Coupling Depth-Averaged and 3D numerical models to study debris flow: Saint-Vincent event

Coupling Depth-Averaged and 3D numerical models to study debris flow: Saint-Vincent event

Andrea Pasqua[1,2*], *Alessandro* Leonardi[2,1], and *Marina* Pirulli[1]

[1] Politecnico di Torino, Corso Duca degli Abruzzi 24, Torino, 10129, Italy
[2] The University of Sheffield, Western Bank, Sheffield, S10 2TN, United Kingdom

Abstract. Debris flows are extremely rapid and unpredictable phenomena whose rheology is poorly understood. Moreover, human settlements are often located in areas prone to debris flows. The combination of these features makes debris flows hazardous phenomena. Barriers are usually installed in debris flow paths to mitigate risk. However, their design is still based on empirical methods. In order to base the design of barriers on a more reliable approach, the understanding of debris flows must be improved. Continuum numerical models have proved to be a helpful tool for studying debris flows. In particular, numerical models can predict the speed and the flow depth in debris flows paths, and roughly estimate the forces and the pressure acting on a mitigation structure. Currently, two main groups of continuum numerical models are available to study debris flows (i) depth-averaged (DA) models and (ii) three-dimensional (3D) models. Although DA models can study a real-scale event, they may over-simplify the flow-structure interaction. On the other hand, 3D models can be very reliable for studying flow-structure interaction but studying a whole phenomenon (from triggering to deposition) would require enormous computational resources. This work aims to show how the coupling of a DA and a 3D model allows an effective and performing analysis of a debris flow dynamics. The study is focused on the 2014 Saint-Vincent event (Aosta Valley, Italy).

1 Introduction

Debris flows consist of poorly sorted material saturated with water and flowing in channelized paths. They are characterized by high speed and the absence of premonitory signs. Moreover, human settlements are often located in areas prone to debris flows. The combination of unpredictability, high speed, and presence of human settlements make debris flows hazardous phenomena.

In order to mitigate the risk, often barriers are installed along debris flows paths (Fig. 1) [1,2]. However, nowadays design of barriers is based on empirical or simplified methods, mainly because of a poor understanding of debris flows dynamics [3] and their interaction with solid obstacles [4]. In want of a better knowledge of the dynamics of debris flow, and thus, to better design the barriers, it is necessary to gain knowledge of this type of event behaviour. Continuum numerical models have proved to be efficient and reliable to study debris flows. In this frame, two groups of models can be efficiently employed: (i) depth-averaged (DA) models and (ii) three-dimensional (3D) models. DA models are based on depth-averaging the flow speed ($\overline{\boldsymbol{u}}$) along the flow depth ($h$) [5]. This leads to very quick analyses even on very large topographies. However, the depth-averaging procedure leads to the loss of information along the direction perpendicular to the flow. Hence, the flow-structure interaction is studied as a flow of depth-averaged velocity impinging on a

Fig. 1 Example of filter barrier, Saint-Vincent, Aosta Valley (Italy)

mitigation structure. Since flow-structure interaction is an intrinsically three-dimensional phenomenon, this approximation may be too simplistic. Thus, DA models should be employed with care when studying flow-structure interaction. On the other hand, 3D models can study flow-structure interaction very accurately, with full resolution of pressure and velocity variation along the direction perpendicular to the flows. Nevertheless, studying a whole flowing process (made of billions of numerical points to be solved) is extremely time-spending and complex.

Since the flow-structure interaction depends on how flows evolve upstream of the barrier, this work aims to merge DA and 3D models to study a flow-structure

* Corresponding author: andrea.pasqua@polito.it

interaction without renouncing of understanding of how the flow develops upstream the barrier. A few studies have been dedicated to coupling two different numerical models. Su et al. (2022) coupled a discrete element method to a depth-averaged method at the site scale [6]. Domnik et al. (2013) coupled a bi-dimensional model to a depth-averaged method at the laboratory scale [7]. Pasqua et al. (2022) coupled a DA model to a 3D model [8]. Unfortunately, these studies do not apply the coupling at the full scale of the event yet.

