradients. The curvature acceleration modifies the friction term (depending on topography curvature and particle velocity). This leads to a mismatch between the energy solution and the DFA simulation. Artificial viscosity can lead to viscous dissipation leading to shorter runouts then what the energy solution predicts. Finally, the effect of the pressure force can be studied, especially the effect of the computation options. The results of this test can also be reused for other purposes or in other test such as in the rotation test described below (*Rotation test*)

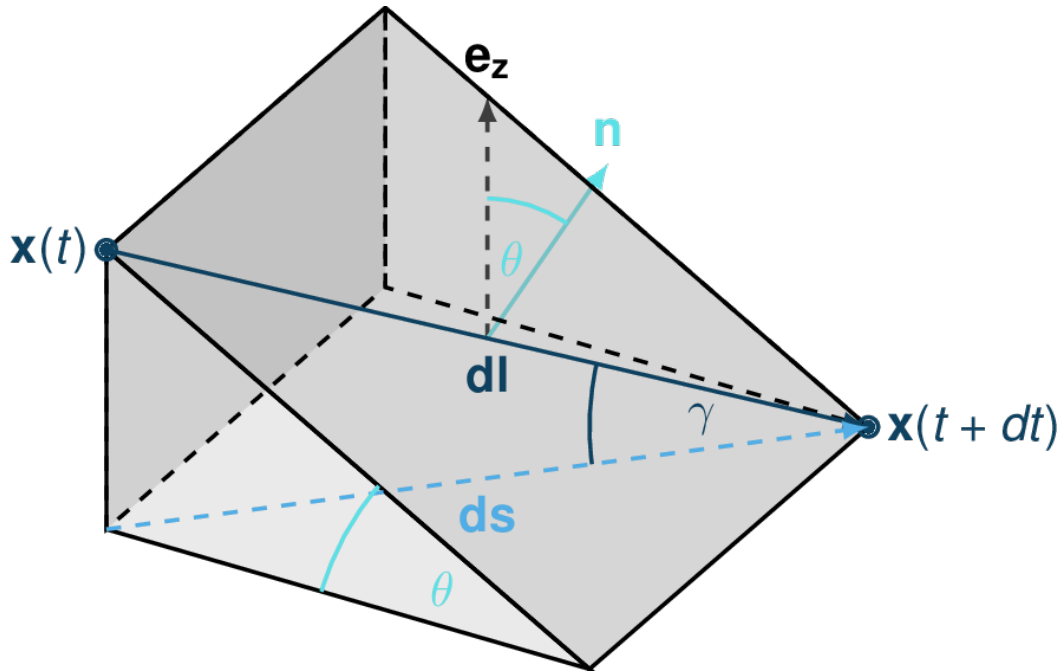


Fig. 5.10: Example of trajectory where the steepest descent path hypothesis fails. The mass point is traveling from  $\mathbf{x}(t)$  to  $\mathbf{x}(t + dt)$ . The slope angle  $\theta$  and travel angle  $\gamma$  are also illustrated. Here  $(\mathbf{e}_z \cdot \mathbf{n})dl = \cos\theta \frac{ds}{\cos\gamma} \neq ds$ .

#### 5.1.4.2 Procedure

First, the DFA simulation is ran (in our case using the com1DFA module) on the desired avalanche, saving the particles (at least the initial and final particles information). Then, the particles mass averaged quantities are computed ( $\bar{x}, \bar{y}, \bar{z}, \bar{s}, \bar{v}^2$ ) to extract a path and path profile. Finally, the mass averaged path profile, the corresponding runout line and the expected  $\alpha$  are displayed and the runout angle and distance errors as well as the velocity altitude error are computed.

#### 5.1.4.3 To run energy line test

A workflow example is given in `runScripts.runEnergyLineTest.py`.

#### 5.1.5 Rotation test

The rotation test aims at verifying that a DFA computation module produces similar results, if not identical, independently of the underlying mesh or grid orientation used for the computation. Indeed, numerical solvers using any sort of mesh based discretization or interpolation method tend to give different results for the same physical problem, boundary conditions and initial conditions if the mesh or grid orientation is different. In this test, the same physical problem is fed to the DFA module changing only the grid orientation. The energy line test is applied to each of the simulations and the runouts are compared. An AIMEC analysis of the peak results (previously rotated so that the results are aligned) is also carried out to give a more detailed idea of the spatial differences between the simulations.

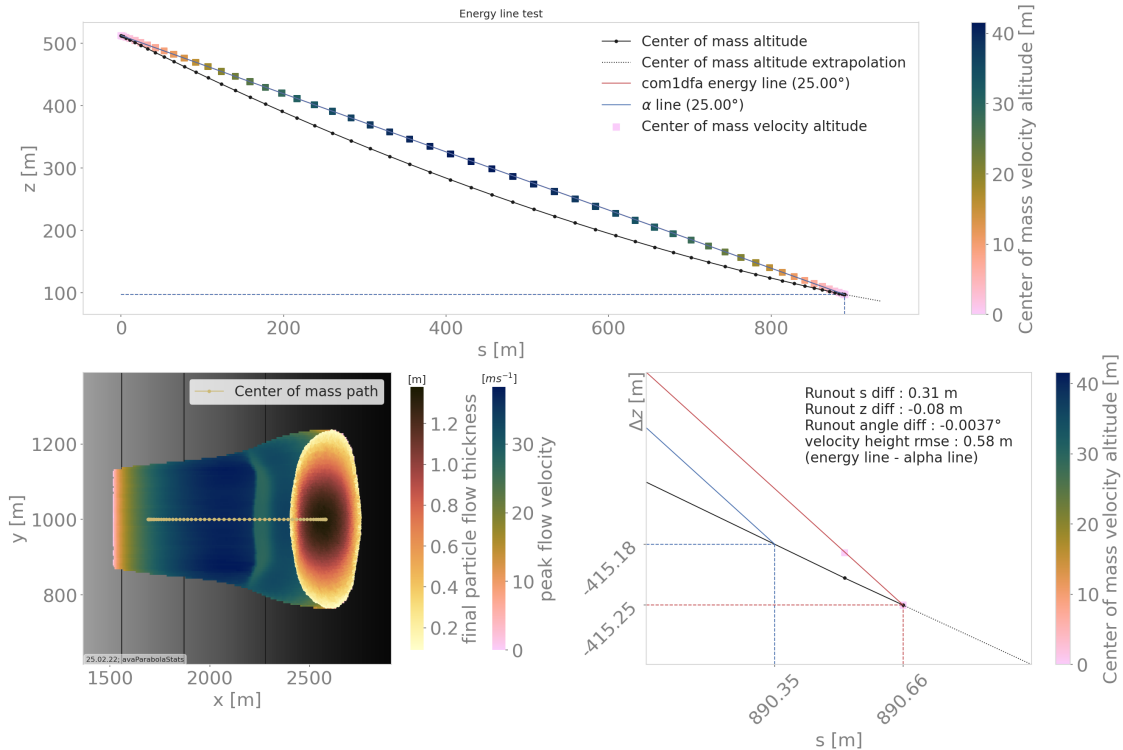


Fig. 5.11: Results from the ana1EnergyLineTest for the avaParabola

### 5.1.5.1 Inputs

This module requires an avalanche directory with inputs for the com1DFA and AIMEC module.

#### com1DFA inputs

- The DEM needs to have a center of symmetry located at the origin (0, 0).
- There needs to be multiple release features also symmetric in regard of the origin. These features need to be named relXXX.shp, where XXX is the rotation angle in the clockwise direction compared to the  $[-\infty, 0]$  x axis.
- There can also be some entrainment or resistance features (also satisfying the symmetry criterion).

#### ana3AIMEC inputs

- The line describing the avalanche path for the reference simulation
- A split point.

The most simple example is a bowl centered on (0, 0) with some circular release features all located at the same distance from the origin and a ring shaped entrainment feature (as shown on Fig. 5.12).

One then needs to specify the the com1DFA configuration (through the com1DFACfg.ini and its local version) as well as the AIMEC configuration (through the ana3AIMECCfg.ini and its local version) and the path generation configuration (through the pathGenerationCfg.ini and its local version).

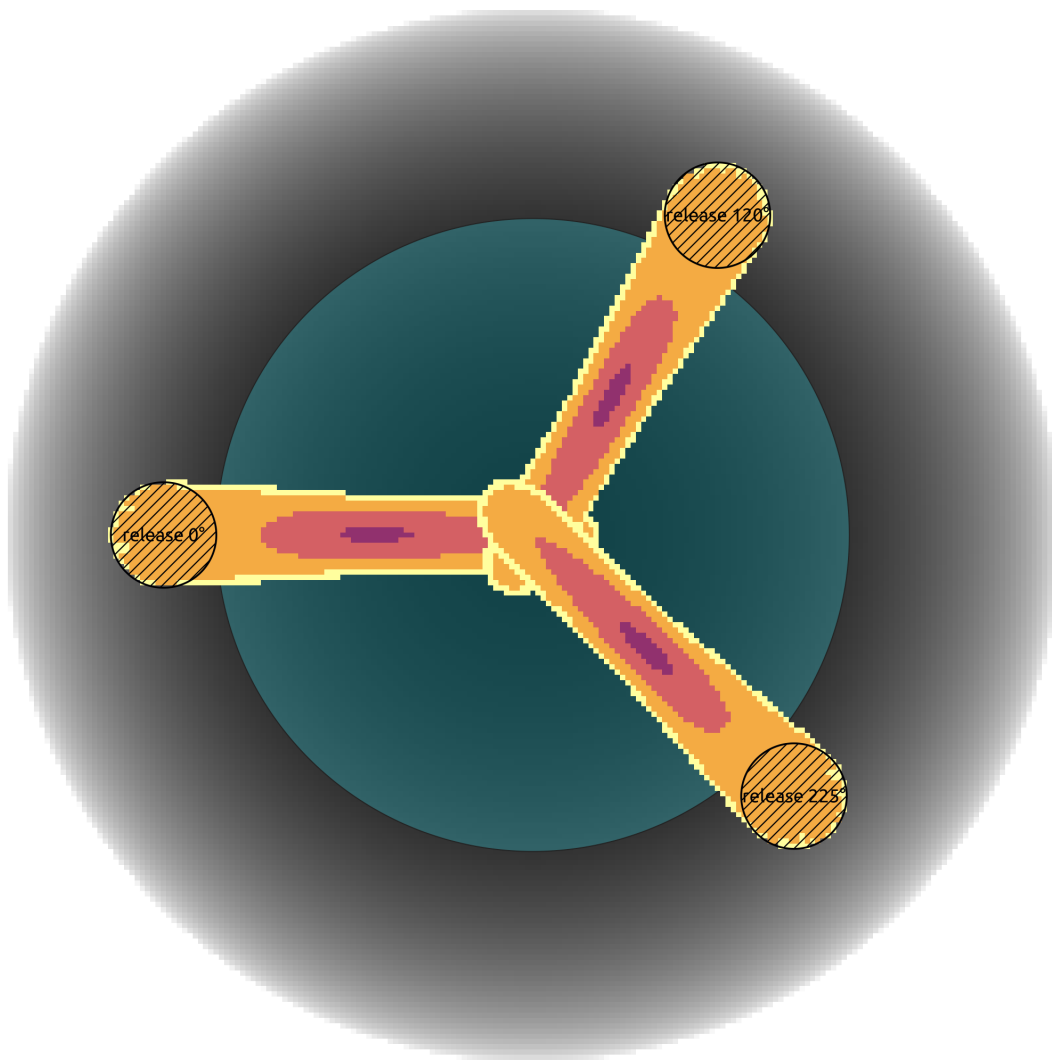


Fig. 5.12: Example of input data used for the rotation test and peak flow thickness result produced by the com1DFA module. In the background the bowl DEM is shown, in dashed areas (small circles) the three release features and in blue (big circle) the entrainment feature. One release feature is aligned with the x axis (rel0), the other is rotated 120° clockwise (rel120) and the last on 225° clockwise (rel225). All three input scenarios are identical (identical in terms of extend and release mass).

### 5.1.5.2 To run rotation test

A workflow example is given in `runScripts.runRotationTest.py`. Some example input data is given in `avaTripleBowl` (bowl topography).

## 5.2 ana3AIMEC: Aimec

**ana3AIMEC** (Automated Indicator based Model Evaluation and Comparison, [Fis13]) is a post-processing module to analyze and compare results from avalanche simulations. For this purpose, avalanche simulation results are transformed into an avalanche *thalweg* following coordinate system (see Fig. 5.13, Fig. 5.14, Fig. 5.15), which enables the comparison of different simulations (for example varying parameter sets, or performed with different models) of the same avalanche catchment, in a standardized way.

In `AvaFrame/avaframe/runScripts`, two different run scripts are provided and show examples on how the post-processing module *ana3AIMEC* can be used:

- full Aimec analysis for simulation results of one computational module (from 1 simulation to x simulations). `runScripts.runAna3AIMEC.runAna3AIMEC()`
- using Aimec to compare the results of two different computational modules (one reference for the reference computational module and multiple simulations in the other computational module). `runScripts.runAna3AIMECCompMods.runAna3AIMECCompMods()`

In all cases, one needs to provide a minimum amount of input data. Below is an example workflow for the full Aimec analysis, as provided in `runScripts/runAna3AIMEC.py`:

### 5.2.1 Inputs

- DEM (digital elevation model) as `.asc` file with [ESRI grid format](#)
- avalanche thalweg in LINES (as a shapefile named `NameOfAvalanche/Inputs/LINES/path_aimec.shp`), the line needs to cover the entire affected area but is not allowed to exceed the DEM extent
- results from avalanche simulation (when using results from `com1DFA`, the helper function `ana3AIMEC.dfa2Aimec.mainDfa2Aimec()` in `ana3AIMEC.dfa2Aimec` fetches and prepares the input for Aimec)
- a method to define the reference simulation. By default, an arbitrary simulation is defined as reference. This can be changed in the `ana3AIMEC/local_ana3AIMECCfg.ini` as explained in [Defining the reference simulation](#).
- consider adjusting the default settings to your application in the Aimec configuration (in your local copy of `ana3AIMEC/local_ana3AIMECCfg.ini`) regarding the domain transformation, result types, runout computation and figures

---

**Note:** The spatial resolution of the DEM and its extent can differ from the result raster data. Spatial resolution can also differ between simulations. If this is the case, the spatial resolution of the reference simulation results raster is used (default) or, if provided, the resolution specified in the configuration file (`cellSizeSL`) is used. This is done to ensure that all simulations will be transformed and analyzed using the same spatial resolution.

---

### 5.2.1.1 Optional inputs

There is the option to define a runout area for the analysis, which is based on the slope angle of the avalanche thalweg. This can be defined in the Aimec configuration by setting the *defineRunoutArea* flag to True. The start of runout area is then determined by finding the first point on the thalweg profile where the slope angle falls below the *startOfRunoutAreaAngle* value and all points upslope of the a additionally provided split point are ignored (see Fig. 5.15). This functionality requires an additional input:

- a splitPoint in POINTS (as a shapefile named NameOfAvalanche/Inputs/POINTS/splitPoint.shp), this point is then snapped onto the thalweg based on the shortest horizontal distance

## 5.2.2 Outputs

- output figures in NameOfAvalanche/Outputs/ana3AIMEC/anaMod/
- csv file with the results in NameOfAvalanche/Outputs/ana3AIMEC/anaMod/ (a detailed list of the results is described in [Analyze results](#))

---

**Note:** *anaMod* refers to the computational module that has been used to create the avalanche simulation results and is specified in the **ana3AIMEC** configuration.

---

## 5.2.3 To run

- first go to AvaFrame/avaframe
- in your local copy of ana3AIMEC/ana3AIMECCfg.ini you can adjust the default settings (if not, the standard settings are used)
- enter path to the desired NameOfAvalanche/ folder in your local copy of avaframeCfg.ini
- run:

```
python3 runScripts/runAna3AIMEC.py
```

---

**Note:** In the default configuration, the analysis is performed on the simulation result files located in NameOfAvalanche/Outputs/anaMod/peakFiles, where anaMod is specified in the aimecCfg.ini. There is also the option to directly provide a path to an input directory to the [ana3AIMEC.ana3AIMEC.fullAimecAnalysis\(\)](#). However, the peak field file names need to have a specific format: *A\_B\_C\_D\_E.asc*, where:

- *A - releaseAreaScenario*: refers to the name of the release shape file
  - *B - simulationID*: needs to be unique for the respective simulation
  - *C - simType*: refers to null (no entrainment, no resistance), ent (with entrainment), res (with resistance), entres (with entrainment and resistance)
  - *D - modelType*: can be any descriptive string of the employed model (here dfa for dense flow avalanche)
  - *E - result type*: is pft (peak flow thickness) and pfv (peak flow velocity)
-

## 5.2.4 Theory

AIMEC (Automated Indicator based Model Evaluation and Comparison, [Fis13]) was developed to analyze and compare avalanche simulations. The computational module presented here is inspired from the original AIMEC code. The simulations are analyzed and compared by projecting the results along a chosen poly-line (same line for all the simulations that are compared) called avalanche *thalweg*. The raster data, initially located on a regular and uniform grid (with coordinates  $x$  and  $y$ ) is transformed based on a regular non uniform grid (grid points are not uniformly spaced) that follows the avalanche *thalweg* (with curvilinear coordinates  $(s,l)$ ). This grid can then be “straightened” or “deskewed” in order to plot it in the  $(s,l)$  coordinates system.

The simulation results (two dimensional fields of e.g. peak flow velocities, pressure or flow thickness) are processed in a way that it is possible to compare characteristic values that are directly linked to the flow variables such as maximum peak flow thickness, maximum peak flow velocity or deduced quantities, for example maximum peak pressure, pressure based runout (including direct comparison to possible references, see *Area indicators*) for different simulations. The following figure illustrates the raster transformation process.

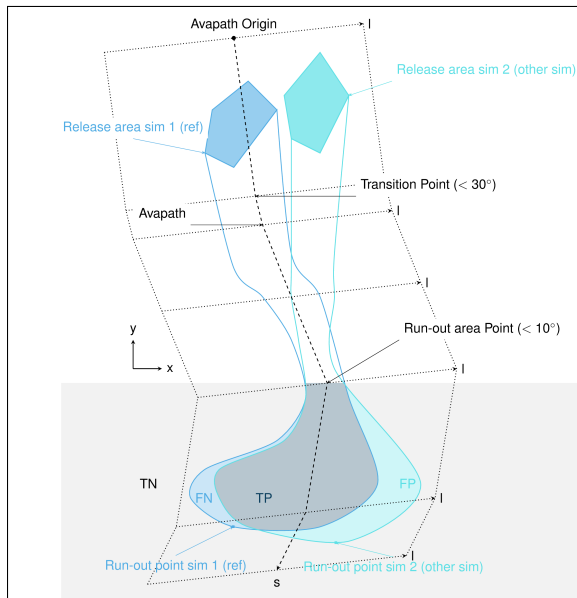


Fig. 5.13: In the real coordinate system  $(x,y)$

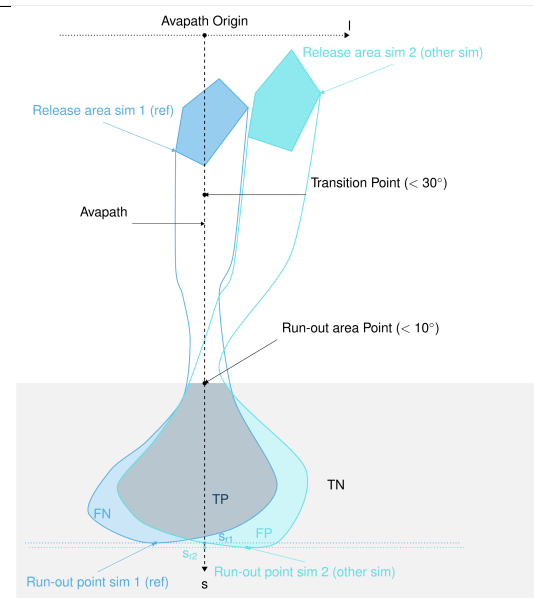


Fig. 5.14: In the new coordinate system  $(s,l)$

Here is the definition of the different indicators and outputs from the AIMEC post-processing process:

### 5.2.4.1 Mean and max values along thalweg

All two dimensional field results (for example peak flow velocities / pressure or flow thickness) can be transformed into the curvilinear system using the previously described method. The maximum and average values of those fields are computed in each cross-section ( $l$  direction) along the thalweg. For example the maximum  $A_{cross}^{max}(s)$  and average  $\bar{A}_{cross}(s)$  of the two dimensional distribution  $A(s,l)$  is:

$$A_{cross}^{max}(s) = \max_{\forall l \in [-\frac{w}{2}, \frac{w}{2}]} A(s,l) \quad \text{and} \quad \bar{A}_{cross}(s) = \frac{1}{w} \int_{-\frac{w}{2}}^{\frac{w}{2}} A(s,l) dl$$

### 5.2.4.2 Runout point

The runout point is always given with respect to a peak result field ( $A(s, l)$  which could be peak pressure or flow thickness, etc.) and a threshold value ( $A_{lim} > 0$ ). The runout point ( $s = s_{runout}$ ) and the respective  $(x_{runout}, y_{runout})$  in the original coordinate system, correspond to the last point in flow direction where the chosen peak result  $A_{cross}^{max}(s)$  is above the threshold value  $A_{lim}$ .

---

**Note:** It is very important to note that the position of the runout point depends on the chosen threshold value and peak result field. It is also possible to use  $\bar{A}_{cross}(s) > A_{lim}$  instead of  $A_{cross}^{max}(s) > A_{lim}$  to define the runout point.

---

### 5.2.4.3 Runout length

This length depends on what is considered to be the beginning of the avalanche  $s = s_{start}$ . It can be related to the release area, to the transition point (first point where the slope angle is below  $30^\circ$ ), to the runout area point (first point where the slope angle is below  $10^\circ$ ) or in a similar way as  $s = s_{runout}$  is defined saying that  $s = s_{start}$  is the first point where  $A_{cross}^{max}(s) > A_{lim}$  (this is the option implemented in `ana3AIMEC.ana3AIMEC.py`). The runout length is then defined as  $L = s_{runout} - s_{start}$ . In the analysis results, this is called *deltaSXY*, whereas  $s_{runout}$  gives the length measured from the start of the thalweg to the runout point.

### 5.2.4.4 Mean and max indicators

From the maximum values along path of the distribution  $A(s, l)$  calculated in module `Ana3AIMEC:Mean` and max values along path, it is possible to calculate the global maximum (MMA) and average maximum (AMA) values of the two dimensional distribution  $A(s, l)$ :

$$MMA = \max_{\forall s \in [s_{start}, s_{runout}]} A_{cross}^{max}(s) \quad \text{and} \quad AMA = \frac{1}{s_{runout} - s_{start}} \int_{s_{start}}^{s_{runout}} A_{cross}^{max}(s) ds$$

### 5.2.4.5 Area indicators

When comparing the runout area (corresponding to a given threshold  $A_{cross}^{max}(s) > A_{Lim}$ ) of two simulations, it is possible to distinguish four different zones. For example, if the first simulation (sim1) is taken as reference and if True corresponds to the assertion that the avalanche reached this zone (reached means  $A_{cross}^{max}(s) > A_{Lim}$ ) and False the avalanche did not reached this zone, those four zones are:

- TP (true positive) zone: green zone on Fig. 5.14 , sim1 = True sim2 = True
- FP (false positive) zone: blue zone on Fig. 5.14 , sim1 = False sim2 = True
- FN (false negative) zone: red zone on Fig. 5.14 , sim1 = True sim2 = False
- TN (true negative) zone: gray zone on Fig. 5.14 , sim1 = False sim2 = False

The two simulations are identical (in the runout zone) when the area of both FP and FN are zero. In order to provide a normalized number describing the difference between two simulations, the area of the different zones is normalized by the area of the reference simulation  $A_{ref} = A_{TP} + A_{FP}$ . This leads to the 4 area indicators:

- $\alpha_{TP} = A_{TP}/A_{ref}$ , which is 1 if sim2 covers at least the reference
- $\alpha_{FP} = A_{FP}/A_{ref}$ , which is a positive value if sim2 covers an area outside of the reference
- $\alpha_{FN} = A_{FN}/A_{ref}$ , which is a positive value if the reference covers an area outside of sim2
- $\alpha_{TN} = A_{TN}/A_{ref}$  (this value may not be of great interest because it depends on the width and length of the entire domain of the result rasters (s,l))



Identical simulations (in the runout zone) lead to  $\alpha_{TP} = 1$ ,  $\alpha_{FP} = 0$  and  $\alpha_{FN} = 0$

#### 5.2.4.6 Mass indicators

From the analysis of the release mass ( $m_r$  at the beginning, i.e  $t = t_{ini}$ ), total mass ( $m_t$  at the end, i.e  $t = t_{end}$ ) and entrained mass ( $m_e$  at the end, i.e  $t = t_{end}$ ) it is possible to calculate the growth index  $GI$  and growth gradient  $GG$  of the avalanche:

$$GI = \frac{m_t}{m_r} = \frac{m_r + m_e}{m_r} \quad \text{and} \quad GG = \frac{m_r + m_e}{t_{end} - t_{ini}}$$

Time evolution of the total mass and entrained one are also analyzed.

### 5.2.5 Procedure

This section describes how the theory is implemented in the ana3AIMEC module.

#### 5.2.5.1 Defining the reference simulation

To apply a complete Aimec analysis, a reference simulation needs to be defined. The analysis of the other simulations will be compared to the one of the reference simulation. The reference simulation can be determined by its name (or part of the name) or based on some configuration parameter and value (to adjust in the local copy of `ana3AIMEC/ana3AIMECCfg.ini`) if it comes from the `com1DFA` module (or any computational module that provides a configuration):

- **based on simulation name**  
one needs to provide a non-empty string in the AIMEC configuration file for the `referenceSimName` parameter. This string can be a part or the full name of the reference simulation. A warning is raised if several simulation match the criterion (can happen if part of the name is given) and the first simulation found is arbitrarily taken as reference.
- **based on some configuration parameter**  
one needs to provide a `varParList` (parameter or list of parameters separated by |) in the AIMEC configuration file as well as the desired sorting order (`ascendingOrder`, True by default) for these parameters and optionally a `referenceSimValue`. The simulations are first going to be sorted according to `varParList` and `ascendingOrder` (this is done by the pandas function `sort_values`). The reference simulation is either the first simulation found after sorting if no `referenceSimValue` or the simulation matching the `referenceSimValue` provided (closest value if the parameter is a float or integer, case insensitive for strings). If multiple simulations match the criterion, the first simulation is taken as reference and a warning is raised.

#### 5.2.5.2 Perform thalweg-domain transformation

First, the transformation from (x,y) coordinate system (where the original rasters lie in) to (s,l) coordinate system is applied given a new domain width (`domainWidth`). This is done by `ana3AIMEC.aimecTools.makeDomainTransfo()`. A new grid corresponding to the new domain (following the avalanche thalweg) is built with a cell size defined by the reference simulation (default) or `cellSizeSL` if provided. The transformation information are stored in a `rasterTransfo` dictionary (see `ana3AIMEC.aimecTools.makeDomainTransfo()` for more details).

### 5.2.5.3 Assign data

The simulation results (for example peak velocities / pressure or flow thickness) are projected on the new grid using the transformation information by `ana3AIMEC.aimecTools.assignData()`. The projected results are stored in the `newRasters` dictionary.

This results in the following plot:

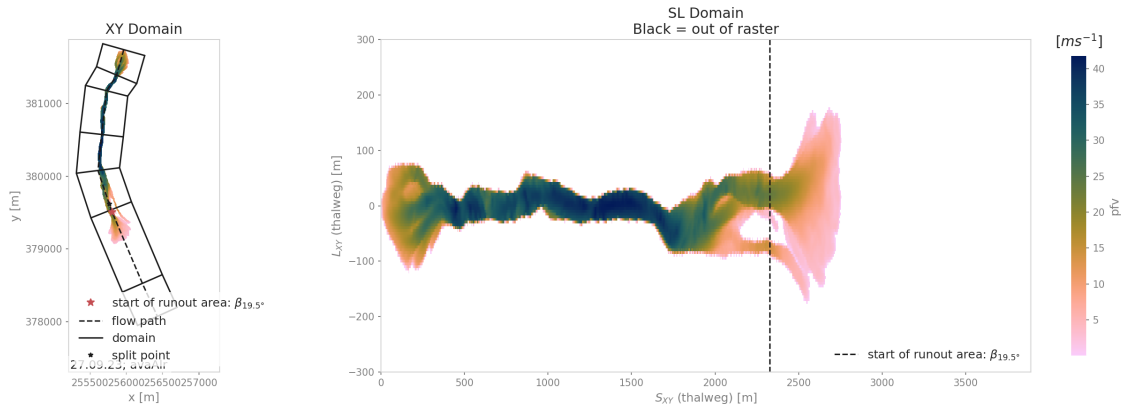


Fig. 5.15: Alr avalanche coordinate transformation and peak pressure field reprojection.

### 5.2.5.4 Analyze results

Calculates the different indicators described in the *Theory* section for a given threshold. The threshold can be based on pressure, flow thickness, ... (this needs to be specified in the configuration file). Returns a `resAnalysisDF` dataFrame with the analysis results (see `ana3AIMEC.ana3AIMEC.postProcessAIMEC()` for more details). In this dataFrame there are multiple columns, one for each result from the analysis (one column for runout length, one for MMA, MAM...) and one row for each simulation analyzed.

### 5.2.5.5 Plot and save results

Plots and saves the desired figures and writes results in `resAnalysisDF` to a csv file. By default, Aimec saves five summary plots plus three plots per simulation comparing the numerical simulations to the reference. The five summary plots are:

### 5.2.5.6 Domain overview plots

- “DomainTransformation” shows the computational domain (xy) on the left and new thalweg-following domain (sl) on the right (Fig. 5.15)
- “referenceFields” shows the peak pressure, flow thickness and flow velocity in the sl domain

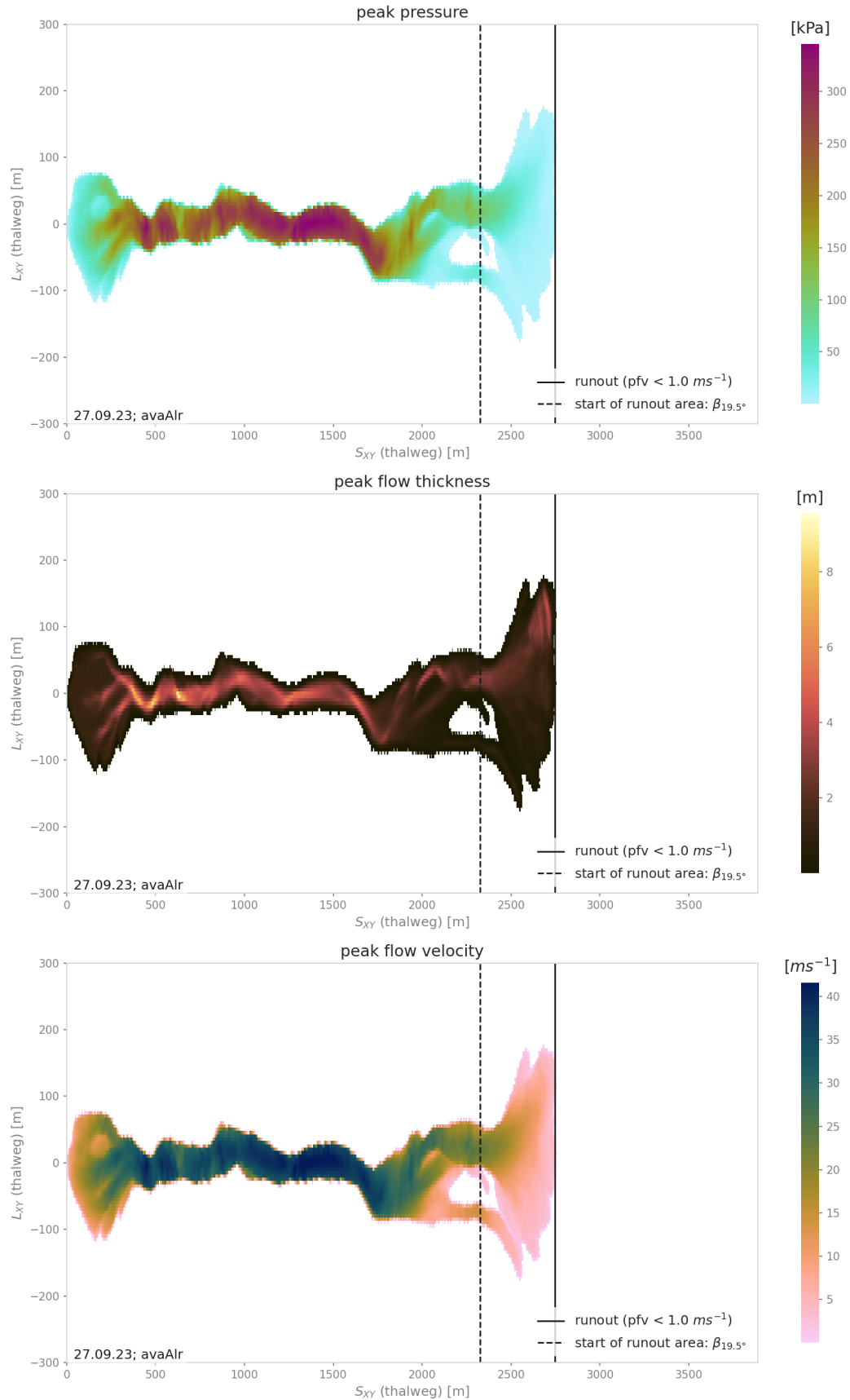


Fig. 5.16: Reference peak fields

### 5.2.5.7 Analysis summary plots

- “slComparisonStats” shows the peak field specified in the configuration and computed runout points in the top panel, whereas in the bottom panel the statistics in terms of cross maximum peak value along profile are plotted (mean, max and quantiles)

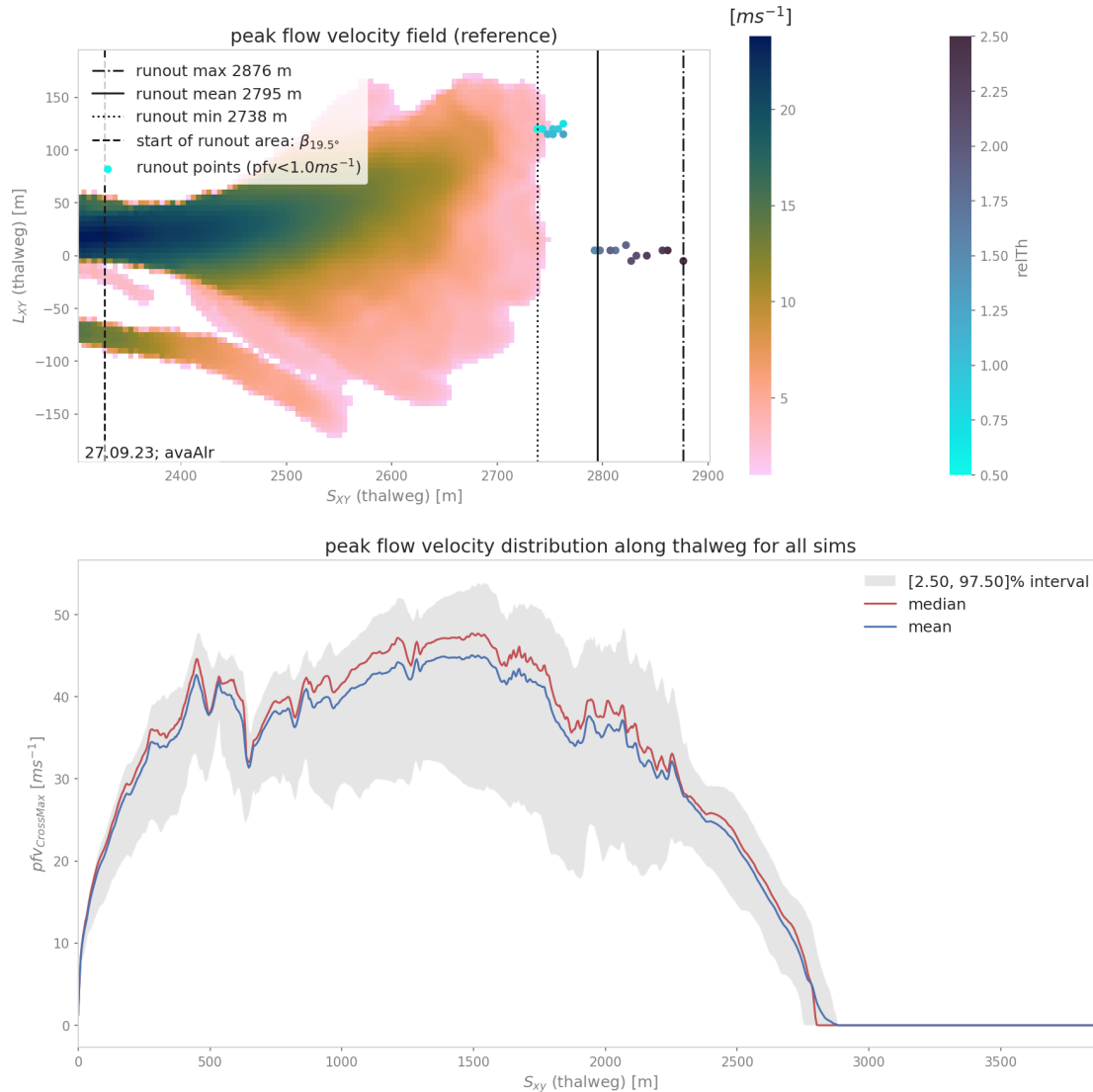


Fig. 5.17: Reference peak field runout area with runout points and distribution of cross max values of peak field along thalweg

- if only two simulations are compared, additionally the difference between the two simulations in terms of peak values along profile are shown for peak pressure, peak flow velocity and peak flow thickness in “slComparison”.
- “ROC” shows the normalized area difference between reference and other simulations.
- “deltaSXvsrunoutAngle” and “pfvFieldMaxvspftFieldMax” show the distribution of these values for all the simulations. There is the option to indicate scenarios in these plots using the available simu-

