Geophysical Research Abstracts
Vol. 17, EGU2015-2919, 2015
EGU General Assembly 2015

Impact and sensitivity of parameters in debris flow models: A Monte Carlo simulation on fluid rheology, geometry and position of release areas

Jörg Robl (1) and Stefan Hergarten (2)

(1) Department of Geography and Geology, University of Salzburg, Salzburg, Austria (joerg.robl@sbg.ac, (2) Institute of Earth and Environmental Sciences - Geology, University of Freiburg, Germany (stefan.hergarten@geologie.uni-freiburg.de)

Debris flows are globally abundant threats for settlements and infrastructure in mountainous regions. Crucial influencing factors for hazard zone planning and mitigation strategies are based on numerical models that describe granular flow on general topography by solving a depth-averaged form of the Navier Stokes equations in combination with an appropriate flow resistance law. In case of debris flows, the Voellmy rheology is a widely used constitutive law describing the flow resistance. It combines a velocity independent Coulomb friction term with a term proportional to the square of the velocity as it is commonly used for turbulent flow.

Parameters of the Vollemy fluid are determined by back analysis from observed events so that modelled events mimic their historical counterparts. Determined parameters characterizing individual debris flows show a large variability (related to fluid composition and surface roughness). However, there may be several sets of parameters that lead to a similar depositional pattern but cause large differences in flow velocity and momentum along the flow path. Fluid volumes of hazardous debris flows are estimated by analyzing historic events, precipitation time series, hydrographs or empirical relationships that correlate fluid volumes and drainage areas of torrential catchments. Beside uncertainties in the determination of the fluid volume the position and geometry of the initial masses of forthcoming debris flows are in general not well constrained but heavily influence the flow dynamics and the depositional pattern even in the run-out zones.

In this study we present a new, freely available numerical description of rapid mass movements based on the GERRIS framework and early results of a Monte Carlo simulation exploring effects of the aforementioned parameters on run-out distance, inundated area and momentum. The novel numerical model describes rapid mass movements on complex topography using the shallow water equations in Cartesian coordinates and appropriate correction terms to compensate large topographic gradients. The numerical model was successfully tested against an analytical solution for fluid flow on an inclined plane and by a comparison of results with another state of the art model (RAMMS) on synthetic and real world topographies.

Numerical models describing rapid mass movements that initiate by a so called "block release" (entire fluid starts at the beginning) show the evolution of characteristic pattern in flow depth and velocity: on uniform slopes (inclined plane) the highest velocity and flow depth is observed at the front and the undeformed body of the fluid layer that approaches rapidly to a steady state velocity. The tail becomes increasingly stretched and thinned accompanied by reduced process celerity. The main body of the rapid mass movement is progressively “consumed” by the tail until the front itself decays and the moving mass decelerates over all. However, even for such a simple (synthetic) topography it is non-trivial to predict whether a given release volume initiated from a small or large initial release area (high versus low initial flow depth) will cause a higher momentum in the run-out zone and a larger run-out distance. The fluid layer of rapid mass movements on general topography featuring curved and twisted flow paths is also progressively stretched over time but the influence of the spatial position and the geometry of the release areas in combination with different sets of rheological parameters and topographic features along the flow path is unpredictable without a series of numerical experiments. Early results indicate that the spatial position and geometry of the release volume in combination with various parameter sets within a realistic range of parameters characterizing the Voellmy fluid heavily influence momentum, inundated areas and run-out distances. Even worse, different parameter sets lead to very similar depositional pattern but may differ in momentum along the flow path by more than one order of magnitude and beyond.

We should be are aware that even state of the art models provide only a crude numerical description of the debris flow dynamics and forthcoming hazardous events may significantly deviate from predictions based on numerical models. This may be caused by limitations of the numerical models itself, by not fully appropriate flow resistance laws or by large uncertainties regarding involved masses, their release position and initial geometry and rheological parameters. Therefore, it is essential that beside of all these uncertainties we have a clear understanding of impact and sensitivity of these parameters on numerical model results that are com-

monly used for the delineation of hazard zone and the development of mitigation strategies against natural hazards.