This work discusses an application where a DA and a 3D model are coupled to study an event at full scale. The event studied is the one that occurred in Saint-Vincent, Aosta Valley (Italy), in 2014. To achieve that, the approach proposed in [8] is modified. Primarily, the Voellmy rheology is employed. The reason is that Voellmy rheology has been widely proven reliable in simulating debris flows at full scale, and it has two parameters to calibrate, which makes back analysis easy [9,10].

This paper is structured as follows. In Section 2, the basis of DA, 3D models, and coupling are discussed. In section 3, the coupling and its application to the real case study of Saint-Vincent (Aosta Valley) are discussed. Finally, some conclusions are drawn.

2 Numerical models

As mentioned in Section1, continuum numerical models can be DA or 3D. Only the fundamental details of the models are here discussed. Please, refer to the following references for a complete discussion of the difference between DA and 3D models.

DA models approximate debris flows, which consist of a heterogeneous mixture of solid and fluid material, as an equivalent fluid [5,11]. DA models solve the depth-averaged version of the Navier-Strokes equations (i.e., mass and momentum conservation). DA models can predict the flow path, speed, and depth evolution in time. However, the depth-averaging process is a critical point. Studying flow-structure interaction with DA implies that every flow property (i.e., velocity, pressure) is depth-averaged along the flow depth. Hence, the design of a mitigation structure is grounded on averaged values. However, flow-structure interaction is intrinsically three-dimensional, and depth-averaging may lead to underestimating the magnitude of the event.

3D models, on the other hand, can study shear deformation within the flow and the flow structure interaction very precisely. In this study, the employed 3D model is Lattice-Boltzmann method (3D LBM), a relatively new fluid solver compared to the more classical fluid solver (finite differences, finite volumes, finite elements). LBM does not discretise the Navier-Strokes equations but relies on a probability density function $f(\boldsymbol{x}, t, \boldsymbol{c})$ representing the probability of finding fluid particles with speed $\boldsymbol{c}$ at location $\boldsymbol{x}$ and time t. LBM discretises the space using a regular grid in which 19 velocities $\boldsymbol{c}_i$ are allowed (D3Q19). The macroscopic variables density and velocity are obtained by summation of $\boldsymbol{c}_i$. The equation governing the evolution of f_i is a discretised form of the Boltzmann equation and leads to thermodynamic equilibrium [12,13].

2.1 Coupling depth-averaged and 3D models

To couple DA and 3D models rationally, it is essential to define which part of the domain each model solves. The DA model solves the domain area, where no structures are assumed to be present. As soon as the flow approaches a structure, the DA results (depth-averaged velocity and flow depth) are converted into an input for the 3D model: a 3D velocity profile.

Coupling a DA and a 3D model is difficult to achieve. The main issue is obtaining a 3D velocity profile from a depth-averaged profile (the velocity along the perpendicular to the flow direction). The 3D velocity profile is unknown in advance and is a rheology function. The approach proposed by [8], where the authors use the μ(I) rheology, is modified in this work. The μ(I) rheology is difficult to calibrate at the site scale since it requires 5 parameters. Moreover, the μ(I) rheology has been validated for dry granular flows at the laboratory scale. To bypass these issues, in this paper, the employed rheology is the Voellmy. This is because the Voellmy rheology has been widely validated to study debris flow events at a site scale. The 3D steady-state Voellmy velocity profile must be imposed at the coupling section to achieve the coupling. However, to our best knowledge, the 3D Voellmy velocity profile has not been proposed yet. This work discusses obtaining such a profile from the depth-averaged velocity and flow depth.

The Voellmy expression in the DA framework is the following:

$$\tau = \mu P + \rho \boldsymbol{g} \frac{\overline{\boldsymbol{u}}^2}{\xi}, \tag{1}$$

where $\tau, \mu, P, \rho, \boldsymbol{g}, \overline{\boldsymbol{u}}$, and ξ are the shear resistance, the friction coefficient, the pressure, the bulk density, the gravity acceleration, the depth-averaged flow velocity, and the turbulent coefficient that considers the dissipation due to the velocity.