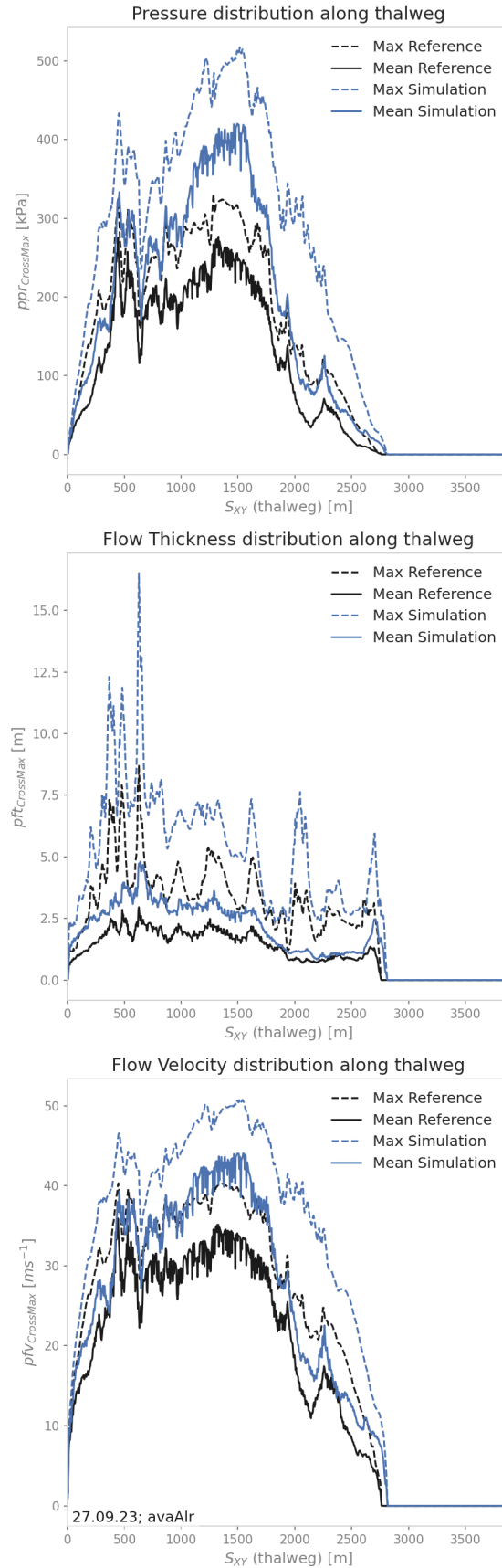


Fig. 5.18: Maximum peak fields comparison between two simulations

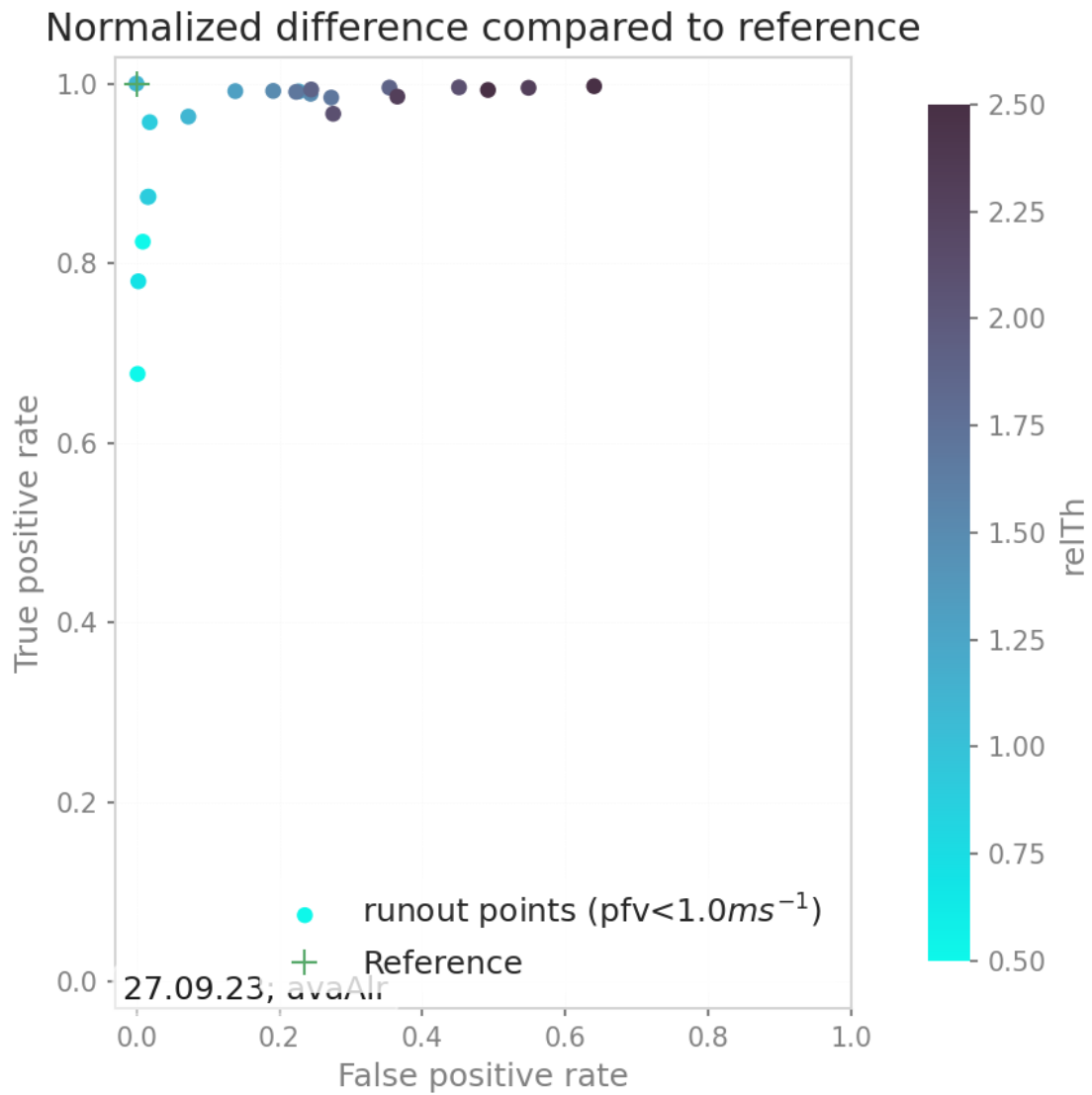


Fig. 5.19: Area analysis plot

lation parameters. Actually other parameter vs parameter plots can be added by setting the parameter names in the **ana3AIMEC** configuration in the section called PLOTS.

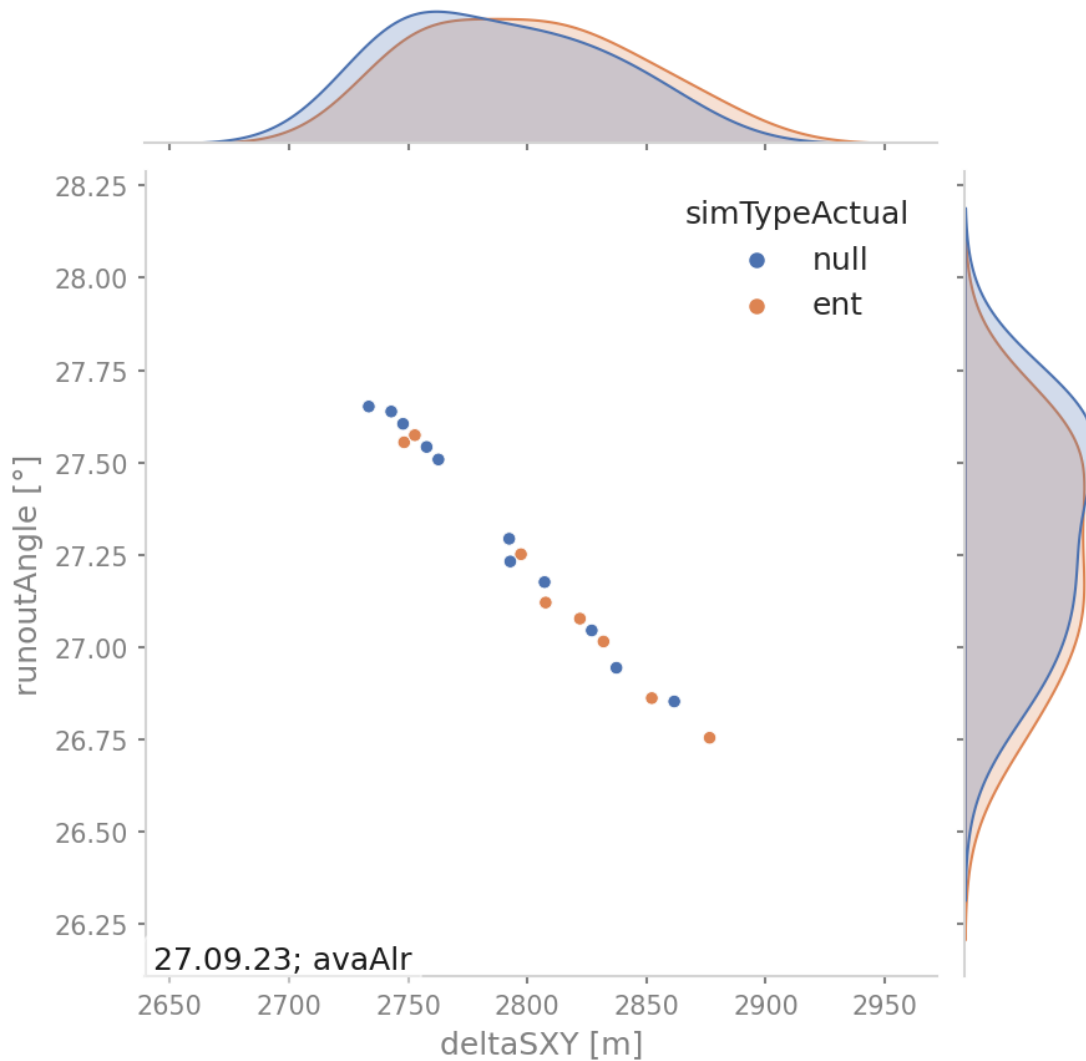


Fig. 5.20: DeltaSXY vs runout angle of all sims colorcoded using the simType

#### 5.2.5.8 Analysis on simulation level plots

- “thalwegAltitude” shows the cross section max values of peak flow thickness as bars on top of the thalweg elevation profile. The colors of the bars correspond to the cross section max values of the respective peak flow velocity. Using the specified *velocityThresholdValue* (aimecCfg.ini section PLOTS), runout length /  $\Delta S_{xy}$ , elevation drop /  $\Delta z$  and the corresponding runout angle  $\alpha$  are computed. The red dots indicate the location of overall max peak flow velocity and thickness and the grey shaded area the runout area.

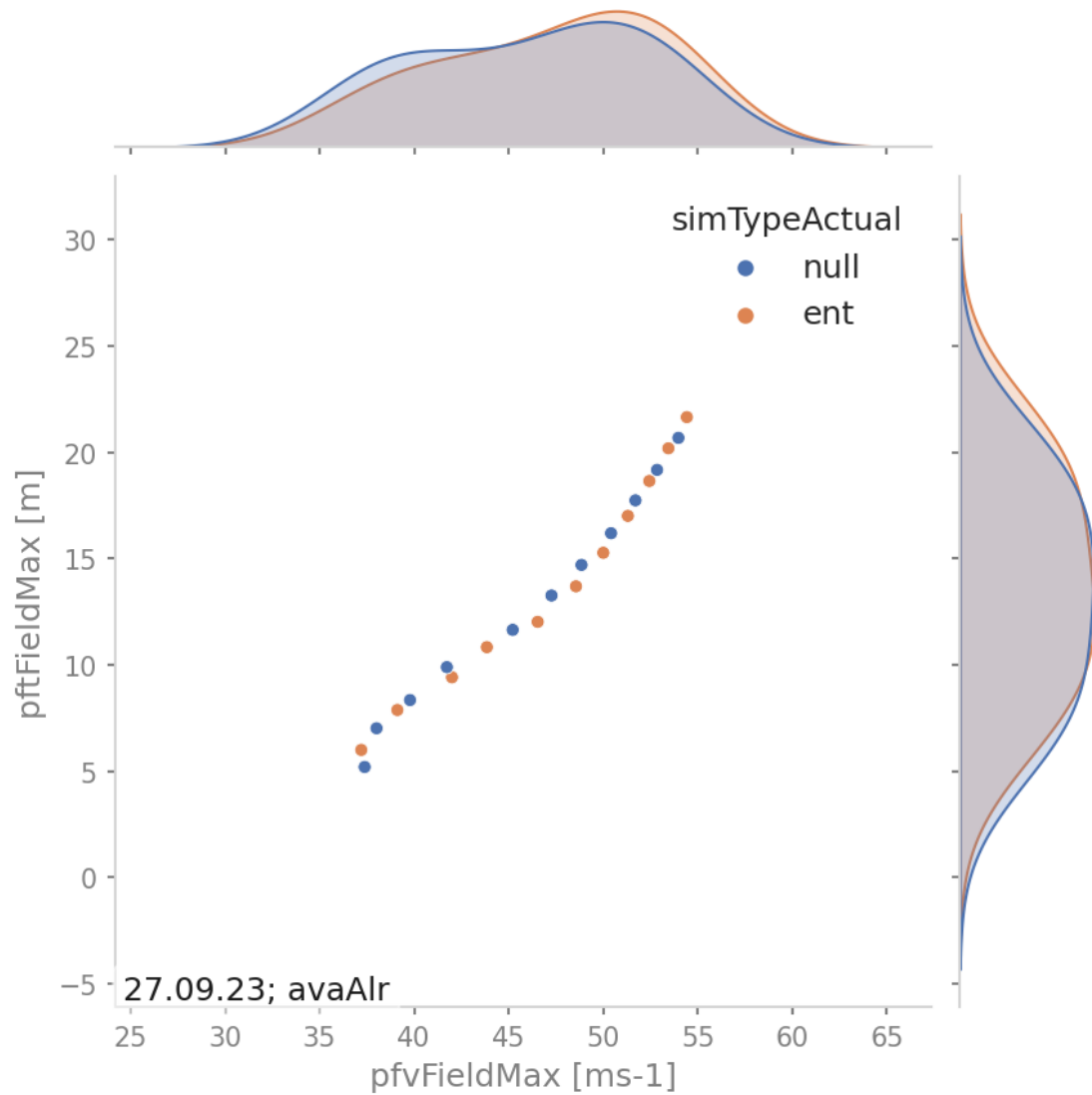


Fig. 5.21: Max field values of peak flow velocity vs flow thickness of all sims colorcoded using the simType



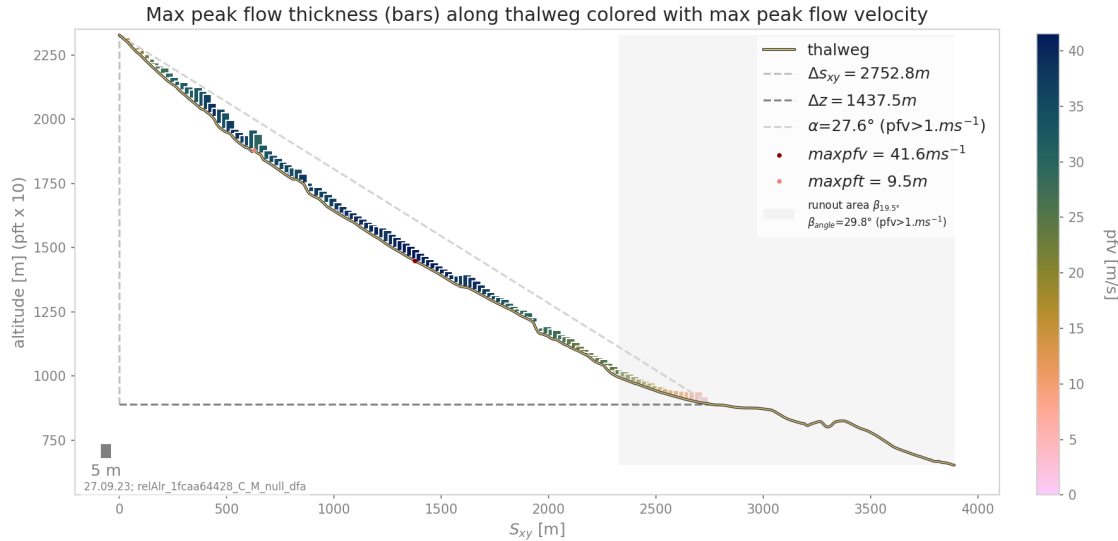


Fig. 5.22: Thalweg altitude plot

### 5.2.5.9 Comparison to reference simulation plots

The plots “\_hashID\_ContourComparisonToReference” and “\_hashID\_massAnalysis” where “hashID” is the name of the simulation show the 2D difference to the reference, the statistics associated and the mass analysis figure (this means these figures are created for each simulation).

### 5.2.5.10 List of Aimec result variables

The result variables listed below are partly included in the above described plots and are also saved to the Aimec results directory as *...resAnalysisDF.csv*, with one row per simulation. In the following, the resType represents the result type(s) set in the *aimecCfg.ini* for the parameter *resTypes*.

- *resTypeFieldMax*: maximum value of *resType* simulation result (simulation coordinate system)
- *resTypeFieldMin*: minimum value of *resType* simulation result (simulation coordinate system), values smaller than a threshold (*minValueField* in *aimecCfg.ini*) are masked
- *resTypeFieldMean*: average value of *resType* simulation result (simulation coordinate system), values smaller than a threshold (*minValueField* in *aimecCfg.ini*) are masked
- *resTypeFieldStd*: standard deviation of *resType* simulation result (simulation coordinate system), values smaller than a threshold (*minValueField* in *aimecCfg.ini*) are masked
- *maxresTypeCrossMax*: maximum value of cross profile maximum values of *resType* field (transformed in thalweg-following coordinate system)
- *sRunout*: the last point along the thalweg ( $S_{XY}$ ) where the transformed *resType* field (cross profile maximum values) still exceeds a threshold value (*thresholdValue* in *aimecCfg.ini*)
- *lRunout*: the cross profile coordinate ( $L_{XY}$ ) of the last point along the thalweg ( $S_{XY}$ ) where the transformed *resType* field (cross profile maximum values) still exceeds a threshold value (*thresholdValue* in *aimecCfg.ini*)
- *xRunout*, *yRunout*: x and y coordinates of the *sRunout* point (simulation coordinate system)
- *sMeanRunout*, *xMeanRunout*, *yMeanRunout*: runout point coordinates derived with cross profile mean values of transformed *resType* field instead of cross profile maximum values

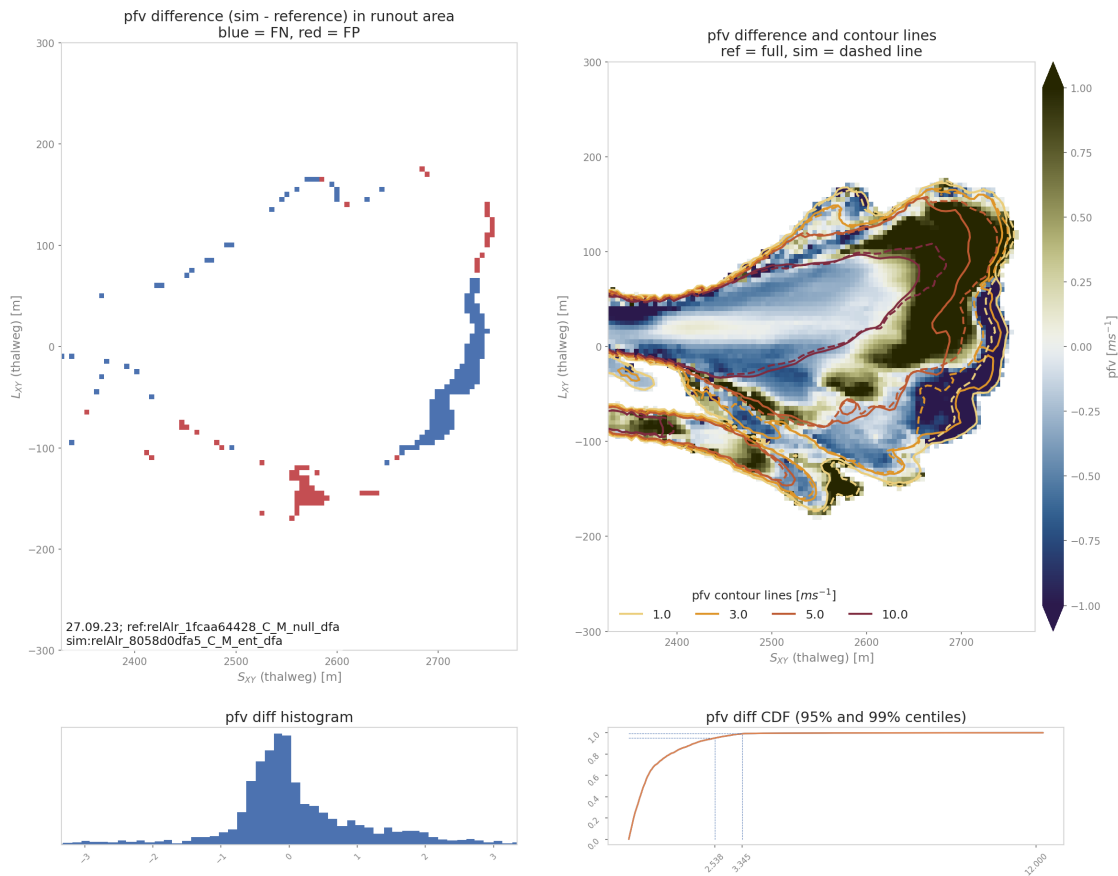


Fig. 5.23: The area comparison panel (top left) shows the false negative (FN in blue, which is where the reference field exceeds the threshold but not the simulation) and true positive (TP in red, which is where the simulation field exceeds the threshold but not the reference) areas. The contour comparison panel (top right) shows the contour lines of the reference (full lines) and the simulation (dashed lines) of the desired result fields in the runout area. It also shows the difference between the reference and simulation and computes the repatriation of this difference (Probability Density Function and Cumulative Density Function of the difference)

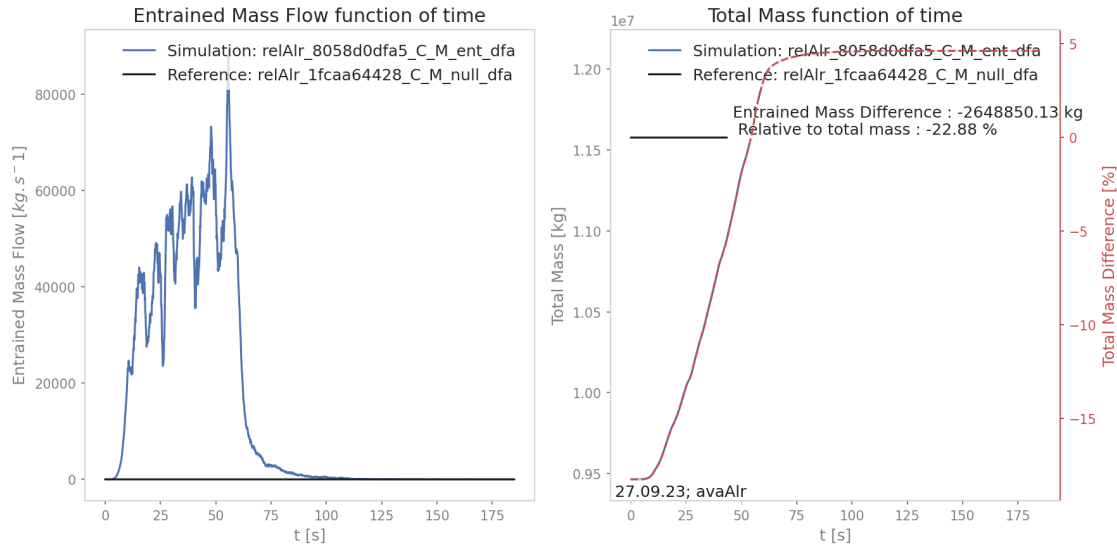


Fig. 5.24: The mass analysis plot shows the evolution of the total and entrained mass during the simulation and compares it to the reference

- `deltaSXY`: distance along thalweg ( $S_{XY}$ ) where the transformed `resType` field exceeds a threshold value (`thresholdValue` in `aimecCf.ini`)
- `zRelease`: altitude along thalweg where the transformed `resType` field first exceeds a threshold value (`thresholdValue` in `aimecCf.ini`)
- `zRunout`: altitude of the `sRunout` point
- `deltaZ`: altitude difference between `zRelease` and `zRunout`
- `runoutAngle`: corresponding runout angle based on `deltaSXY` and `deltaZ`

## 5.2.6 Configuration parameters

All configuration parameters are explained in `ana3AIMEC/ana3AIMECCfg.ini` (and can be modified in a local copy `ana3AIMEC/local_ana3AIMECCfg.ini`):

```
### Config File - This file contains the main settings for the ana3AIMEC run
## Set your parameters
# This file is part of Avaframe.
# This file will be overridden by local_ana3AIMEC.ini if it exists
# So copy this file to local_ana3AIMEC.ini, adjust your variables there

# Optional settings-----
[AIMECSETUP]
# width of the domain around the avalanche path in [m]
domainWidth = 600

# The cell size for the new (s,l) raster is automatically computed from the reference.
↳ result file (leave the following field empty)
# It is possible to force the cell size to take another value, then specify this new.
↳ value below, otherwise leave empty (default).
cellSizeSL =
```

(continues on next page)

(continued from previous page)

```

# define a runout area based on an angle of the thalweg profile (requires splitPoint in_
↳avaName/Inputs/POINTS as shpFile)
defineRunoutArea = True
# only used if defineRunoutArea=True - angle for the start of the run-out zone
startOfRunoutAreaAngle = 10

# data result type for general analysis (ppr/pft/pfd). If left empty takes the result_
↳types available for all simulations
resTypes = ppr|pft|pfd

# data result type for runout analysis (ppr, pft, pfd)
runoutResType = ppr

# limit value for evaluation of runout (depends on the runoutResType chosen)
thresholdValue = 1

# contour levels value for the difference plot (depends on the runoutResType chosen)
# use | delimiter (for ppr 1/3/5/10, for pft 0.1/0.25/0.5/0.75/1)
contourLevels = 1|3|5|10

# max of runoutResType difference for contour lines plus capped difference in_
↳runoutResType plot (for example 1 (pft), 5 (ppr))
diffLim = 1

# percentile to display when analyzing the peak values along profile
# for example 5 (corresponding to 5%) will lead to the interval [2.5, 97.5]%
percentile = 5
# chose interpolation method between 'nearest' and 'bilinear'
interpMethod = bilinear

# threshold distance [m]. When looking for the beta point make sure at least
# dsMin meters after the beta point also have an angle below 10°
dsMin = 30

# computational module that was used to produce avalanche simulations (to locate_
↳peakFiles)
anaMod = com1DFA

# two computational module that were used to produce avalanche simulations (to locate_
↳peakFiles) for comparison separated by |
comModules =

# if a computation module is benchmark, specify the test name (we assume that the_
↳testName folder is in AvaFrame/benchmarks/)
testName =

# parameter used for ordering the simulations - multiple possible; (e.g.relTh/deltaTh)
varParList =
# True if ascending ordering of simulations for varPar, False if descending order
ascendingOrder = True

```

(continues on next page)

(continued from previous page)

```

# parameter value (first parameter in varParList) that should be used as reference,
↳simulation (e.g. 1.0)
referenceSimValue =
# OR directly set reference simulation by its name (name of simulation result file or,
↳parts of it that definitively
# identify one particular simulation)
referenceSimName =
# unit of desired result parameter (e.g. m)
unit =

#-----
## Uncomment this section FILTER in your local copy of the ini file and add filter,
↳parameter and parameter values
## see the example provided below for release thickness
#[FILTER]
## define parameter and corresponding values from the simulation configuration to filter,
↳simulations
## multiple parameters are possible, just add them in a new line each
##relTh = 0.75|0.9|0.8

[LOTS]
# if extraPlots true additional analysis plots are created
extraPlots = True
# for one to one comparison of two result variables or derived quantities with option to,
↳distinguish scenarios using scenario
# comparison result variables options: resTypeFieldMax (or Min or Mean),,
↳maxresTypeCrossMax, sRunout, deltaSXY, zRelease,
# zRunout, deltaH, relMass, finalMass, entMass, thalwegTravelAngle
# comparison result variable 1
compResType1 = pfvFieldMax|deltaSXY
# comparison result variable 2
compResType2 = pftFieldMax|runoutAngle
# scenario parameter name used to colorcode comparison plots
scenarioName =
# interval of cross max values of pft to create bars in profile plot
barInterval = 25
# threshold of velocity to compute alpha angle
velocityThreshold = 1.

# plot save results flags-----
[FLAGS]
# Which value to plot in resultVisu
# 1 = mean pressure data
# 2 = growth index
# 3 = max pressure data
typeFlag = 3
# number of simulations above which a (additional) density plot is created
nDensityPlot = 100

```

(continues on next page)

```
# Mass analysis
flagMass = True

#-----
```

## 5.3 ana4Stats: Statistical analysis tools

### 5.3.1 probAna - Probability maps

probAna is used to derive simple probability maps for a set of simulations for one avalanche track. These maps show for each point in space the fraction of simulations where a chosen parameter has exceeded a given threshold. For example, it is possible to compute the probability map for an avalanche with respect to a peak pressure threshold of 1kPa, but it is also possible to choose other result variables and threshold values.

A set of multiple avalanche simulations is required to generate these maps. The simulations can be generated with *com1DFA* using a parameter variation, different release-, entrainment- or resistance scenarios. An example run script is given by *runAna4ProbAna*: where firstly, *com1DFA* is used to perform avalanche simulations varying parameters set in the *ana4Stats/probAnaCfg.ini* file. Using these simulations, a probability map is generated. The output is a raster file (.asc) with values ranging from 0-1. 0 meaning that no simulation exceeded the threshold in this point in space. 1 on the contrary means that all simulations exceeded the threshold. Details on this function, as for example required inputs can be found in: *ana4Stats.probAna*.

#### 5.3.1.1 To run - via QGIS Connector

Since version 1.6 it is possible to generate probability maps via the QGIS connector. This is provided in the experimental folder of the QGIS processing plugin via **Probability run**. A standard setup is used in which mu and release thickness are varied. Please note: *samosAT* friction is used instead of the default *samosATAuto* one.

The input requirements are the same as for *com1DFA* with one important difference: an additional *ci95* attribute is needed for the release shapes. This describes the confidence interval for the release thickness, using the same units. I.e. if you are using a release thickness of 1.5m, you need to provide the ci in [m] as well, for example ci=0.3m. The release thickness is then automatically varied within the release thickness plus/minus the confidence interval.

#### 5.3.1.2 To run - example run scripts

In *runProbAnaCom1DFA.py*, avalanche simulations are performed with the settings defined in the configuration file of *com1DFA* and in addition a parameter variation is performed according to the parameters set in *ana4Stats/probAnaCfg.ini* in the section *PROBRUN*. The parameters to be varied are set in **varParList**, the type of variation in **variationType** (options are: percent, range, rangefromci) and the value of the variation in **variationValue**. Then there are two sampling strategies to choose from, for performing the parameter variation: (1) a latin hypercube sample of all the parameters to be varied (provided in *varParList*) is generated and simulations are performed using sets of parameters drawn from this sample. Using sampling strategy (2) all the parameters set in *PROBRUN* are varied on at a time, i.e. simulations are performed for the standard settings of all parameters, except the one parameter to be varied, subsequently the other variations are performed. One probability map is created for all the different simulations and in case of sampling strategy (2), also one map per parameter that is varied once at a time, is created in addition. In order to run this example:

- first go to *AvaFrame/avaframe*
- copy *avaframeCfg.ini* to *local\_avaframeCfg.ini* and set your desired avalancheDir

- copy `ana4Stats/probAnaCfg.ini` to `ana4Stats/local_probAnaCfg.ini` and optionally adjust variation settings
- uncomment 'FILTER' section in `local_probAnaCfg.ini` and insert filter parameters if you want to first filter simulations
- run:

```
python3 runAna4ProbAna.py
```

Another example on how to generate probability maps for avalanche simulations performed with *com1DFA* is given in `runScripts.runProbAna`, where for *avaHockeyChannel* simulations are performed with varying release thickness values ranging from 0.75 to 1.75 meters in steps of 0.05 meters. The resulting simulations are then used to generate the probability map with `out3Plot.statsPlots.plotProbMap()`. There is also the option to filter the simulations further - using the function `in3Utils.fileHandlerUtils.getFilterDict()` which generates a parameter dictionary for filtering according to the filter criteria set in the configuration file (`ana4Stats/probAnaCfg.ini`) of the `ana4Stats.probAna` function. In order to run this example:

- first go to `AvaFrame/avaframe`
- copy `ana4Stats/probAnaCfg.ini` to `ana4Stats/local_probAnaCfg.ini`
- uncomment 'FILTER' section in `local_probAnaCfg.ini` and insert filter parameters if you want to first filter simulations
- run:

```
python3 runScripts/runProbAna.py
```

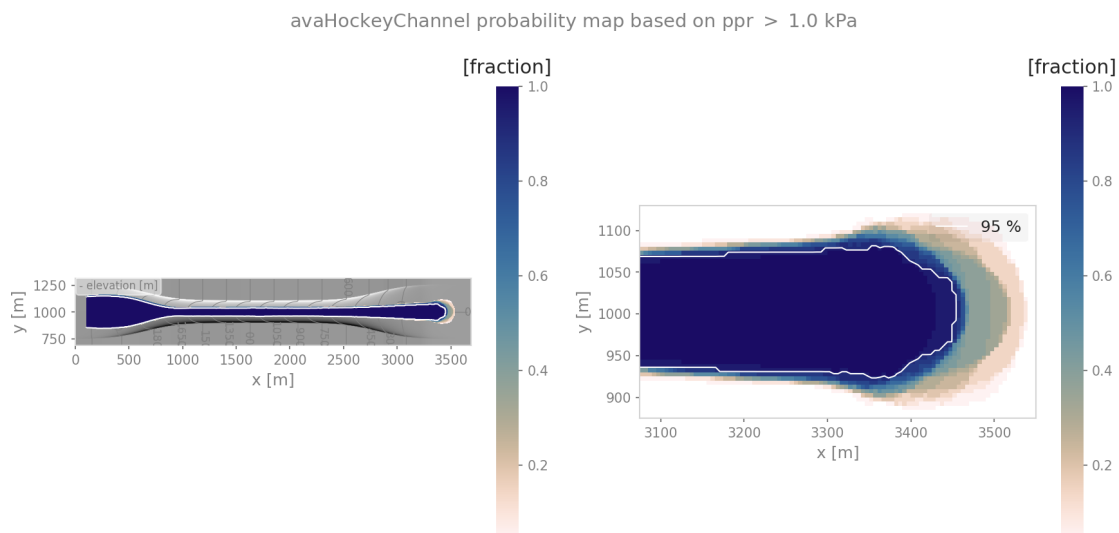


Fig. 5.25: Probability map example.

### 5.3.1.3 Theory

This point-wise probability is expressed by the relative frequency of avalanche peak flow field exceeding a certain threshold for a set of deterministic avalanche simulations derived from a range of input parameters (see [HBB19]).

## 5.3.2 getStats

In `ana4Stats.getStats`, functions that help to compute statistical properties of simulation results are gathered. `ana4Stats.getStats.extractMaxValues()` can be used to determine the maximum peak values of the simulation results. These values can then be plotted using the functions in `out3Plot.statsPlots` in order to visualise the statistics of a set of avalanche simulations. For further details on the specific functions, have a look at: `ana4Stats.getStats`.

### 5.3.2.1 To run

An example on how to use these statistical functions is given in `runScripts.runStatsExample`, where for `avaHockeyChannel` simulations are performed for two different release area scenarios and the release thickness is varied from 0.75 to 1.75 meters in steps of 0.05 meters. The resulting simulations are then analysed using the `ana4Stats.getStats.extractMaxValues()` function and plots are generated using the plotting routines from `out3Plot.statsPlots`. If in the configuration file `ana4Stats/getStats.ini` the flag `aimec` is set to `True`, additionally an *ana3AIMEC: Aimec* analysis is performed.

- first go to `AvaFrame/avaframe`
- copy `ana4Stats/getStats.ini` to `ana4Stats/local_getStatsCfg.ini`
- uncomment 'FILTER' section in `ana4Stats/local_getStatsCfg.ini` and insert filter parameters if you want to first filter simulations
- run:

```
python3 runScripts/runStatsExample.py
```

## 5.4 ana5Utils

### 5.4.1 distanceTimeAnalysis: Visualizing the temporal evolution of flow variables

With the functions gathered in this module, flow variables of avalanche simulation results can be visualized in a distance versus time diagram, the so called **thalweg-time diagram**. The **tt-diagram** provides a way to identify main features of the temporal evolution of flow variables along the avalanche *path*. This is based on the ideas presented in [FFGS13] and [RK20], where avalanche simulation results have been transformed into the radar coordinate system to facilitate direct comparison, combined with the attempt to analyze simulation results in an avalanche path dependent coordinate system ([Fis13]). In addition to the **tt-diagram**, `ana5Utils.distanceTimeAnalysis` also offers the possibility to produce simulated **range-time diagrams** of the flow variables with respect to a radar field of view. With this, simulation results can be directly compared to radar measurements (for example moving-target-identification (MTI) images from [KMS18]) in terms of front position and inferred approach velocity. The colorcoding of the simulated **range-time** diagrams show the average values of the chosen flow parameter (e.g. flow thickness (FT), flow velocity (FV)) at specified range gates. This colorcoding is not directly comparable to the MTI intensity given in the range-time diagram from radar measurements.