Unfortunately, Eq. (1) cannot be employed as it is in a 3D framework because the ξ parameter has been developed and validated only in DA contexts. To overcome this issue, see the following expression of Voellmy rheology in a 3D framework is proposed:

$$\tau = \mu P + \frac{\rho g l^2 \dot{\gamma}^2}{\xi}, \tag{2}$$

where l, and $\dot{\gamma}$ are a length scale related and the internal shear rate, respectively. Assuming steady-state from Eq. (2), one can obtain both the depth-averaged Voellmy velocity expression:

$$\bar{u} = \frac{2}{5} \frac{h^{\frac{3}{2}}}{d} \sqrt{(\tan\theta - \mu) g \cos\theta} \tag{4}$$

and the 3D Voellmy velocity profile:

$$u(z) = \frac{2}{3} \frac{h^{\frac{3}{2}}}{d} \sqrt{(\tan\theta - \mu) g \cos\theta} \cdot \left[1 - \left(1 - \frac{z}{h}\right)^{3/2}\right]. \tag{5}$$

By substituting Eq. (4) in Eq. (5) and rearranging the terms, the 3D velocity profile can be rewritten as a function of $\bar{u}$:

$$u(z,\bar{u}) = \frac{5}{3}\bar{u}\left[1-\left(1-\frac{z}{h}\right)^{3/2}\right]. \qquad (6)$$

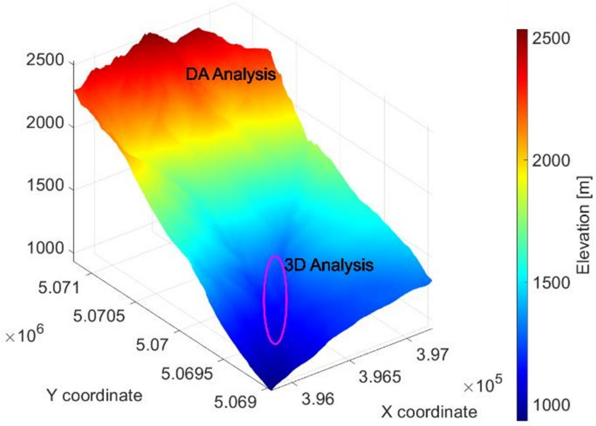

Fig. 2 Drainage area of Saint-Vincent, Aosta Valley (Italy). The DA model resolves the domain outside the purple oval. In contrast, the 3D model solves the volume inside the purple oval. The 3D volume representation, for graphical reasons, is not represented to scale.

The velocity profile in Eq. (6) can be used in a 3D model if the flow is in or close to a stationary condition.

3 Saint-Vincent event and results

In this section, the DA-3D coupling is applied at the event that occurred in Saint-Vincent, Aosta Valley, Italy, in 2014. A complete description of the event is reported in Leonardi et al. (2021) [14]. Since this area is prone to debris flows, local authorities installed a concrete rack dam. In the rack dam, 3 m long steel profiles IPE 270 are imbedded in a 1 m concrete base. The steel profiles are 0.5 m spaced. Fig. 3 sketches the barrier characteristics mentioned.

The drainage basin extends for 5.22 km^2 (Fig. 2). The DA model carries out the analyses where no structure exists (jet-coloured area in Fig. 2). The 3D model run analyses where the rack dam is present (volume within the purple oval in Fig. 2). The 3D model converts the results of the DA model (depth-averaged velocity and flow depth) into a 3D velocity profile (Eq. 6). The 3D model studies a rectangular natural channel 18.0 m long and 9.0 m wide (see Fig. 4). Because the barrier is 3.5 m high, the numerical domain is 4.0 m high to contain the whole barrier.

The DA model here employed is RASH3D [15]. RASH3D uses an unstructured grid whose dimension is 5.0 m, guaranteeing a good approximation of the topography and short computing time. On the other hand, the 3D model uses a squared grid of $8.5 \cdot 10^{-2}$ m, and the time discretisation is $1 \cdot 10^{-4}$ s. These two parameters guarantee the stability of LMB method and a good approximation of the natural channel and the barrier.

As stated above, the Voellmy rheology fits the study of debris flow event. However, to rationally couple the DA and the 3D model, the rheological parameters must be the same in both frameworks. We propose to calibrate first the DA model and then obtain the rheological parameters in the 3D model. Since μ has the same physical meaning in both frameworks, its value does not

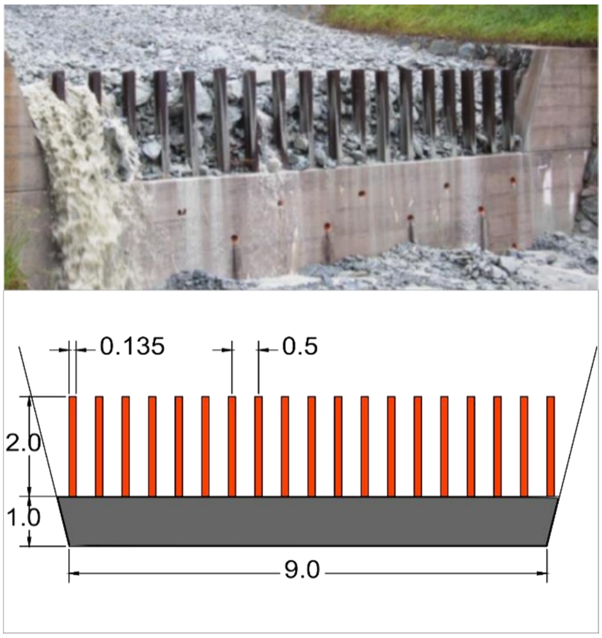

Fig. 3 Rack dam and its numerical schematisation. All the dimensions are in meters

Table 1 Depth-Averaged and 3D Voellmy rheological parameters

Depth- Averaged Voellmy	3D Voellmy
$\mu = \tan 11.3°$	$\mu = \tan 11.3°$
$\xi = 500\ m/s^2$	$l = 0.285\ m$

change. The value of l in the 3D framework is related to the particle diameter and to ξ. In this study the l value was calibrated with a back-analysis process, and its best fit-value is 0.285 m. Table 1 shows the rheological parameters for both frameworks.

In Fig. 4, the reader can see the results downwards: (i) the 3D domain when the flow front reaches the coupling section, (ii) the flow impinging on the rack dam, and (iii) the flow crossing the rack dam. At the initial stage, the flow enters the 3D domain; the DA analysis is over, and the depth-averaged velocity and flow depth are converted through Eq. 6 in a 3D velocity profile. In the second frame, the flow impinges against the barrier. This frame may be the most important. Simulating impact as accurately as possible to better design barriers is crucial. The DA-3D coupling here shows all its potential. In their original paper, Leonardi and Pirulli (2020) [14] studied the flow-structure interaction simulating a dam break. The model employed was based on the discrete element method (DEM). Although the study [14] is solid, some variables must be assumed, i.e., particle numbers and their velocity. These assumptions must be made with care and verifying them with data field is difficult. On the other hand, coupling DA-3D can bypass this issue. Since the studied flow first fills the rack dam and then crosses the barrier. The impact with the barrier is likely similar to what was observed on the field, the run-up mechanism appears realistic, and no unphysical behaviour appears in the simulation. Eventually, the flow crosses the rack dam as expected. Stopping a debris flow event with only one mitigation structure would be hard to achieve due to the magnitude of the velocity and pressure. Common risk mitigation practice prescribes installing more mitigation structures. Each mitigation structure slows down the flow and retains boulders. This multi-structure strategy reduces the dimension of each mitigation structure.