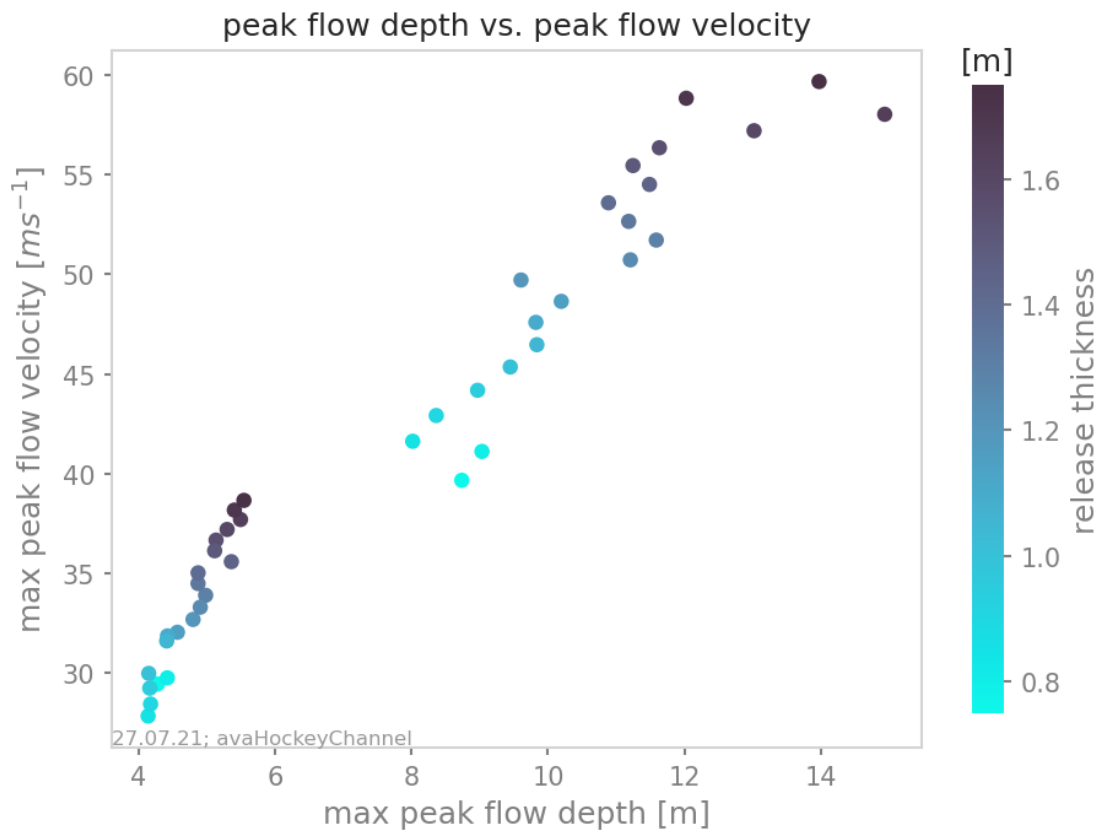


Fig. 5.26: Scatter plot of the hockey example with color-coded release thickness values.

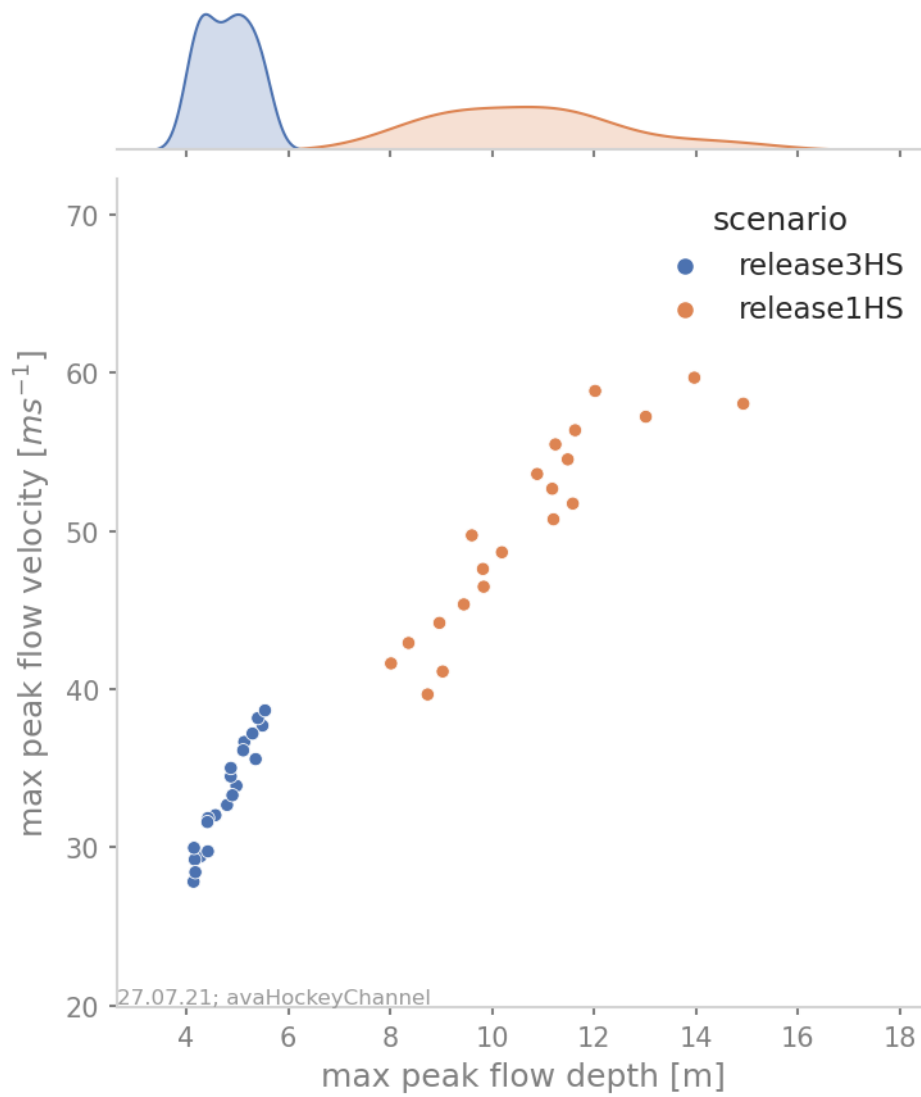


Fig. 5.27: Scatter plot of the hockey example including a marginal kde plot and color coded with release area scenario.

---

**Note:** The data processing for the **tt-diagram** and the **range-time diagram** can be done *during run time* of *com1DFA*, or as a postprocessing step. However, the second option requires first saving and then reading all the required time steps of the flow variable fields, which is much more computationally expensive compared to the first option.

---

#### 5.4.1.1 To run

During run-time of *com1DFA*:

- in your local copy of *com1DFA/com1DFACfg.ini* in [VISUALISATION] set *createRangeTimeDiagram* to True and choose if you want a *TTdiagram* by setting this flag to True or in the case of a simulated range-time diagram to False
- in your local copy of *ana5Utils/distanceTimeAnalysisCfg.ini* you can adjust the default settings for the generation of the diagrams
- run *runCom1DFA.py* to calculate *mtiInfo* dictionary (saved as pickle in *avalancheDir/Outputs/com1DFA/distanceTimeAnalysis/mtiInfo\_simHash.p*) that contains the required data for producing the **tt-diagram** or **range-time diagram**
- run *runScripts.runThalwegTimeDiagram.py* or *runScripts.runRangeTimeDiagram.py* and set the *preProcessedData* flag to *True*

As a postprocessing step:

- first you need to run *com1DFA* to produce fields of the desired flow variable (e.g. FT, FV) of sufficient temporal resolution (every second), for this in your local copy of *com1DFACfg.ini* add e.g. FT to the *resType* and change the *tSteps* to *0:1*
- have a look at *runScripts.runThalwegTimeDiagram.py* and *runScripts.runRangeTimeDiagram.py*
- in your local copy of *ana5Utils/distanceTimeAnalysisCfg.ini* you can adjust the default settings for the generation of the diagrams

The resulting figures are saved to *avalancheDirectory/Outputs/ana5Utils*.

---

**Note:** The **tt-diagram** requires info on an avalanche path (see *ana3AIMEC: Aimec*). The simulated **range-time diagram** requires info on the coordinates of the radar location (*x0*, *y0*), a point in the direction of the field of view (*x1*, *y1*), the aperture angle and the width of the range gates. The maximum approach velocity is indicated in the distance-time diagrams with a red star and is computed as the ratio of the distance traveled by the front and the respective time needed for a time step difference of at least *minVelTimeStep* which is set to 2 seconds as default. The approach velocity is a projection on the horizontal plane since the distance traveled by the front is also measured in this same plane.

---

#### 5.4.1.2 Theory

#### 5.4.1.3 Thalweg-time diagram

First, the flow variable result field is transformed into a path-following coordinate system, of which the centerline is the avalanche *path*. For this step, functions from *ana3AIMEC* are used. The distance of the avalanche front to the *start of runout area point* is determined using a user defined threshold of the flow variable. The front positions defined with this method for all the time steps are shown as black dots in the **tt-diagram**. The mean values of the flow variable are computed at cross profiles along the avalanche path for each time step and included in the **tt-diagram** as colored field. When computing the mean values, all the area where the flow variable is bigger than zero is taken into account. For this analysis, all available flow variables can be chosen, but the interpretation of the **tt-diagram** structures and the corresponding meaning of avalanche front may be different for flow thickness or flow velocity.

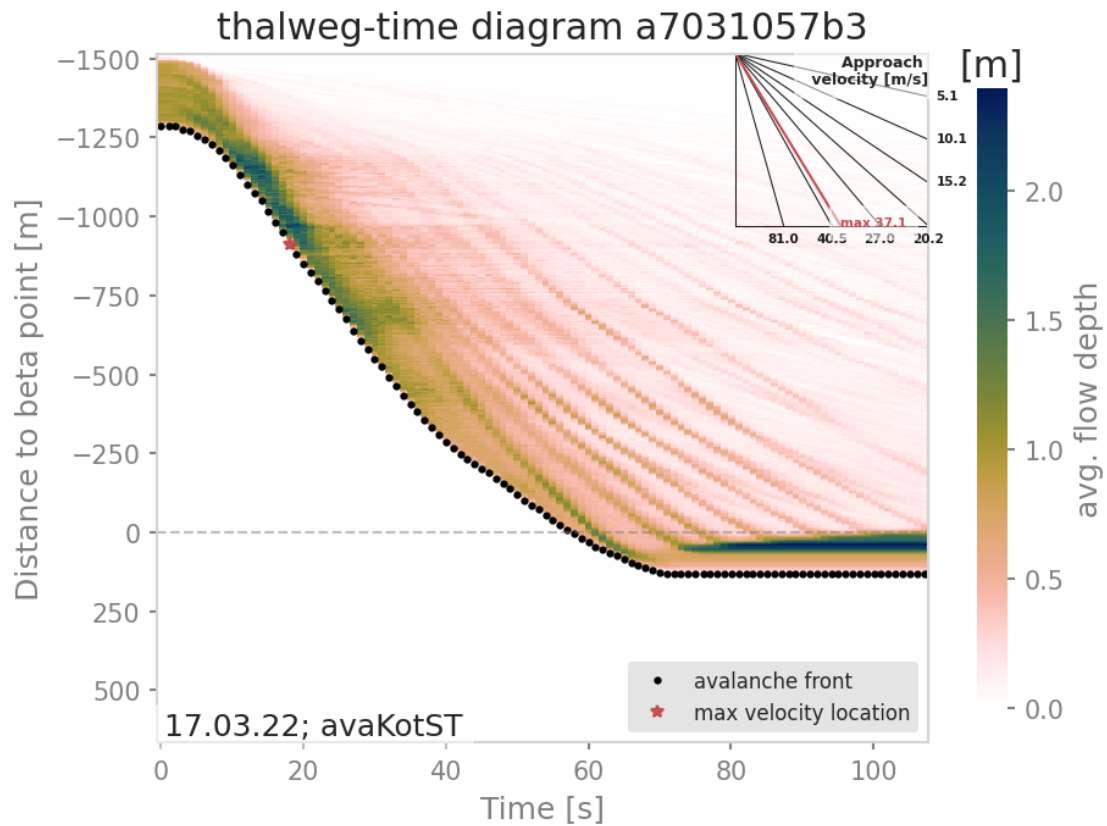


Fig. 5.28: Thalweg-time diagram example: The y-axis contains the distance from the beta point along the avalanche path (projected on the horizontal plane), e.g. the thalweg. Dots represent the avalanche front with the slope being the approach velocity. Red star marks the maximal approach velocity (this approach velocity is also projected on the horizontal plane).

#### 5.4.1.4 Simulated Range-Time diagram

The radar's field of view is determined using its location, a point in the direction of the field of view and the horizontal (azimuth) aperture angle of the antenna. The elevation or vertical aperture angle is not yet included. The line-of-sight distance of every grid point in the simulation results to the radar location is computed. The simulation results which lie outside the radar's field of view are masked. The distance of the avalanche front with respect to the radar location is determined for a user defined threshold in the flow variable and the average values of the result field for each range gate along the radar's line of sight are computed. This data is plotted in a range-time diagram, where the black dots indicate the avalanche front, and the colored field indicates the mean values of the flow variable for the range gates at each time step.

### 5.4.2 Automated path generation

Computational modules like  $\alpha\beta$  (*com2AB: Alpha Beta Model*) or analysis modules like the Thalweg-time diagram (*ana5Utils*) or Aimec (*ana3AIMEC: Aimec*) require an avalanche path and split point as input. This avalanche path and split point are usually created manually based on an expert opinion. The objective of this module is to automatically generate an avalanche path from a dense flow avalanche (DFA) simulation and placing a split point. The path is generated from the center of mass position of the dense material, so it is called the mass averaged path. It is extended towards the top of the release area and at the bottom. Therefore it covers the entire length of the avalanche with some buffer in the runout area. The split point is extracted from the parabola that is fitted on to the avalanche path profile.

#### 5.4.2.1 Input

The automatic path generation needs dense flow simulation results as input. These can be flow mass and flow thickness or particles for multiple time steps. *com1DFA* provides these already in the correct way.

We provide *runComputeDFAPath*, in which two options exist:

1. DFA simulation results already exist: in this case, you want to provide these as inputs to the path generation function. The flag *runDFAModule* in *runComputeDFAPath* is set to *False*. You need to provide the avalanche directory in your *local\_avaframeCfg.ini* file. This avalanche directory should already have *Outputs/com1DFA* with one or multiple simulation results. The simulation DEM is also required.
2. No DFA simulation results exist: use *com1DFA* to generate the simulation results before generating a path. Change the *runDFAModule* flag in *runComputeDFAPath* to *True*. The default configuration for *com1DFA* is read. *tSteps* are adjusted, *resType* and *simTypeList* are modified before running *com1DFA*.

#### 5.4.2.2 Outputs

A mass averaged path is produced for each *com1DFA* simulation. The path is/are saved in *avalancheDir/Outputs/DFAPath*

#### 5.4.2.3 To run automated path

- go to *AvaFrame/avaframe*
- copy *ana5Utils/DFAPathGenerationCfg.ini* to *ana5Utils/local\_DFAPathGenerationCfg.ini* and edit (if not, default values are used)
- run:

```
python3 runScripts/runComputeDFAPath.py
```

#### 5.4.2.4 Theory automated path

##### Mass average path

Any DFA simulation should be able to produce information about mass distribution for different time steps of the simulation (either flow thickness, mass, velocities... rasters or particles). This information is used to compute time dependent mass average quantities such as position (center of mass), velocity... For a flow quantity  $\mathbf{a}(\mathbf{x}, t)$ , the associated mass averaged quantity is defined by:

$$\bar{\mathbf{a}}(t) = \int_V \rho \mathbf{a}(\mathbf{x}, t) dV \approx \sum_k m_k \mathbf{a}_k(t)$$

where  $m_k$  respectively  $\mathbf{a}_k(t)$  defines the mass respectively flow quantity of particle or raster cell  $k$ . Applying the mass averaging to  $(x, y, z)$  gives the mass average path profile.

---

**Note:** The mass average path profiles does not necessarily lie on the topography

---

It is also possible to compute the mass averaged velocity squared  $\overline{\mathbf{u}^2}(t)$ , kinetic energy  $\frac{1}{2}m\overline{\mathbf{u}^2}(t)$  or travel distance  $s$  (which are used in the *Energy line test*).

The path is resampled at `nCellsResample` x cellsize and is extended towards the release area top to produce meaningful results when used in the com2AB module. Since results from the  $\alpha\beta$  analysis depend on the path profile start, moving the starting point of the profile will shift the  $\alpha$  upwards or downwards and affect the runout value.

##### Extending path towards the top (release)

There are two options available to extend the mass averaged path profile in the release area (`extTopOption` in the configuration file):

0. Extend the path up to the highest point in the release (highest particle or highest cell depending on which particles or rasters are available).
1. Extend the path towards the point that will lead to the longest runout. This point does not necessarily coincide with the highest point in the release area and corresponds to the point for which  $(\Delta z - \Delta s \tan \alpha)$  is maximum.  $\alpha$  corresponds to the angle of the runout line going from first to last point of the mass averaged line.  $\Delta z$  and  $\Delta s$  represent the vertical and horizontal distance between a point in the release and the first point of the mass averaged path profile.

##### Extending path towards the bottom (runout)

It is also necessary to extend the profile in the runout area. This is done by finding the direction of the path given by the few last points in the path in (x,y) (all points at a distance `nCellsMinExtend` x cellSize < distance < `nCellsMaxExtend` x cellSize)) and extending in this direction for a given percentage (`factBottomExt`) of the total length of the path  $s$ .

## Split point generation

A parabolic curve is fitted to the avalanche path profile extracted from the DFA simulation (non-extended profile), where the first and last point of the parabolic profile match the avalanche path profile. To find the best fitting parabolic profile, an additional constraint is needed. Two options are available: the default one (`fitOption` = 0`) minimises the distance between the parabolic profile and the avalanche path profile. The second option (`fitOption` = 1`) matches the end slope of the parabola to the profile. This parabolic fit determines the split point location. It is the first point for which the slope is lower than the `slopeSplitPoint` angle. This point is then projected on the avalanche path profile.

## Resampling

If the center of mass positions are derived in an equal time interval from the simulations, derived points will not be spaced equally due to variations in flow velocity. Especially in the release and runout area, lower velocities result in a denser spacing of extracted centers of mass, which can cause a crossing of grid lines that are drawn perpendicularly to the thalweg over the width of the domain. In order to reduce these overlaps, a the resampling function `in3Utils.geoTrans.prepareLine()` can be used, where the thalweg is generated based on a spline of degree `k` `scipy.splprep` and a user defined approximate distance between points along the spline.

## 5.5 out1Peak: Peak plots

### 5.5.1 Plot all peak fields

`out1Peak.outPlotAllPeak.plotAllPeakFields()` generates one plot for each peak field in `Outputs/modName/peakFiles`, constrained to existing data including a buffer specified in `out3Plot/plotUtilsCfg.ini`. These peak fields represent the peak values of the simulation result parameters (*dynamic peak pressure*, *peak flow thickness*, *peak velocity*), and `modName` corresponds to the name of the computational module that has been used to perform the simulations. Details on this function, as for example required inputs, can be found in `out1Peak.outPlotAllPeak.plotAllPeakFields()`.

### 5.5.2 Plot all fields

`out1Peak.outPlotAllPeak.plotAllFields()` generates one plot for each simulation result field provided in the specified input directory. This function is designed to work for result fields that follow a certain naming convention in order to provide full functionality: *releaseAreaName\_simulationType\_modelType\_simulationIdentifier\_resultType.asc*

One plot for each field is saved using the name of the input field. By default, the plot is constrained to where there is data, however this can be turned off by setting `constrainData=False` in the inputs. Details on this function, as for example required inputs, can be found in `out1Peak.outPlotAllPeak.plotAllField()`.

## 5.6 out3Plot: Plots

This module gathers various functions for creating visualisations of simulation results, analysis results, etc. Details on these functions can be found in [out3Plot](#). General plot settings can be found in `out3Plot.plotUtils` and the respective configuration file `out3Plot/plotUtilsCfg.ini`.

In the following sections, some of these plotting functions are described in more detail.

### 5.6.1 plotUtils

`out3Plot.plotUtils` gathers the general settings used for creating plots as well as small utility functions.

#### 5.6.1.1 makeColorMap

When plotting raster data, `out3Plot.plotUtils.makeColorMap()` is useful for deriving a suitable colormap. As inputs, this function requires:

```
makeColorMap(colormapDict, minimum level, maximum level, continuous flag)
```

By setting the continuous flag, there is the option to return a continuous or discrete colormap. Within this function, several default colormaps are available, as for example the ones used to visualize flow thickness, flow velocity or pressure. These can be initialised by:

```
from avaframe.out3Plot import plotUtils
plotUtils.cmapThickness
```

A colormap dictionary is loaded with information about the colormap, specific colors and levels corresponding to the default option of visualising pressure in `com1DFA`. If the flag *continuous* is set to *True*, this function returns a colormap object and a norm object fitted to the minimum and maximum values provided as input parameters. If the *continuous* flag is set to *False*, it will return a colormap object and a norm object representing the discrete levels defined in the dictionary. The maximum level provided in the inputs will be added as maximum level of the colorscale. See the two example plots below, where the predefined levels correspond to 1, 10, 25, and 50 kPa and the maximum level provided as input is 320 kPa.

In order to define customized discrete colorbars, there are different options. Firstly, provide a colorbar dictionary that follows the structure in the example dictionary:

```
from cmcrameri import cm as cmapCameri
colormapDict = {'cmap': cmapCameri.hawaii.reversed(),
                'colors': ["#B0F4FA", "#75C165", "#A96C00", "#8B0069"],
                'levels': [1.0, 10.0, 25.0, 50.0]}
```

It is also possible to provide key colors only. In this case the number of levels will match the number of colors and are equally distributed between the provided maximum and minimum value. If neither colors nor levels are provided in the dictionary, a default of six levels will be used. Another option is to just provide a colormap object instead of a dictionary. Here the *continuous* flag will be ignored and a colormap object as well as a norm object fitted to the maximum and minimum value will be returned.



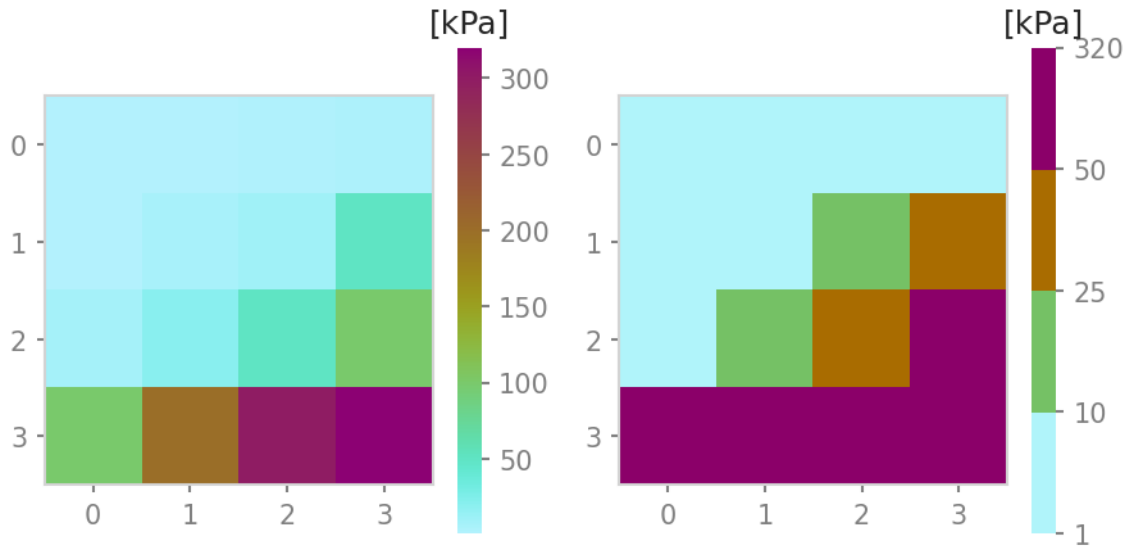


Fig. 5.29: left panel: continuous colormap. right panel: default discrete colormap used for pressure in com1DFA

## 5.6.2 outQuickPlot

`out3Plot.outQuickPlot` is used to generate plots of raster datasets, as for example the simulation results of `com1DFA`.

### 5.6.2.1 generatePlot

`out3Plot.outQuickPlot.generatePlot()` creates two plots, one plot with four panels, first dataset, second dataset, the absolute difference of the two datasets and the absolute difference capped to a smaller range of differences (ppr:  $\pm 100$  kPa, pft:  $\pm 1$  m, pfv:  $\pm 10$  ms $^{-1}$ ). The difference plots also include an insert showing the histogram and the cumulative density function of the differences. The second plot shows a cross- and along profile cut of the two datasets. In addition to the plots, a dictionary is returned with information on the plot paths, as well as the statistical measures of the difference plots, such as mean, max and min difference. Details on the required inputs for this function can be found in `out3Plot.outQuickPlot.generatePlot()`.

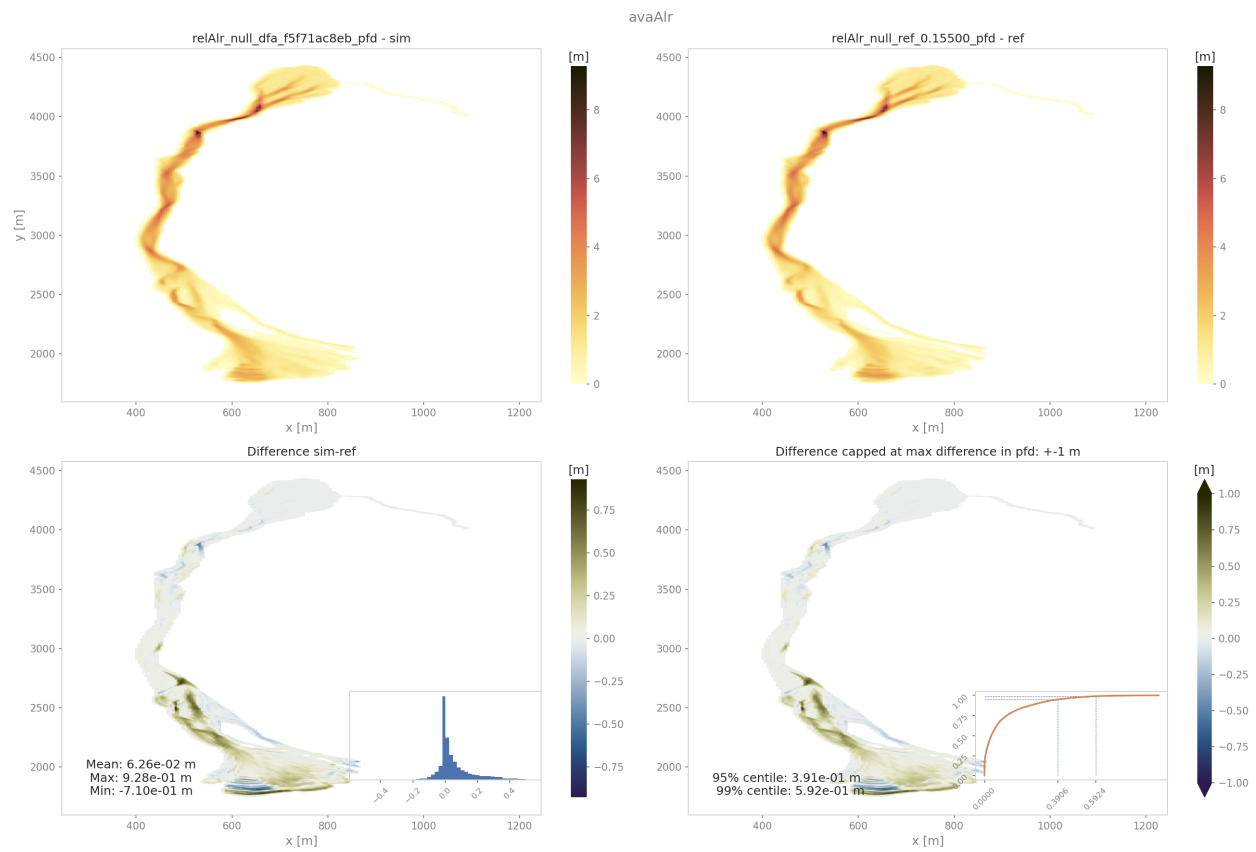


Fig. 5.30: Output plot from generatePlot on peak flow thickness results

### 5.6.2.2 quickPlotBench

`out3Plot.outQuickPlot.quickPlotBench()` calls `out3Plot.outQuickPlot.generatePlot()` to generate all comparison plots between the results of two simulations. This requires information on simulation names and paths to the simulation results and the desired result type. For further details have a look at `out3Plot.outQuickPlot.quickPlotBench()`.

### 5.6.2.3 quickPlotSimple

`out3Plot.outQuickPlot.quickPlotSimple()` is a bit more general, as it calls `out3Plot.outQuickPlot.generatePlot()` to generate the comparison plots between of two raster datasets of identical shape in a given input directory, without requiring further information. For further details have a look at `out3Plot.outQuickPlot.quickPlotSimple()`.

### To run

An example on how to create the difference plots for two raster datasets of identical shape is provided in `runScript/runQuickPlotSimple`

- first go to `AvaFrame/avaframe`
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your avalanche directory and the flag `showPlot`
- specify input directory, default is `data/NameOfAvalanche/Work/simplePlot`
- run:

```
python3 runScripts/runQuickPlotSimple.py
```

### 5.6.2.4 generateOnePlot

`out3Plot.outQuickPlot.generateOnePlot()` creates one plot of a single raster dataset. The first panel shows the dataset and the second panel shows a cross- or along profile of the dataset. The function returns a list with the file path of the generated plot. For further details have a look at `out3Plot.outQuickPlot.generateOnePlot()`.

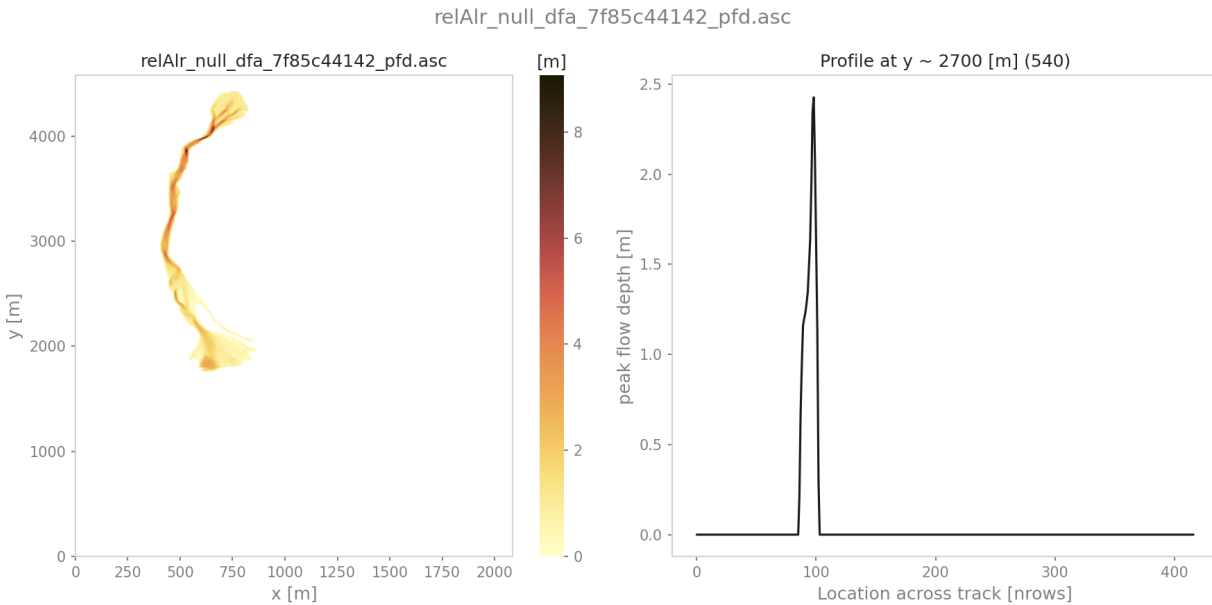


Fig. 5.31: Output plot from generateOnePlot on peak flow thickness results

### 5.6.2.5 quickPlotOne

`out3Plot.outQuickPlot.quickPlotOne()` calls `out3Plot.outQuickPlot.generateOnePlot()` to generate the plot corresponding to the input data. For information on the required inputs have a look at `out3Plot.outQuickPlot.quickPlotOne()`.

#### To run quickPlotOne

An example on how to create this plot from a given input directory or from the default one `data/NameOfAvalanche/Work/simplePlot`, is provided in `runScript/runQuickPlotOne`

- first go to `AvaFrame/avaframe`
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your avalanche directory and the flag `showPlot`
- copy `out3Plot/outQuickPlotCfg.ini` to `out3Plot/local_outQuickPlotCfg.ini` and optionally specify input directory
- run:

```
python3 runScripts/runQuickPlotOne.py
```

### 5.6.3 in1DataPlots

`out3Plot.in1DataPlots` can be used to plot a sample and its characteristics derived with `in1Data.computeFromDistribution`, such as: cumulative distribution function (CDF), bar plot of sample values, probability density function (PDF) of the sample, comparison plot of empirical- and desired CDF and comparison of empirical- and desired PDF.

### 5.6.4 statsPlots

`out3Plot.statsPlots` can be used to create scatter plots using a peak dictionary where information on two result parameters of avalanche simulations is saved. This peak dictionary can be created using the function `ana4Stats.getStats.extractMaxValues()` of `ana4Stats.getStats`. This can be used to visualize results of avalanche simulations where a parameter variation has been used or for e.g. in the case of different release area scenarios. If a parameter variation was used to derive the simulation results, the plots indicate the parameter values in color. If the input data includes information about the ‘scenario’ that was used, for example different release scenarios, the plots use different colors for each scenario. There is also the option to add a kde (kernel density estimation) plot for each result parameter as marginal plots. An example on how these plotting functions are used and exemplary plots can be found in `getStats`

Additionally, a plotting function for visualising probability maps is provided by `out3Plot.statsPlots.plotProbMap()`, where probability maps can be plotted including contour lines. An example on how these plotting function is used and an exemplary plot can be found in module `Ana4Stats:probAna`.

#### 5.6.4.1 plotValuesScatter

`out3Plot.statsPlots.plotValuesScatter()` produces a scatter plot of result type 1 vs result type 2 with color indicating values of the varied parameter.

#### 5.6.4.2 plotValuesScatterHist

`out3Plot.statsPlots.plotValuesScatterHist()` produces a scatter plot with marginal kde plots of result type 1 vs result type 2 with color indicating different scenarios (optional).

#### 5.6.4.3 plotHistCDFDiff

`out3Plot.statsPlots.plotHistCDFDiff()` generates the histogram plot and CDF plot of a input dataset.

### 5.6.5 particle analysis plots

`out3Plot.particleAnalysisPlots` can be used to create plots of particle properties for a `com1DFA` simulation, where particles refer to two-dimensional numerical columns (see [Discretization](#)). The particle properties can be analyzed over time or transformed into a thalweg following coordinate system using `ana3AIMEC`. Additional functions to compute velocity envelopes, i.e. the min and max values of the particle properties over time but also along the thalweg are used (see `out3Plot.outParticleAnalysis.velocityEnvelope()` and `out3Plot.outParticleAnalysis.velocityEnvelopeThalweg()`). The provided run script `runScripts/runParticleAnalysis.py`, provides an example of calling `com1DFA` to perform an avalanche simulation and then perform the respective particle analysis including a coordinate transformation and producing the final plots, which are exemplary shown here:

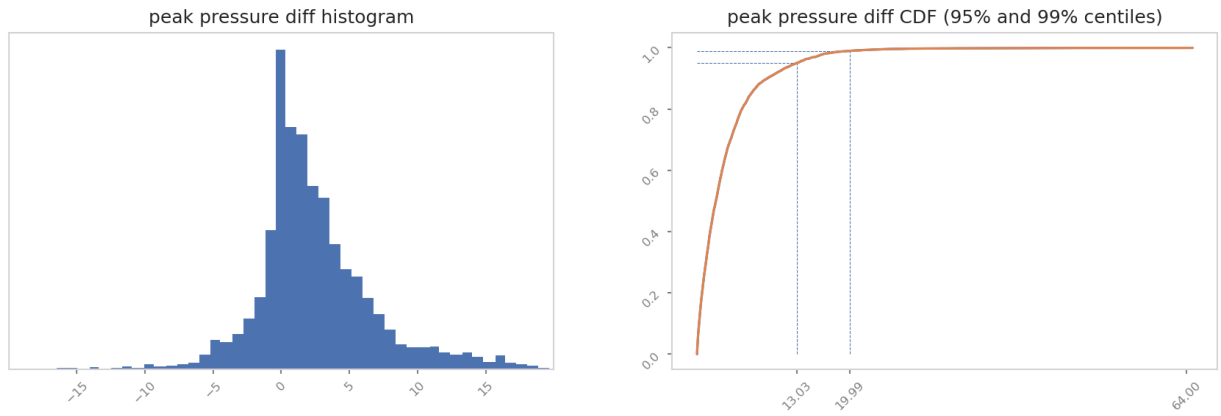


Fig. 5.32: Output plot from plotHistCDFDiff on peak pressure results from two simulations of avaAlr

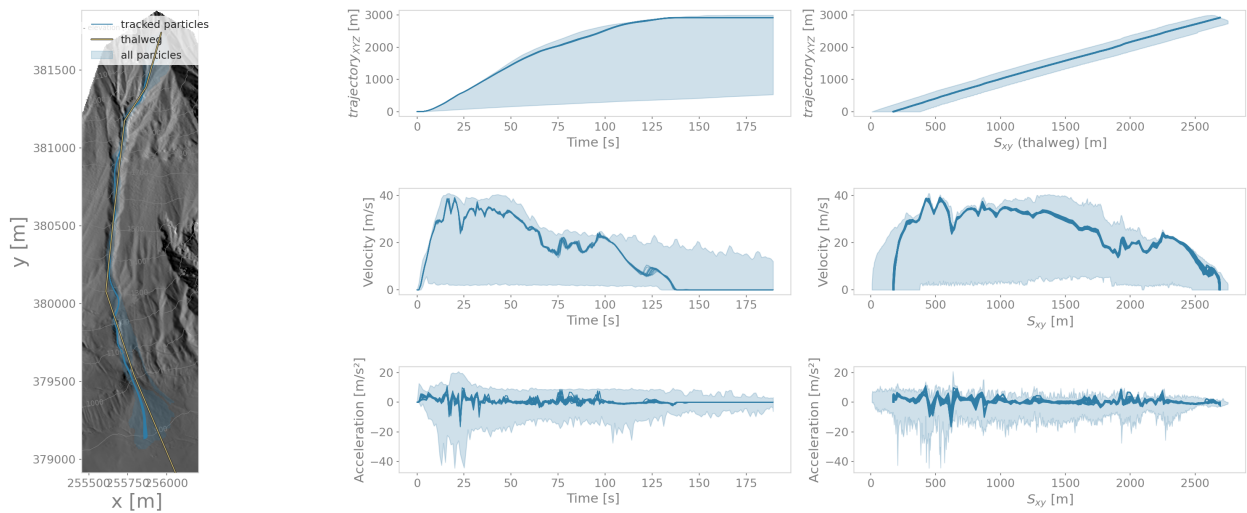


Fig. 5.33: Particle properties summary plot showing a map view of the affected area (by all particles) based on the peak flow velocity field, the tracked particles trajectories in dark blue with the superimposed thalweg line. The panels in the middle row show the evolution of particles' trajectory lengths, velocity and acceleration over time. In the rightmost panels, the particle data has been transformed into a thalweg following coordinate system and particle properties are shown along the thalweg coordinate  $S_{XY}$ . The light blue area extends from the min value found for all particles to the max value and the blue solid lines indicate the values of the tracked particles.

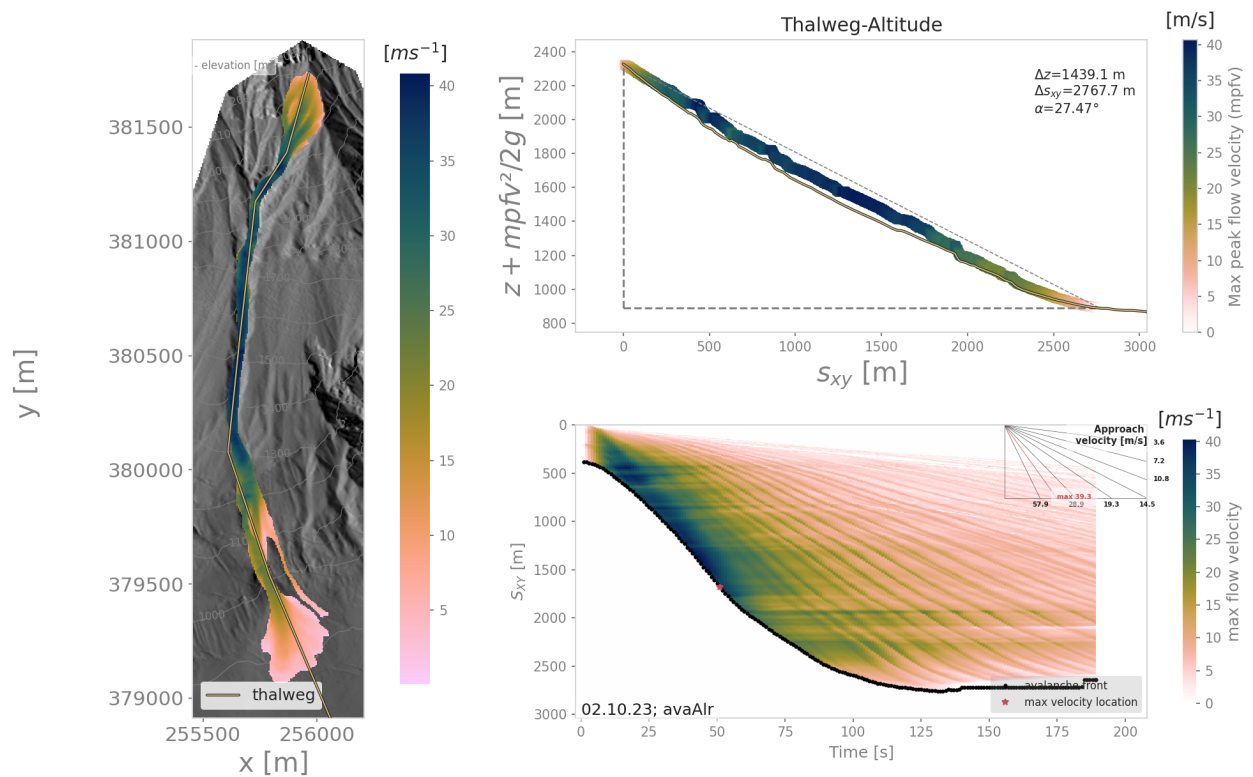


Fig. 5.34: The left panel shows a map view of the peak flow velocity field for the avalanche simulation with superimposed thalweg line. The top right plot shows the thalweg profile and the ‘velocity altitude’, i.e. the thalweg elevation plus the peak flow velocity cross max values along the thalweg to the power of two divided by two times the gravity acceleration, these values are then colored using the peak flow velocity cross max values. In the legend, the runout length  $\Delta s_{xy}$ , altitude difference  $\Delta z$  and corresponding runout angle  $\alpha$ , measured using the peak flow velocity field and a threshold of  $1\text{ m s}^{-1}$  (default setting) are provided. The lower right panel shows the thalweg-time diagram for the respective simulation.

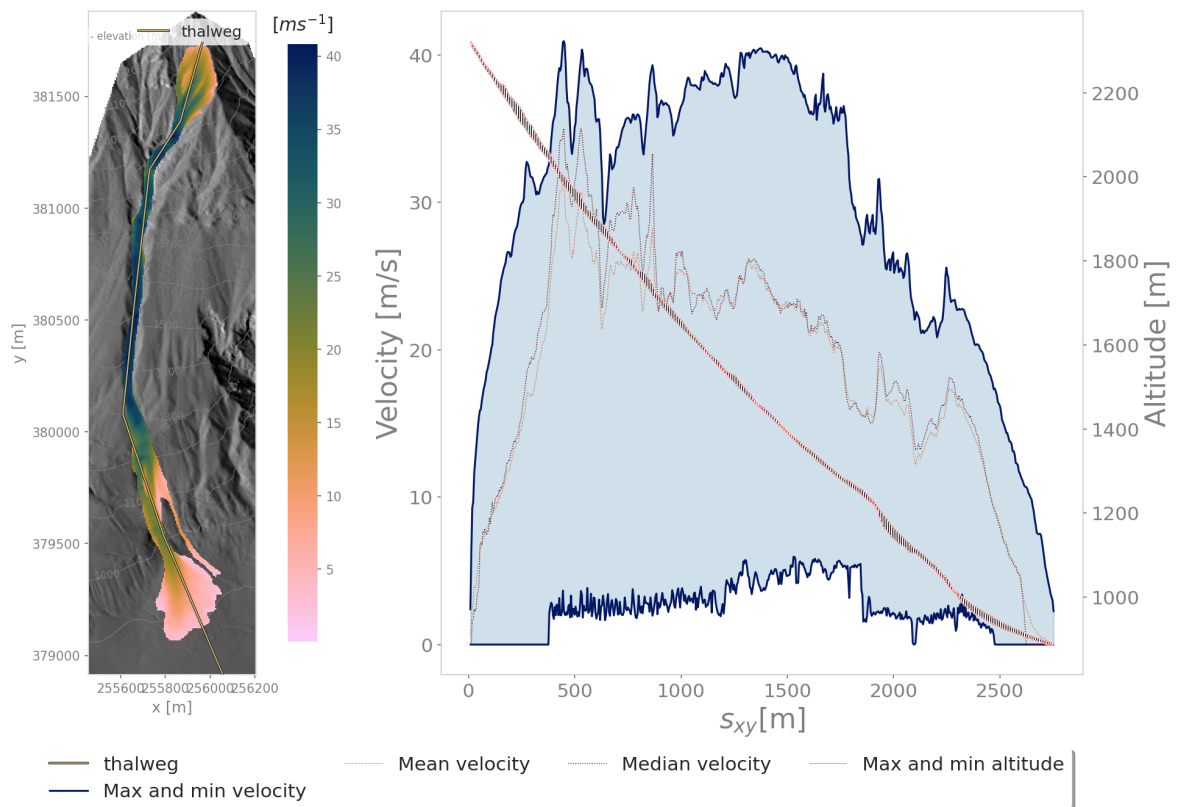


Fig. 5.35: The left panel shows a map view of the peak flow velocity field for the avalanche simulation with superimposed thalweg line. The right panel shows the velocity envelope for all particles, the mean and median particle velocity and the spread of altitudes covered by the particle positions along the thalweg coordinate.



---

**Note:** In order to create the presented plots, in addition to the particle data also result fields of peak flow velocity are required.

---

### 5.6.5.1 To run

- first go to AvaFrame/avaframe
- copy `avaframeCfg.ini` to `local_avaframeCfg.ini` and set your desired avalanche directory name
- create an avalanche directory with required input files - for this task you can use *Initialize Project*
- in `AvaFrame/avaframe/out3Plot`, copy `outParticleAnalysisCfg.ini` to `local_outParticleAnalysisCfg.ini` and set your desired configuration for the analysis, the avalanche simulation run in `com1DFA_override`, the aimec analysis in `ana3AIMEC_override` and the range time diagram in `distanceTimeAnalysis_override` section
- run:

```
python3 runScripts/runParticleAnalysis.py
```

## 5.7 log2Reports: Create Reports

### 5.7.1 Generate Report

`log2Report.generateReport` creates a markdown style report corresponding to the data provided in the python input dictionary. The report is structured in blocks according to the types provided in the dictionary. Currently, the report generation supports the following types:

- title - creates a title header
- avaName - includes a line with the avalanche directory
- simName - includes a line with Simulation name in the header
- time - includes a line with the date
- list - creates a table with two columns where the keys and values are listed and the dictionary name is set as title
- columns - creates a table with two rows and one column for each key and value pair and the dictionary name is set as title
- image - creates a block with the dictionary name as title and prints the key as heading with the prefix *Figure* and plots the image provided as value
- text - creates a text block with the dictionary name as title and prints the key as header with the prefix *Topic* and prints the text provided as value

For the input dictionary, a certain structure is required as illustrated for the following example:

```
reportDict = {'title Block' : {'type' : 'title', 'title' : 'This is my title'},
              'Simulation Name' : {'type' : 'simName', 'name' : 'This is my desired_
↳simulation name'},
              'Simulation Parameters' : {'type' : 'list', 'Parameter 1' : 0.155,
↳'Parameter 2' : 'red'},
              'Release Area' : {'type' : 'columns', 'scenario' : 'rell', 'area' : '40000
```

(continues on next page)

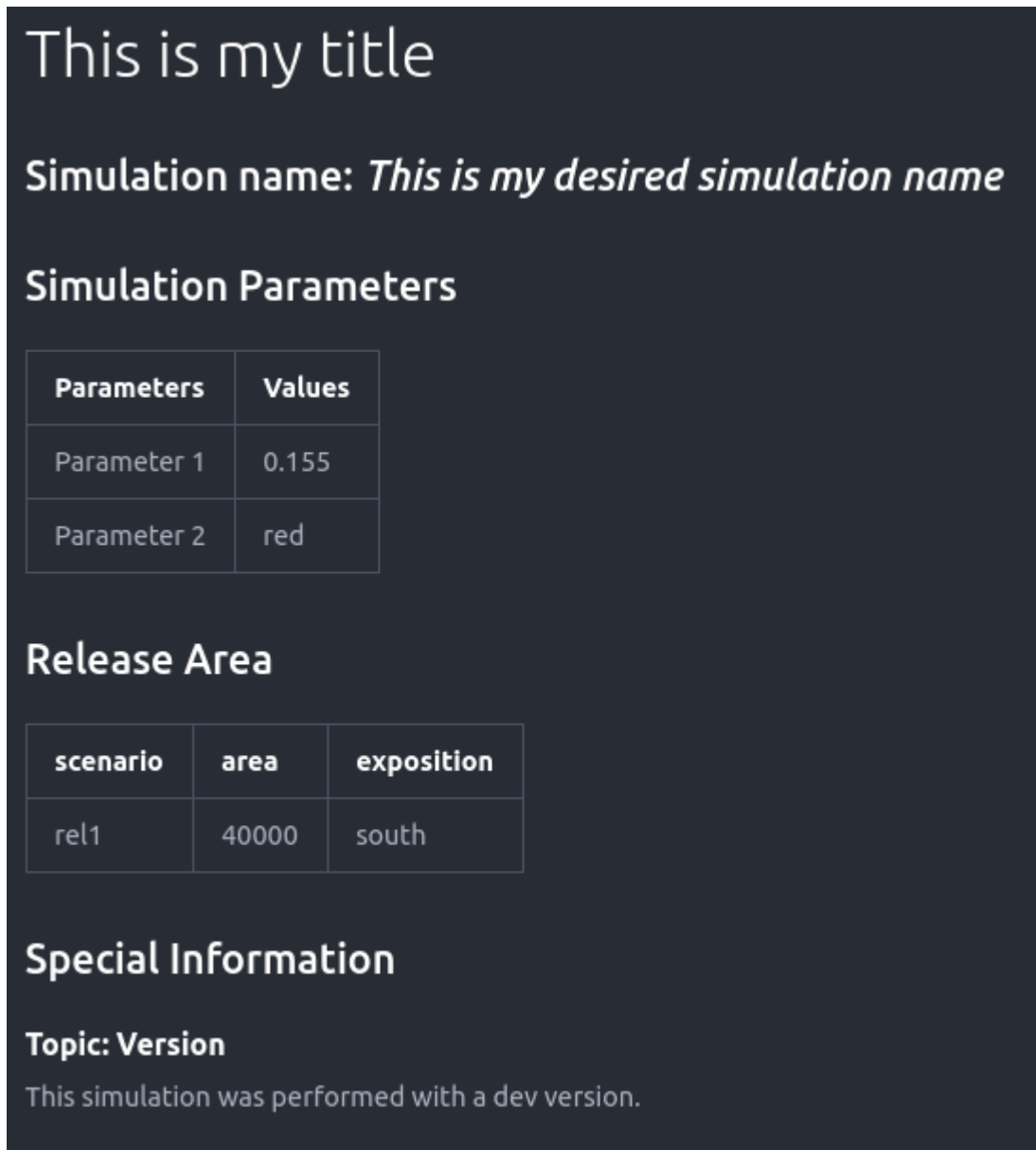
(continued from previous page)

```

↪', 'exposition' : 'south'},
    'Special Information' : {'type' : 'text', 'Version' : 'This simulation was_
↪performed with a dev version.'}
    }

```

where the keys and values do not have to follow a predefined form, only the key *type* has to be provided. This example dictionary results in a report of the form:



This is my title

Simulation name: *This is my desired simulation name*

Simulation Parameters

Parameters	Values
Parameter 1	0.155
Parameter 2	red

Release Area

scenario	area	exposition
rel1	40000	south

Special Information

**Topic: Version**

This simulation was performed with a dev version.

If a separate plotDictionary shall be included, in addition the key *simName* is required so that it can be appended to the correct simulation dictionary.

The default setting is to produce one report file, *fullSimulationReport.md*, however if one wants to receive one report for each *simName*, set the flag *reportOneFile = False* in *avaframeCfg.ini* or preferably your local copy

local\_com1DFACfg.ini. For further details have a look at [log2Report.generateReport](#).

## 5.7.2 Generate Compare Report

[log2Report.generateCompareReport](#) creates a markdown style report where the simulation results are compared to the benchmark results. The report is structured as follows:

- name of avalanche
- simulation name
- text block with info on test
- table listing simulation parameter name, value (reference) and value (simulation) -> if value of simulation run differs from reference value, it is highlighted in red
- table listing aimec analysis measure, value (reference) and value (simulation) -> if value of simulation run differs from reference value, it is highlighted in red
- block with plots including a header with plot title -> if differences exceed threshold a Warning is printed for each plot

For this purpose, python dictionaries are used to read the required input data. In order to produce the structure described above, the dictionaries require a certain structure too, for example:

```
dictionary = {
    '*simName*': 'name of simulation',
    '*Simulation Parameters*': {
        '*type*': '*list*',
        'name of parameter': 'value of parameter'
        ...
    },
    '*Test Info*': {
        '*type*': '*text*',
        'Title of text box': 'text, text',
        ...
    },
    '*Simulation Results*': {
        'desired plot title': '*file path*',
        ...
    }
}
```

where the required **keys** are indicated by the asterisk sign. The settings can be found in the respective configuration file `log2Report/generateCompareReportCfg.ini`. For further details visit [log2Report.generateCompareReport](#).

An example of creating a comparison report can be found in `runStandardTestsCom1DFA.py`. There, all the standard tests are run sequentially and a report is created where the com1DFA simulation results and the benchmark results are compared.



## THEORY

### 6.1 com1DFA DFA-Kernel theory

#### 6.1.1 Governing Equations for the Dense Flow Avalanche

The governing equations of the dense flow avalanche are derived from the incompressible mass and momentum balance on a Lagrange control volume ([Zwi00, ZKS03]).

##### 6.1.1.1 Mass balance:

$$\frac{d}{dt} \int_{V(t)} \rho_0 dV = \rho_0 \frac{dV(t)}{dt} = \oint_{\partial V(t)} q^{\text{ent}} dA \quad (6.1)$$

Where  $q^{\text{ent}}$  represents the snow entrainment rate.

##### 6.1.1.2 Momentum balance:

$$\rho_0 \frac{d}{dt} \int_{V(t)} u_i dV = \oint_{\partial V(t)} \sigma_{ij}^{\text{tot}} n_j dA + \rho_0 \int_{V(t)} g_i dV, \quad i = (1, 2, 3) \quad (6.2)$$

We introduce the volume average of a quantity  $P(\mathbf{x}, t)$ :

$$\bar{P}(\mathbf{x}, t) = \frac{1}{V(t)} \int_{V(t)} P(\mathbf{x}, t) dV$$

and split the area integral into :

$$\oint_{\partial V(t)} \sigma_{ij}^{\text{tot}} n_j dA = \oint_{\partial V(t)} \sigma_{ij} n_j dA + F_i^{\text{ent}} + F_i^{\text{res}}, \quad i = (1, 2, 3)$$

$F_i^{\text{ent}}$  represents the force required to break the entrained snow from the ground and to compress it (since the dense-flow bulk density is usually larger than the density of the entrained snow, i.e.  $\rho_{\text{ent}} < \rho$ ) and  $F_i^{\text{res}}$  represents the resistance force due to obstacles (for example trees). This leads to in Eq.6.2:

$$\rho_0 \frac{dV(t) \bar{u}_i}{dt} = \rho_0 V \frac{d\bar{u}_i}{dt} + \rho_0 \bar{u}_i \frac{dV}{dt} = \oint_{\partial V(t)} \sigma_{ij} n_j dA + \rho_0 V g_i + F_i^{\text{ent}} + F_i^{\text{res}}, \quad i = (1, 2, 3)$$

Using the mass balance equation Eq.6.1, we get:

$$\rho_0 V \frac{d\bar{u}_i}{dt} = \oint_{\partial V(t)} \sigma_{ij} n_j dA + \rho_0 V g_i + F_i^{\text{ent}} + F_i^{\text{res}} - \bar{u}_i \oint_{\partial V(t)} q^{\text{ent}} dA, \quad i = (1, 2, 3) \quad (6.3)$$

### 6.1.1.3 Boundary conditions:

The free surface is defined by :

$$F_s(\mathbf{x}, t) = z - s(x, y, t) = 0$$

The bottom surface is defined by :

$$F_b(\mathbf{x}) = z - b(x, y) = 0$$

The boundary conditions at the free surface and bottom of the flow read:

$$\left\{ \begin{array}{ll} \frac{dF_s}{dt} = \frac{\partial F_s}{\partial t} + u_i \frac{\partial F_s}{\partial x_i} = 0 & \text{at } F_s(\mathbf{x}, t) = 0 \quad \text{Kinematic BC (Material boundary)} \\ \sigma_{ij} n_j = 0 & \text{at } F_s(\mathbf{x}, t) = 0 \quad \text{Dynamic BC (Traction free surface)} \\ u_i n_i = 0 & \text{at } F_b(\mathbf{x}, t) = 0 \quad \text{Kinematic BC (No detachment)} \\ \tau_i^{(b)} = f(\sigma^{(b)}, \bar{u}, \bar{h}, \rho_0, t, \mathbf{x}) & \text{at } F_b(\mathbf{x}, t) = 0 \quad \text{Dynamic BC (Chosen friction law)} \end{array} \right. \quad (6.4)$$

$\sigma_i^{(b)} = (\sigma_{kl} n_l n_k) n_i$  represents the normal stress at the bottom and  $\tau_i^{(b)} = \sigma_{ij} n_j - \sigma_i^{(b)}$  represents the shear stress at the bottom surface.  $f$  describes the chosen friction model and are described in [Friction Model](#). The normals at the free surface ( $n_i^{(s)}$ ) and bottom surface ( $n_i^{(b)}$ ) are:

$$n_i^{(s,b)} = \frac{\partial F_{s,b}}{\partial x_i} \left( \frac{\partial F_{s,b}}{\partial x_j} \frac{\partial F_{s,b}}{\partial x_j} \right)^{-1/2}$$

### 6.1.1.4 Choice of the coordinate system:

The previous equations will be developed in the orthonormal coordinate system  $(B, \mathbf{v}_1, \mathbf{v}_2, \mathbf{v}_3)$ , further referenced as Natural Coordinate System (NCS). In this NCS,  $\mathbf{v}_1$  is aligned with the velocity vector at the bottom and  $\mathbf{v}_3$  with the normal to the slope, i.e.:

$$\mathbf{v}_1 = \frac{\mathbf{u}}{\|\mathbf{u}\|}, \quad \mathbf{v}_2 = \mathbf{v}_3 \wedge \mathbf{v}_1, \quad \mathbf{v}_3 = \mathbf{n}^{(b)}$$

The origin  $B$  of the NCS is attached to the slope. This choice leads to:

$$n_i^{(b)} = \delta_{i3}, \quad \left. \frac{\partial b}{\partial x_i} \right|_0 = 0 \quad \text{for } i = (1, 2), \quad \text{and} \quad u_2^{(b)} = u_3^{(b)} = 0$$

### 6.1.1.5 Thickness averaged equations:

In this NCS and considering a prism-like Control volume, the volume content  $V(t) = A_b(t) \bar{h}$  is obtained by multiplication of the basal area of the prism,  $A_b$ , with the averaged value of the flow thickness,

$$\bar{h} = \frac{1}{A_b(t)} \int_{A_b(t)} [s(\mathbf{x}) - b(\mathbf{x})] dA = \frac{1}{A_b(t)} \int_{A_b(t)} h(\mathbf{x}) dA, \quad \bar{u}_i = \frac{1}{V(t)} \int_{V(t)} u_i(\mathbf{x}) dV \quad (6.5)$$

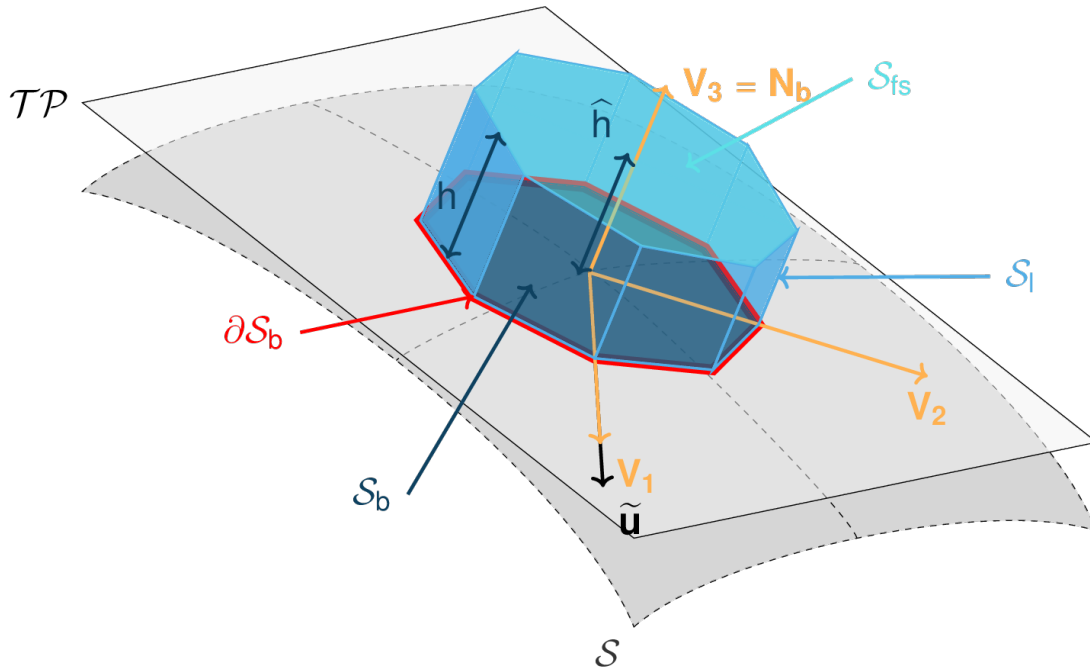


Fig. 6.1: Small Lagrangian prism-like Control volume

### Entrainment:

The snow entrainment is either due to plowing at the front of the avalanche or to erosion at the bottom. The entrainment rate at the front  $\dot{q}^{\text{plo}}$  can be expressed as a function of the properties of the entrained snow (density  $\rho_{\text{ent}}$  and snow thickness  $h_{\text{ent}}$ ), the velocity of the avalanche at the front  $\bar{\mathbf{u}}$  and length  $w_f$  of the front (measured perpendicularly to the flow velocity  $\bar{\mathbf{u}}$ ). It obviously only happens on the front of the avalanche:

$$\oint_{\partial V(t)} \dot{q}^{\text{plo}} dA = \int_{l_{\text{front}}} \int_b^s \dot{q}^{\text{plo}} dl dz = \rho_{\text{ent}} w_f h_{\text{ent}} \|\bar{\mathbf{u}}\| \quad (6.6)$$

The entrainment rate at the bottom  $\dot{q}^{\text{ero}}$  can be expressed as a function of the bottom area  $A_b$  of the control volume, the velocity of the avalanche  $\bar{\mathbf{u}}$ , the bottom shear stress  $\tau^{(b)}$  and the specific erosion energy  $e_b$ :

$$\oint_{\partial V(t)} \dot{q}^{\text{ero}} dA = \int_{A_b} \dot{q}^{\text{ero}} dA = A_b \frac{\tau^{(b)}}{e_b} \|\bar{\mathbf{u}}\| \quad (6.7)$$

This leads in the mass balance Eq.6.1 to :

$$\frac{dV(t)}{dt} = \frac{d(A_b \bar{h})}{dt} = \frac{\rho_{\text{ent}}}{\rho_0} w_f h_{\text{ent}} \|\bar{\mathbf{u}}\| + \frac{A_b}{\rho_0} \frac{\tau^{(b)}}{e_b} \|\bar{\mathbf{u}}\| \quad (6.8)$$

The force  $F_i^{\text{ent}}$  required to break the entrained snow from the ground and to compress it is expressed as a function of the required breaking energy per fracture surface unit  $e_s$  ( $J.m^{-2}$ ), the deformation energy per entrained mass element  $e_d$  ( $J.kg^{-1}$ ) and the entrained snow thickness ([FFGS13, SFF+08, Sam07]):

$$F_i^{\text{ent}} = -w_f (e_s + q^{\text{ent}} e_d), \quad (6.9)$$

where  $q^{\text{ent}}$  refers to the entrainable mass per surface area ( $kg.m^{-2}$ ) defined by  $q^{\text{ent}} := \rho^{\text{ent}} h^{\text{ent}}$  which depending on whether entrainment is due to ploughing or erosion, is derived using the integral of  $\dot{q}^{\text{plo}}$ , or respectively  $\dot{q}^{\text{ero}}$ , over time.

### Resistance:

The force  $F_i^{\text{res}}$  due to obstacles is expressed as a function of the characteristic diameter  $\bar{d}$  and height  $h_{\text{res}}$  of the obstacles, the spacing  $s_{\text{res}}$  between the obstacles and an empirical coefficient  $c_w$  (see Fig. 6.2). The effective height  $h^{\text{eff}}$  is defined as  $\min(\bar{h}, h_{\text{res}})$ :

$$F_i^{\text{res}} = -\left(\frac{1}{2} \bar{d} c_w / s_{\text{res}}^2\right) \rho_0 A h^{\text{eff}} \bar{u}^2 \frac{\bar{u}_i}{\|\bar{u}\|}$$

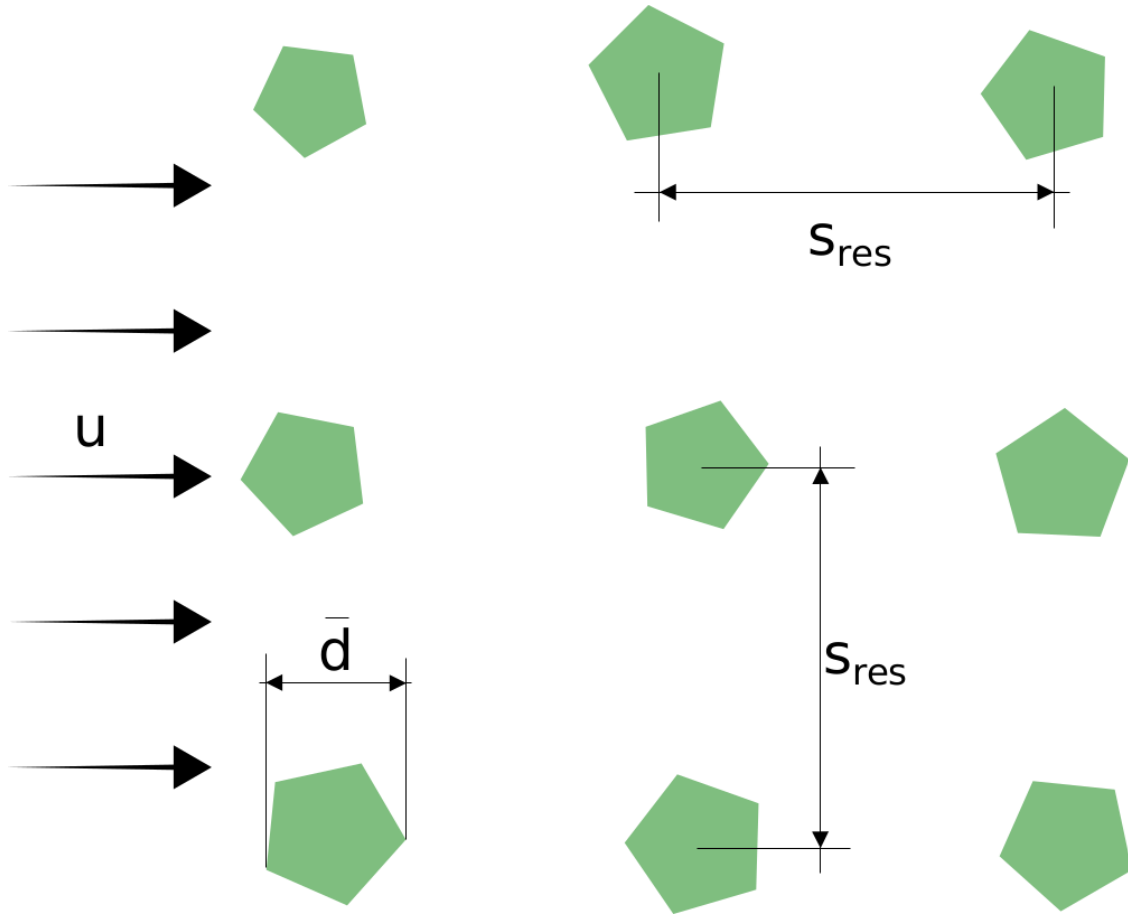


Fig. 6.2: Resistance force due to obstacles (from [FK13])

### Surface integral forces:

The surface integral is split in three terms, an integral over  $A_b$  the bottom  $x_3 = b(x_1, x_2)$ ,  $A_s$  the top  $x_3 = s(x_1, x_2, t)$  and  $A_h$  the lateral surface. Introducing the boundary conditions Eq.6.4 leads to:

$$\begin{aligned} \oint_{\partial V(t)} \sigma_{ij} n_j \, dA &= \int_{A_b} \underbrace{\sigma_{ij} n_j^{(b)}}_{-\sigma_{i3}} \, dA + \int_{A_s} \underbrace{\sigma_{ij} n_j^{(s)}}_0 \, dA + \int_{A_h} \sigma_{ij} n_j \, dA \\ &= -A_b \bar{\sigma}_{i3}^{(b)} + \oint_{\partial A_b} \left( \int_b^s \sigma_{ij} n_j \, dx_3 \right) dl \end{aligned}$$



Which simplifies the momentum balance Eq.6.3 to:

$$\rho_0 V \frac{d\bar{u}_i}{dt} = \oint_{\partial A_b} \left( \int_b^s \sigma_{ij} n_j dx_3 \right) dl - A_b \bar{\sigma}_{i3}^{(b)} + \rho_0 V g_i + F_i^{\text{ent}} + F_i^{\text{res}} - \bar{u}_i \oint_{\partial V(t)} q^{\text{ent}} dA, \quad (6.10)$$

$$i = (1, 2, 3)$$

The momentum balance in direction  $x_3$  (normal to the slope) is used to obtain a relation for the vertical distribution of the stress tensor ([Sam07]). Due to the choice of coordinate system and because of the kinematic boundary condition at the bottom, the left side of Eq.6.10 can be expressed as a function of the velocity  $\bar{u}_1$  in direction  $x_1$  and the curvature of the terrain in this same direction  $\frac{\partial^2 b}{\partial x_1^2}$  ([Zwi00]):

$$\rho A_b \bar{h} \frac{d\bar{u}_3}{dt} = \rho A_b \bar{h} \frac{\partial^2 b}{\partial x_1^2} \bar{u}_1^2,$$

rearranging the terms in the momentum equation leads to:

$$\bar{\sigma}_{33}(x_3) = \rho_0(s - x_3) \left( g_3 - \frac{\partial^2 b}{\partial x_1^2} \bar{u}_1^2 \right) + \frac{1}{A_b} \oint_{\partial A_b} \left( \int_{x_3}^s \sigma_{3j} n_j \, dx_3 \right) \, dl \quad (6.11)$$

### 6.1.1.6 Non-dimensional Equations

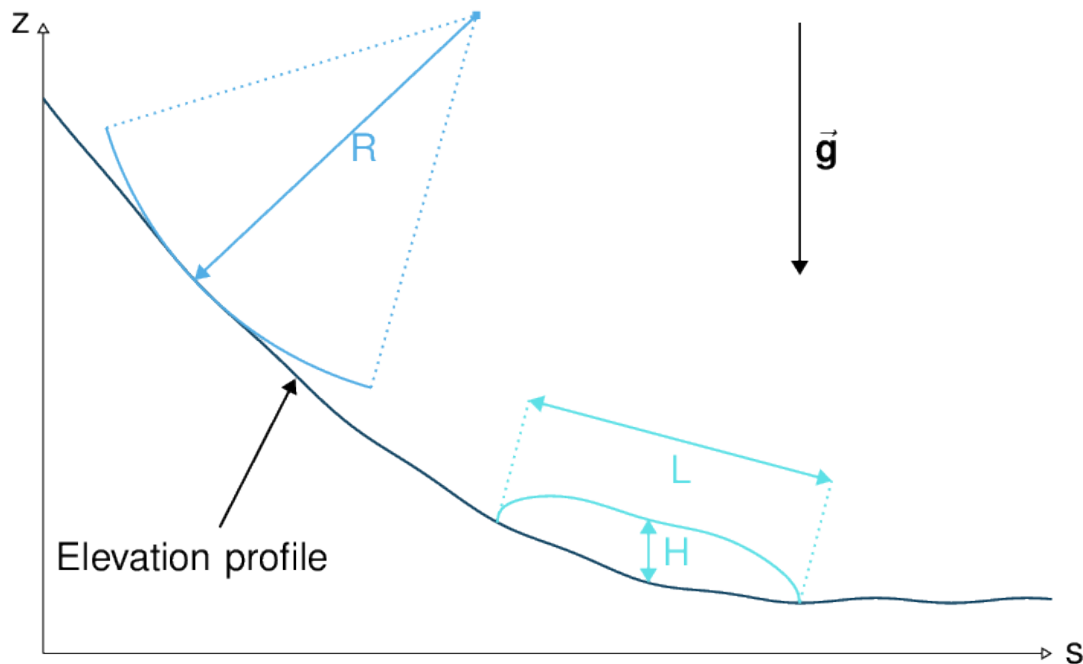


Fig. 6.3: Characteristic size of the avalanche along its path (from [Zwi00], modified)

The previous equations Eq.6.10 and Eq.6.11 can be further simplified by introducing a scaling based on the characteristic values of the physical quantities describing the avalanche. The characteristic length  $L$ , the thickness  $H$ , the acceleration due to gravity  $g$  and the characteristic radius of curvature of the terrain  $R$  are the chosen quantities. From those values, it is possible to form two non dimensional parameters that describe the flow:

- Aspect ratio:  $\varepsilon = H/L$

- Curvature:  $\lambda = L/R$

The different properties involved are then expressed in terms of characteristic quantities  $L$ ,  $H$ ,  $g$ ,  $\rho_0$  and  $R$  (see Fig. 6.3):

$$\begin{aligned} x_i &= L x_i^* \\ (dx_3, h, \bar{h}) &= H (dx_3^*, h^*, \bar{h}^*) \\ A_b &= L^2 A_b^* \\ t &= \sqrt{L/g} t^* \\ \bar{u}_i &= \sqrt{gL} \bar{u}_i^* \\ g_i &= g g_i^* \\ \frac{\partial^2 b}{\partial x_1^2} &= \frac{1}{R} \frac{\partial^2 b^*}{\partial x_1^{*2}} \end{aligned}$$

The normal part of the stress tensor is directly related to the hydro-static pressure:

$$\sigma_{ii} = \rho_0 g H \sigma_{ii}^*$$

The dimensionless properties are indicated by a superscripted asterisk. Introducing those properties in Eq.6.11, leads to :

$$\bar{\sigma}_{33}^* = \left( g_3^* - \lambda \frac{\partial^2 b^*}{\partial x_1^{*2}} \bar{u}_1^{*2} \right) (s^* - x_3^*) + \underbrace{\varepsilon \oint_{\partial A_b^*} \left( \int_{x_3^*}^{s^*} \sigma_{31}^* dx_3^* \right) dl^*}_{O(\varepsilon)}. \quad (6.12)$$

The height,  $H$  of dense flow avalanches is assumed to be small compared to its length,  $L$ . Meaning that the equations are examined in the limit  $\varepsilon \ll 1$ . It is then possible to neglect the last term in Eq.6.12 which leads to (after reinserting the dimensions):

$$\bar{\sigma}_{33}(x_3) = \rho_0 \underbrace{\left( g_3 - \bar{u}_1^2 \frac{\partial^2 b}{\partial x_1^2} \right)}_{g_{\text{eff}}} [\bar{h} - x_3] \quad (6.13)$$

And at the bottom of the avalanche, with  $x_3 = 0$ , the normal stress can be expressed as:

$$\bar{\sigma}_{33}^{(b)} = \rho_0 \left( g_3 - \bar{u}_1^2 \frac{\partial^2 b}{\partial x_1^2} \right) \bar{h} \quad (6.14)$$

Calculating the surface integral in equation Eq.6.10 requires to express the other components of the stress tensor. Here again a magnitude consideration between the shear stresses  $\sigma_{12} = \sigma_{21}$  and  $\sigma_{13}$ . The shear stresses are based on a generalized Newtonian law of materials, which controls the influence of normal stress and the rate of deformation through the viscosity.

$$\tau_{ij} = \eta \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad i \neq j$$

Because  $\partial x_1$  and  $\partial x_2$  are of the order of  $L$ , whereas  $\partial x_3$  is of the order of  $H$ , it follows that:

$$O\left(\frac{\sigma_{12}}{\sigma_{13}}\right) = \frac{H}{L} = \varepsilon \ll 1$$

and thus  $\sigma_{12} = \sigma_{21}$  is negligible compared to  $\sigma_{13}$ .  $\sigma_{13}$  is expressed using the bottom friction law  $\tau_i^{(b)} = f(\sigma^{(b)}, \bar{u}, \bar{h}, \rho_0, t, \mathbf{x})$  introduced in Eq.6.4.