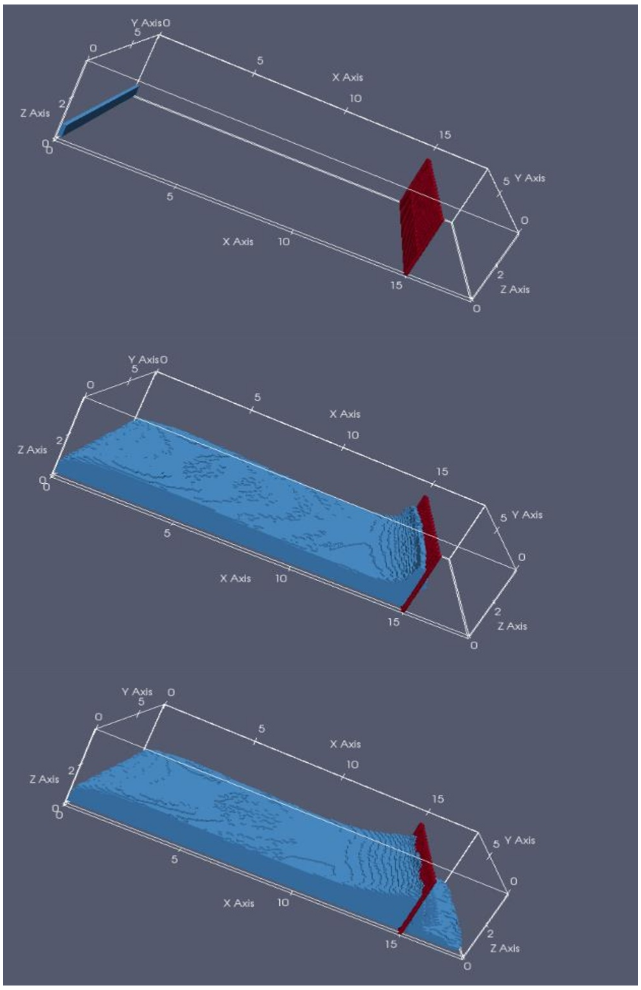

Fig. 4 Evolution of the flow against the rack dam (initial state, impact, and overflow)

4 Conclusions

In this work, a DA-3D coupling to study debris flows at full scale was proposed.

The present work studied the event that occurred in 2014 in Saint Vincent (Italy). The study showed that it is feasible to couple a DA model with a 3D model. Moreover, it was possible to study the event at the full scale. Especially where the absence of mitigation structure is assumed, the domain is solved with a DA model. By contrast, the 3D model converts the depth-averaged velocity and flow depth where a mitigation structure is present into a 3D velocity profile. The flow-structure interaction is thus studied via a 3D model without renouncing to understanding how the flow evolves upstream.

The results reported in the present work are encouraging. As mentioned in Section 3, DA-3D coupling may bypass the difficulties linked to more classical approach methods like dam break simulated with DEM. Nonetheless, further developments should be carried out. Multi-barrier numerical models should be compared to field.

References

1. Leonardi, A., Wittel, F.K., Mendoza, M., Vetter, R., Herrmann, H.J., 2016. Comput.-Aided Civ. Infrastruct. Eng. **31** (5), 323–333 (2016)
2. Leonardi, A., Pirulli, M., 2020. Comput. Geotech. **118**, 103311 (2020)
3. Iverson, R.M. International Conference on Debris-Flow Hazards Mitigation. Vol. **1**. pp. 303–314 (2003)
4. Iverson, R.M., Rev. Geophys. **35** (3), 245–296 (2007)
5. Savage, S.B., Hutter, K., J. Fluid Mech. **199**, 177–215 (1989)
6. Su X., Xia X., Liang Q., Hou J., Comput. Geotech. **141** 104537 (2022)
7. Domnik B., Pudasaini S.P., Katzenbach R., Miller S.A., Journal of Non-Newtonian Fluid Mechanics, **201** 56-58 (2013).
8. Pasqua A., Leonardi A., Pirulli M., Comput. Geotech **149** 104879 (2022)
9. Pirulli, M., Sorbino G., Natural Hazards and Earth System Sciences 8.4 (2008): 961-971.
10. Pirulli, Marina, et al. Geoenvironmental Disasters 4.1 (2017): 1-15.
11. Hungr O Canadian Geotechnical Journal 32 (1995)
12. Mohammed A A New York: Springer ISBN 978-0-85729-454-8 (2011)
13. Succi S Oxford University Press ISBN 9780198503989 (2001)
14. Leonardi A., Pirulli M., (2020), Comp & Geo , **118**, 103311.
15. Pirulli, M., and A. Mangeney. Rock Mechanics and Rock Engineering 41.1 (2008): 59-84.