In addition, a relation linking the horizontal normal stresses,  $\sigma_{ii}$ ,  $i = (1, 2)$ , to the vertical pressure distribution given by Eq.6.14 is introduced. In complete analogy to the arguments used by Savage and Hutter ([SH89]) the horizontal normal stresses are given as:

$$\sigma_{ii} = K_{(i)} \sigma_{33}$$

Where  $K_{(i)}$  are the earth pressure coefficients (cf. [Sal04, ZKS03]):

$$\begin{aligned}\sigma_{11} &= K_{x \text{ akt/pass}} \sigma_{33} \\ \sigma_{22} &= K_{y \text{ akt/pass}}^{(x \text{ akt/pass})} \sigma_{33}\end{aligned}$$

With the above specifications, the integral of the stresses over the flow height is simplified in equation Eq.6.10 to:

$$\int_b^s \sigma_{ij} dx_3 = \int_b^s K_{(i)} \sigma_{33} dx_3 = K_{(i)} \frac{\bar{h} \sigma^{(b)}}{2}$$

and the momentum balance can be written:

$$\begin{aligned}\rho_0 A \bar{h} \frac{d\bar{u}_i}{dt} &= \rho_0 A \bar{h} g_i + \underbrace{K_{(i)} \oint_{\partial A} \left( \frac{\bar{h} \sigma^{(b)}}{2} \right) n_i dl}_{F_i^{\text{lat}}} \underbrace{- \delta_{i1} A \tau^{(b)}}_{F_i^{\text{bot}}} \underbrace{- \rho_0 A h_{\text{eff}} C_{\text{res}} \bar{\mathbf{u}}^2 \frac{\bar{u}_i}{\|\bar{\mathbf{u}}\|}}_{F_i^{\text{res}}} \\ &\quad - \bar{u}_i \rho_0 \frac{d(A \bar{h})}{dt} + F_i^{\text{ent}}\end{aligned}\tag{6.15}$$

with

$$C_{\text{res}} = \frac{1}{2} \bar{d} \frac{c_w}{s_{\text{res}}^2}.$$

The mass balance Eq.6.8 remains unchanged:

$$\frac{dV(t)}{dt} = \frac{d(A_b \bar{h})}{dt} = \frac{\rho_{\text{ent}}}{\rho_0} w_f h_{\text{ent}} \|\bar{\mathbf{u}}\| + \frac{A_b}{\rho_0} \frac{\tau^{(b)}}{e_b} \|\bar{\mathbf{u}}\|\tag{6.16}$$

The unknown  $\bar{u}_1$ ,  $\bar{u}_2$  and  $\bar{h}$  satisfy Eq.6.14, Eq.6.15 and Eq.6.16. In equation Eq.6.15 the bottom shear stress  $\tau^{(b)}$  remains unknown, and a constitutive equation has to be introduced in order to completely solve the equations.

### 6.1.1.7 Friction Model

The problem can be solved by introducing a constitutive equation which describes the basal shear stress tensor  $\tau^{(b)}$  as a function of the flow state of the avalanche.

$$\tau_i^{(b)} = f(\sigma^{(b)}, \bar{u}, \bar{h}, \rho_0, t, \mathbf{x})\tag{6.17}$$

With

$\sigma^{(b)}$	normal component of the stress tensor
$\bar{u}$	average velocity
$\bar{h}$	average flow thickness
$\rho_0$	density
$t$	time
$\mathbf{x}$	position vector

Several friction models already implemented in the simulation tool are described here.

### Mohr-Coulomb friction model

The Mohr-Coulomb friction model describes the friction interaction between two solids. The bottom shear stress simply reads:

$$\tau^{(b)} = \tan \delta \sigma^{(b)}$$

$\tan \delta = \mu$  is the friction coefficient (and  $\delta$  the friction angle). The bottom shear stress linearly increases with the normal stress component  $\sigma^{(b)}$  ([BSG99, Sam07, WHP04, Zwi00]).

With this friction model, an avalanche starts to flow if the slope inclination is steeper than the friction angle  $\delta$ . In the case of an infinite slope of constant inclination, the avalanche velocity would increase indefinitely. This is unrealistic to model snow avalanches because it leads to over prediction of the flow velocity. The Mohr-Coulomb friction model is on the other hand well suited to model granular flow. Because of its relative simplicity, this friction model is also very convenient to derive analytic solutions and validate the numerical implementation.

### Chezy friction model

The Chezy friction model describes viscous friction interaction. The bottom shear stress then reads:

$$\tau^{(b)} = c_{\text{dyn}} \rho_0 \bar{u}^2$$

$c_{\text{dyn}}$  is the viscous friction coefficient. The bottom shear stress is a quadratic function of the velocity. ([BSG99, Sam07, WHP04, Zwi00]).

This model enables to reach more realistic velocities for avalanche simulations. The draw back is that the avalanche doesn't stop flowing before the slope inclination approaches zero. This implies that the avalanche flows to the lowest local point.

### Voellmy friction model

Anton Voellmy was a Swiss engineer interested in avalanche dynamics [Voe55]. He first had the idea to combine both the Mohr-Coulomb and the Chezy model by summing them up in order to take advantage of both. This leads to the following friction law:

$$\tau^{(b)} = \tan \delta \sigma^{(b)} + \frac{g}{\xi} \rho_0 \bar{u}^2$$

where  $\xi$  is the turbulent friction term. This model is described as Voellmy-Fluid [Sal04, Sam07].

### VoellmyMinShear friction model

In order to increase the friction force and make the avalanche flow stop on steeper slopes than with the Voellmy friction relation, a minimum shear stress can be added to the Voellmy friction relation. This minimum value defines a shear stress under which the snowpack doesn't move, and induces a strong flow deceleration. This expression of the basal layer friction model also resembles the one used in the swiss RAMMS model, where the Voellmy model is modified by adding a yield stress supposed to account for the snow cohesion (<https://ramms.slf.ch/en/modules/debrisflow/theory/friction-parameters.html>).

$$\tau^{(b)} = \tau_0 + \tan \delta \sigma^{(b)} + \frac{g}{\xi} \rho_0 \bar{u}^2$$

## SamosAT friction model

SamosAT friction model is a modification of some more classical models such as Voellmy model [Voellmy friction model](#). The basal shear stress tensor  $\tau^{(b)}$  is expressed as ([Sam07]):

$$\tau^{(b)} = \tau_0 + \tan \delta \left( 1 + \frac{R_s^0}{R_s^0 + R_s} \right) \sigma^{(b)} + \frac{\rho_0 \bar{u}^2}{\left( \frac{1}{\kappa} \ln \frac{\bar{h}}{R} + B \right)^2}$$

With

$\tau_0$	minimum shear stress
$R_s$	relation between friction and normal pressure (fluidization factor)
$R$	empirical constant
$R_s^0$	empirical constant
$B$	empirical constant
$\kappa$	empirical constant

The minimum shear stress  $\tau_0$  defines a lower limit below which no flow takes place with the condition  $\rho_0 \bar{h} g \sin \alpha > \tau_0$ .  $\alpha$  being the slope.  $\tau_0$  is independent of the flow thickness, which leads to a strong avalanche deceleration, especially for avalanches with low flow heights.  $R_s$  is expressed as  $R_s = \frac{\rho_0 \bar{u}^2}{\sigma^{(b)}}$ . Together with the empirical parameter  $R_s^0$  the term  $\frac{R_s^0}{R_s^0 + R_s}$  defines the Coulomb basal friction. Therefore lower avalanche speeds lead to a higher bed friction, making avalanche flow stop already at steeper slopes  $\alpha$ , than without this effect. This effect is intended to avoid lateral creep of the avalanche mass ([SG09]).

The default configuration also provides two additional calibrations for small- (< 25.000  $m^3$  release volume) and medium-sized (< 60.000  $m^3$  release volume) avalanches. A further constraint is the altitude of runout below 1500m msl for both.

## Wet snow friction type

**Note:** This is an experimental option to account for wet snow conditions, still under development and not yet tested. Also the parameters are not yet calibrated.

In addition, com1DFA provides an optional friction model implementation to account for wet snow conditions. This approach is based on the Voellmy friction model but with an enthalpy dependent friction parameter.

$$\tau^{(b)} = \mu \sigma^{(b)} + c_{\text{dyn}} \rho_0 \bar{u}^2$$

where,

$$\mu = \mu_0 \exp(-\text{enthalpy}/\text{enthRef})$$

The total specific enthalpy of the particles is initialized based on their initial temperature, specific heat capacity, altitude and their velocity (which is zero for the initial time step). Throughout the computation, the particles specific enthalpy is then computed following:

$$\text{enthalpy} = \text{totalEnthalpy} - g z - 0.5 \bar{u}^2$$

### 6.1.1.8 Dam

The dam is described by a crown line, that is to say a series of  $x, y, z$  points describing the crown of the dam (the dam wall is located on the left side of the line), by the slope of the dam wall (slope measured from the horizontal,  $\beta$ ) and a restitution coefficient (describing if we consider more elastic or inelastic collisions between the particles and the dam wall, varying between 0 and 1).

The geometrical description of the dam is given on the figure Fig. 6.4. The dam crown line ( $\mathbf{x}_{\text{crown}}$ ) is projected onto the topography, which provides us with the dam center line ( $\mathbf{x}_{\text{center}}$ ). We compute the tangent vector to the center line ( $\mathbf{t}_f$ ). From this tangent vector and the dam slope, it is possible to compute the wall tangent vector ( $\mathbf{t}_w$ ). Knowing the wall tangent vector and height, it is possible to determine normal vector to the wall ( $\mathbf{n}_w$ ) and the foot line which is the intersection between the dam wall and the topography ( $\mathbf{x}_{\text{foot}}$ ).

When the dam fills up (flow thickness increases), the foot line is modified ( $\mathbf{x}_{\text{foot}}^{\text{filled}} = \mathbf{x}_{\text{foot}} + \frac{h_v}{2} \mathbf{e}_z$ ). The normal and tangent vectors to the dam wall are readjusted accordingly.

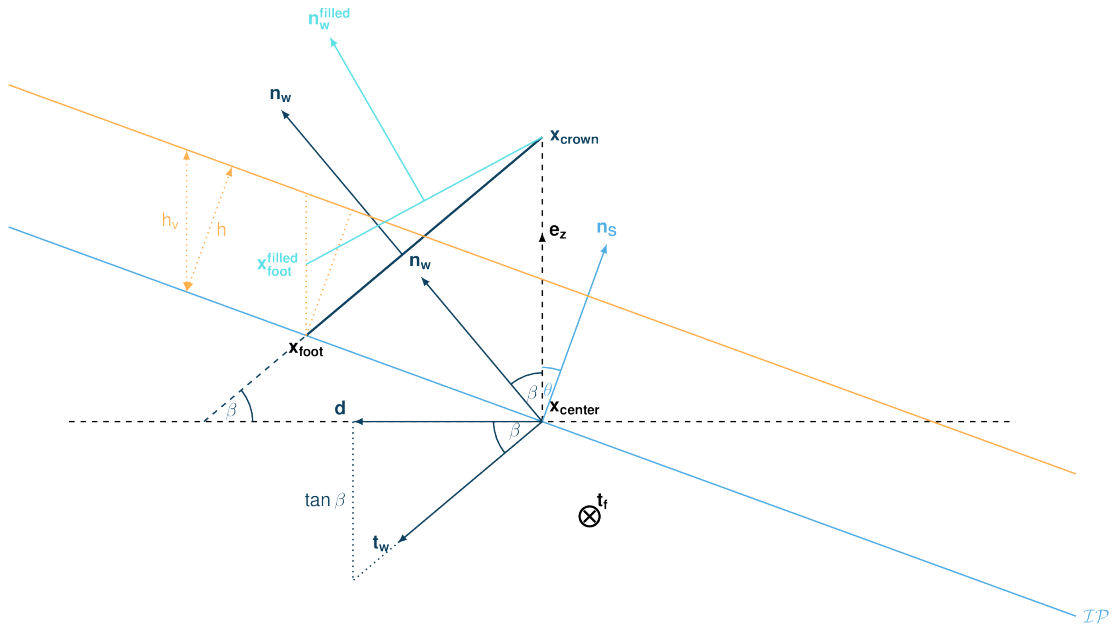


Fig. 6.4: Side view of the dam (cut view).  $\mathbf{x}_{\text{crown}}$  describes the crown of the dam,  $\mathbf{x}_{\text{center}}$  is the vertical projection of the crown on the topography (here the light blue line represents the topography). The tangent vector to the center line ( $\mathbf{t}_f$ ) is computed from the center line points. The tangent vector to the center line with the dam slope angle enable to compute the tangent ( $\mathbf{t}_w$ ) and normal ( $\mathbf{n}_w$ ) vector to the dam wall. Finally, this normal vector is adjusted depending on the snow thickness at the dam location (filling of the dam,  $\mathbf{n}_w^{\text{filled}}$ )

In the initialization of the simulation, the dam tangent vector to the center line ( $\mathbf{t}_f$ ), foot line ( $\mathbf{x}_{\text{foot}}$ ) and normal vector to the wall ( $\mathbf{n}_w$ ) are computed. The grid cells crossed by the dam as well as their neighbor cells are memorized (tagged as dam cells).

## 6.2 com1DFA DFA-Kernel numerics

The numerical method used in com1DFA mixes particle methods and mesh methods. Mass and momentum are tracked using particles but flow thickness is tracked using the mesh. The mesh is also used to access topographic information (surface elevation, normal vector) as well as for displaying results.

Mass Eq.6.16 and momentum Eq.6.15 balance equations as well as basal normal stress Eq.6.14 are computed numerically using a SPH method (Smoothed Particle Hydrodynamics) ([Mon92]) for the variables  $\bar{\mathbf{u}} = (\bar{u}_1, \bar{u}_2)$  and  $\bar{h}$  by discretization of the released avalanche volume in a large number of mass elements. SPH in general, is a mesh-less numerical method for solving partial differential equations. The SPH algorithm discretizes the numerical problem within a domain using particles ([Sam07, SG09]), which interact with each-other in a defined zone of influence. Some of the advantages of the SPH method are that free surface flows, material boundaries and moving boundary conditions are considered implicitly. In addition, large deformations can be modeled due to the fact that the method is not based on a mesh. From a numerical point of view, the SPH method itself is relatively robust.

### 6.2.1 Discretization

#### 6.2.1.1 Space discretization

The domain is discretized in particles. The following properties are assigned to each particle  $p_k$ : a mass  $m_{p_k}$ , a thickness  $h_{p_k}$ , a density  $\rho_{p_k} = \rho_0$  and a velocity  $\mathbf{u}_{p_k} = (u_{p_k,1}, u_{p_k,2})$  (**those quantities are thickness averaged, note that we dropped the overline from Eq.6.5 for simplicity reasons**). In the following paragraphs,  $i$  and  $j$  indexes refer to the different directions in the NCS, whereas  $k$  and  $l$  indexes refer to particles.

The quantities velocity, mass and flow thickness are also defined on the fixed mesh. It is possible to navigate from particle property to mesh property using the interpolation methods described in *Mesh and interpolation*.

#### 6.2.1.2 Time step

A fixed time step can be used or an adaptive time step that depends on the sph kernel radius as well as the particle size.

### 6.2.2 Mesh and interpolation

Here is a description of the mesh and the interpolation method that is used to switch from particle to mesh values and the other way around.

#### 6.2.2.1 Mesh

For practical reasons, a 2D rectilinear mesh (grid) is used. Indeed the topographic input information is read from 2D raster files (with  $N_y$  and  $N_x$  rows and columns) which correspond exactly to a 2D rectilinear mesh. Moreover, as we will see in the following sections, 2D rectilinear meshes are very convenient for interpolations as well as for particle tracking. The 2D rectilinear mesh is composed of  $N_y$  and  $N_x$  rows and columns of square cells (of side length  $csz$ ) and  $N_y + 1$  and  $N_x + 1$  rows and columns of vertices as described in Fig. 6.5. Each cell has a center and four vertices. The data read from the raster file is assigned to the cell centers. Note that although this is a 2D mesh, as we use a terrain-following coordinate system to perform our computations, this 2D mesh is oriented in 3D space and hence the projected side length corresponds to  $csz$ , whereas the actual side length and hence also the *Cell area*, depend on the local slope, expressed by the *Cell normals*.

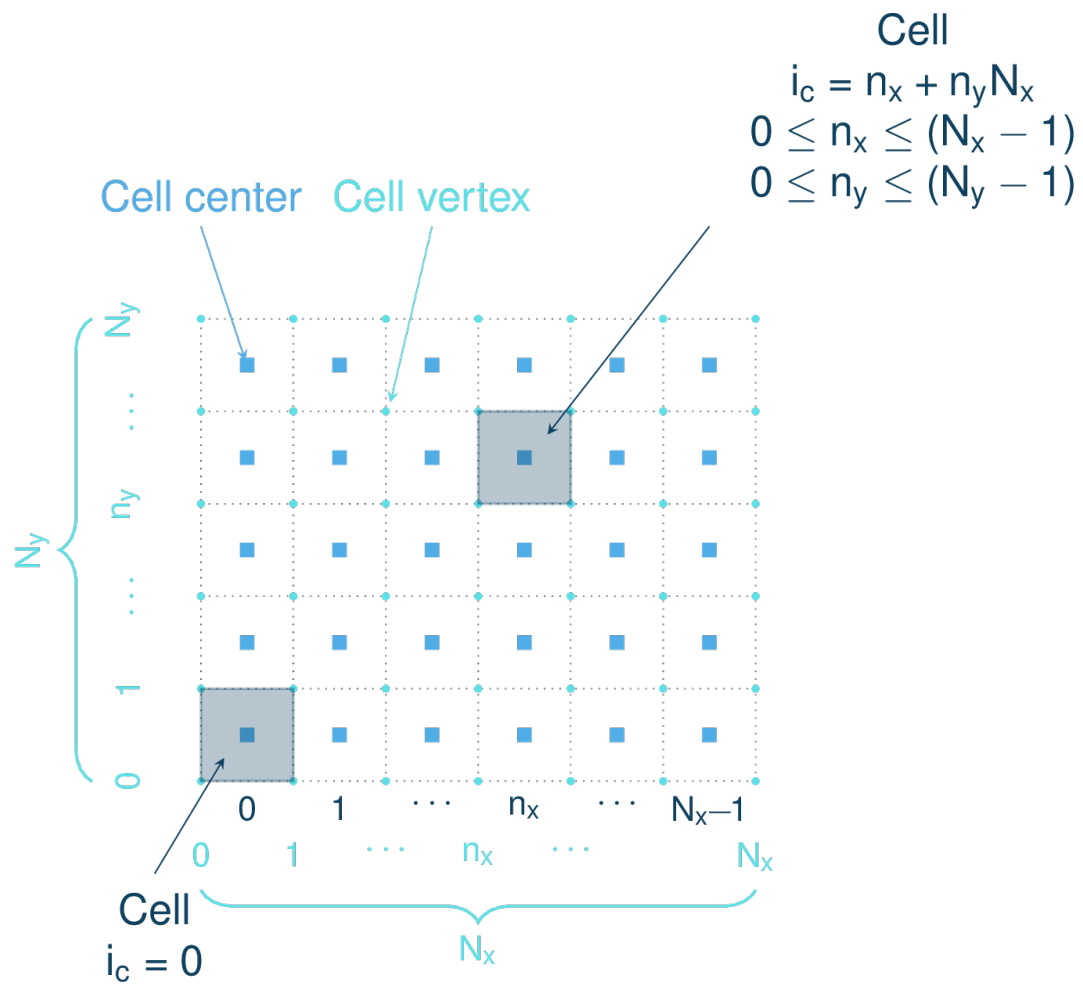


Fig. 6.5: Rectangular grid



## Cell normals

There are many different methods available for computing normal vectors on a 2D rectilinear mesh. Several options are available in com1DFA.

The first one consists in computing the cross product of the diagonal vectors between four cell centers. This defines the normal vector at the vertices. It is then possible to interpolate the normal vector at the cell centers from the ones at the vertices.

The other methods use the plane defined by different adjacent triangles to a cell center. Each triangle has a normal and the cell center normal is the average of the triangles normal vectors.

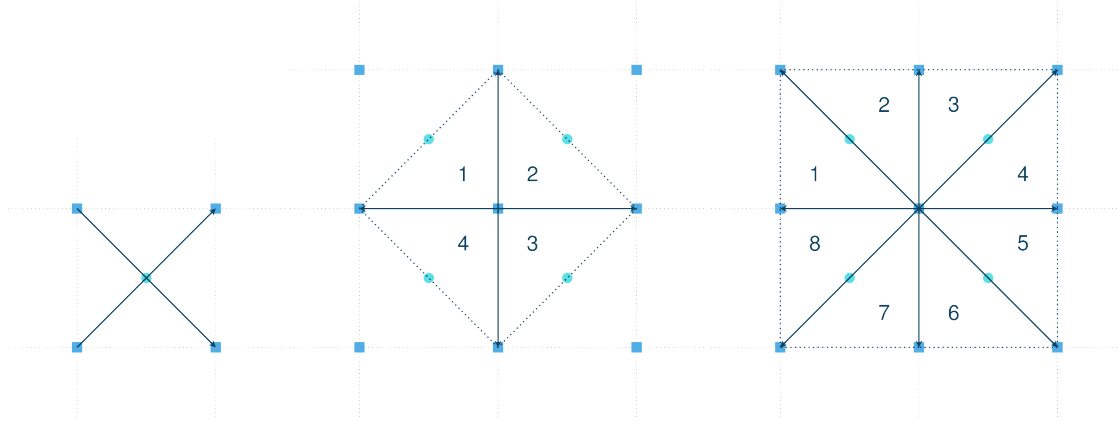


Fig. 6.6: Grid normal computation

## Cell area

The cell area can be deduced from the grid cellsize and the cell normal. A cell is a plane ( $z = ax + by + c$ ) of same normal as the cell center:

$$\mathbf{n} = \frac{1}{\sqrt{1 + a^2 + b^2}} \begin{pmatrix} -a \\ -b \\ 1 \end{pmatrix}$$

Surface integration over the cell extent leads to the area of the cell:

$$A_{cell} = \iint_S dS = \int_0^{csz} \int_0^{csz} \sqrt{1 + \frac{\partial z^2}{\partial x} + \frac{\partial z^2}{\partial y}} dx dy = csz^2 \sqrt{1 + \frac{\partial z^2}{\partial x} + \frac{\partial z^2}{\partial y}} = \frac{csz^2}{n_z}$$

### 6.2.2.2 Interpolation

In the DFA kernel, mass, flow thickness and flow velocity can be defined at particle location or on the mesh. We need a method to be able to go from particle properties to mesh (field) values and from mesh values to particle properties.

## Mesh to particle

On a 2D rectilinear mesh, scalar and vector fields defined at cell centers can be evaluated anywhere within the mesh using a bilinear interpolation between mesh cell centers. Evaluating a vector field simply consists in evaluating the three components as scalar fields.

The bilinear interpolation consists in successive linear interpolations in both  $x$  and  $y$  direction using the four nearest cell centers, two linear interpolations in the first direction (in our case in the  $y$  direction in order to evaluate  $f_{0v}$  and  $f_{1v}$ ) followed by a second linear interpolation in the second direction ( $x$  in our case to finally evaluate  $f_{uv}$ ) as shown on Fig. 6.7:

$$\begin{aligned} f_{0v} &= (1 - v)f_{00} + vf_{01} \\ f_{1v} &= (1 - v)f_{10} + vf_{11} \end{aligned}$$

and

$$\begin{aligned} f_{uv} &= (1 - u)f_{0v} + uf_{1v} \\ &= (1 - u)(1 - v)f_{00} + (1 - u)v f_{01} + u(1 - v)f_{10} + uv f_{11} \\ &= w_{00}f_{00} + w_{01}f_{01} + w_{10}f_{10} + w_{11}f_{11} \end{aligned}$$

the  $w$  are the bilinear weights. The example given here is for a unit cell. For no unit cells, the  $u$  and  $v$  simply have to be normalized by the cell size.

## Particles to mesh

Going from particle property to mesh value is also based on bilinear interpolation and weights but requires a bit more care in order to conserve mass and momentum balance. Flow thickness and velocity fields are determined on the mesh using, as intermediate step, mass and momentum fields. First, mass and momentum mesh fields can be evaluated by summing particles mass and momentum. This can be done using the bilinear weights  $w$  defined in the previous paragraph (here  $f$  represents the mass or momentum and  $f_{uv}$  is the particle value.  $f_{nm}$ ,  $n, m \in \{0, 1\} \times \{0, 1\}$ , are the cell center values):

$$\begin{aligned} f_{00} &= w_{00}f_{uv} \\ f_{01} &= w_{01}f_{uv} \\ f_{10} &= w_{10}f_{uv} \\ f_{11} &= w_{11}f_{uv} \end{aligned}$$

The contribution of each particle to the different mesh points is summed up to finally give the mesh value. This method ensures that the total mass and momentum of the particles is preserved (the mass and momentum on the mesh will sum up to the same total). Flow thickness and velocity mesh fields can then be deduced from the mass and momentum fields and the cell area (actual area of each grid cell, not the projected area).

### 6.2.3 Neighbor search

The lateral pressure forces are computed via the SPH flow thickness gradient. This method is based on particle interactions within a certain neighborhood, meaning that it is necessary to keep track of all the particles within the neighborhood of each particle. Computing the gradient of the flow thickness at a particle location, requires to find all the particles in its surrounding. Considering the number of particles and their density, computing the gradient ends up in computing a lot of interactions and represents the most computationally expensive part of the dense flow avalanche simulation. It is therefore important that the neighbor search is fast and efficient. [IOS+14] describe different rectilinear mesh neighbor search methods. In com1DFA, the simplest method is used. The idea is to locate each particle in a cell, this way, it is possible to keep track of the particles in each cell. To find the neighbors of a particle, one only needs to read the cell in which the particle is located (dark blue cell in Fig. 6.8), find the direct adjacent cells in all directions (light blue cells) and simply read all particles within those cells. This is very easily achieved on rectilinear meshes because locating a particle in a cell is straightforward and finding the adjacent cells is also easily done.

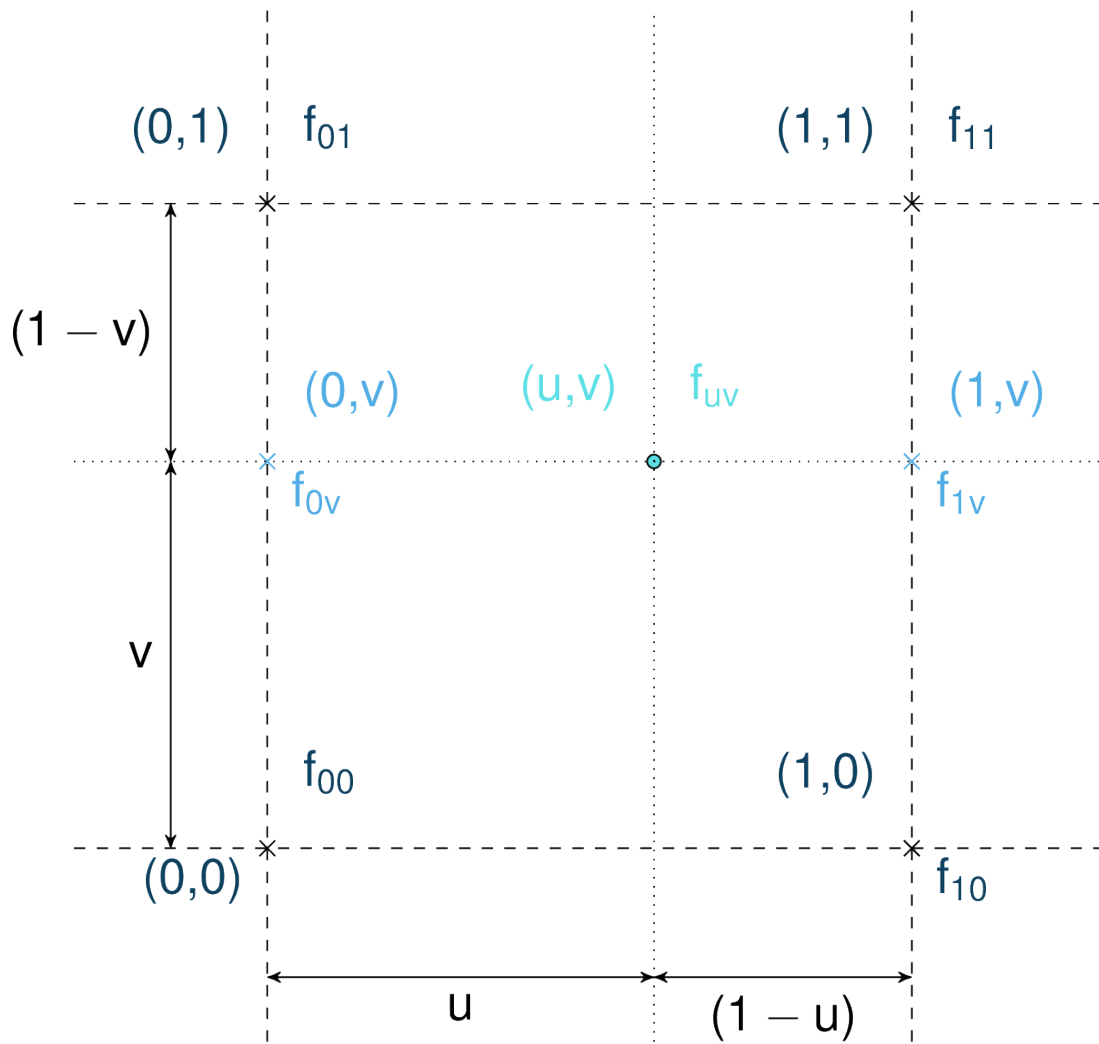


Fig. 6.7: Bilinear interpolation in a unit mesh (cell size is 1).

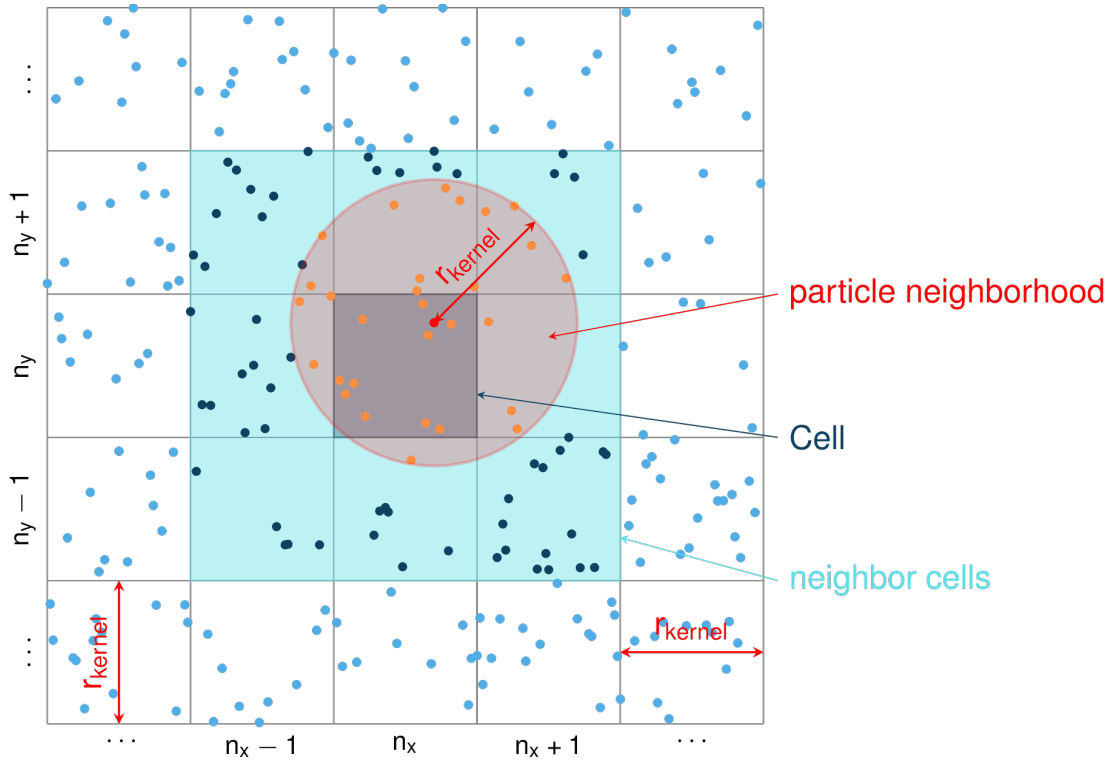


Fig. 6.8: Support mesh for neighbor search: if the cell side is bigger than the kernel length  $r_{kernel}$  (red circle in the picture), the neighbors for any particle in any given cell (dark blue square) can be found in the direct neighborhood of the cell itself (light blue squares)

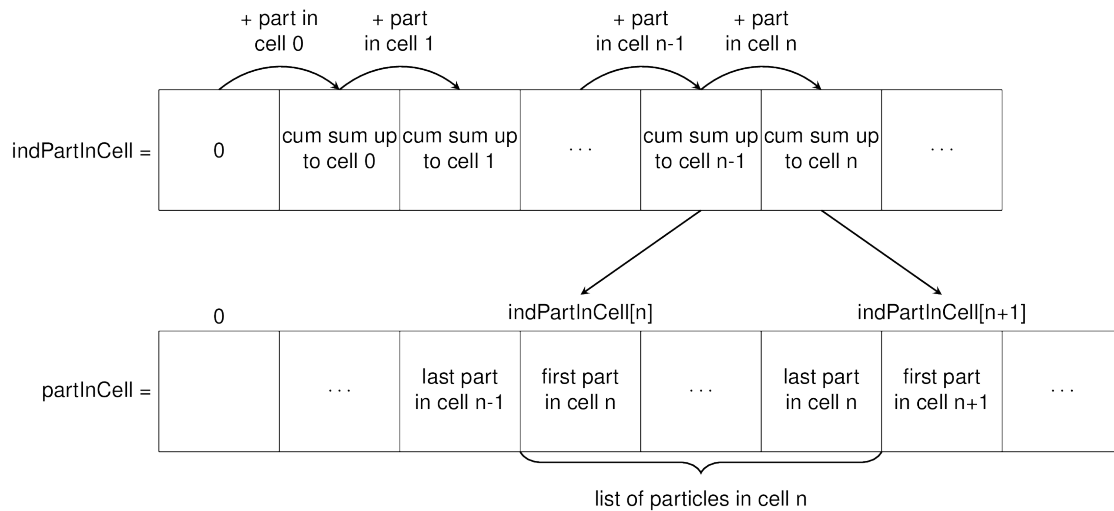


Fig. 6.9: The particles are located in the cells using two arrays. `indPartInCell` of size number of cells + 1 which keeps track of the number of particles in each cell and `partInCell` of size number of particles + 1 which lists the particles contained in the cells.

## 6.2.4 SPH gradient

SPH method can be used to solve thickness integrated equations where a 2D (respectively 3D) equation is reduced to a 1D (respectively 2D) one. This is used in ocean engineering to solve shallow water equations (SWE) in open or closed channels for example. In all these applications, whether it is 1D or 2D SPH, the fluid is most of the time, assumed to move on a horizontal plane (bed elevation is set to a constant). In the case of avalanche flow, the “bed” is sloped and irregular. The aim is to adapt the SPH method to apply it to thickness integrated equations on a 2D surface living in a 3D world.

### 6.2.4.1 Method

The SPH method is used to express a quantity (the flow thickness in our case) and its gradient at a certain particle location as a weighted sum of its neighbors properties. The principle of the method is well described in [LL10]. In the case of thickness integrated equations (for example SWE), a scalar function  $f$  and its gradient can be expressed as following:

$$\begin{aligned} f_k &= \sum_l f_l A_l W_{kl} \\ \nabla f_k &= - \sum_l f_l A_l \nabla W_{kl} \end{aligned} \quad (6.18)$$

Which gives for the flow thickness:

$$\begin{aligned} \bar{h}_k &= \frac{1}{\rho_0} \sum_l m_l W_{kl} \\ \nabla \bar{h}_k &= - \frac{1}{\rho_0} \sum_l m_l \nabla W_{kl} \end{aligned} \quad (6.19)$$

Where  $W$  represents the SPH-Kernel function.

The computation of its gradient depends on the coordinate system used.

### Standard method

Let us start with the computation of the gradient of a scalar function  $f: \mathbb{R}^2 \rightarrow \mathbb{R}$  on a horizontal plane. Let  $P_k = \mathbf{x}_k = (x_{k,1}, x_{k,2})$  and  $Q_l = \mathbf{x}_l = (x_{l,1}, x_{l,2})$  be two points in  $\mathbb{R}^2$  defined by their coordinates in the Cartesian coordinate system  $(P_k, \mathbf{e}_1, \mathbf{e}_2)$ .  $\mathbf{r}_{kl} = \mathbf{x}_k - \mathbf{x}_l$  is the vector going from  $Q_l$  to  $P_k$  and  $r_{kl} = \|\mathbf{r}_{kl}\|$  the length of this vector. Now consider the kernel function  $W$ :

$$\begin{aligned} W: \mathbb{R}^2 \times \mathbb{R}^2 \times \mathbb{R} &\rightarrow \mathbb{R} \\ (P_k, Q_l, r_0) &\mapsto W(P_k, Q_l, r_0) \end{aligned} \quad , r_0 \in \mathbb{R} \text{ is the smoothing kernel length}$$

In the case of the spiky kernel,  $W$  reads (2D case):

$$\begin{aligned} W_{kl} &= W(\mathbf{x}_k, \mathbf{x}_l, r_0) = W(\mathbf{x}_k - \mathbf{x}_l, r_0) = W(\mathbf{r}_{kl}, r_0) \\ &= \frac{10}{\pi r_0^5} \begin{cases} (r_0 - \|\mathbf{r}_{kl}\|)^3, & 0 \leq \|\mathbf{r}_{kl}\| \leq r_0 \\ 0, & r_0 < \|\mathbf{r}_{kl}\| \end{cases} \end{aligned} \quad (6.20)$$

$\|\mathbf{r}_{kl}\| = \|\mathbf{x}_k - \mathbf{x}_l\|$  represents the distance between particle  $k$  and  $l$  and  $r_0$  the smoothing length.

Using the chain rule to express the gradient of  $W$  in the Cartesian coordinate system  $(x_1, x_2)$  leads to:

$$\nabla W_{kl} = \frac{\partial W}{\partial r} \cdot \nabla r, \quad r = \|\mathbf{r}\| = \sqrt{(x_{k,1} - x_{l,1})^2 + (x_{k,2} - x_{l,2})^2} \quad (6.21)$$

with,

$$\frac{\partial W}{\partial r} = -3 \frac{10}{\pi r_0^5} \begin{cases} (r_0 - \|\mathbf{r}_{kl}\|)^2, & 0 \leq \|\mathbf{r}_{kl}\| \leq r_0 \\ 0, & r_0 < \|\mathbf{r}_{kl}\| \end{cases}$$

and

$$\frac{\partial r}{\partial x_{k,i}} = \frac{(x_{k,i} - x_{l,i})}{\sqrt{(x_{k,1} - x_{l,1})^2 + (x_{k,2} - x_{l,2})^2}}, \quad i = \{1, 2\}$$

which leads to the following expression for the gradient:

$$\nabla W_{kl} = -3 \frac{10}{\pi r_0^5} \begin{cases} (r_0 - \|\mathbf{r}_{kl}\|)^2 \frac{\mathbf{r}_{kl}}{r_{kl}}, & 0 \leq \|\mathbf{r}_{kl}\| \leq r_0 \\ 0, & r_0 < \|\mathbf{r}_{kl}\| \end{cases} \quad (6.22)$$

The gradient of  $f$  is then simply:

$$\nabla f_k = - \sum_l f_l A_l \nabla W_{kl} \quad (6.23)$$

## 2.5D SPH method

We now want to express a function  $f$  and its gradient on a potentially curved surface and express this gradient in the 3 dimensional Cartesian coordinate system  $(P_k, \mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3)$ .

Let us consider a smooth surface  $\mathcal{S}$  and two points  $P_k = \mathbf{x}_k = (x_{k,1}, x_{k,2}, x_{k,3})$  and  $Q_l = \mathbf{x}_l = (x_{l,1}, x_{l,2}, x_{l,3})$  on  $\mathcal{S}$ . We can define  $\mathcal{TP}$  the tangent plane to  $\mathcal{S}$  in  $P_k$ . If  $\mathbf{u}_k$  is the (none zero) velocity of the particle at  $P_k$ , it is possible to define the local orthonormal coordinate system  $(P_k, \mathbf{V}_1, \mathbf{V}_2, \mathbf{V}_3 = \mathbf{n})$  with  $\mathbf{V}_1 = \frac{\mathbf{u}_k}{\|\mathbf{u}_k\|}$  and  $\mathbf{n}$  the normal to  $\mathcal{S}$  at  $P_k$ . Locally,  $\mathcal{S}$  can be assimilated to  $\mathcal{TP}$  and  $Q_l$  to its projection  $Q'_l$  on  $\mathcal{TP}$ . The vector  $\mathbf{r}'_{kl} = \mathbf{x}_k - \mathbf{x}'_l$  going from  $Q'_l$  to  $P_k$  lies in  $\mathcal{TP}$  and can be express in the plane local basis:

$$\mathbf{r}'_{kl} = \mathbf{x}_k - \mathbf{x}'_l = v_{kl,1} \mathbf{V}_1 + v_{kl,2} \mathbf{V}_2$$

It is important to define  $f$  properly and the gradient that will be calculated:

$$\begin{aligned} f: \mathcal{TP} \subset \mathbb{R}^3 &\rightarrow \mathbb{R} \\ (x_1, x_2, x_3) &\mapsto f(x_1, x_2, x_3) = f(x_1(v_1, v_2), x_2(v_1, v_2)) = \tilde{f}(v_1, v_2) \end{aligned}$$

Indeed, since  $(x_1, x_2, x_3)$  lies in  $\mathcal{TP}$ ,  $x_3$  is not independent of  $(x_1, x_2)$ :

$$\begin{aligned} \tilde{f}: \mathcal{TP} \subset \mathbb{R}^2 &\rightarrow \mathbb{R} \\ (v_1, v_2) &\mapsto \tilde{f}(v_1, v_2) = \tilde{f}(v_1(x_1, x_2), v_2(x_1, x_2)) = f(x_1, x_2, x_3) \end{aligned}$$

The target is the gradient of  $\tilde{f}$  in terms of the  $\mathcal{TP}$  variables  $(v_1, v_2)$ . Let us call this gradient  $\nabla_{\mathcal{TP}}$ . It is then possible to apply the *Standard method* to compute this gradient:

$$\nabla_{\mathcal{TP}} W_{kl} = \frac{\partial W}{\partial r} \cdot \nabla_{\mathcal{TP}} r, \quad r = \|\mathbf{r}\| = \sqrt{v_{kl,1}^2 + v_{kl,2}^2} \quad (6.24)$$

Which leads to:

$$\nabla_{\mathcal{TP}} W_{kl} = -3 \frac{10}{\pi r_0^5} \frac{(r_0 - \|\mathbf{r}'_{kl}\|)^2}{r'_{kl}} \begin{cases} v_{kl,1} \mathbf{V}_1 + v_{kl,2} \mathbf{V}_2, & 0 \leq \|\mathbf{r}'_{kl}\| \leq r_0 \\ 0, & r_0 < \|\mathbf{r}'_{kl}\| \end{cases} \quad (6.25)$$

$$\nabla_{\mathcal{TP}} \tilde{f}_k = - \sum_l \tilde{f}_l A_l \nabla W_{kl} \quad (6.26)$$

This gradient can now be expressed in the Cartesian coordinate system. It is clear that the change of coordinate system was not needed:

$$\nabla_{\mathcal{TP}} W_{kl} = -3 \frac{10}{\pi r_0^5} \frac{(r_0 - \|\mathbf{r}'_{kl}\|)^2}{r'_{kl}} \begin{cases} r_{kl,1} \mathbf{e}_1 + r_{kl,2} \mathbf{e}_2 + r_{kl,3} \mathbf{e}_3, & 0 \leq \|\mathbf{r}'_{kl}\| \leq r_0 \\ 0, & r_0 < \|\mathbf{r}'_{kl}\| \end{cases}$$

The advantage of computing the gradient in the local coordinate system is if the components (in flow direction or in cross flow direction) need to be treated differently.

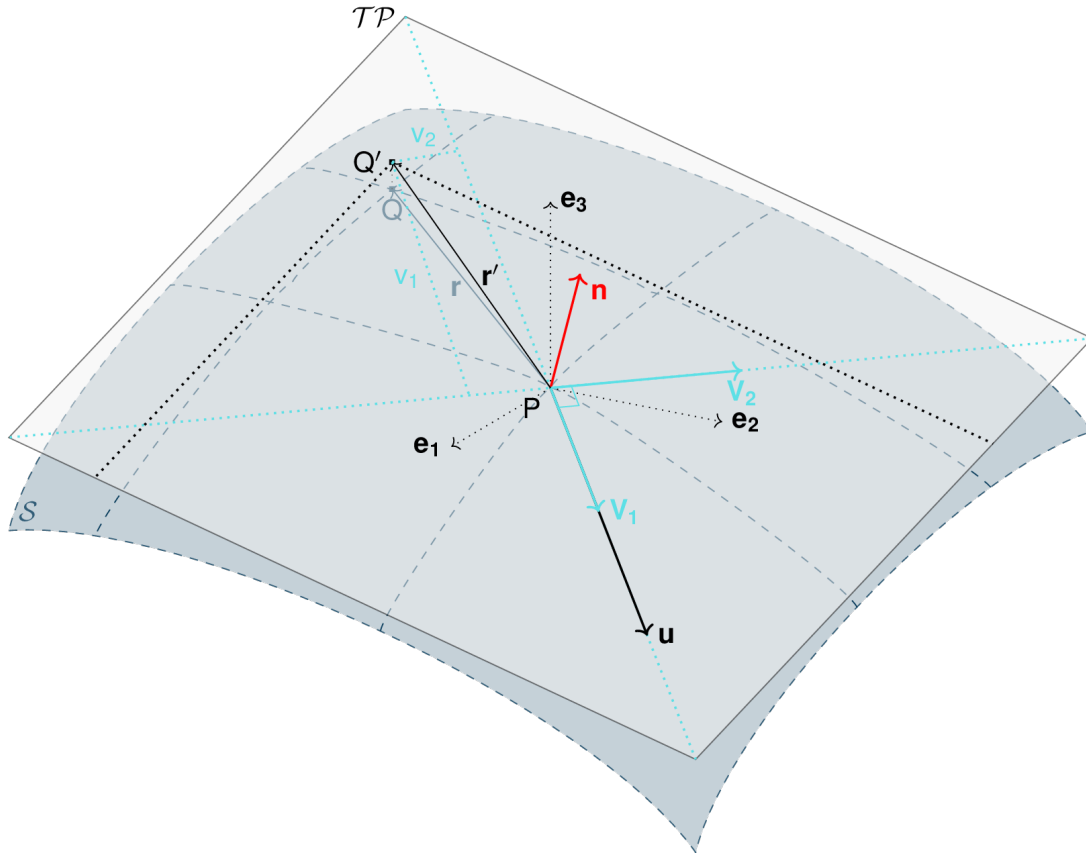


Fig. 6.10: Tangent plane and local coordinate system used to apply the SPH method

### 6.2.5 Particle splitting and merging

There are two different approaches treating splitting of particles in com1DFA. The first one only deals with splitting of particles with too much mass('split only'). The second approach, "split/merge" approach aims at keeping a stable amount of particles within a given range. This is done in order to guaranty a sufficient accuracy of the sph flow thickness gradient computation.

### 6.2.5.1 Split (default)

If the `splitOption` is set to 0, particles are split because of snow entrainment. In this case, particles that entrain snow grow, i.e. their mass increases. At one point the mass of the particles is considered to be too big and this particle is split in two. The splitting operation happens if the mass of the particle exceeds a threshold value ( $m_{Part} > massPerPart \times thresholdMassSplit$ ), where `thresholdMassSplit` is specified in the configuration file and `massPerPart` depends on the chosen `massPerParticleDeterminationMethod` as defined here: *Initialize particles*. When a particle is split a new child particle is created with the same properties as the parent apart from mass and position. Both parent and child get half of the parent mass. The parent and child's position are adjusted: the first / second is placed forward / backward in the direction of the velocity vector at a distance  $distSplitPart \times r_{Part}$  of the initial parent position. Particles are considered to have a circular basal surface  $A = \frac{m}{\rho} = \pi r^2$ .

### 6.2.5.2 Split and merge

If the `splitOption` is set to 1 particles are split or merged in order to keep the particle count as constant as possible within the kernel radius. Assessing the number of particles within one kernel radius is done based on the particle area. Particles are assumed to be cylindrical, i.e. the base is a circle. For particle  $k$  we have  $A_k = \frac{m_k}{\rho}$ . The area of the support domain of the sph kernel function is  $\pi r_0^2$ . The aim is to keep  $n_{PPK}$  particles within the kernel radius. The particles are split if the estimated number of particles per kernel radius  $\frac{\pi r_0^2}{A_k}$  falls below a given value of  $n_{PPK}^{min} = C_{n_{PPK}}^{min} n_{PPK}$ . Particles are split using the same method as in *Split (default)*. Similarly, particles are merged if the estimated number of particles per kernel radius exceeds a given value  $n_{PPK}^{max} = C_{n_{PPK}}^{max} n_{PPK}$ . In this case particles are merged with their closest neighbor. The new position and velocity is the mass averaged one. The new mass is the sum. Here, two coefficients  $C_{\{n_{PPK}\}^{min}}$  and  $C_{\{n_{PPK}\}^{max}}$  were introduced. A good balance needs to be found for the coefficients so that the particles are not constantly split or merged but also not too seldom. The split and merge steps happen only once per time step and per particle.

## 6.2.6 Artificial viscosity

Two options are available to add viscosity to stabilize the numerics. The first option consists in adding artificial viscosity (`viscOption` = 1). The second option attempts to adapt the Lax-Friedrich scheme (usually applied to meshes) to the particle method (`viscOption` = 2). Finally, `viscOption` = 0 deactivates any viscosity force.

### 6.2.6.1 SAMOS Artificial viscosity

In *Governing Equations for the Dense Flow Avalanche*, the governing equations for the DFA were derived and all first order or smaller terms were neglected. Among those terms is the lateral shear stress. This term leads toward the homogenization of the velocity field. It means that two neighbor elements of fluid should have similar velocities. The aim behind adding artificial viscosity is to take this phenomena into account. The following viscosity force is added:

$$\begin{aligned} \mathbf{F}_{viscosity} &= -\frac{1}{2} \rho C_{Lat} \|\mathbf{du}\|^2 A_{Lat} \frac{\mathbf{du}}{\|\mathbf{du}\|} \\ &= -\frac{1}{2} \rho C_{Lat} \|\mathbf{du}\| A_{Lat} \mathbf{du} \end{aligned}$$

Where the velocity difference reads  $\mathbf{du} = \mathbf{u} - \bar{\mathbf{u}}$  ( $\bar{\mathbf{u}}$  is the mesh velocity interpolated at the particle position).  $C_{Lat}$  is a coefficient that rules the viscous force. It would be the equivalent of  $C_{Drag}$  in the case of the drag force. The  $C_{Lat}$  is a numerical parameter that depends on the mesh size. Its value is set to 100 and should be discussed and further tested.



## Adding the viscous force

The viscous force acting on particle  $k$  reads:

$$\begin{aligned} \mathbf{F}_k^{\text{viscosity}} &= -\frac{1}{2}\rho C_{Lat} \|\mathbf{du}_k^{old}\| A_{Lat} \mathbf{du}_k^{new} \\ &= -\frac{1}{2}\rho C_{Lat} \|\mathbf{u}_k^{old} - \bar{\mathbf{u}}_k^{old}\| A_{Lat} (\mathbf{u}_k^{new} - \bar{\mathbf{u}}_k^{old}) \end{aligned}$$

Updating the velocity is done in two steps. First adding the explicit term related to the mean mesh velocity and then the implicit term which leads to:

$$\mathbf{u}_k^{new} = \frac{\mathbf{u}_k^{old} - C_{vis} \bar{\mathbf{u}}_k^{old}}{1 + C_{vis}}$$

With  $C_{vis} = \frac{1}{2}\rho C_{Lat} \|\mathbf{du}_k^{old}\| A_{Lat} \frac{dt}{m}$

### 6.2.6.2 Ata Artificial viscosity

#### An upwind method based on Lax-Friedrichs scheme

Shallow Water Equations are well known for being hyperbolic transport equations. They have the particularity of carrying discontinuities or shocks which will cause numerical instabilities.

A decentering in time allows to better capture the discontinuities. This can be done in the manner of the Lax-Friedrichs scheme as described in [AS05], which is formally the same as adding a viscous force. Implementing it for the SPH method, this viscous force applied on a given particle  $k$  can be expressed as follows:

$\mathbf{u}_{kl} = \mathbf{u}_k - \mathbf{u}_l$  is the relative velocity between particle  $k$  and  $l$ ,  $\mathbf{r}_{kl} = \mathbf{x}_k - \mathbf{x}_l$  is the vector going from particles  $l$  to particle  $k$  and  $\lambda_{kl} = \frac{c_k + c_l}{2}$  with  $c_k = \sqrt{gh_k}$  the wave speed. The  $\lambda_{kl}$  is obtained by turning expressions related to time and spatial discretization parameters into an expression on maximal speed between both particles in the Lax Friedrich scheme.

Due to the expression of the viscosity force, it makes sense to compute it at the same place where the SPH pressure force are computed (for this reason, the `viscOption = 2` corresponding to the ‘‘Ata’’ viscosity option is only available in combination with the `sphOption = 2`).

## 6.2.7 Forces discretization

### 6.2.7.1 Lateral force

The SPH method is introduced when expressing the flow thickness gradient for each particle as a weighted sum of its neighbors ([LL10, Sam07]). From now on the  $p$  for particles in  $p_k$  is dropped (same applies for  $p_l$ ).

The lateral pressure forces on each particle are calculated from the compression forces on the boundary of the particle. The boundary is approximated as a square with the base side length  $\Delta s = \sqrt{A_p}$  and the respective flow height. This leads to (subscript  $|\cdot, i$  stands for the component in the  $i^{th}$  direction,  $i = 1, 2$ ):

$$F_{k,i}^{\text{lat}} = K_{(i)} \oint_{\partial A_k} \left( \int_b^s \sigma_{33} n_i dx_3 \right) dl$$

From equation Eq.6.15

$$F_{k,i}^{\text{lat}} = K_{(i)} \frac{\Delta s}{2} \left( (\bar{h} \bar{\sigma}_{33}^{(b)})_{x_i - \frac{\Delta s}{2}} - (\bar{h} \bar{\sigma}_{33}^{(b)})_{x_i + \frac{\Delta s}{2}} \right) = K_{(i)} \frac{\Delta s^2}{2} \left. \frac{d \bar{h} \bar{\sigma}^{(b)}}{d x_i} \right|_k$$

The product of the average flow thickness  $\bar{h}$  and the basal normal pressure  $\bar{\sigma}_{33}^{(b)}$  reads (using equation Eq.6.14 and dropping the curvature acceleration term):

$$\bar{h} \bar{\sigma}^{(b)} = \bar{h}^2 \rho_0 \left( g_3 - \bar{u}_1^2 \frac{\partial^2 b}{\partial x_1^2} \right) \approx \bar{h}^2 \rho_0 g_3$$

Which leads to, using the relation Eq.6.18:

$$F_{k,i}^{\text{lat}} = K_{(i)} \rho_0 g_3 A_k \bar{h}_k \cdot \left. \frac{d \bar{h}}{d x_i} \right|_k = -K_{(i)} m_i g_3 \cdot \frac{1}{\rho_0} \sum_l m_l \left. \frac{d W_{kl}}{d x_i} \right|_l \quad (6.27)$$

### 6.2.7.2 Bottom friction force

The bottom friction forces on each particle depend on the chose friction model. Using the SamosAT friction model (using equation Eq.6.14 for the expression of  $\sigma_k^{(b)}$ ) the formulation of the bottom friction forec reads:

$$F_{k,i}^{\text{bot}} = -\frac{\bar{u}_{k,i}}{\|\bar{\mathbf{u}}_k\|} A_k \tau_k^{(b)} = -\delta_{k1} A_k \bar{h}_k \cdot \left( \tau_0 + \tan \delta \left( 1 + \frac{R_s^0}{R_s^0 + R_s} \right) \sigma_k^{(b)} + \frac{\rho_0 \bar{\mathbf{u}}_k^2}{\left( \frac{1}{\kappa} \ln \frac{\bar{h}}{R} + B \right)^2} \right) \quad (6.28)$$

### 6.2.7.3 Added resistance force

The resistance force on each particle reads (where  $h_k^{\text{eff}}$  is a function of the average flow thickness  $\bar{h}_k$ ):

$$F_{k,i}^{\text{res}} = -\rho_0 A_k h_k^{\text{eff}} C_{\text{res}} \|\bar{\mathbf{u}}_k\|^2 \frac{\bar{u}_{k,i}}{\|\bar{\mathbf{u}}_k\|} \quad (6.29)$$

Both the bottom friction and resistance force are friction forces. The expression above represent the maximal friction force that can be added. This maximal force is added if the particles are flowing. If not, the friction force equals the driving forces. See [MCVB+03] for more information.

### 6.2.7.4 Entrainment force

The term  $-\bar{u}_i \rho_0 \frac{d(A \bar{h})}{dt}$  related to the entrained mass in Eq.6.3 now reads:

$$-\bar{u}_{k,i} \rho_0 \frac{d}{dt} (A_k \bar{h}_k) = -\bar{u}_{k,i} A_k^{\text{ent}} q_k^{\text{ent}}$$

The mass of entrained snow for each particle depends on the type of entrainment involved (plowing or erosion) and reads:

$$\rho_0 \frac{d}{dt} (A_k \bar{h}_k) = \frac{d m_k}{dt} = A_k^{\text{ent}} q_k^{\text{ent}}$$

with

$$\begin{aligned} A_k^{\text{plo}} &= w_f h_k^{\text{ent}} = \sqrt{\frac{m_k}{\rho_0 \bar{h}_k}} h_k^{\text{ent}} & \text{and} & \quad q_k^{\text{plo}} = \rho_{\text{ent}} \|\bar{\mathbf{u}}_k\| & \text{for plowing} \\ A_k^{\text{ero}} &= A_k = \frac{m_k}{\rho_0 \bar{h}_k} & \text{and} & \quad q_k^{\text{ero}} = \frac{\tau_k^{(b)}}{e_b} \|\bar{\mathbf{u}}_k\| & \text{for erosion} \end{aligned}$$

Finally, the entrainment force reads:

$$F_{k,i}^{\text{ent}} = -w_f (e_s + q_k^{\text{ent}} e_d)$$

## 6.2.8 Adding forces

The different components are added following an operator splitting method. This means particle velocities are updated successively with the different forces.

### 6.2.8.1 Adding artificial viscosity

If the viscosity option (`viscOption`) is set to 1, artificial viscosity is added first, as described in [Artificial viscosity](#) (this is the default option). With `viscOption` set to 0, no viscosity is added. Finally, if `viscOption` is set to 2, artificial viscosity is added during SPH force computation. (TODO add link to description)

### 6.2.8.2 Adding entrainment

Entrainment is taken into account by first adding the component representing the loss of momentum due to acceleration of the entrained mass  $-\bar{u}_{k,i} A_k^{\text{ent}} q_k^{\text{ent}}$ . Second by adding the force due to the need to break and compact the entrained mass ( $F_{k,i}^{\text{ent}}$ ) as described in [Entrainment force](#).

### 6.2.8.3 Adding driving forces

The driving forces -gravity force and lateral forces- are taken into account next. The velocity is updated explicitly.

### 6.2.8.4 Adding friction forces

Both the bottom friction and resistance forces act against the flow. Two methods are available to add these forces in `com1DFA`.

#### An implicit method:

$$\mathbf{u}_k^{\text{new}} = \frac{\mathbf{u}_k^{\text{old}}}{1 + \frac{C_k^{\text{fric}} \Delta t}{m_k}}$$

where  $F_{k,i}^{\text{fric}} = C_k^{\text{fric}} u_{k,i}^{\text{new}} = F_{k,i}^{\text{res}} + F_{k,i}^{\text{bot}}$  (the two forces are described in [Bottom friction force](#) and [Added resistance force](#)).

This implicit method has a few draw-backs. First the flow does not start properly if the friction angle  $\delta$  is too close to the slope angle. Second, the flow never properly stops, even if the particles physically should, i.e. particles keep oscillating back and forth around their end position.

#### An explicit method:

The method based on [MCVB+03] addresses these two issues. The idea is that the friction forces only modify the magnitude of velocity and not the direction. This means dissipation, so the friction force can not become a driving force. Moreover, the friction force magnitude depends on the particle state, i.e. if it is flowing or at rest. The friction force is expressed:

$$\mathbf{F}_k^{\text{fric}} = -\|\mathbf{F}_k^{\text{fric}}\| \frac{\mathbf{u}_k}{\|\mathbf{u}_k\|}$$

with:

$$\|\mathbf{F}_k^{\text{fric}}\| \leq \|\mathbf{F}_k^{\text{fric}}\|_{\text{max}}$$

If the velocity of the particle  $k$  reads  $\mathbf{u}_k^{\text{old}}$  after adding the driving forces, adding the friction force leads to :

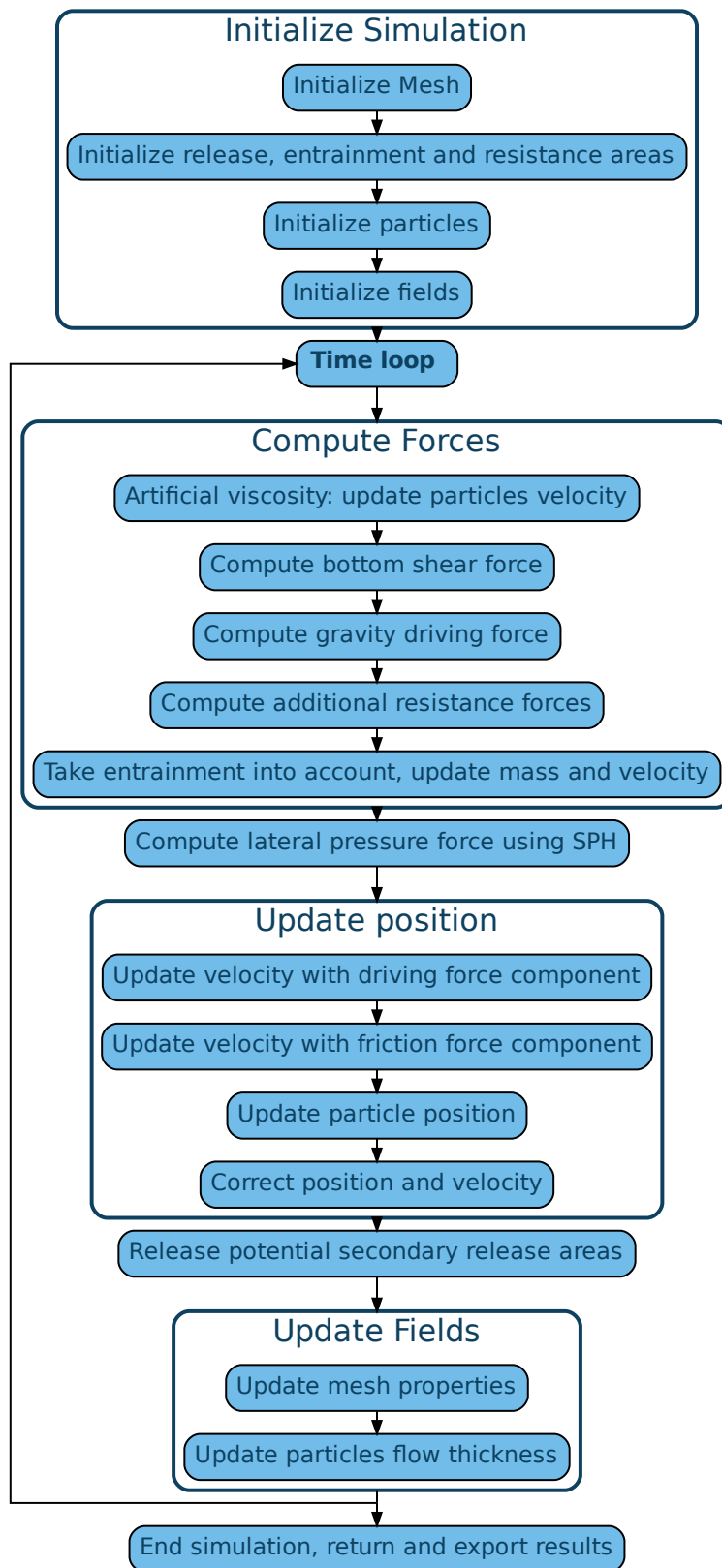
$$\mathbf{u}_k = \mathbf{u}_k^{\text{old}} \left(1 - \frac{\Delta t}{m} \frac{\|\mathbf{F}_k^{\text{fric}}\|}{\|\mathbf{u}_k^{\text{old}}\|}\right), \quad \|\mathbf{F}_k^{\text{fric}}\| = \|\mathbf{F}_k^{\text{fric}}\|_{\text{max}}$$

at the condition that  $1 \geq \frac{\Delta t}{m} \frac{\|\mathbf{F}_k^{\text{fric}}\|_{\text{max}}}{\|\mathbf{u}_k^{\text{old}}\|}$ . If on the contrary  $1 < \frac{\Delta t}{m} \frac{\|\mathbf{F}_k^{\text{fric}}\|_{\text{max}}}{\|\mathbf{u}_k^{\text{old}}\|}$ , the friction would change the velocity direction which is nonphysical. In this case, the particle will stop before the end of the time step. This allows the particles to start and stop flowing properly.

## 6.3 com1DFA Algorithm and workflow

### 6.3.1 Algorithm graph

The following graph describes the Dense Flow Avalanche simulation workflow (the different nodes are clickable and link to the detailed documentation)



### 6.3.2 Initialization:

At the beginning of the simulation, the avalanche folder and the configuration are read (configuration:Configuration). Input data is fetched according to the chosen configuration. Mesh, particles and fields are subsequently initialized.

#### 6.3.2.1 Initialize Mesh

Read DEM ascii file provided in the Input folder (only one DEM ascii file allowed). If the DEM cell size is different from the `meshCellSize` specified in the configuration from more than `meshCellSizeThreshold` [m] the DEM is remeshed (`in3Trans.geoTrans.remesh()`).

Prepare DEM for simulation, compute surface normals vector field, cell area (*Mesh*).

This is done in the `com1DFA.com1DFA.initializeMesh()` function.

Go back to *Algorithm graph*

#### 6.3.2.2 Initialize release, entrainment and resistance areas

Read and check shapefiles according to the configuration (check consistency between what is required by the configuration file and what is available in the Inputs folder). Convert shapefile features (polygons) to rasters (*in3Utils.geoTrans.prepareArea()*). Check consistency of rasters according to the following rules:

- multiple release features in the release and secondary release shapefiles are allowed but they should not overlap. If they do, simulation terminates with an error message.
- secondary release and entrainment rasters should no overlap between each other or with the main release. If they do, the overlapping part is removed. Order of priority is: main release, secondary release, entrainment area.

Go back to *Algorithm graph*

#### 6.3.2.3 Initialize Dam

If the dam option is activated (which is the case in the default configuration `dam` is `True`) AND the dam input shapefile is provided, the dam is initialized and will be take into account for in the DFA computation.

#### 6.3.2.4 Initialize particles

Particles are initialized according to the release raster extracted from the release shapefile and the mass per particle determination method (`massPerParticleDeterminationMethod`) specified in the configuration. The mass per particle determination method can be chosen between:

- MPPDIR= mass per particle direct. The `massPerPart` value is taken from the configuration and is the same for all cells.
- MPPDH= mass per particles through release thickness. The `massPerPart` value is computed using the release thickness per particle `deltaTh` value given in the configuration and the area of the release mesh cell:  $\text{massPerPart} = \rho \times \text{cellArea} \times \text{deltaTh}$ .
- MPPKR= mass per particles through number of particles per kernel radius. There is no `massPerPart` since it can vary from one cell to another depending on the release thickness of the cells. The aim of this method is to ensure a constant density of particles within the snow domain (nPPK particles per kernel radius is the target value). This is related to the SPH method used for computing the flow thickness gradient. It requires a sufficient number of particles to properly approximate the flow thickness gradient. It makes the most sense to combine the MPPKR particle initialization method with the *splitOption* 1. In this combination, the particles will be merged or split to keep a constant density of particles per kernel radius (*Particle splitting and merging*).

---

**Note:** If MPPDIR is used, consider adapting the mass per particle value when changing the mesh cell size from the default. This is important because, when using MPPDIR, the total number of particles is independent of the cell size. Hence, reducing the cell size results in less particles per cell, whereas when using MPPDH, the number of particles per cell is fixed (considering the respective release thickness and deltaTh value). Reducing the cell size will increase the total number of particles but not the number of particles per cell. Finally, using the MPPKR method, the number of particles per cell is independent from both cell size and release thickness (nPPK particles per kernel radius is the target value).

---

The number of particles placed in each release cell `nPartPerCell` is computed according to the `massPerPart` or `nPPK` depending on the `massPerParticleDeterminationMethod` chosen and the area and/or release thickness of the cell. The number should be an integer meaning that the float is rounded up or down with a probability corresponding to the decimal part (i.e. 5.7 will be rounded to 6 with a probability of 0.7 and 5 with a probability of 0.3). This ensures a better match with the desired `massPerPart` value.

There are then different ways to place the particles in the cells. This is decided by the `initPartDistType` parameter in the configuration file:

- **random** initialization: The `nPartPerCell` particles are placed randomly within each release cell. This initialization method allows steps of ones in the choice of `nPartPerCell`, which enables to avoid large jumps in the mass of the particles or the number of particles per Cell between one release cell to another. Due to the random initialization process, some particles cluster can appear. For operational applications, this does not seem to have a significant impact since the particles will redistribute in the first few time steps due to the pressure gradient. For some specific research applications (e.g. the dam break test), clusters might disturb the results. In this case, applying an initialization reprocessing can help (see `iniStep` in the configuration file). The random number generator is controlled by a random seed which ensures the possibility to reproduce the DFA results.
- **uniform** initialization: The release (square) cell is divided into (square) subcells (as many as `nPartPerCell`). The particles are placed in the center of each subcell. This method allows only steps of 1, 4, 9, 16, ... due to the square subdivision of the cells. This can lead, in some cases, to significant variations in the number of particles per cell or in the mass of the particles. This can then lead to spurious numerical artifacts. The particles are aligned with the grid, which can also lead to some spurious numerical artifacts.
- **semirandom** initialization: This method uses the uniform division of the release cell into subcells but the particles are placed randomly within each subcell. This method has the same disadvantage as the **uniform** method, it allows only steps of 1, 4, 9, 16, ... due to the square subdivision of the cells. This can lead, in some cases, to significant variations in the number of particles per cell or in the mass of the particles. But due to the random position of particles in the subcells particles are not aligned anymore with the grid.
- **triangular** initialization: The particles are initialized along a regular triangular mesh within the release area. This allows a regular distribution of particles with steps of one (like in the case of **random** initialization). The disadvantages are that the same triangle size is used within the whole release leading in some cases to significant differences in the mass of the particles. The same comment applies if multiple release features are used. Finally, this method only works for constant release thickness.

---

**Note:** This initialization option is meant to be used when the snow slide option is activated (`cohesion` activated). The behavior in the standard case (`cohesion` deactivated) has not been tested.

---

## Particle properties

Other particles properties are also initialized here:

- **x, y, z** - coordinates
- **m** - mass of particle [kg]
- **h** - flow thickness [m]
- **ux, uy, uz** - velocity components [ms<sup>-1</sup>]
- **uAcc** - approximation for particle acceleration between each computational time step (simply computed by  $(\text{velocityMagnitude\_t0} - \text{velocityMagnitude\_t1}) / \text{dt}$ )
- **trajectoryLengthXY** - traveled length of particle accumulated over time in xy plane
- **trajectoryLengthXYCor** - trajectoryLengthXY corrected with the angle difference between the slope and the normal
- **trajectoryLengthXYZ** - traveled length of a particle accumulated over time in xyz
- **travelAngle** - travel angle computed using  $\arctan((z_0 - z) / \text{trajectoryLengthXY})$
- **ID, parentID** - particle IDs and parentID required if splitting, merging
- **t** - corresponding time step

For more details, see `com1DFA.com1DFA.initializeParticles()`.

Go back to [Algorithm graph](#)

### 6.3.2.5 Initialize fields

All fields (mesh values defined as a raster) are initialized. Flow velocity, pressure, peak flow velocity and peak pressures are set to zero. Flow thickness and peak flow thickness are set according to the initial particle distribution. See `com1DFA.com1DFA.initializeFields()`

Go back to [Algorithm graph](#)

### 6.3.3 Time scheme and iterations:

The mass and momentum equations described in *Governing Equations for the Dense Flow Avalanche* are solved numerically in time using an operator splitting method. The different forces involved are sequentially added to update the velocity (see [Adding forces](#)). Position is then updated using a centered Euler scheme. The time step can either be fixed or dynamically computed using the Courant–Friedrichs–Lewy (CFL) condition (in the second case one must set `cflTimeStepping` to `True` and set the desired CFL coefficient).

Go back to [Algorithm graph](#)



### 6.3.4 Compute Forces:

This section gives an overview of the different steps to compute the forces acting on the snow particles. Those forces are separated in several terms: A gravity driving force ( $F_{drive}$ ), a friction force ( $F_{fric}$ ), an entrainment force (related to the entrained mass of snow) and an artificial viscous force. Those forces are computed by the two following functions `com1DFA.DFAfunctionsCython.computeForceC()` and `com1DFA.DFAfunctionsCython.computeForceSPHC()`.

Go back to [Algorithm graph](#)

#### 6.3.4.1 Artificial viscosity

This viscous friction force is artificially added to the numerical computation. The aim of this force is to stabilize the simulation and prevent neighbor particles to have too significantly different velocities. Physically, this force also makes sense and corresponds to some second order forces that were neglected (lateral shear stress) as explained in [Artificial viscosity](#). This force is controlled by the `subgridMixingFactor` in the configuration file. Setting this parameter to 0 deactivates the artificial viscosity term. The default value (set to 100) does not have any physical foundation yet. Future work will help defining this parameter in a more physical way. Remember that the artificial viscosity is dependent on the grid cell size.

The velocity is updated immediately after using an explicit/implicit formulation.

Go back to [Algorithm graph](#)

#### 6.3.4.2 Compute friction forces

The friction force encompasses all forces that oppose the motion of the particles. One of those forces is the bottom shear force. The other is an optional resistance force. Both components are added to the  $F_{fric}$  force term.

This force accounts for the friction between the snow particles and the bottom surface. The expression of the bottom shear stress depends on the friction model chosen but can be written in the following general form,  $\tau_i^{(b)} = f(\sigma^{(b)}, \bar{u}, \bar{h}, \rho_0, t, \mathbf{x})$ . The friction model is set by the `frictModel` value and the corresponding parameters can be set in the configuration file. More details about the different friction models are given in [Friction Model](#). Be aware that the normal stress on the bottom surface  $\sigma^{(b)}$  is composed of the normal component of the gravity force and the curvature acceleration term as shown in Eq.6.14. It is possible to deactivate the curvature acceleration component of the shear stress by setting the `curvAcceleration` coefficient to 0 in the configuration file.

An additional friction force called resistance can be added. This force aims to model the added resistance due to the specificity of the terrain on which the avalanche evolves, for example due to forests. To add a resistance force, one must provide a resistance shapefile in the `Inputs/RES` folder and switch the `simType` to `res`, `entres` or `available` to take this resistance area into account. Then, during the simulation, all particles flowing through this resistance area will undergo an extra resistance force. More details about how this force is computed and the different parameters chosen are found in [Resistance](#).

Go back to [Algorithm graph](#)

#### 6.3.4.3 Compute driving force

This force takes into account the gravity force, which is the driving force of the snow motion. The expression of this force is rather simple, it represents the tangential (tangent to the surface) part of the gravity force (the normal part of the force is accounted for in the friction term).

Go back to [Algorithm graph](#)

#### 6.3.4.4 Take entrainment into account

Snow entrainment can be added to the simulation. One must provide an entrainment shapefile in `Inputs/ENT` and set the `simType` to `ent`, `entres` or `available` (see [Initialize release, entrainment and resistance areas](#)). In the entrainment areas defined by the entrainment shapefile, particles can entrain mass through erosion or plowing. In both mechanisms, one must account for three things:

- change of mass due to the entrainment
- change of momentum - entrained snow was accelerated from rest to the speed of the avalanche
- loss of momentum due to the plowing or erosion processes -entrained mass bounds with the ground needs to be broken

These three terms are further detailed in [Entrainment](#). The parameters used to compute these processes can be set in the configuration file.

In the numerics, the mass is updated according to the entrainment model in `com1DFA.DFAfunctionsCython.computeEntMassAndForce()`. The velocity is updated immediately after using an implicit formulation.

Go back to [Algorithm graph](#)

#### 6.3.4.5 Compute lateral pressure forces

The lateral pressure forces ( $F_{SPH}$ ) are related to the gradient of the flow thickness ([Forces discretization](#)). This gradient is computed using a smoothed particle hydrodynamic method ([SPH gradient](#)).

Go back to [Algorithm graph](#)

### 6.3.5 Update position

Driving force, lateral pressure force and friction forces are subsequently used to update the velocity. Then the particle position is updated using a centered Euler scheme. These steps are done in `com1DFA.DFAfunctionsCython.updatePositionC()`.

#### 6.3.5.1 Take gravity and lateral pressure forces into account

$F_{drive}$  and  $F_{SPH}$  are summed up and taken into account to update the velocity. This is done via an explicit method.

### 6.3.5.2 Take friction into account

$F_{fric}$  is taken into account to update the velocity. This is done via an implicit method.

### 6.3.5.3 Update particle position

The particles position is updated using the new velocity and a centered Euler scheme:

$$\mathbf{x}_{new} = \mathbf{x}_{old} + dt0.5(\mathbf{u}_{old} + \mathbf{u}_{new})$$

### 6.3.5.4 Take dam interaction into account

If the dam option is activated, the interaction between the particles and the dam is taken into account. During the computation of the DFA simulation, at each time step, if a particle enters one of the dam cells, the dam algorithm is called. The first step is to check if the particle crosses the dam during the time step. If not, the particle position and velocity are updated as if there was no dam. If yes, the intersection point between the particle trajectory and the dam line is computed. The dam properties are interpolated at the intersection point (dam tangent and normal vectors). The wall tangent and normal vectors are updated taking the flow thickness into account. The particle position and velocity are updated taking the dam into account.

Let  $\mathbf{x}_{foot}$  and  $\mathbf{x}_{new}$  be the intersection point with the dam and the new particle position vector if there was no dam.  $\mathbf{u}_{new}$  is the new particle velocity with no dam. First the position  $\mathbf{x}_b$  after elastic bouncing of the particle on the dam is computed:

$$\mathbf{x}_b = \mathbf{x}_{new} - 2 \{ \mathbf{n}_w^{filled} \cdot (\mathbf{x}_{new} - \mathbf{x}_{foot}) \} \mathbf{n}_w^{filled}$$

Which gives the direction  $\mathbf{e}_b$ :

$$\mathbf{e}_b = \mathbf{x}_b - \mathbf{x}_{foot}$$

Next, the restitution coefficient is accounted for which leads to the new velocity:

$$\mathbf{x}_{new}^* = \mathbf{x}_{foot} - \alpha_{rest} \mathbf{e}_b = \alpha_{rest} \mathbf{x}_b + (1 - \alpha_{rest}) \mathbf{x}_{foot}$$

The velocity vector  $\mathbf{u}_{new}^*$  after the dam interaction reads:

$$\mathbf{u}_{new}^* = (\mathbf{u}_{new} - 2 \{ \mathbf{n}_w^{filled} \cdot \mathbf{u}_{new} \} \mathbf{n}_w^{filled}) \alpha_{rest}$$

Finally, the new velocity and position are re-projected onto the topography.

### 6.3.5.5 Correction step:

The particles z coordinate is readjusted so that the particles lie on the surface of the slope. There are two reasons why the particles might not lie on the surface anymore after updating their position according to the computed velocities:

- 1) because of the inaccuracy related to the time and space discretization. This can lead to a particle position being slightly above or under the surface. We want to correct this inaccuracy and therefore reproject the particle on the surface using its x and y coordinates.
- 2) because of the curvature of the slope and the particle velocity, particles can become detached from the ground in - in this case, the particle is located above the surface. In the current state, the com1DFA kernel does not allow this. If a particle becomes detached, the particle is also reprojected onto the surface using its x and y coordinates.

Similarly, the particles velocity is corrected in order to ensure that it lies in the tangent plane to the surface (the velocity vector magnitude is preserved, only the direction is changed).

The way the particles position is reprojected onto the surface does not allow both the velocity magnitude and the particle displacement to match perfectly. This is amplified by highly curved topographies or abrupt changes in slope.

Go back to [Algorithm graph](#)

### 6.3.6 Add secondary release area

If a secondary release area is provided, the flow thickness field from the previous time step is used to release a potential secondary release area. To do so, the flow thickness field is compared to the secondary release area rasters. If they overlap, the secondary release area is triggered and the secondary release particles are initialized and added to the flowing particles.

Go back to [Algorithm graph](#)

### 6.3.7 Update fields

This steps are done in `com1DFA.DFAfunctionsCython.updateFieldsC()`.

The mesh values are updated with the particles properties using [particles to mesh interpolation](#) methods. This is used to compute flow thickness, flow velocity and pressure fields from the particle properties.

#### 6.3.7.1 Update particles flow thickness

The mesh flow thickness is finally used to update the particle flow thickness value using [mesh to particle interpolation](#) methods.

Go back to [Algorithm graph](#)

### 6.3.8 Simulation outputs

At the end of the simulation, the result fields are exported to .asc files in `com1DFA.com1DFA.exportFields()` and the information gathered in an info dictionary is used to create a markdown report (`log2Report.generateReport.writeReport()`). Further details on the available outputs can be found in [Output](#).

Go back to [Algorithm graph](#)

## 6.4 com4FlowPy Theory

![[Image]](img/Motivation\_2d.png)

*Fig. 1: Definition of angles and geometric measures for the calculation of  $Z^{\sup}&\delta;$ , where  $s$  is the projected distance along the path and  $z(s)$  the corresponding altitude.*

The model equations that determine the run out in three dimensional terrain are mainly motivated with respect to simple, geometric, two dimensional concepts [0,3,4] in conjunction with ideas existing algorithms for flow routing in three dimensional terrain [1,2], controlling the main routing and final stopping of the flow.

Figure 1 summarizes the basic concept of a constant run out angle ( $\alpha$ ) with the corresponding geometric relations in two dimensions along a possible process path.

![[tan\_alpha]](img/tan\_alpha.png)

The local travel angle  $\gamma$  is defined by the altitude difference and projected distance along the path, from the release point to the current location.

![tan\_gamma](img/tan\_gamma.png)

The angle  $\delta$  is the difference between the local travel angle  $\gamma$  and the runout angle  $\alpha$  and is related to  $Z^{\delta}$ , so when  $\delta$  equals zero or  $\gamma$  equals  $\alpha$ , the maximum runout distance is reached.

![z\_alpha](img/z\_alpha.png)

$Z^{\alpha}$  can be interpreted as dissipation energy.

![z\_gamma](img/z\_gamma.png)

$Z^{\gamma}$  is the altitude difference between the starting point and the current calculation step at the projected distance  $s$ .

$Z^{\delta}$  is the difference between  $Z^{\gamma}$  and  $Z^{\alpha}$ , so when  $Z^{\delta}$  is lower or equal zero the stopping criterion is met and the flow stops.  $Z^{\delta}$  is associated to the process magnitude and can be interpreted as the kinetic energy or velocity of the process.

![z\_delta](img/z\_delta.png)

The major drawback of implementing the geometric runout angle concepts is that they require a predefined flow path in two dimensional terrain. To allow for an enhanced routing in three dimensional terrain without prior knowledge of the flow path we combine these concepts [4] with extensions of existing algorithms [1,2,3] that are described in the following sections.

### ## Spatial Input and Iterative Calculation Steps on the Path

In nature a GMF has one or more release areas that span over single or multiple release cells. Flow-Py computes the so called path, which is defined as the spatial extent of the routing from each release cell. Each release area (single raster cell in release area layer) has its own unique path (collection of raster cells), and a location on the terrain (a single raster cell) can belong to many paths. Flow-Py identifies the path with spatial iterations starting with a release area raster cell and only iterating over cells which receive routing flux. The corresponding functions are implemented in the code in the `flow_class.calc_distribution()` function.

To route on the surface of the three dimensional terrain, operating on a quadrilateral grid, we implement the geometric concepts that have been sketched in the model motivation utilizing the following cell definitions:

![grid\_overview](img/Neighbours.png)

*Fig. 2: Definition of parent, base, child and neighbors, as well as the indexing around the base.*

Each path calculation starts with a release cell and operates on the raster, requiring the definition of parent, base, child and neighbor cells (see Fig. 2). The base cell is the cell being calculated on the current spatial iteration step. The 8 raster cells surrounding the base cell are called neighbor cells ( $n, i$ ) which have the potential to be parents (supplying flux to base cell), or a child (receive flux from the base cell). In 2d the base cell corresponds to the cell/location at the distance  $s$  along the path in Fig. 1.

Every base has at least one parent cell, except in the first calculation step from the release cell, where we start our calculation, this would be at  $s = s_{>0}$  in Fig. 1.

During an iteration step a raster cell from the iteration list is identified as the current base cell. The routing flux is calculated across the base cell from the parent cell to possible child cells. The goal is to keep the spatial iteration steps to a minimum, which is achieved by only adding neighbor cells to the iteration list that have flux routed to them from the base cell and do not meet either of the stopping conditions. These cells are called child cells. Child cells that are not already on the iteration list are added to the list and `flow_class` python object is created for the raster cell. The child cells `flow_class` has the parent added to it as a source for routing flux. By being added to the iteration list the cell has been recognized as being part of the GMF path and will be the base cell for a future iteration step.

When the iteration list is empty and all potential children fulfill one of the stopping criteria:

- $Z^{\Delta}$  has to be smaller than zero:  $Z^{\Delta} < 0$ ,
- Routing Flux has to be smaller than the flux cut off:  $R_i < R_{\text{Stop}}$ ,

the path calculation is finished. The required information is saved from the cell class to the summarizing output raster files. Then the calculation starts again for the next release cell and respective flow path. The spatial extent and magnitude for all release cells are summarized in the output raster files, which represent the overlay of all paths.

Every path is independent from the other, but depending on the information we want to extract, we save the highest values (e.g.  $Z^{\Delta}$ ) or sums (e.g. Cell Counts) of different paths to the output raster file.

###  $Z^{\Delta}$

For each base cell in a path we solve the equations (6,7 and 8) for every neighbor  $n$ , if  $Z_{bn}^{\Delta}$  is higher than zero, this neighbor is defined as a potential child of this base, and routing in this direction is possible.

![z\_delta\_i](img/z\_delta\_array.png)

Here  $S_{bn}$  is the projected distance between the base and the neighbor.

As  $Z_{bn}^{\Delta}$  can be interpreted as process magnitude (and kinetic energy or velocity respectively) it is possible to limit this value to a maximum. In comparison to process based modeling approaches this would correspond to maximum velocity induced by a velocity dependent turbulent friction term.

![z\_delta\_max](img/z\_delta\_max.png)

The projected path lengths, or total travel distance to one of the neighbors ( $S_n$ ) equals the path length to the base ( $S_b$ ) plus the path from base to the neighbor ( $S_{bn}$ ), which reads:

![S\_bn](img/S\_bn.png)

As there are many possibilities for the path from the starting point to the actual cell or base, the shortest path is taken into account, corresponding to the highest  $Z^{\Delta}$  in the base. If  $Z^{\Delta}_{\text{max}}$  is set to infinity, or as in the code to 8848 m (= Mount Everest), we can calculate the shortest path from the starting point to the base and yields the total projected travel distance:

![S\_n\_eq1](img/S\_n\_eq1.png)

This equations determine the routing and corresponding run out distance for the process, the next steps demonstrate how spreading is handled on the surface of the three dimensional terrain.

### Persistence based routing

The persistence contribution  $P_i$  aims to reproduce the behavior of inertia, and takes the change in flow direction into account [3]. The direction contribution is scaled with the process magnitude  $Z^{\Delta}_{\text{parent}}$ , such that the direction from a parent cell with higher process magnitude has more effect on the path routing and direction.



The direction contributions  $D_n$  are defined by the cosine of the angle between parent, base and child/neighbor minus  $\pi$ :



Therefore the direction contribution limits the maximum number of potential children to three, getting input via the persistence function from one parent.

In the first calculation step, at the release or start cell no parent cells are defined and the persistence is set to one. So the first calculation step is solely determined by the terrain contribution.

### Terrain based routing

The terrain based routing is solely dependent on the slope angle  $\phi$ . The exponent  $\exp$  allows to control the divergence of the spreading. The Holmgren (1994) algorithm [1] is used in different kind of models and works well for avalanches but also rockfall or soil slides. For avalanches an exponent of 8 shows good results. To reach a single flow in step terrain (rockfall, soil slides, steepest descend), an exponent of 75 is considered.

![Holmgren](img/flow\_direction.png) *Holmgren Algorithm from 1994 [1]*

To overcome the challenge of routing in flat or uphill terrain, we adapted the slope angle  $\phi$  for the normalized terrain contribution to:

![Phi\_Formula](img/Phi.png)

### ### Routing Flux

The routing flux summarizes the persistence and terrain contributions according to Eq.(16):



where  $i$  is the direction and  $n$  are the neighbors from 1 to 8.  $R_{<sub>i</sub>}$  is then the routing flux in direction  $i$ .  $R_{<sub>b</sub>}$  is the flux in the base, for a release cell or starting cell the flux of the base equals one. The result of Eq. (16) is a  $3 \times 3$  array with assigned flux values. A normalization stage is then required to bring the sum of the  $R_{<sub>i</sub>}$ 's to the value of  $R_{<sub>b</sub>}$ . This aims at avoiding loss of flux [2].

### ### Flow Chart / Overview

In Fig. 3 the algorithm of the computational implementation is sketched, including function and files names with respect to the code in the repository.

The file `main.py` handles the input for the computation and splits the release layer in tiles and saves them in a release list. Then the `main.py` starts one process per tile, which calls the `flow_core.py` and starts the calculation for one release cell and the corresponding path. The number of processes is depending on the hardware setting (CPU and RAM). Whenever a new cell is created `flow_core.py` calls `flow_class.py` and makes a new instance of this class, which is saved in the path. When the calculation in `flow_core.py` is finished it returns the path to `main.py` which saves the result to the output rasters.

![Flow\_Chart](img/Flow-Py\_chart.png)

*Fig.3: Flow chart of the Flow-Py computational process and an overview of the files and what they manage.*

### ### References

[0] [Heim, A. (1932).]: Bergstürze und Menschenleben

[1] [Holmgren, P. (1994).]([https://www.researchgate.net/publication/229484151\\_Multiple\\_flow\\_direction\\_algorithms\\_for\\_runoff\\_modelling\\_in\\_grid\\_based\\_elevation\\_models\\_An\\_empirical\\_evaluation](https://www.researchgate.net/publication/229484151_Multiple_flow_direction_algorithms_for_runoff_modelling_in_grid_based_elevation_models_An_empirical_evaluation)) Multiple flow direction algorithms for runoff modelling in grid based elevation models: an empirical evaluation. Hydrological Processes, 8:327–334.

[2] [Horton, P., Jaboyedoff, M., Rudaz, B., and Zimmermann, M. (2013).](<https://nhess.copernicus.org/articles/13/869/2013/nhess-13-869-2013.pdf>) Flow-R, a model for susceptibility mapping of debris flows and other gravitational hazards at a regional scale. Natural Hazards and Earth System Science, 13:869–885.

[3] [Gamma, P. (1999).]([https://www.researchgate.net/publication/34432465\\_dfwalk-Ein\\_Murgang-Simulationsprogramm\\_zur\\_Gefahrenzonierung](https://www.researchgate.net/publication/34432465_dfwalk-Ein_Murgang-Simulationsprogramm_zur_Gefahrenzonierung)) dfwalk - Ein Murgang-Simulationsprogramm zur Gefahrenzonierung. PhD thesis, Universität Bern.

[4] [Huber, A., Fischer, J. T., Kofler, A., and Kleemayr, K. (2016).] Using spatially distributed statistical models for avalanche runout estimation. In International Snow Science Workshop, Breckenridge, Colorado, USA - 2016.

### ## Contact and acknowledgment

For Questions contact: Michael Neuhauser, Austrian Research Centre for Forest: [Michael.Neuhauser@bfw.gv.at](mailto:Michael.Neuhauser@bfw.gv.at)  
Christopher D'Amboise, Austrian Research Centre for Forest: [Christopher.DAmboise@bfw.gv.at](mailto:Christopher.DAmboise@bfw.gv.at)

This study was carried out in the framework of the GreenRisk4Alps project ASP635, funded by the European Regional Development Fund through the Interreg Alpine Space programme. Additional financial support from the AvaRange ([www.AvaRange.org](http://www.AvaRange.org), international cooperation project “AvaRange - Particle Tracking in Snow Avalanches” supported by the German Research Foundation (DFG) and the Austrian Science Fund (FWF, project number I 4274-N29) and the AvaFrame ([www.AvaFrame.org](http://www.AvaFrame.org), AvaFrame - The open Avalanche Framework is a cooperation between the Austrian Research Centre for Forests (Bundesforschungszentrum für Wald; BFW) and Austrian Avalanche and Torrent Service (Wildbach- und Lawinenverbauung; WLW) in conjunction with the Federal Ministry Republic of Austria: Agriculture, Regions and Tourism (BMLRT)) projects are greatly acknowledged.

**## Citation:**

Michael Neuhauser, Christopher D’Amboise, Michaela Teich, Andreas Kofler, Andreas Huber, Reinhard Fromm, & Jan Thomas Fischer. (2021, June 24). Flow-Py: routing and stopping of gravitational mass flows (Version 1.0). Zenodo. <http://doi.org/10.5281/zenodo.5027275>



## REFERENCES

- *References*
- *Data sources*
- *Glossary*

### 7.1 Data sources

Here is a list of external data we used, with links and sources if available:

- Release , entrainment and resistance geodata for the realistic test cases **Gar, Mal, Hit, Alr, Kot, Wog, Mal** was kindly provided by the Austrian Torrent and Avalanche Service (<https://die-wildbach.at>; via FZGL, Fachbereich Lawinen; Nov 2020).
- Topography data for for the realistic test cases **Gar, Mal, Hit, Alr, Kot, Wog, Mal** was obtained through *Open data Austria*

**Datenquelle: Land Tirol - data.tirol.gv.at**

[https://www.data.gv.at/katalog/dataset/land-tirol\\_tiroelnde](https://www.data.gv.at/katalog/dataset/land-tirol_tiroelnde)

### 7.2 Glossary

---

**Note:** This is still under construction

---

---

**Note:** This glossary has the main purpose to introduce and consolidate an unambiguous language and definitions with respect to avalanche dynamics, avalanche simulations and modelling for the digital toolbox AvaFrame and particularly the Avalanche Modelling Atlas (AMA). Terms are organized in alphabetical order and the terminology builds on the UNESCO Avalanche atlas [DeQuervain+81] and relates to existing guidelines provided and used by Avalanche Associations and Warning Services (Canadian Avalanche Association [CAA16] , American Avalanche Association (AAA) [AmericanAAssociation16] or the European Avalanche Warning Services (EAWS)). The glossary has the goal to follow common conventions used in publications and related projects.

---

#### **alpha point**

#### **alpha angle**

The alpha point is the point of the furthest reach of an avalanche (visible deposition or affected area). The alpha angle is the angle between the line connecting the release point and the alpha point and the horizontally projected

*travel length* along the *thalweg* (see *travel length* and *runout angle*). The alpha angle is also termed the angle of reach or the *runout angle*.

**beta point****beta angle**

The boundary between the *transition* and *deposition* zone (start of deposition) is often called beta point and associated to a certain slope threshold of the *thalweg*. The corresponding *travel angle* is referred to as beta angle. For major avalanche *path* (s) that may produce avalanches of *size* 4-5, the beta point is associated to the 10° Point (or beta\_10), i.e. the point where the slope angle of the *thalweg* decreases below 10°. For avalanche *path* (s) with *event* (s) of avalanche *size* 1-3 the beta point may accordingly be associated to larger *thalweg* slope angles (i.e. up to 30°, see start of *transition*).

**coordinate transformation**

Coordinate transformations refer to the operation of changing coordinates, e.g. between a fixed, Eulerian, global coordinate system with east and north orientation, to an avalanche *path* dependent coordinate system along the *thalweg*, or even a Lagrangian coordinate system, moving along particle trajectories.

**cycle**

An avalanche cycle describes a series of *event* (s) that occur across a region over a relatively short time span.

**danger scale**

The avalanche danger scale refers to the avalanche hazard and is an inherent part of avalanche warning.

**dense flow**

Dense flow is a *form of movement* in the *transition* zone of the avalanche. Dense flow avalanches (DFA) are flowing along the ground and are rather associated to warm flow. Mixed types of movement are often observed, combining different flow regimes and their partial or complete transitions, e.g. 'mixed flow and powder avalanches' or 'flow avalanche with powder component', towards the evolution of a fluidized layer in the avalanche flow.

**density**

Release, *entrainment*, flow, or deposition density. Important quantity relating mass and volume, influencing impact pressure and particular friction relations.

**deposition**

see *zone of deposition*

**depth**

Release, *entrainment*, flow, or deposition depth refers to the extent of the avalanche measured in the direction of gravity.

**entrainment**

Entrainment describes the process of mass intake during the avalanche flow.

**event**

see *scenario*

**flow variables**

Flow variables include *thickness*, *velocity*, or *density* and are determined by the form of movement. Flow models that are implemented usually calculate the spatio-temporal evolution of these variables and where the maximum over the whole flow or computational duration, i.e., their peak values are the most used results. The flow variables are used to derive other variables such as impact pressure or kinetic energy of the flow.

**form of movement**

Is an avalanche criterion in the zone of *transition* and has *powder snow* or *dense flow* as characteristics.

**manner of starting**

Is an avalanche criterion in the zone of *origin* and has the possible characteristics loose, slab, or gliding.

**origin**

see *zone of origin*

**path**

The avalanche path summarizes the total catchment and is divided into different zones (zone of *origin*, *transition*, *deposition*) with different criteria and characteristics. An inherent property of the avalanche path is the *thalweg* and the associated avalanche *event*.

**powder snow**

Powder snow avalanches (PSA) refer to the *form of movement* in the zone of *transition*, referring to the dust or suspension cloud in avalanches. PSA are associated with cold, dry cohesionless snow. Mixed types of movement are often observed, combining different flow regimes and their transitions, e.g., ‘powder avalanche with flow component’.

**projection**

The projection refers to the projection of coordinates within a coordinate system, I.e. the projection of the *runout point* (as furthest reach of the avalanche) to the *thalweg*. Or projecting a 3D travel length (xyz) to a 2D travel length measured only along xy.

**release area**

Potential release areas are located in the zone of *origin*. Each documented *event* or simulation *scenario* is associated to one or more primary and/or secondary release areas, that can further be described by the *manner of starting*.

**return period**

Return periods are related to return levels describing the *size* or magnitude of design or recorded *event* (s) on a respective avalanche *path*. The return level is often determined by the run out length of historically documented avalanche *event* (s) accompanied with return period estimates, which are associated to the occurrence probability.

**runout area****runout angle****runout length****runout point**

Runout lengths and angles are intricately linked to the *alpha point*, utilizing the *projection* to the *thalweg*. In the same manner as *travel length*, run out lengths are measured as horizontally projected lengths along the *thalweg*, from the uppermost point of the *release area* to furthest reach of the runout area. The runout may refer to visible deposition (associated to dense flow), damages or the impacted and affected area (associated to air blast or *powder snow*) in the zone of *deposition* and is usually defined via flow *thickness*, velocity, kinetic energy or impact pressure thresholds.

**scenario**

One or multiple avalanche events or corresponding simulation scenarios are associated to a certain avalanche *path* and have distinct criteria and characteristics such as avalanche *size*, *release area* (s), or *runout area*. These properties are morphologically connected to the different zones (*origin*, *transition*, *deposition*) of an avalanche *path* and allow to define other associated properties, such as *alpha angle* or *runout length* that are defined in combination with the avalanche *thalweg*. Besides observed and documented avalanche events, design events of particular *return period* (s) are of particular interest for engineering applications.

**size**

Avalanche size refers to the magnitude or intensity of an *event*, classified by destructive potential, *runout length* and dimension according to the EAWS size classification, which is closely related to the CAA destructive size.

**terrain classification**

Terrain may be classified according to the Avalanche Terrain Exposure Scale (ATES) into simple (low angle or primarily forested terrain with some openings that may involve the *deposition* zone of infrequent avalanche *path* (s)), challenging (well defined avalanche *path* (s), starting zones, or terrain traps), and complex (exposure to multiple overlapping avalanche *path* (s), large expanses of steep, open terrain, multiple starting zones, and terrain traps below).

**thalweg**

The thalweg is defined by the main flow direction of an avalanche *path* of one or multiple, i.e. not regarding

a specific *event* or *scenario*. avalanche *event* (s). Technically it is the two-dimensional terrain representation, displaying the terrain altitude along the horizontally projected *travel length*.

**thickness**

Release, *entrainment*, flow, or deposition thickness refers to the extent (distance) of the avalanche measured perpendicular to the slope.

**trajectory length**

Used in com1DFA particle dictionaries, where the trajectory length is computed as the distance traveled by a particle from one time step to the next and then accumulated over time. Three different trajectory lengths are computed (1) trajectoryLengthXY - computed in the x, y plane, (2) trajectoryLengthXYZ - also taking the slope of the topography into account and (3) trajectoryLengthXYCor - same as trajectoryLengthXY but corrected for the potential angle difference of the slope and the normal.

**transition**

see *zone of transition*

**travel angle****travel length**

Travel lengths are measured as horizontally projected travel length ( $s_{XY}$ ) along the *thalweg* and are associated with the corresponding travel angle, measured between the line connecting the current location with the uppermost point of the release and the horizontal plane. Alternatively, the surface parallel travel length ( $s_{XYZ}$ ) may be defined as the three-dimensional length travelled by the avalanche.

**velocity**

Flow velocities are usually measured in a surface parallel direction. Alternatively approach velocities are measured along the line of sight.

**wet snow**

A wet snow avalanche (WSA) implies the presence of liquid water within the avalanche and is usually associated to *dense flow* type of movement in the *transition* zone of the avalanche.

**zone of deposition**

The zone of deposition is where the *runout area* of the avalanche is located and where the avalanche stops due to frictional energy dissipation. The boundary with the *transition* zone (start of deposition) is often called *beta point*.

**zone of origin**

The zone of origin delineates the area, in which typical *release area* (s) are located, and an avalanche's appearance is characterized by the *manner of starting*. The uppermost possible point is referred to as start of origin.

**zone of transition**

The zone of transition is the area between zone of *origin* and zone of *deposition* along the *thalweg*. The *form of movement* is linked to the *flow variables*. The start of transition links the zone of *origin* and transition and is usually associated with a slope inclination of about 28-30°.

## BIBLIOGRAPHY

- [AS05] R. Ata and A. Soulaïmani. A stabilized sph method for inviscid shallow water flows. *International Journal for Numerical Methods in Fluids*, 47:139–159, 2005.
- [BDL83] S. Bakkehoi, U. Domaas, and K. Lied. Calculation of snow avalanche runout distance. *Annals of Glaciology*, 4:24–29, 1983.
- [BSG99] P. Bartelt, B. Salm, and U. Gruber. Calculating dense-snow avalanche runout using a voellmy-fluid model with active/passive longitudinal straining. *Journal of Glaciology*, 45(150):242–254, 1999.
- [CAA16] Canadian Avalanche Association CAA. Observation guidelines and recording standards for weather snow-pack and avalanches. Technical Report, Canadian Avalanche Association, Revelstoke, BC, Canada, 2016.
- [FM13] Gloria Faccanoni and Anne Mangeney. Exact solution for granular flows. *International Journal for Numerical and Analytical Methods in Geomechanics*, 37(10):1408–1433, 2013.
- [FFGS13] J. T. Fischer, R. Fromm, P. Gauer, and B. Sovilla. Evaluation of probabilistic snow avalanche simulation ensembles with doppler radar observations. *Cold Regions Science and Technology*, 2013. doi:10.1016/j.coldregions.2013.09.011.
- [Fis13] J.-T. Fischer. A novel approach to evaluate and compare computational snow avalanche simulation. *Natural Hazards and Earth System Science*, 13(6):1655–1667, 2013. URL: <http://www.nat-hazards-earth-syst-sci.net/13/1655/2013/>, doi:10.5194/nhess-13-1655-2013.
- [FK13] J.-T. Fischer and A. Kofler. Samosat cosica. Technical Report, Federal Research and Training Centre for Forests, Natural Hazards and Landscape, Innsbruck, Austria, 2013.
- [HSSN93] Columban Hutter, M. Siegel, Stuart Savage, and Y. Nohguchi. Two-dimensional spreading of a granular avalanche down an inclined plane part i. theory. *Acta Mechanica*, 100:37–68, 01 1993. doi:10.1007/BF01176861.
- [HBB19] David M Hyman, Andrea Bevilacqua, and Marcus I Bursik. Statistical theory of probabilistic hazard maps: a probability distribution for the hazard boundary location. *Natural Hazards and Earth System Sciences*, 19(7):1347–1363, 2019.
- [IOS+14] Markus Ihmsen, Jens Orthmann, Barbara Solenthaler, Andreas Kolb, and Matthias Teschner. Sph fluids in computer graphics. In Sylvain Lefebvre and Michela Spagnuolo, editors, *Eurographics 2014 - State of the Art Reports*. The Eurographics Association, 2014. doi:10.2312/egst.20141034.
- [KMS18] A Köhler, JN McElwaine, and B Sovilla. Geodar data and the flow regimes of snow avalanches. *Journal of geophysical research: earth surface*, 123(6):1272–1294, 2018.
- [LB80] K. Lied and K. Bakkehoi. Empirical calculations of snow-avalanche run-out distance based on topographic parameters. *Journal of Glaciology*, 26(94):165–177, 1980. doi:10.3189/S0022143000010704.

- [LL10] M. Liu and G.R. Liu. Smoothed particle hydrodynamics (sph): an overview and recent developments. *Archives of Computational Methods in Engineering*, 17:25–76, 03 2010. doi:10.1007/s11831-010-9040-7.
- [MCVB+03] Anne Mangeney-Castelnau, Jean-Pierre Vilotte, Marie-Odile Bristeau, Benoit Perthame, François Bouchut, Chiara Simeoni, and Sudhakar Yerneni. Numerical modeling of avalanches based on saint venant equations using a kinetic scheme. *Journal of Geophysical Research: Solid Earth*, 108:2527–2544, 11 2003. doi:10.1029/2002JB002024.
- [Mon92] J.J. Monaghan. Smoothed particle hydrodynamics. *Annual review of astronomy and astrophysics*, 30:543–574, 1992.
- [RK20] Matthias Rauter and Anselm Köhler. Constraints on entrainment and deposition models in avalanche simulations from high-resolution radar data. *Geosciences*, 2020. URL: <https://www.mdpi.com/2076-3263/10/1/9>, doi:10.3390/geosciences10010009.
- [SFF+08] R. Sailer, W. Fellin, R. Fromm, P. Jörg, L. Rammer, P. Sampl, and A. Schaffhauser. Snow avalanche mass-balance calculation and simulation-model verification. *Annals of Glaciology*, 48(1):183–192, 2008.
- [Sal04] B. Salm. A short and personal history of snow avalanche dynamics. *Cold Regions Science and Technology*, 39(2-3):83–92, 2004.
- [Sam07] P. Sampl. Samosat modelltheorie und numerik. Technical Report, AVL List GMBH, 2007.
- [SG09] P. Sampl and M. Granig. Avalanche simulation with samos-at. In *Proceedings of the International Snow Science Workshop, Davos*. 2009.
- [SH89] S. B. Savage and K. Hutter. The motion of a finite mass of granular material down a rough incline. *Journal of Fluid Mechanics*, 199(1):177–215, 1989.
- [Voe55] A. Voellmy. Über die zerstörungskraft von lawinen. *Schweizerische Bauzeitung*, Sonderdruck aus dem 73. Jahrgang(12, 15, 17, 19 und 37):1–25, 1955.
- [Wag16] P.M. Wagner. Kalibrierung des --modells für das ermitteln der auslauflänge von kleinen und mittleren lawinen. Master's thesis, Institut für Alpine Naturgefahren (IAN), BOKU-Universität für Bodenkultur, 2016.
- [WHP04] Y. Wang, K. Hutter, and S.P. Pudasaini. The savage-hutter theory: a system of partial differential equations for avalanche flows of snow, debris, and mud. *ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik*, 84(8):507–527, 2004.
- [Zwi00] T. Zwinger. *Dynamik einer Trockenschneelawine auf beliebig geformten Berghängen*. PhD Thesis, Technischen Universitaet Wien, 2000.
- [ZKS03] T. Zwinger, A. Kluwick, and P. Sampl. Numerical simulation of dry-snow avalanche flow over natural terrain. *Dynamic Response of Granular and Porous Materials under Large and Catastrophic Deformations*, Hutter, K. and Kirchner, N., Springer Verlag, 11:161–194, 2003.
- [AmericanAAssociation16] AAA American Avalanche Association. Snow, weather and avalanches: observation guidelines for avalanche programs in the united states. Technical Report, American Avalanche Association, 2016.
- [DeQuervain+81] R De Quervain and others. *Avalanche atlas*. Unesco, Paris, 1981. URL: <http://unesdoc.unesco.org/images/0004/000480/048004MB.pdf>.

## PYTHON MODULE INDEX

### a

- analTests, 109
- analTests.analysisTools, 110
- analTests.damBreak, 112
- analTests.energyLineTest, 115
- analTests.FPtest, 109
- analTests.rotationTest, 117
- analTests.simiSolTest, 119
- analTests.testUtilities, 127
- ana3AIMEC, 130
- ana3AIMEC.ana3AIMEC, 130
- ana4Stats, 136
- ana4Stats.getStats, 136
- ana4Stats.probAna, 137
- ana5Utils, 144
- ana5Utils.DFAPathGeneration, 144
- ana5Utils.distanceTimeAnalysis, 150

### C

- com1DFA, 36
- com1DFA.deriveParameterSet, 40
- com1DFA.DFAtools, 37
- com1DFA.particleInitialisation, 47
- com1DFA.timeDiscretizations, 49
- com2AB, 50
- com2AB.com2AB, 50
- com3Hybrid, 53

### i

- in1Data, 53
- in1Data.computeFromDistribution, 53
- in1Data.getInput, 55
- in2Trans, 62
- in2Trans.ascUtils, 62
- in2Trans.shpConversion, 64
- in3Utils, 68
- in3Utils.cfgHandling, 69
- in3Utils.cfgUtils, 75
- in3Utils.fileHandlerUtils, 83
- in3Utils.generateTopo, 88
- in3Utils.geoTrans, 91
- in3Utils.getReleaseArea, 104

- in3Utils.initialiseDirs, 106
- in3Utils.initializeProject, 106
- in3Utils.logUtils, 108

### l

- log2Report, 157
- log2Report.generateCompareReport, 157
- log2Report.generateReport, 158

### O

- out1Peak, 160
- out1Peak.outPlotAllPeak, 160
- out3Plot, 161
- out3Plot.amaPlots, 162
- out3Plot.in1DataPlots, 163
- out3Plot.outAIMEC, 165
- out3Plot.outAna1Plots, 171
- out3Plot.outCom1DFA, 177
- out3Plot.outCom3Plots, 181
- out3Plot.outContours, 182
- out3Plot.outDebugPlots, 184
- out3Plot.outDistanceTimeAnalysis, 186
- out3Plot.outParticlesAnalysis, 191
- out3Plot.outQuickPlot, 195
- out3Plot.outTopo, 198
- out3Plot.statsPlots, 199

### t

- tmp1Ex.tmp1Ex, 203





## A

- addConstrainedDataField() (in module *out1Peak.outPlotAllPeak*), 160
- addContour2Plot() (in module *out3Plot.outAna1Plots*), 171
- addDem2Plot() (in module *out3Plot.outCom1DFA*), 178
- addDrop() (in module *in3Utils.generateTopo*), 89
- addErrorTime() (in module *out3Plot.outAna1Plots*), 172
- addInfoToSimName() (in module *in3Utils.cfgHandling*), 71
- addLineBlock() (in module *log2Report.generateReport*), 159
- addLinePlot() (in module *out3Plot.outAIMEC*), 166
- addParticles2Plot() (in module *out3Plot.outCom1DFA*), 178
- addPeakFieldConstrained() (in module *out3Plot.outParticlesAnalysis*), 192
- addRangeTimePlotToAxes() (in module *out3Plot.outDistanceTimeAnalysis*), 187
- addResult2Plot() (in module *out3Plot.outCom1DFA*), 179
- addSLToParticles() (in module *ana3AIMEC.ana3AIMEC*), 131
- addThalwegAltitude() (in module *out3Plot.outAIMEC*), 166
- addTitleBox() (in module *out3Plot.outDistanceTimeAnalysis*), 187
- addTrOrMe() (in module *out3Plot.outParticlesAnalysis*), 192
- addVelocityValues() (in module *out3Plot.outDistanceTimeAnalysis*), 188
- aimecRes2ReportDict() (in module *ana3AIMEC.ana3AIMEC*), 131
- aimecTransform() (in module *ana3AIMEC.ana3AIMEC*), 132
- alpha angle, 317
- Alpha Beta (*com2*), 6
- alpha point, 317
- analTests
  - module, 109
- analTests.analysisTools
  - module, 110
- analTests.damBreak
  - module, 112
- analTests.energyLineTest
  - module, 115
- analTests.FPtest
  - module, 109
- analTests.rotationTest
  - module, 117
- analTests.simiSolTest
  - module, 119
- analTests.testUtilities
  - module, 127
- ana3AIMEC
  - module, 130
- ana3AIMEC.ana3AIMEC
  - module, 130
- ana4Stats
  - module, 136
- ana4Stats.getStats
  - module, 136
- ana4Stats.probAna
  - module, 137
- ana5Utils
  - module, 144
- ana5Utils.DFAPathGeneration
  - module, 144
- ana5Utils.distanceTimeAnalysis
  - module, 150
- analysisPlots() (in module *out3Plot.outDebugPlots*), 184
- analyzeResults() (in module *analTests.damBreak*), 113
- analyzeResults() (in module *analTests.simiSolTest*), 121
- animationPlot() (in module *out3Plot.outDistanceTimeAnalysis*), 188
- appendAverageStd() (in module *ana5Utils.DFAPathGeneration*), 145
- appendCgf2DF() (in module *in3Utils.cfgUtils*), 77
- appendShpThickness() (in module *com1DFA.deriveParameterSet*), 42

appendTcpu2DF() (in module *in3Utils.cfgUtils*), 77  
 applyCfgOverride() (in module *in3Utils.cfgHandling*), 71  
 approachVelocity() (in module *ana5Utils.distanceTimeAnalysis*), 151  
 areaPoly() (in module *in3Utils.geoTrans*), 92

## B

beta angle, 318  
 beta point, 318  
 bowl() (in module *in3Utils.generateTopo*), 89  
 buildRotationTestReport() (in module *ana1Tests.rotationTest*), 118

## C

calcABAngles() (in module *com2AB.com2AB*), 50  
 calcABDistances() (in module *com2AB.com2AB*), 51  
 calcEarlySol() (in module *ana1Tests.simiSolTest*), 121  
 cartToSpherical() (in module *in3Utils.geoTrans*), 93  
 cfgFilesGlobalApproach() (in module *ana4Stats.probAna*), 138  
 cfgFilesLocalApproach() (in module *ana4Stats.probAna*), 138  
 cfgHash() (in module *in3Utils.cfgUtils*), 77  
 checkCommonSims() (in module *in3Utils.fileHandlerUtils*), 83  
 checkForMultiplePartsShpArea() (in module *in1Data.getInput*), 56  
 checkForNumberOfReferenceValues() (in module *ana4Stats.probAna*), 138  
 checkIfFileExists() (in module *in3Utils.fileHandlerUtils*), 84  
 checkMakeDir() (in module *in3Utils.initializeProject*), 107  
 checkOverlap() (in module *in3Utils.geoTrans*), 93  
 checkParameterSettings() (in module *ana4Stats.probAna*), 139  
 checkParticlesInRelease() (in module *in3Utils.geoTrans*), 93  
 checkPathlib() (in module *in3Utils.fileHandlerUtils*), 84  
 checkProfile() (in module *in3Utils.geoTrans*), 94  
 checkRasterMeshSize() (in module *com1DFA.deriveParameterSet*), 42  
 checkResType() (in module *com1DFA.deriveParameterSet*), 42  
 checkThicknessSettings() (in module *com1DFA.deriveParameterSet*), 43  
 cleanModuleFiles() (in module *in3Utils.initializeProject*), 107  
 cleanRemeshedDir() (in module *in3Utils.initializeProject*), 107  
 cleanSingleAvaDir() (in module *in3Utils.initializeProject*), 107

com1DFA  
 module, 36  
 com1DFA.deriveParameterSet  
 module, 40  
 com1DFA.DFAtools  
 module, 37  
 com1DFA.particleInitialisation  
 module, 47  
 com1DFA.timeDiscretizations  
 module, 49  
 com2AB  
 module, 50  
 com2AB.com2AB  
 module, 50  
 com2ABKern() (in module *com2AB.com2AB*), 51  
 com2ABMain() (in module *com2AB.com2AB*), 51  
 com3Hybrid  
 module, 53  
 compareTwoConfigs() (in module *in3Utils.cfgUtils*), 78  
 computeAlongLineDistance() (in module *in3Utils.geoTrans*), 94  
 computeAreasFromLines() (in module *in1Data.getInput*), 56  
 computeAreasFromRasterAndLine() (in module *in1Data.getInput*), 56  
 computeCoordGrid() (in module *in3Utils.generateTopo*), 89  
 computeEarthPressCoeff() (in module *ana1Tests.simiSolTest*), 122  
 computeErrorAndNorm() (in module *ana1Tests.analysisTools*), 111  
 computeFCoeff() (in module *ana1Tests.simiSolTest*), 122  
 computeH() (in module *ana1Tests.simiSolTest*), 123  
 computeLengthOfLine2D() (in module *in3Utils.geoTrans*), 94  
 computeParameters() (in module *in1Data.computeFromDistribution*), 54  
 computePert() (in module *in1Data.computeFromDistribution*), 54  
 computeRelStats() (in module *in1Data.getInput*), 57  
 computeS() (in module *in3Utils.geoTrans*), 95  
 computeSLParticles() (in module *ana3AIMEC.ana3AIMEC*), 132  
 computeU() (in module *ana1Tests.simiSolTest*), 123  
 computeV() (in module *ana1Tests.simiSolTest*), 124  
 computeXC() (in module *ana1Tests.simiSolTest*), 124  
 convertConfigParserToDict() (in module *in3Utils.cfgUtils*), 78  
 convertDF2numerics() (in module *in3Utils.cfgUtils*), 78  
 convertDictToConfigParser() (in module *in3Utils.cfgUtils*), 78  
 convertToCfgList() (in module *in3Utils.cfgUtils*), 79

coordinate transformation, [318](#)  
 copyAimecPlots() (in module *log2Report.generateCompareReport*), [158](#)  
 copyPlots2ReportDir() (in module *log2Report.generateReport*), [159](#)  
 copyQuickPlots() (in module *log2Report.generateCompareReport*), [158](#)  
 correctOrigin() (in module *in3Utils.getReleaseArea*), [105](#)  
 createCfgFiles() (in module *ana4Stats.probAna*), [139](#)  
 createComModConfig() (in module *ana4Stats.probAna*), [139](#)  
 createConfigurationInfo() (in module *in3Utils.cfgUtils*), [79](#)  
 createContourPlot() (in module *out3Plot.outCom1DFA*), [179](#)  
 createDesDictTemplate() (in module *ana1Tests.testUtilities*), [128](#)  
 createFolderStruct() (in module *in3Utils.initializeProject*), [107](#)  
 createParabolaAxis() (in module *in3Utils.generateTopo*), [89](#)  
 createRasterContourDict() (in module *out3Plot.outContours*), [182](#)  
 createReleaseBuffer() (in module *com1DFA.particleInitialisation*), [48](#)  
 createReleaseStats() (in module *in1Data.getInput*), [57](#)  
 createSample() (in module *ana4Stats.probAna*), [140](#)  
 createSampleFromConfig() (in module *ana4Stats.probAna*), [140](#)  
 createSampleWithVariationForThParameters() (in module *ana4Stats.probAna*), [140](#)  
 createSampleWithVariationStandardParameters() (in module *ana4Stats.probAna*), [141](#)  
 createSimDict() (in module *com1DFA.deriveParameterSet*), [43](#)  
 createThalwegTimeInfoFromSimResults() (in module *ana5Utils.distanceTimeAnalysis*), [152](#)  
 crossProd() (in module *com1DFA.DFAtools*), [37](#)  
 cycle, [318](#)

## D

damBreakSol() (in module *ana1Tests.damBreak*), [113](#)  
 danger scale, [318](#)  
 defineEarthPressCoeff() (in module *ana1Tests.simiSolTest*), [124](#)  
 dense flow, [318](#)  
 Dense Flow (*com1*), [6](#)  
 density, [318](#)  
 deposition, [318](#)  
 depth, [318](#)

## E

entrainment, [318](#)  
 errorDuplicateListEntry() (in module *in3Utils.cfgHandling*), [71](#)  
 event, [318](#)  
 exportcom1DFAOrigOutput() (in module *in3Utils.fileHandlerUtils*), [84](#)  
 exportData() (in module *ana5Utils.distanceTimeAnalysis*), [152](#)  
 extendDFAPath() (in module *ana5Utils.DFAPathGeneration*), [145](#)  
 extendProfileBottom() (in module *ana5Utils.DFAPathGeneration*), [146](#)  
 extendProfileTop() (in module *ana5Utils.DFAPathGeneration*), [146](#)  
 extractFeature() (in module *in2Trans.shpConversion*), [66](#)  
 extractFromCDF() (in module *in1Data.computeFromDistribution*), [54](#)  
 extractFrontAndMeanValuesRadar() (in module *ana5Utils.distanceTimeAnalysis*), [153](#)  
 extractFrontAndMeanValuesTT() (in module *ana5Utils.distanceTimeAnalysis*), [153](#)  
 extractLogInfo() (in module *in3Utils.fileHandlerUtils*), [84](#)  
 extractMaxValues() (in module *ana4Stats.getStats*), [136](#)  
 extractNormalDist() (in module *in1Data.computeFromDistribution*), [54](#)  
 extractParameterInfo() (in module *in3Utils.fileHandlerUtils*), [85](#)  
 extractUniform() (in module *in1Data.computeFromDistribution*), [54](#)

## F

fetchAndOrderSimFiles() (in module *in3Utils.cfgHandling*), [72](#)  
 fetchBenchmarkResults() (in module *ana1Tests.testUtilities*), [128](#)  
 fetchContCoors() (in module *out3Plot.outCom1DFA*), [179](#)  
 fetchContourLines() (in module *out3Plot.outAIMEC*), [166](#)  
 fetchFlowFields() (in module *in3Utils.fileHandlerUtils*), [85](#)  
 fetchProbConfigs() (in module *ana4Stats.probAna*), [142](#)  
 fetchRangeTimeInfo() (in module *ana5Utils.distanceTimeAnalysis*), [153](#)  
 fetchReleaseFile() (in module *in1Data.getInput*), [57](#)  
 fetchStartCfg() (in module *ana4Stats.probAna*), [142](#)  
 fetchThicknessInfo() (in module *ana4Stats.probAna*), [143](#)

fetchThThicknessLists() (in module *ana4Stats.probAna*), 142  
 fetchTimeStepFromName() (in module *ana5Utils.distanceTimeAnalysis*), 154  
 fetchValidationString() (in module *in3Utils.cfgHandling*), 72  
 Ffunction() (in module *ana1Tests.simiSolTest*), 120  
 fileNotFoundMessage() (in module *in3Utils.fileHandlerUtils*), 85  
 filterBenchmarks() (in module *ana1Tests.testUtilities*), 128  
 filterCom1DFAThicknessValues() (in module *in3Utils.cfgHandling*), 72  
 filterSims() (in module *in3Utils.cfgHandling*), 73  
 findAngleProfile() (in module *in3Utils.geoTrans*), 95  
 findClosestPoint() (in module *in3Utils.geoTrans*), 95  
 findPointOnDEM() (in module *in3Utils.geoTrans*), 96  
 findSplitPoint() (in module *in3Utils.geoTrans*), 96  
 first\_nonzero() (in module *out3Plot.outAna1Plots*), 172  
 flatplane() (in module *in3Utils.generateTopo*), 89  
 flow variables, 318  
 form of movement, 318  
 fullAimecAnalysis() (in module *ana3AIMEC.ana3AIMEC*), 133

## G

generateAveragePath() (in module *ana5Utils.DFAPathGeneration*), 147  
 generateCom1DFAEnergyPlot() (in module *ana1Tests.energyLineTest*), 115  
 generateCom1DFAPathPlot() (in module *out3Plot.outCom3Plots*), 181  
 generateOnePlot() (in module *out3Plot.outQuickPlot*), 195  
 generatePlot() (in module *out3Plot.outQuickPlot*), 195  
 generateTopo() (in module *in3Utils.generateTopo*), 89  
 getAlphaProfileIntersection() (in module *ana1Tests.energyLineTest*), 116  
 getAndCheckInputFiles() (in module *in1Data.getInput*), 58  
 getAreaMesh() (in module *com1DFA.DFAtools*), 37  
 getCellsAlongLine() (in module *in3Utils.geoTrans*), 96  
 getCornersFP() (in module *in3Utils.getReleaseArea*), 105  
 getCornersHS() (in module *in3Utils.getReleaseArea*), 105  
 getCornersIP() (in module *in3Utils.getReleaseArea*), 105  
 getDamExtend() (in module *ana1Tests.damBreak*), 114  
 getDefaultModuleConfig() (in module *in3Utils.cfgUtils*), 79  
 getDEMFromConfig() (in module *in1Data.getInput*), 58  
 getDEMPath() (in module *in1Data.getInput*), 58  
 getDFAPathFromField() (in module *ana5Utils.DFAPathGeneration*), 147  
 getDFAPathFromPart() (in module *ana5Utils.DFAPathGeneration*), 148  
 getEmpiricalCDF() (in module *in1Data.computeFromDistribution*), 54  
 getEmpiricalCDFNEW() (in module *in1Data.computeFromDistribution*), 55  
 getEnergyInfo() (in module *ana1Tests.energyLineTest*), 116  
 getFilterDict() (in module *in3Utils.fileHandlerUtils*), 85  
 getGeneralConfig() (in module *in3Utils.cfgUtils*), 79  
 getGridDefs() (in module *in3Utils.generateTopo*), 90  
 getIndicesVel() (in module *out3Plot.outAIMEC*), 167  
 getIniPosition() (in module *com1DFA.particleInitialisation*), 48  
 getInputData() (in module *in1Data.getInput*), 59  
 getInputDataCom1DFA() (in module *in1Data.getInput*), 59  
 getInputPaths() (in module *in1Data.getInput*), 60  
 getLabel() (in module *out3Plot.outAna1Plots*), 172  
 getMaxVelocityPoint() (in module *out3Plot.outDistanceTimeAnalysis*), 188  
 getModPathName() (in module *in3Utils.cfgUtils*), 80  
 getModuleConfig() (in module *in3Utils.cfgUtils*), 80  
 getNeighborCells() (in module *in3Utils.geoTrans*), 97  
 getNormalArray() (in module *com1DFA.DFAtools*), 38  
 getNormalMesh() (in module *com1DFA.DFAtools*), 38  
 getNumberOfProcesses() (in module *in3Utils.cfgUtils*), 80  
 getParabolaParams() (in module *in3Utils.generateTopo*), 90  
 getParabolicFit() (in module *ana5Utils.DFAPathGeneration*), 148  
 getParameterVariationInfo() (in module *com1DFA.deriveParameterSet*), 43  
 getPlotLimits() (in module *out3Plot.outAna1Plots*), 172  
 getRadarLocation() (in module *ana5Utils.distanceTimeAnalysis*), 154  
 getReleaseArea() (in module *in3Utils.getReleaseArea*), 105  
 getReleaseThickness() (in module *ana1Tests.FPtest*), 110  
 getReleaseThickness() (in module *ana1Tests.simiSolTest*), 125  
 getRunOutAngle() (in module *ana1Tests.energyLineTest*), 117  
 getSHPPProjection() (in module *in2Trans.shpConversion*), 66  
 getSimiSolParameters() (in module

`ana1Tests.simiSolTest`), 125  
`getSphKernelRadiusTimeStep()` (in module `com1DFA.timeDiscretizations`), 49  
`getSplitPoint()` (in module `ana5Utils.DFAPathGeneration`), 148  
`getTestAvaDirs()` (in module `ana1Tests.testUtilities`), 129  
`getThicknessInputSimFiles()` (in module `in1Data.getInput`), 60  
`getThicknessValue()` (in module `com1DFA.deriveParameterSet`), 44  
`getTitleError()` (in module `out3Plot.outAna1Plots`), 173  
`getVariationDict()` (in module `com1DFA.deriveParameterSet`), 44  
`getVelocityPoints()` (in module `out3Plot.outDistanceTimeAnalysis`), 189  
`GetVersion`, 6

## H

`helix()` (in module `in3Utils.generateTopo`), 90  
`hockey()` (in module `in3Utils.generateTopo`), 90  
`hybridPathPlot()` (in module `out3Plot.outCom3Plots`), 182  
`hybridProfilePlot()` (in module `out3Plot.outCom3Plots`), 182

## I

`importMTIData()` (in module `ana5Utils.distanceTimeAnalysis`), 154  
`in1Data`  
     module, 53  
`in1Data.computeFromDistribution`  
     module, 53  
`in1Data.getInput`  
     module, 55  
`in2Trans`  
     module, 62  
`in2Trans.ascUtils`  
     module, 62  
`in2Trans.shpConversion`  
     module, 64  
`in3Utils`  
     module, 68  
`in3Utils.cfgHandling`  
     module, 69  
`in3Utils.cfgUtils`  
     module, 75  
`in3Utils.fileHandlerUtils`  
     module, 83  
`in3Utils.generateTopo`  
     module, 88  
`in3Utils.geoTrans`  
     module, 91  
`in3Utils.getReleaseArea`  
     module, 104  
`in3Utils.initialiseDirs`  
     module, 106  
`in3Utils.initializeProject`  
     module, 106  
`in3Utils.logUtils`  
     module, 108  
`inclinedplane()` (in module `in3Utils.generateTopo`), 90  
`initialiseRunDirs()` (in module `in3Utils.initialiseDirs`), 106  
`initializeDEM()` (in module `in1Data.getInput`), 60  
`initializeFolderStruct()` (in module `in3Utils.initializeProject`), 108  
`initializeRangeTime()` (in module `ana5Utils.distanceTimeAnalysis`), 155  
`initializeRelTh()` (in module `in1Data.getInput`), 61  
`initializeRotationTestReport()` (in module `ana1Tests.rotationTest`), 118  
`initiateLogger()` (in module `in3Utils.logUtils`), 108  
`insertIntoSimName()` (in module `in3Utils.cfgHandling`), 73  
`isCounterClockWise()` (in module `in3Utils.geoTrans`), 97  
`isEqualASChheader()` (in module `in2Trans.ascUtils`), 63

## L

`L2Norm()` (in module `ana1Tests.analysisTools`), 110  
`last_nonzero()` (in module `out3Plot.outAna1Plots`), 173  
`Layer` rename, 6  
`log2Report`  
     module, 157  
`log2Report.generateCompareReport`  
     module, 157  
`log2Report.generateReport`  
     module, 158

## M

`mainAIMEC()` (in module `ana3AIMEC.ana3AIMEC`), 133  
`mainEnergyLineTest()` (in module `ana1Tests.energyLineTest`), 117  
`mainRotationTest()` (in module `ana1Tests.rotationTest`), 118  
`mainSimilaritySol()` (in module `ana1Tests.simiSolTest`), 126  
`makeADir()` (in module `in3Utils.fileHandlerUtils`), 86  
`makeContourSimiPlot()` (in module `out3Plot.outAna1Plots`), 173  
`makeCoordGridFromHeader()` (in module `in3Utils.geoTrans`), 97



makeCoordinateGrid() (in module *in3Utils.geoTrans*), 98  
 makeDictFromVars() (in module *ana4Stats.probAna*), 143  
 makeLists() (in module *log2Report.generateCompareReport*), 158  
 makeSimDF() (in module *in3Utils.fileHandlerUtils*), 86  
 makeSimFromResDF() (in module *in3Utils.fileHandlerUtils*), 86  
 makexyPoints() (in module *in3Utils.getReleaseArea*), 105  
 manner of starting, 318  
 maskRangeFull() (in module *ana5Utils.distanceTimeAnalysis*), 155  
 module  
   analTests, 109  
   analTests.analysisTools, 110  
   analTests.damBreak, 112  
   analTests.energyLineTest, 115  
   analTests.FPtest, 109  
   analTests.rotationTest, 117  
   analTests.simiSolTest, 119  
   analTests.testUtilities, 127  
   ana3AIMEC, 130  
   ana3AIMEC.ana3AIMEC, 130  
   ana4Stats, 136  
   ana4Stats.getStats, 136  
   ana4Stats.probAna, 137  
   ana5Utils, 144  
   ana5Utils.DFAPathGeneration, 144  
   ana5Utils.distanceTimeAnalysis, 150  
   com1DFA, 36  
   com1DFA.deriveParameterSet, 40  
   com1DFA.DFAtools, 37  
   com1DFA.particleInitialisation, 47  
   com1DFA.timeDiscretizations, 49  
   com2AB, 50  
   com2AB.com2AB, 50  
   com3Hybrid, 53  
   in1Data, 53  
   in1Data.computeFromDistribution, 53  
   in1Data.getInput, 55  
   in2Trans, 62  
   in2Trans.ascUtils, 62  
   in2Trans.shpConversion, 64  
   in3Utils, 68  
   in3Utils.cfgHandling, 69  
   in3Utils.cfgUtils, 75  
   in3Utils.fileHandlerUtils, 83  
   in3Utils.generateTopo, 88  
   in3Utils.geoTrans, 91  
   in3Utils.getReleaseArea, 104  
   in3Utils.initialiseDirs, 106  
   in3Utils.initializeProject, 106

in3Utils.logUtils, 108  
 log2Report, 157  
 log2Report.generateCompareReport, 157  
 log2Report.generateReport, 158  
 out1Peak, 160  
 out1Peak.outPlotAllPeak, 160  
 out3Plot, 161  
 out3Plot.amaPlots, 162  
 out3Plot.in1DataPlots, 163  
 out3Plot.outAIMEC, 165  
 out3Plot.outAna1Plots, 171  
 out3Plot.outCom1DFA, 177  
 out3Plot.outCom3Plots, 181  
 out3Plot.outContours, 182  
 out3Plot.outDebugPlots, 184  
 out3Plot.outDistanceTimeAnalysis, 186  
 out3Plot.outParticlesAnalysis, 191  
 out3Plot.outQuickPlot, 195  
 out3Plot.outTopo, 198  
 out3Plot.statsPlots, 199  
 tmp1Ex.tmp1Ex, 203

## N

norm() (in module *com1DFA.DFAtools*), 39  
 norm2() (in module *com1DFA.DFAtools*), 39  
 normalize() (in module *com1DFA.DFAtools*), 40  
 normL2Scal() (in module *analTests.analysisTools*), 111  
 normL2Vect() (in module *analTests.analysisTools*), 112

## O

odeSolver() (in module *analTests.simiSolTest*), 126  
 Operational Run (*com1* and *com2*), 6  
 orderSimFiles() (in module *in3Utils.cfgHandling*), 74  
 orderSimulations() (in module *in3Utils.cfgHandling*), 74  
 origin, 318  
 out1Peak  
   module, 160  
 out1Peak.outPlotAllPeak  
   module, 160  
 out3Plot  
   module, 161  
 out3Plot.amaPlots  
   module, 162  
 out3Plot.in1DataPlots  
   module, 163  
 out3Plot.outAIMEC  
   module, 165  
 out3Plot.outAna1Plots  
   module, 171  
 out3Plot.outCom1DFA  
   module, 177  
 out3Plot.outCom3Plots  
   module, 181

out3Plot.outContours  
    module, 182  
out3Plot.outDebugPlots  
    module, 184  
out3Plot.outDistanceTimeAnalysis  
    module, 186  
out3Plot.outParticlesAnalysis  
    module, 191  
out3Plot.outQuickPlot  
    module, 195  
out3Plot.outTopo  
    module, 198  
out3Plot.statsPlots  
    module, 199

## P

parabola() (in module in3Utils.generateTopo), 90  
parabolaRotation() (in module in3Utils.generateTopo), 90  
path, 319  
path2domain() (in module in3Utils.geoTrans), 98  
plotAllContours() (in module out3Plot.outQuickPlot), 196  
plotAllFields() (in module out1Peak.outPlotAllPeak), 161  
plotAllPartAcc() (in module out3Plot.outCom1DFA), 180  
plotAllPeakFields() (in module out1Peak.outPlotAllPeak), 161  
plotAreaDebug() (in module out3Plot.outDebugPlots), 185  
plotAreaShpError() (in module out3Plot.in1DataPlots), 164  
plotBondsSnowSlideFinal() (in module out3Plot.outDebugPlots), 185  
plotBoxPlot() (in module out3Plot.amaPlots), 162  
plotBufferRelease() (in module out3Plot.outDebugPlots), 185  
plotComparisonDam() (in module out3Plot.outAna1Plots), 173  
plotContours() (in module out3Plot.outDebugPlots), 185  
plotContours() (in module out3Plot.outQuickPlot), 197  
plotContoursFromDict() (in module out3Plot.outContours), 183  
plotContoursTransformed() (in module out3Plot.outAIMEC), 167  
plotDamAnaResults() (in module out3Plot.outAna1Plots), 174  
plotDamBreakSummary() (in module out3Plot.outAna1Plots), 174  
plotDEM3D() (in module out3Plot.outTopo), 199  
plotDist() (in module out3Plot.in1DataPlots), 164  
plotDistFromDF() (in module out3Plot.statsPlots), 200  
plotECDF() (in module out3Plot.in1DataPlots), 164  
plotEmpCDF() (in module out3Plot.in1DataPlots), 164  
plotEmpPDF() (in module out3Plot.in1DataPlots), 164  
plotErrorConvergence() (in module out3Plot.outAna1Plots), 175  
plotErrorTime() (in module out3Plot.outAna1Plots), 175  
plotFindAngle() (in module out3Plot.outDebugPlots), 185  
plotGeneratedDEM() (in module out3Plot.outTopo), 199  
plotHist() (in module out3Plot.amaPlots), 163  
plotHistCDFDiff() (in module out3Plot.statsPlots), 200  
plotMaskForMTI() (in module out3Plot.outDistanceTimeAnalysis), 189  
plotMaxValuesComp() (in module out3Plot.outAIMEC), 167  
plotPartAfterRemove() (in module out3Plot.outDebugPlots), 185  
plotParticleMotionTracking() (in module out3Plot.outParticlesAnalysis), 192  
plotParticles() (in module out3Plot.outCom1DFA), 180  
plotParticleThalwegAltitudeVelocity() (in module out3Plot.outParticlesAnalysis), 193  
plotPartIni() (in module out3Plot.outDebugPlots), 185  
plotPathAngle() (in module out3Plot.amaPlots), 163  
plotPathExtBot() (in module out3Plot.outDebugPlots), 185  
plotPathExtTop() (in module out3Plot.outDebugPlots), 186  
plotPosition() (in module out3Plot.outDebugPlots), 186  
plotPresentation() (in module out3Plot.outAna1Plots), 176  
plotProbMap() (in module out3Plot.statsPlots), 200  
plotProfile() (in module out3Plot.outDebugPlots), 186  
plotProfilesFPtest() (in module ana1Tests.FPtest), 110  
plotRangeRaster() (in module out3Plot.outDistanceTimeAnalysis), 190  
plotRangeTime() (in module out3Plot.outDistanceTimeAnalysis), 190  
plotReleasePoints() (in module out3Plot.outTopo), 199  
plotReleaseScenarioView() (in module out3Plot.outCom1DFA), 180  
plotRemovePart() (in module out3Plot.outDebugPlots), 186  
plotSample() (in module out3Plot.in1DataPlots), 164

- plotSample() (in module out3Plot.statsPlots), 201  
 plotSamplePDF() (in module out3Plot.in1DataPlots), 164  
 plotSimiSolSummary() (in module out3Plot.outAna1Plots), 176  
 plotSlopeAngle() (in module out3Plot.outDebugPlots), 186  
 plotThalwegAltitude() (in module out3Plot.outAIMEC), 168  
 plotThalwegTimeAltitudes() (in module out3Plot.outParticlesAnalysis), 193  
 plotThSample() (in module out3Plot.statsPlots), 201  
 plotThSampleFromVals() (in module out3Plot.statsPlots), 201  
 plotTimeCPULog() (in module out3Plot.outAna1Plots), 177  
 plotTrackParticle() (in module out3Plot.outCom1DFA), 180  
 plotTrackParticleAcceleration() (in module out3Plot.outCom1DFA), 181  
 plotValuesScatter() (in module out3Plot.statsPlots), 201  
 plotValuesScatterHist() (in module out3Plot.statsPlots), 202  
 plotVelThAlongThalweg() (in module out3Plot.outAIMEC), 168  
 plotVolumeRelease() (in module out3Plot.outDebugPlots), 186  
 pointInPolygon() (in module in3Utils.geoTrans), 99  
 polygon2Raster() (in module in3Utils.geoTrans), 99  
 postProcessAIMEC() (in module ana3AIMEC.ana3AIMEC), 134  
 postProcessDamBreak() (in module ana1Tests.damBreak), 114  
 postProcessFPcom1DFA() (in module ana1Tests.FPtest), 110  
 postProcessSimiSol() (in module ana1Tests.simiSolTest), 127  
 powder snow, 319  
 prepareAngleProfile() (in module in3Utils.geoTrans), 100  
 prepareArea() (in module in3Utils.geoTrans), 100  
 prepareLine() (in module in3Utils.geoTrans), 101  
 prepareParticlesFieldscom1DFA() (in module ana1Tests.simiSolTest), 127  
 Probability analysis for directory (ana4), 6  
 Probability run (ana4, com1), 6  
 probAnalysis() (in module ana4Stats.probAna), 143  
 projection, 319  
 projectOnGrid() (in module in3Utils.geoTrans), 101  
 projectOnRaster() (in module in3Utils.geoTrans), 101  
 pyramid() (in module in3Utils.generateTopo), 91  
 Q  
 quickPlotBench() (in module out3Plot.outQuickPlot), 197  
 quickPlotOne() (in module out3Plot.outQuickPlot), 198  
 quickPlotSimple() (in module out3Plot.outQuickPlot), 198  
 R  
 radarFieldOfViewPlot() (in module out3Plot.outDistanceTimeAnalysis), 190  
 radarMask() (in module ana5Utils.distanceTimeAnalysis), 156  
 rangeTimeVelocityLegend() (in module out3Plot.outDistanceTimeAnalysis), 191  
 readABinputs() (in module com2AB.com2AB), 52  
 readAllBenchmarkDesDicts() (in module ana1Tests.testUtilities), 129  
 readAllConfigurationInfo() (in module in3Utils.cfgUtils), 81  
 readASCdata2numpyArray() (in module in2Trans.ascUtils), 63  
 readAScheader() (in module in2Trans.ascUtils), 63  
 readCfgFile() (in module in3Utils.cfgUtils), 81  
 readCompareConfig() (in module in3Utils.cfgUtils), 81  
 readDEM() (in module in1Data.getInput), 61  
 readDesDictFromJson() (in module ana1Tests.testUtilities), 129  
 readLine() (in module in2Trans.shpConversion), 66  
 readLogFile() (in module in3Utils.fileHandlerUtils), 87  
 readMeasuredParticleData() (in module out3Plot.outParticlesAnalysis), 194  
 readPoints() (in module in2Trans.shpConversion), 67  
 readRaster() (in module in2Trans.ascUtils), 63  
 readShpLines() (in module out3Plot.outContours), 183  
 readThickness() (in module in2Trans.shpConversion), 67  
 release area, 319  
 Release area stats(in1, com1), 6  
 remeshData() (in module in3Utils.geoTrans), 102  
 remeshRaster() (in module in3Utils.geoTrans), 102  
 removeFeature() (in module in2Trans.shpConversion), 67  
 removeSimsNotMatching() (in module in3Utils.cfgHandling), 74  
 resamplePath() (in module ana5Utils.DFAPathGeneration), 149  
 resetMassPerParticle() (in module com1DFA.particleInitialisation), 49  
 resizeData() (in module in3Utils.geoTrans), 103  
 resultHistPlot() (in module out3Plot.statsPlots), 202  
 resultVisu() (in module out3Plot.outAIMEC), 168  
 resultWrite() (in module out3Plot.outAIMEC), 169



return period, [319](#)  
 rewriteLocalCfgs() (in module *in3Utils.cfgHandling*), [75](#)  
 Rock Avalanche (*com6*), [6](#)  
 rotate() (in module *in3Utils.geoTrans*), [103](#)  
 rotateDFAResults() (in module *ana1Tests.rotationTest*), [119](#)  
 rotateRaster() (in module *in3Utils.geoTrans*), [103](#)  
 runout angle, [319](#)  
 runout area, [319](#)  
 runout length, [319](#)  
 runout point, [319](#)

## S

saveSimiSolProfile() (in module *out3Plot.outAna1Plots*), [177](#)  
 saveSplitAndPath() (in module *ana5Utils.DFAPathGeneration*), [149](#)  
 scalProd() (in module *com1DFA.DFAtools*), [40](#)  
 scenario, [319](#)  
 searchRemeshedRaster() (in module *in3Utils.geoTrans*), [104](#)  
 selectReleaseFile() (in module *in1Data.getInput*), [61](#)  
 setDemOrigin() (in module *ana5Utils.distanceTimeAnalysis*), [156](#)  
 setEqParameters() (in module *com2AB.com2AB*), [52](#)  
 setPercentVariation() (in module *com1DFA.deriveParameterSet*), [44](#)  
 setRangeFromCiVariation() (in module *com1DFA.deriveParameterSet*), [45](#)  
 setRangeVariation() (in module *com1DFA.deriveParameterSet*), [45](#)  
 setStrnanToNan() (in module *in3Utils.cfgUtils*), [82](#)  
 setThicknessValueFromVariation() (in module *com1DFA.deriveParameterSet*), [46](#)  
 setupRangeTimeDiagram() (in module *ana5Utils.distanceTimeAnalysis*), [156](#)  
 setupThalwegTimeDiagram() (in module *ana5Utils.distanceTimeAnalysis*), [157](#)  
 setVariationForAllFeatures() (in module *com1DFA.deriveParameterSet*), [46](#)  
 SHP2Array() (in module *in2Trans.shpConversion*), [65](#)  
 size, [319](#)  
 snapPtsToLine() (in module *in3Utils.geoTrans*), [104](#)  
 Snow slide (*com5*), [6](#)  
 splitIniValueToArraySteps() (in module *in3Utils.fileHandlerUtils*), [87](#)  
 splitTimeValueToArrayInterval() (in module *in3Utils.fileHandlerUtils*), [87](#)  
 splitVariationToArraySteps() (in module *com1DFA.deriveParameterSet*), [46](#)

## T

terrain classification, [319](#)  
 thalweg, [319](#)  
 thickness, [320](#)  
 tmp1Ex.tmp1Ex module, [203](#)  
 tmp1ExMain() (in module *tmp1Ex.tmp1Ex*), [203](#)  
 trajectory length, [320](#)  
 transition, [320](#)  
 travel angle, [320](#)  
 travel length, [320](#)

## U

UpdateAvaFrame, [6](#)  
 updateCfgRange() (in module *ana4Stats.probaAna*), [144](#)  
 updateThicknessCfg() (in module *in1Data.getInput*), [62](#)  
 updateTrackPart() (in module *out3Plot.outCom1DFA*), [181](#)

## V

validateVarDict() (in module *com1DFA.deriveParameterSet*), [47](#)  
 velocity, [320](#)  
 velocityEnvelope() (in module *out3Plot.outParticlesAnalysis*), [194](#)  
 velocityEnvelopeThalweg() (in module *out3Plot.outParticlesAnalysis*), [194](#)  
 visuComparison() (in module *out3Plot.outAIMEC*), [169](#)  
 visuMass() (in module *out3Plot.outAIMEC*), [169](#)  
 visuRunoutComp() (in module *out3Plot.outAIMEC*), [169](#)  
 visuRunoutStat() (in module *out3Plot.outAIMEC*), [170](#)  
 visuSimple() (in module *out3Plot.outAIMEC*), [170](#)  
 visuTransfo() (in module *out3Plot.outAIMEC*), [170](#)

## W

weightedAvgAndStd() (in module *ana5Utils.DFAPathGeneration*), [150](#)  
 wet snow, [320](#)  
 writeAllConfigurationInfo() (in module *in3Utils.cfgUtils*), [82](#)  
 writeCfg2Log() (in module *in3Utils.logUtils*), [108](#)  
 writeCfgFile() (in module *in3Utils.cfgUtils*), [82](#)  
 writeCompareReport() (in module *log2Report.generateCompareReport*), [158](#)  
 writeDEM() (in module *in3Utils.generateTopo*), [91](#)  
 writeDesDictToJson() (in module *ana1Tests.testUtilities*), [130](#)  
 writeDictToJson() (in module *in3Utils.cfgUtils*), [83](#)  
 writeLine2SHPfile() (in module *in2Trans.shpConversion*), [68](#)

`writePoint2SHPfile()` (in module *in2Trans.shpConversion*), 68  
`writeReleaseArea()` (in module *in3Utils.getReleaseArea*), 106  
`writeReport()` (in module *log2Report.generateReport*), 159  
`writeReportFile()` (in module *log2Report.generateReport*), 160  
`writeResultToAsc()` (in module *in2Trans.ascUtils*), 64  
`writeToCfgLine()` (in module *com1DFA.deriveParameterSet*), 47

## Z

zone of deposition, 320  
zone of origin, 320  
zone of transition, 320